Resolving PSCAD Issues

Written for PSCAD X4

November 25, 2019
Revision 64
Resolving PSCAD Issues

Contents

1. INTRODUCTION .......................................................................................... 1
  1.1 OVERVIEW ............................................................................................... 1
  1.2 RELATED SUPPORT DOCUMENTS .......................................................... 1
2. ISSUES WHEN LAUNCHING PSCAD .......................................................... 2
  2.1 RECEIVING A "ROOT ELEMENT IS MISSING" ERROR MESSAGE ................. 2
  2.2 THE PSCAD LAUNCH IS PROLONGED .................................................... 4
  2.3 RECEIVING A "STOP RUNNING THIS SCRIPT?" ERROR MESSAGE .......... 5
  2.4 RECEIVING AN "UNABLE TO LOCATE THE MASTER LIBRARY FROM THE GIVEN FILE PATH" ERROR MESSAGE .............................................. 6
  2.5 RECEIVING A "SCRIPT ERROR" ERROR MESSAGE ............................... 8
  2.6 RECEIVING AN "MFC100.DLL IS MISSING" ERROR MESSAGE ............ 10
  2.7 RECEIVING A "THE PROCEDURE ENTRY POINT ... DYNAMIC LINK LIBRARY SHELL32.DLL" ERROR MESSAGE ........................................... 11
  2.8 RECEIVING A "THE PROCEDURE ENTRY POINT ... COULD NOT BE LOCATED IN THE DYNAMIC LINK LIBRARY ZSLIB2.DLL" ERROR MESSAGE 12
  2.9 PSCAD CRASHES UPON STARTUP ......................................................... 13
  2.10 THE PSCAD BETA EDITION CRASHES UPON STARTUP ......................... 14
  2.11 PSCAD V4.6.0 OR V4.6.1 64-BIT DOES NOT LAUNCH .......................... 15
  2.12 RECEIVING A "THE COMPUTER MUST BE TRUSTED FOR DELEGATION..." ERROR MESSAGE ................................................................. 16
  2.13 HOW TO LAUNCH PSCAD WITHOUT WINDOWS ADMINISTRATOR PRIVILEGES ................................................................. 16
  2.14 RECEIVING A "THE APPLICATION WAS UNABLE TO START CORRECTLY (0x00007b)" ERROR MESSAGE ........................................... 18
  2.15 RECEIVING AN "ENTRY POINT NOT FOUND" ERROR MESSAGE .......... 19
  2.16 RECEIVING MISSING CRITICAL FILE MESSAGES WHEN TRYING TO LAUNCH PSCAD ........................................................................... 20
3. ISSUES WHEN LICENSING PSCAD – LEGACY LOCK-BASED ..................... 22
  3.1 LICENSE CANNOT BE ACTIVATED .......................................................... 22
  3.2 ERRORS WITHIN THE PSCADLmgr.TXT FILE .................................... 23
  3.3 ERRORS WITHIN THE Lmgrd-log.TXT FILE ........................................ 25
  3.4 ERRORS WITHIN THE GETINFO UTILITY ........................................... 26
  3.5 RECEIVING A "CLIENT HAS LOCALHOST AND REMOTEHOST IPS" ERROR MESSAGE .......................................................... 29
  3.6 RECEIVING A "FAILURE: UNABLE TO UPDATE THE LICENSE DB FILE" ERROR MESSAGE .......................................................... 30
  3.7 UNABLE TO LICENSE PERIODICALLY .................................................... 31
  3.8 PSCAD CLIENT CANNOT CONTACT THE SERVER .................................... 32
  3.9 LICENSING IS ACTIVATED, BUT PSCAD CLAIMS LICENSING IS EXPIRED .................................................................................. 33
  3.10 RECEIVING A "NO LICENSES FOUND ON THE SELECTED LICENSE HOST" ERROR MESSAGE .............................................................. 34
  3.11 LICENSE CANNOT BE ACTIVATED ON WINDOWS 10 ................................ 36
  3.12 RECEIVING A "(193) APPLICATION IS NOT VALID WIN32 APPLICATION" ERROR MESSAGE .......................................................... 37
  3.13 LICENSING IS ACTIVATED SUCCESSFULLY FOR TWO INSTANCES OF PSCAD, BUT THE FIRST INSTANCE LATER BECOMES DE-ACTIVATED .... 38
  3.14 LICENSING FAILS WHEN USING THE STANDALONE LICENSE MANAGER ........................................................................................ 39
  3.15 UNABLE TO START OR RESTART THE LICENSE MANAGER ................. 40
  3.16 RECEIVING AN "UNABLE TO READ LICENSE DATABASE FILE" ERROR MESSAGE .......................................................... 41
  3.17 LICENSING ISSUES ON WINDOWS 10 Enterprise or Windows 10 Educational Edition ........................................................................... 42
  3.18 UNABLE TO ACTIVATE A LICENSE FROM THE SERVER ....................... 44
  3.19 RECEIVING ERROR "USB ERROR: DRIVER BINARY NOT SIGNED BY A VALID AUTHORITY, STATUS = 104" ............................................. 45
  3.20 RECEIVING A "USING AN INVALID COMMUNICATION PROTOCOL: < GNUTELLA" ERROR MESSAGE .................................................. 46
  3.21 RECEIVING AN "UNABLE TO REPAIR CORRUPTED LOCK [1A]" ERROR MESSAGE ................................................................................ 47
4. ISSUES WHEN LICENSING PSCAD – CERTIFICATE-BASED .................... 48
5. ISSUES WHEN LICENSING PSCAD – LEGACY LOCKLESS ....................... 49
  5.1 RECEIVING AN "UNABLE TO OBTAIN A LICENSE WITH THE CURRENT LICENSE SETTINGS" ERROR MESSAGE ........................................ 49
  5.2 RECEIVING AN "ERROR: PRODUCT MISMATCH" MESSAGE ................ 52
  5.3 RECEIVING A "WINDOWS OS MISMATCH" ERROR MESSAGE ................ 56
6. **ISSUES WHEN USING PSCAD** ........................................................................................................ 57

6.1 PSCAD CRASHES WHEN THE “ASSOCIATIONS” TAB IS SELECTED ................................................. 57
6.2 THE COMPONENT WIZARD PANE IS NOT DISPLAYING CORRECTLY .................................................. 58
6.3 RECEIVING A “THE PSCAD AUTOMATED EMAIL SYSTEM HAS FAILED TO SEND YOUR REQUEST” MESSAGE ............................................................................................................. 60
6.4 RECEIVING AN “UNABLE TO DISPLAY START PAGE” ERROR MESSAGE ............................................ 62
6.5 COPYING A CONTROL WILL SWITCH ITS VALUE TO THE DEFAULT SETTING ..................................... 63
6.6 RECEIVING AN “UNABLE TO CONNECT TO MYCENTRE SERVER” ERROR MESSAGE ........................... 64
6.7 RECEIVING A “REVOCATION INFORMATION FOR THE SECURITY CERTIFICATE FOR THIS SITE IS NOT AVAILABLE” ERROR MESSAGE ......................................................... 65
6.8 RECEIVING A “A PROGRAM IS TRYING TO SEND AN E-MAIL MESSAGE ON YOUR BEHALF” ERROR MESSAGE .................................................................................................................. 66
6.9 A PANES APPEARS TO BE MISSING ...................................................................................................... 67
6.10 GRAPHS DO NOT DISPLAY .................................................................................................................. 69
6.11 ISSUES WITH DEBUGGING PSCAD .................................................................................................... 70
6.12 RECEIVING A “SERVER BUSY” ERROR MESSAGE ............................................................................. 71
6.13 THE “OPEN EXAMPLES” MENU OPTION DOES NOT WORK ............................................................. 72
6.14 CANNOT LOAD A PSCAD PROJECT .................................................................................................... 73
6.15 MATLAB IS NOT DETECTED BY PSCAD ............................................................................................ 74
6.16 RECEIVING A “SECURITY ALERT” MESSAGE .................................................................................... 76
6.17 MENU TOOLS ARE NOT ACTIVE ....................................................................................................... 78
6.18 RECEIVING A “POWER SYSTEMS SIMULATOR HAS STOPPED WORKING” ERROR MESSAGE ............ 79
6.19 RECEIVING A “POWER SYSTEMS SIMULATOR HAS STOPPED WORKING” ERROR MESSAGE ........ 80
6.20 64-BIT EDITS OF INTEL FORTRAN ARE NOT DETECTED BY PSCAD ............................................. 81
6.21 USER DOES NOT HAVE PERMISSIONS FOR LocalAppData folder ..................................................... 82

7. **ISSUES WHEN COMPILING CASES IN PSCAD** .............................................................................. 83

7.1 RECEIVING A “1 PROJECT(s) WAS UNSUCCESSFUL WHEN BUILDING” ERROR MESSAGE ...................... 83
7.2 RECEIVING AN “UNRESOLVED EXTERNAL SYMBOL” ERROR Message ............................................... 85
7.3 RECEIVING A “FORTRAN COMPILER...IS NOT DETECTED” ERROR Message ..................................... 86
7.4 RECEIVING A "CANNOT EXECUTE COMMAND" ERROR ......................................................................... 88
7.5 RECEIVING A “gfortran.exe: Main.f: Invalid Argument” ERROR......................................................... 90
7.6 RECEIVING AN “ERROR 1 – MULTIPLE DEFINITION OF ‘...’ - FIRST DEFINED HERE” ERROR ......... 92
7.7 RECEIVING AN “ERROR 1 – SYNTAX ERROR IN ARGUMENT LIST AT [...]” ERROR ......................... 93
7.8 RECEIVING A “MAKE FAILED TO GENERATE A SIMULATION EXECUTABLE FOR NAMESPACE "****", BINARY FILE WAS NOT FOUND” ERROR ................................. 94
7.9 RECEIVING A “MAKE: *** [SomeFile.exe] ERROR 1” ERROR ............................................................ 97
7.10 RECEIVING AN "Error U1052 – "file' data. mak' not found " ERROR .................................................. 99
7.11 RECEIVING A "MAKE FILE ERROR" ERROR ....................................................................................... 101
7.12 RECEIVING AN “UNABLE TO SOLVE LINE CONSTANTS” BUILD MESSAGE ..................................... 102
7.13 RECEIVING A “cl.exe” IS NOT RECOGNIZED AS AN INTERNAL OR EXTERNAL COMMAND” BUILD MESSAGE ............................................................................................................... 103
7.14 RECEIVING A “THE NUMBER OF PROJECTS INCLUDED IN THE SIMULATION LIST MUST NOT EXCEED 95” ERROR Message .......................................................... 104
7.15 RECEIVING AN “UNABLE TO OPEN FILE...” ERROR Message .......................................................... 105
7.16 RECEIVING A “COULD NOT READ SYMBOLS; MALFORMED ARCHIVE” ............................................. 106
7.17 PSCAD STOPS UNEXPECTEDLY, WITH NO RUNTIME ERROR ................................................................ 107
7.18 RECEIVING “ERROR code = 112” Message ......................................................................................... 109
7.19 RECEIVING A “TRANSMITTER **" CANNOT BE ADDED" BUILD MESSAGE ........................................... 110
7.20 RECEIVING A “EMTDC: ERROR WHEN OPENING FILE FOR READ” BUILD MESSAGE ..................... 111
7.21 RECEIVING A “MAKE: *** [Station.0] ERROR 126” ERROR Message .................................................. 112
7.22 RECEIVING A “WSNINOCERROR #10048” ERROR Message .............................................................. 113
7.23 RECEIVING A “CONNECT FUNCTION FAILED WITH ERROR: 10061” ERROR Message ..................... 115
7.24 RECEIVING AN “UNABLE TO EXCEUTE A SIMULATION EXECUTABLE FOR NAMESPACE "**", ERROR Message .............................................................................................................. 116
7.25 RECEIVING A “SIMULATION SET "**" HAS XX PROJECT TASKS INCLUDED , EXCEEDING THE WORKSPACE SELECTED LIMIT OF * TASKS” ERROR Message ....................................................... 118
7.26 RUNNING A PSCAD V4.6 SIMULATION ON WINDOWS 10 IS SLOWER THAN RUNNING EARLIER VERSIONS OF PSCAD ............................................................................................................. 119
7.27 RECEIVING AN “‘NMAKE’ IS NOT RECOGNIZED AS AN INTERNAL OR EXTERNAL COMMAND...” ERROR Message ........................................................................................................ 123
8. Issues when running cases in PSCAD

8.1 Receiving a "Project output storage requirements are xx MB" Warning .................................................. 180
8.2 Receiving an "EMTDC Runtime Error" Message .................................................................................. 181
8.3 Receiving an "Abnormal termination of EMTDC by *" error message ............................................... 183
Resolving PSCAD Issues

8.4 Receiving a “Server Busy” Error Message ................................................................. 184
8.5 Receiving a “Result Too Large” Runtime Error Message ........................................... 185
8.6 Receiving an “Error: 10013” Runtime Error Message ............................................. 186
8.7 Receiving a “Singularity (a zero diagonal) encountered” Runtime Error Message .... 187
8.8 Unable to Run any PSCAD Cases - The Regional Language Issue ............................... 188
8.9 Receiving a “Process...Cannot execute command...Error #5” Error Message ................ 193

9. Issues with MyCentre .................................................................................................... 194

10. Issues with Diagnostic Tools ........................................................................................ 195

10.1 Receiving a “GetInfo32.exe – System Error” Message (Get Info Tool) ....................... 195
10.2 Receiving an “Unable to proceed” error message when using the License Update Utility .......................................................... 197
10.3 Receiving an “Unable to update the lock. Unable to initialize interface due to: Comm error,” Error Message ................ 198
10.4 Receiving an “Unable to update the lock” Error Message ........................................ 199

11. Resolving Fortran Coding Issues .................................................................................... 200

11.1 Receiving an “Unable to generate a simulation executable” Error Message ................ 200

Appendix A Using the Fortran Medic Utility ...................................................................... 201

A.1 Overview .................................................................................................................... 201
A.2 Running the Fortran Medic Utility .............................................................................. 201
A.3 Generating the Log File ............................................................................................ 201
A.4 Fixing Issues Using the Utility .................................................................................. 201
A.5 Errors Listed in the Utility ....................................................................................... 202
A.6 Functions Listed in the Medic .................................................................................. 218

Appendix B Using the Get Info Utility ................................................................................ 220

B.1 Overview .................................................................................................................... 220
B.2 Running the Get Info Utility ..................................................................................... 220
B.3 Using the Utility to Detect Licensing ....................................................................... 220

Appendix C Lock-Based Legacy Licensing - License Manager Requirements ..................... 221

Appendix D Certificate Licensing – Requirements ............................................................. 222

Appendix E How to Display a Hidden Folder ..................................................................... 223

Appendix F File Creation During a Simulation .................................................................... 224

Appendix G Testing Connectivity for Certificate Licensing .............................................. 227
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, which includes version 4.3.0 and later, and covers the following topics:

- Section 2: Launching PSCAD (Section 2)
- Section 3: Licensing PSCAD – Legacy Lock-based Licensing
- Section 4: Content (certificate licensing issues) moved to new manual “Resolving Certificate Licensing Issues” (see Section 1.2)
- Section 5: Licensing PSCAD – Legacy Lockless Licensing
- 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: Issues with Diagnostic tools
- Section 11: Resolving FORTRAN coding issues
- Appendix A: Using the Fortran Medic utility
- Appendix B: Using the Get Info utility
- Appendix C: Requirements for the License Manager (legacy lock-based licensing)
- 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:

```
Stop running this script?
A script on this page is causing your web browser to run slowly.
If it continues to run, your computer might become unresponsive.
```

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:

![Error Message]

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.

![Application Options]

Check the specified folder path.
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:

![Script Error Message]

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:

  ![Application Options Dialog Box]

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

  ![Workspace Options Dialog Box]

  - Select “Ok” to apply the change:

  ![Start Page Options Dialog Box]

  - 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:

![Error Message]

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.dll” Error Message

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

![Error Message]

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:

![Image of error message]

The procedure entry point ??0ZGraphPanelAnalog@@QAE@XZ could not be located in the dynamic link library ZSlib2.dll.

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:

```
Runtime Error!
Program: C...
R6030
- CRT not initialized
```

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.

“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:

![pscad.exe – Application Error](image)

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:

![pscad.exe – Entry Point Not Found](image)

The procedure entry point
?componentResized@ZCanvas@@UEAAXZ
could not be located in the dynamic link library
ZSLib2.dll.

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.
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.
3. Issues when Licensing PSCAD – Legacy Lock-Based

3.1 License Cannot be Activated

Problem

When the “Workspace Settings” dialog box is configured, and the “Activate” button is selected...

...licensing is not activated, and the following error message displays:

Cause

There are many possible causes for a licensing issue.

Solution

Proceed as follows to resolve the licensing issue:

- If the license is installed on the machine running PSCAD, view the causes and solutions in Sections 3.2 and 3.4.
- If the license is installed on a Server, view the causes and solutions in Sections 3.2, 3.3 and 3.4.
3.2 Errors within the PscadLmgr.txt File

When there is a licensing issue, the log file “PscadLmgr.txt” may be used to help identify the cause.

The PscadLmgr.txt file is located on the machine that is running PSCAD, as follows:

- **On Windows XP:**
  - C:\Documents and Settings\All Users\Application Data\Manitoba HVDC Research Centre\License Manager\PscadLmgr.txt

- **On Windows Vista and Windows 7:**
  - C:\Users\All Users\Manitoba HVDC Research Centre\License Manager\PscadLmgr.txt
  - C:\Users\YOUR-USER-ID\AppData\Local\Manitoba HVDC Research Centre\License Manager\PscadLmgr.txt
    (if the appdata folder is hidden, it may be displayed as per Appendix L)
  - C:\Program Data\Manitoba HVDC Research Centre\License Manager\PscadLmgr.txt

The following table lists some of the common error messages listed in the log file, as well as their possible causes and solutions.

<table>
<thead>
<tr>
<th>Error</th>
<th>Cause</th>
<th>Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. USB Error: Sentinel driver not installed, status = 524</td>
<td>USB lock is not plugged in.</td>
<td>Plug in USB lock</td>
</tr>
<tr>
<td>2. Sentinel Drivers not installed.</td>
<td></td>
<td>Install Sentinel Drivers per the manual posted here: <a href="https://hvdc.ca/knowledge-base/read_article/360/sentinel-drivers-installation/v">https://hvdc.ca/knowledge-base/read_article/360/sentinel-drivers-installation/v</a>:</td>
</tr>
<tr>
<td>3. Checking for lock on USB USB Error: all licenses are in use, status = 0</td>
<td>User is remotely logging in to machine running PSCAD.</td>
<td>Remotely logging in to the machine running PSCAD is not enabled in this version; you must be directly logged into the machine. This includes running PSCAD on a virtual machine, running PSCAD in Windows XP compatibility mode on Windows 7, and logging in via virtual private network. Remotely logging in is enabled in higher versions (e.g. v4.5).</td>
</tr>
<tr>
<td>4. ERROR: Detected a second instance of &lt; pscad.exe &gt;</td>
<td>On a machine with a valid trial license, this error occurs if a second instance of PSCAD is run. A trial license permits only one instance.</td>
<td>Ensure only one instance of PSCAD is running</td>
</tr>
</tbody>
</table>

**NOTES**

1. This limitation is applicable when trying to license a second instance with either Trial or USB-based licensing
2. This limitation is not applicable when trying to license a second instance with Certificate Licensing; a second instance of PSCAD using Certificate Licensing may be run
<table>
<thead>
<tr>
<th>Error</th>
<th>Cause</th>
<th>Solution</th>
</tr>
</thead>
</table>
| 5. ERROR: Invalid semaphore value  
ERROR: Refused to renew license [No.] for [machine name] for <PSCAD X4 Professional>.  
License not active. | An unexpected second instance of PSCAD was being licensed.  
This would occur for licensing that is only programmed to allow a single instance of PSCAD, either because the license is a legacy lock-based license with one PSCAD seat, or it is a legacy trial license. | Ensure only one instance of PSCAD is running  
NOTES  
1. This limitation is applicable when trying to license a second instance with either a lockless trial license or with lock-based licensing with only one PSCAD seat.  
2. This limitation is not applicable when trying to license a second instance with Certificate Licensing; a second instance of PSCAD using Certificate Licensing may be run. |
| 6. To be determined | PSCAD v4.5.0 (64-bit) was launched, and an Educational License is being used.  
The 64-bit version may only be licensed using a Professional License. | There are three options:  
1. Use the Educational License with PSCAD v4.5.0 (32-bit).  
2. Use the Educational License with the 64-bit version of PSCAD v4.5.2 or later; the 64-bit version of PSCAD is enabled for later versions.  
3. If available, switch to a Professional License and use PSCAD v4.5.0.
3.3 Errors within the Lmgrd-log.txt File

When there is a licensing issue, the log file “Lmgrd-log.txt” may be used to help identify the cause.

The Lmgrd-log.txt file is located on the machine hosting the License Manager, as follows:

- **On Windows XP:**
  - C:\Documents and Settings\All Users\Application Data\Manitoba HVDC Research Centre\License Manager\Lmgrd-log.txt

- **On Windows Vista and Windows 7 and better, the file is located in one or more of the following locations:**
  - C:\Users\All Users\Manitoba HVDC Research Centre\License Manager\Lmgrd-log.txt
  - C:\Users\YOUR-USER-ID\AppData\Local\Manitoba HVDC Research Centre\License Manager\Lmgrd-log.txt
    (if the appdata folder is hidden, it may be displayed as per Appendix L)
  - C:\ProgramData\Manitoba HVDC Research Centre\LicenseManager\Lmgrd-log.txt

The following table lists some of the common error messages listed in the log file, as well as their possible causes and solutions.

<table>
<thead>
<tr>
<th>Error</th>
<th>Cause</th>
<th>Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>USB Error: unable to find a Sentinel lock</td>
<td>USB lock is not plugged in (required in order to authorize licensing)</td>
<td>Plug in USB lock</td>
</tr>
<tr>
<td>ERROR: Unable to open and read Licence Manager database C:\Windows\lmgr-hvdc if the database file is missing, run the Enter License Key utility to create it</td>
<td>Sentinel Drivers not installed.</td>
<td>Install Sentinel Drivers per manual posted here: <a href="https://hvdc.ca/knowledge-base/read/article/360/sentinel-drivers-installation/v">https://hvdc.ca/knowledge-base/read/article/360/sentinel-drivers-installation/v</a>;</td>
</tr>
<tr>
<td>ERROR: Client [machine name] [IP address] is not on the same host or net as the LM [IP address]</td>
<td>The client machine must be on the same network as the server. For example, the following are on the same network: • Server: 172.17.<em>.</em> • Client: 172.17.<em>.</em> The following are not on the same network: • Server: 172.17.<em>.</em> • Client: 10.100.<em>.</em></td>
<td>Options: 1. Move the client to the same network as the server. 2. Request “expanded licensing” from our support desk (<a href="mailto:support@pscad.com">support@pscad.com</a>). 3. Consider switching licensing from “legacy” to “Certificate” (consult <a href="mailto:support@pscad.com">support@pscad.com</a>).</td>
</tr>
</tbody>
</table>
### 3.4 Errors within the Get Info Utility

When there is a licensing issue, the “Get Info” utility may be used to help identify the cause. The Get Info Utility may be run as per Appendix B, Section B.2.

The following table lists some of the common error messages displayed in the Get Info utility, as well as their possible causes and solutions.

<table>
<thead>
<tr>
<th>Message</th>
<th>Possible Impact</th>
<th>Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>User level: User</td>
<td>User does not have administrator rights: Therefore, the user may not install or update a license. Note Administrator rights are not required to run PSCAD</td>
<td>1. Obtain temporary administrator rights, then install the license 2. Administrator to install the license</td>
</tr>
<tr>
<td>License Info &lt;localhost&gt; Licence DB: &lt;no licenses installed&gt;</td>
<td>No license database file is installed on this machine, which is hosting the license: Therefore the licensing will not work</td>
<td>Install the license database file as per the manual posted here: <a href="https://hvdc.ca/knowledge-base/read_article/312/installing-a-license-database-file/">https://hvdc.ca/knowledge-base/read_article/312/installing-a-license-database-file/</a></td>
</tr>
</tbody>
</table>
| Local License Manager: 1.XX [installed] | The License Manager is installed on a machine that is running a Single-User License: Therefore, the licensing might not work | Uninstall the License Manager:  
  a. Stop the License Manager Service (from the Windows Start menu, browse to “Stop License Manager Service”).  
  b. Go to “Add or remove program” in the Windows Control Panel, and remove “Manitoba HVDC Research Centre Licence Manager”. |
<table>
<thead>
<tr>
<th>Message</th>
<th>Possible Impact</th>
<th>Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>ERROR: Unable to read the lock [5] ... Sentinel Driver Info USB Error: unable to find a sentinel lock USB driver version 7.150.10 &lt;driver NOT detected, or SUB lock not yet inserted&gt; &lt;driver NOT detected&gt; ... Lock Info (direct access) customer ID XXXX (expected) customer ID unknown cause(s) &lt;Dongle absent, not communicating, or wrong port in Lmgrd.ini&gt;</td>
<td>USB lock is not plugged in on the machine.</td>
<td>Plug in the USB lock.</td>
</tr>
<tr>
<td>Installed Sentinel Driver Software: &lt;non detected&gt; ... USB error: USB Error: Sentinel driver not istalled, status = 524 USB info: USB Error: Sentinel driver not istalled, status = 524 USB port: &lt;driver not detected, or USB lock not yet inserted&gt; Par port: &lt;driver not detected&gt; Par port: &lt;unknown hardware address or disabled &gt; Par port: &lt;unknown software address or disabled &gt; ... ERROR: the License Manager appears to be unresponsive.</td>
<td>USB lock will not be detected</td>
<td>Install the Sentinel Driver</td>
</tr>
<tr>
<td>Message</td>
<td>Possible Impact</td>
<td>Solution</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>---------------------------------------------------------------------------------</td>
<td>---------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Lock Info (from active LM on localhost)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- customer ID: xxxxx (expected)</td>
<td>The USB lock does not appear to have ever been programmed, and licensing will not work.</td>
<td>Send your Get Info log file to <a href="mailto:support@pscad.com">support@pscad.com</a> (location of log file will be listed within the utility). We will review your file and forward the solution to you.</td>
</tr>
<tr>
<td>- customer ID: xxxxx (Licence DB)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- ERROR: the Licence Manager appears to be unresponsive.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[The above License Manager error only appears if it is running]</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Lock Info (direct access)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Expected on: &lt;no port specified, or Lmgrd.ini does not exist =&gt; assume USB&gt;</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- Expected on: USB &lt;default&gt;</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- com port: USB</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- device: USB</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- expected port: xxx (expected)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- customer ID: xxx (expected)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- customer ID: 0 &lt;corrupted&gt;</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- lock ID: xxxx</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- user date: &lt;error in date value&gt;</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Licence Host</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- LM registry: &lt;<em>&gt;.</em>.<em>.</em> 2053</td>
<td>Ping Echo and Ping Reply are not enabled on the server; they must be enabled to allow licensing.</td>
<td>Enable Ping Echo and Ping Reply on the server (see Appendix C).</td>
</tr>
<tr>
<td>- LM hostname: <em>.</em>.<em>.</em></td>
<td></td>
<td></td>
</tr>
<tr>
<td>- ping LM host: icmpSendEcho failed: IP request timed out</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- LM port: 2053</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
3.5 Receiving a "Client has localhost and remotehost IPs" Error Message

Problem
The license cannot be activated, and the following error message displays in the log file:

ERROR: Client [machine name] <[some IP address]>, <127.0.0.1> is not on same net as LM <[some IP address]>. Client has localhost and remotehost IPs

Cause
This is a bug in earlier versions of License Manager (v1.28, v1.29).

Solution
Update your License Manager to v1.30 or later.
3.6 Receiving a "FAILURE: unable to update the licence DB file" Error Message

Problem

When using the License Update Utility to license PSCAD, the following message displays in the utility:

Unable to save license database file
File: C:\Users\Public\Documents\Manitoba HVDC Research Centre\LicenseManager\lmgr-hvdc
Error: 'C:\Users\Public\Documents\Manitoba HVDC Research Centre\LicenseManager\lmgr-hvdc' Cannot find part of the path.
FAILURE: unable to update the licence DB file

This issue occurs when the following requirements are both met:

- The License Update Utility used is v1.36.0.0 (comes bundled with PSCAD v4.6.0)
- A PSCAD license has not been installed previously on this machine.

Cause

The License Update Utility released with v4.6.0 fails on a new license installation because the expected folder for the license database file does not exist, and the utility fails to create it.

Solution (1)

- Create the following folder:
  
  C:\Users\Public\Documents\Manitoba HVDC Research Centre\License Manager

- Retry the license installation using the License Update Utility.

Solution (2)

Use a later version of the License Update Utility to license PSCAD (v1.36.2.0 or later):

- Shut down all instances of PSCAD.
- Ensure that the license.txt file is saved to your machine.
- Download the latest License Update Utility from the following link:
  
  http://updater.pscad.com/utilities/LicenseUpdate.zip

- Extract the files to a local folder, and launch the “LicenseUpdate.exe” file.
- From the “Actions” menu, select “Enter license keys”.
- When prompted, browse to and select the “license.txt” file, and select “Open”.
- The license will be installed. Close the utility.
### 3.7 Unable to License Periodically

**Problem**

Sometimes a PSCAD license cannot be activated (lock-based). On those occasions, the lmgrd-log.txt file indicates that a USB lock is not detected:

USB Error: unable to find a Sentinel lock

**Cause**

This issue can occur when running PSCAD on a laptop, and sometimes using a docking station:

- If your laptop is in a docking station, your USB ports are remapped to the docking station USB ports. If you install the Sentinel drivers while your laptop is docked, the USB key will only be detected when your laptop is docked. If you install the Sentinel drivers while your laptop is not docked, the USB key will only be detected when your laptop is not docked.

**Note**

This is a third party Sentinel driver issue, not a PSCAD or License Manager issue.

**Solution**

The work-around is to always run PSCAD with your laptop in the same manner (docked or undocked). If you need to re-install the Sentinel Drivers, software and instructions may be found in the manual posted here:

https://hvdc.ca/knowledge-base/read_article/360/sentinel-drivers-installation/v:
3.8 PSCAD Client Cannot Contact the Server

Problem

PSCAD cannot ping the Server, nor activate a license from the Server.

Cause 1

This issue can occur if the QuickHeal Antivirus is installed, and blocking the licensing.

Solution 1

Uninstall QuickHeal Antivirus.
3.9 Licensing is Activated, but PSCAD claims Licensing is Expired

Problem
Immediately following a license activation, the license is returned, and the following errors display:

When “OK” is selected, the following dialog might display:

See Appendix A.5, Item 22 to see the corresponding Fortran Medic error.

Cause
This licensing issue is due to an incompatible computer setting: The computer is set to use a calendar that is incompatible with PSCAD legacy licensing, for example the Arabic Regional and Language settings. As a result, the PSCAD licensing perceives an issue with the date, and denies licensing.

System
This issue affects PSCAD v4.5.4, v4.5.5, and v4.6.0, and has been fixed in later versions.

Solution 1
Switch to PSCAD v4.6.1 or later, as the licensing appears to work when the computer is set to the Arabic Regional Language settings.

Solution 2
Or, continue to use PSCAD v4.6.0, but switch your computer to the English language, so that the computer will use the Gregorian calendar (from the Windows Start button, display the Control Panel, browse to “Language and Region”, and change the setting to “English”).
3.10 Receiving a “No licenses found on the selected license host” Error Message

Problem
When a license host IP address or name is entered into the PSCAD System Settings dialog box, the following errors display:

Causes and Solutions

Cause 1
The license is not correctly installed on the host machine: The license file and/or USB lock are missing.

Solution 1
Redirect PSCAD to use the license from a different host machine, or
Correctly set up the licensing on the host machine. Instructions may be found in the manual posted here: https://hvdc.ca/knowledge-base/read.article/304/installing-and-activating-a-lock-based-license/

Cause 2
This issue occurs when an incompatible version of OpenVPN client on Windows 10 is used to access the license server.

Solution 2
Ensure that a Windows 10 compatible version of OpenVPN client is used to access the license server.

Cause 3
The License Manager setup tool is configured improperly, to only allow self-licensing. Licensing therefore will not be shared with other machines.

This configuration can be viewed in either of the following manners on the server machine:

- Launch “Setup Tool” from the Windows Start menu, and review the Access tab. If set to “This machine only…”, as shown below, then licensing will not be shared with other machines:
Display the lmgrd-log.txt file on the server machine (Section 3.3). If the following text is listed in this file, then licensing is configured only for self-licensing:

Specified: Localhost  
Actual: Localhost

**Solution 3**

Configure the license manager to share licensing other machines:

- Launch “Setup Tool” from the Windows Start menu.
- In the “Access” tab, select “Authorized networks” (to specify authorized networks) or “Everyone” (to allow machines from any network).
- If “Authorized networks” is selected in the above step, then specify the networks on the “Networks” tab (see Appendix A of this manual for details).
3.11 License cannot be activated on Windows 10

Problem
When the “Workspace Settings” dialog box is configured, and the “Activate” button is selected...

...licensing is not activated, and the following error message displays:

Cause
On a machine running Windows 10, the PSCAD USB lock can not be detected using an earlier version of Sentinel USB Drivers (v7.5.8).

Solution
- Uninstall current Sentinel Drivers.
- Install Sentinel Driver 7.5.9 from the following link (or find a later version):
  \[http://updater.pscad.com/utilities/USBDriver_7.5.9.zip\]
- Log out then log back in to your machine to apply the change.
- Retry licensing PSCAD.
3.12 Receiving a “(193) Application is not valid Win32 application” Error Message

Problem
When attempting to start the License Manager, either manually or automatically upon restarting a computer, the following error displays, and the License Manager fails to start:

Cause
This issue occurs when one or both of the following conditions is true:

- In addition to the standard “Program Files” folders on your system drive (e.g. “C:\Program Files” and “C:\Program Files (x86)”), there is another folder or file named ‘Program’ (e.g. “C:\Program for Calculating Taxes”), and/or
- In addition to the standard “Common Files” folders in your “Program Files” folders (e.g. “C:\Program Files\Common Files” and “C:\Program Files (x86)\Common Files”), there is another folder or file containing the word “Common” (e.g. “C:\Program Files (x86)\Common Tasks at Work”).

When the Windows Service Control Manager (Services.exe) tries to start the Server service, the program verifies the following paths:

- SystemDrive\Program
- SystemDrive\Program Files\Common

If the Program or Common paths exist as folders or files, the Service Control Manager cannot locate the required folders, and the error message will display.

For more details, please refer to the following article from Microsoft®: [http://support.microsoft.com/kb/812486](http://support.microsoft.com/kb/812486)

Solution
Rename, move, or delete the offending files or folders.
3.13 Licensing is activated successfully for two instances of PSCAD, but the first instance later becomes de-activated

Problem
One instance of PSCAD is opened and licensed. When a second instance of PSCAD is opened, licensing is also successful, however, licensing on the first instance becomes de-activated.

The following error message displays in the PscadLmgr.txt file:


Note that the License Manager is not running, PSCAD is self-licensing.

Cause
When using self-licensing, only one instance of PSCAD may be run.

Solution
If your license is a multi-user license with more than one PSCAD seat (e.g. “n” seats), then you may run up to “n” instances of PSCAD as long as the standalone License Manager is installed and running. Instructions for setting up the latest License Manager may be found here.

Or, if your license only has a single PSCAD seat, you may only run a single instance of PSCAD.

For both above solutions, if you convert your license from “lock-based” to “certificate”, there would be no limit to the number of instances of PSCAD that may be run on a machine. A document posted here provides a comparison of lock-based and certificate licensing. To proceed with this conversion, please contact our Sales Desk (sales@pscad.com).
3.14 Licensing fails when using the standalone License Manager

Problem
When trying to activate a license being managed by the standalone License Manager, licensing fails, and the following error is listed in the lmgd-log.txt file:

ERROR: Cannot run multiple instances of Lmgrd-hvdc.exe

Cause
A process on the machine is running, which is causing a conflict with the License Manager.

Solution
Try closing all instances of the following processes from the Windows “Task Manager” if any are open:

- The License Manager (named “Lmgrd-hvdc”, listed in the “Services” tab of the Task Manager)
- Windows browsers
- Get Info utility
- Enter License Key utility
- Notepad
- Setup Tool

Restart the License Manager from the Windows Start menu (“Restart License Manager Service”).

Retry the licensing.
3.15 Unable to Start or Restart the License Manager

Problem
When trying to start or restart the License Manager from the Windows Start menu ("Start License Manager Service", Restart License Manager Service”), the following error is displayed:

![Error Message]

Cause
Anti-virus software might be quarantining the license manager service, which prevents it from being started or restarted. Specifically, an anti-virus software named Cylance has been shown to have quarantined the license manager service in this manner.

Solution
If possible, white-list the license manager service in the anti-virus software.
3.16 Receiving an “Unable to read license database file” Error Message

Problem
When using the License Update tool to install a license database file, the following message displays in the tool, and the new license file will not install:

Database appears to be corrupted. Computed crc: ******** Actual crc: 00000000
Unable to read license database file

Cause
A license database file that is pre-existing on this machine is corrupted. The new license file cannot be installed until the pre-existing license database file is removed.

Solution (1)
Use License Update tool v1.43.1 and later. The latest one is posted to the following link:

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

Download the files from the above link, extract the files to a temporary folder, then run the extracted LicenseUpdate.exe file. From the “Actions” menu, select “Enter license keys”, then browse to the license text file. This tool will over-write the corrupted file with the new license database file.

Solution (2)
For License Update tool v1.36 and later:

- Launch the License Update Tool.
- From the “Actions” menu, select “Delete license databases”, and the pre-existing files will be deleted.
- From the “Actions” menu, select “Enter license keys”, then browse to the license text file. The license database file will be installed.

Solution (3)
For License Update tool 1.28 and later:

- Manually delete the license database files (“lmgr-hvdc”) from the following two locations as applicable:

  C:\Windows

  C:\Users\Public\Documents\Manitoba HVDC Research Centre\LicenseManager

- Launch the License Update Tool.
- From the “Actions” menu, select “Enter license keys”, then browse to the license text file. The license database file will be installed.
3.17 Licensing issues on Windows 10 Enterprise or Windows 10 Educational Edition

Problem
When running on Windows 10 Enterprise Edition, neither PSCAD nor the License Manager can use lock-based licensing when using Sentinel System Driver 7.5.9 while Credential Guard is running.

Cause
Windows 10 Enterprise with CredentialGuard/SecurityGuard enabled will block the loading of the Sentinel System Driver (version 7.5.9 and earlier), blocking all licensing by the License Manager, and by PSCAD when using self-licensing.

Evidence
1. In Windows Settings | Devices | Connected Devices, the following should be displayed:

   ![USB UltraPro Driver error]

2. In the Windows Device Manager, if you browse to the USB devices section, the following should be displayed, which indicates a Sentinel driver issue:

   ![Universal Serial Bus controllers]
   \[Generic USB Hub
   \[Generic USB Hub
   \[Intel(R) 7 Series/C216 Chipset Family USB Enhanced Host Controller - 1E26
   \[Intel(R) 7 Series/C216 Chipset Family USB Enhanced Host Controller - 1E2D
   \[Intel(R) USB 3.0 extensible Host Controller - 1.0 (Microsoft)
   \[SafeNet USB SuperPro/UltraPro
   \[USB Composite Device
   \[USB Mass Storage Device
   \[USB Root Hub
   \[USB Root Hub
   \[USB Root Hub (xHCI)

3. When PSCAD is launched, you should not be able to acquire a lock-based license.
4. When the Fortran Medic tool is run (Appendix A.2), if the tool detects that Credential Guard is running, a warning is displayed as shown in Appendix A.5 Item 28.

Affected PSCAD Versions
- All versions

Affected Windows Platforms
- Windows 10 Enterprise Edition: affected
- Windows 10 Educational Edition: affected
- Windows 10 Professional Edition: not affected
- Windows 10 Home Edition: not affected
Solution (1)
Update your Sentinel Driver to v7.6.0, as this version can run on Windows 10 Enterprise regardless whether Credential Guard is installed:

1. Download the updated Sentinel Driver installer from the following link:
   http://updater.pscad.com/utilities/USBDriver_7.6.0.zip
2. Open the ZIP file, and extract the three files to a temporary folder.
3. Launch the installation in one of the following two methods:
   - Double-click the “Install.bat” file to begin the installation, or
   - Open a Command Prompt with Windows administrator privileges (from the Windows Start menu, browse to and right-click on “Command Prompt”, and select “Run as administrator”) and type in the following:

   Setup.exe /s /v"/qn ADDLOCAL=USB_Driver,Parallel_Driver CONFIRMUPGRADE=TRUE"
4. When prompted to allow the program to make changes to your computer, select “Yes”.
5. When the installation is complete, the command prompt will close.
6. Log out, then log in on the machine.
7. If using a License Manager for PSCAD, ensure that the License Manager is started (from the Windows Start menu, browse to and select “Start License Manager Service”).
8. Launch PSCAD and ensure a license may be activated.

Solution (2)
Continue to use Sentinel Driver 7.5.9 on Windows 10 Enterprise, but remove Credential Guard.

Note
Lock-based licensing does not work at all when Sentinel System Driver 7.5.8 is used on Windows 10 Enterprise Edition.
3.18 Unable to activate a License from the Server

Problem

PSCAD clients are unable to activate a license over the network from the Server.

Cause

ICMP Echo, required for communication between the server and client machines, is being blocked by anti-virus (e.g. McAfee).

Solution (1)

Configure the firewall to allow Echo Requests. For example, to enable this in Windows Firewall, the following are the general steps:

- Launch the Windows Firewall, and select Advanced settings
- Select Inbound Rules, enable Echo Request – ICMPv4-In
- Select Outbound Rules, enable Echo Request – ICMPv4-Out

Solution (2)

Move the License Manager and USB lock and license database file to a machine on which the anti-virus is not running.
3.19 Receiving Error “USB Error: driver binary not signed by a valid authority, status = 104”

Problem
When trying to activate a license, the following error message displays:

![License Manager Error](image1)

When you click on “OK” on the above dialog box, the following error message then displays:

![PSCAD Error](image2)

The following error displays in the “lmgrd-log.txt” file on the machine hosting the license:

```
USB Error: driver binary not signed by a valid authority, status = 104
```

Cause
There is an issue with the USB driver software, and it should be re-installed.

Solution
- From the Windows Start menu, uninstall the Sentinel Drivers.
- Log off then log back in to the machine (or restart the machine) to fully apply the changes.
- Re-install the Sentinel Drivers. A download and setup instructions are available in Section 2.7.2 (Step 2) of the Resolving Issues document: [https://hvdc.ca/uploads/knowledge_base/software_installation_and_licensing.pdf?t=1531946006](https://hvdc.ca/uploads/knowledge_base/software_installation_and_licensing.pdf?t=1531946006)
- Restart the License Manager (from the Windows Start menu, browse to “Restart License Manager Service”).
3.20 Receiving a "using an invalid communication protocol: < GNUTELLA" Error Message

Problem
When trying to activate a license, the following messages display in the lmgrd-log.txt file:

```
ERROR: System [IP ADDRESS] using an invalid communication protocol: < GNUTELLA CONNECT/0.4
```

Cause
The use of GNUTELLA may block or interfere with PSCAD communications with the License Manager

Solution
The following are two options to resolve this:

- Uninstall GNUTELLA, and do not use this software on your network.
- Or, set the GNUTELLA port to something other than 2053

[c]ase 11184, 185724, 22352]
Resolving PSCAD Issues

3.20 Receiving an "Unable to repair corrupted lock [1a]" Error Message

Problem
When trying to activate a license, licensing fails, and the following message displays in the PscadLmgr.txt file:

   ERROR: Unable to repair corrupted lock [1a]:

Cause
The cause is due to trying to license PSCAD v4.6.0 or older with a type of license that does not support that version. Specifically, a lock-based license containing additional parallel simulations is only supported with PSCAD v4.6.1 and later.

   Note
   This limitation does not exist with certificate licensing, which supports additional parallel simulations with PSCAD v4.5.3 and later.

Solution (1)
Update the PSCAD software to v4.6.1 or newer.

Solution (2)
Continue to use the older version of PSCAD, but convert your license from lock-based licensing to certificate licensing.

To employ either solution, please contact our sales desk (sales@pscad.com).
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

5.1 Receiving an "Unable to obtain a license with the current license settings" Error Message

Problem

When trying to activate a license, the following error message displays:

![License Manager dialog box]

When you click on “OK” on the above dialog box, the following error message then displays:

![PSCAD dialog box]

Unable to obtain a license with the current license settings.
Please review and modify your license settings.
Cause 1

If the following error message is listed in the PscadLmgr.txt file (this log file may be displayed as instructed in Section 3.2):

    Trial Licence Manager V1.34 starting on Windows 7
    Found 1 licence for <Educational v4/x4>
    ERROR: Damage after Normal block (#196) at <0x00000000:000>

...then in a Lockless Trial License (i.e. v4.2.0 to v4.5.3), licensing is halted because an incorrect time setting on the computer has been detected – the clock has been set back.

Solution 1

Set the clock to the correct time, and forward the following to our Support Desk (support@pscad.com):

- A brief description of the issue
- Your PscadLmgr.txt file
- The trial.txt file that was provided by our Sales Desk

You will receive a fix lock utility to run to fix your license.

Cause 2

If the following error message is listed in the PscadLmgr.txt file (this log file may be displayed as instructed in Section 3.2):

    Machine1 requesting Professional X4
    ================================================
    Trial License Manager V1.34 starting on Windows 7
    Rejected 1 licence: Cause: Trial licence for <Professional 4> expired on Sat Apr 01 23:59:59 2006
    ERROR: No valid trial licenses detected

...then the lockless trial license is expired.

Solution 2

Contact our Sales Desk (sales@pscad.com) to purchase PSCAD.
**Cause 3**

If the following messages are listed in the PscadLmgr.txt file (this log file may be displayed as instructed in Section 3.2):

```
ERROR: Invalid semaphore value
ERROR: Refused to renew license [No.] for [machine name] for <PSCAD X4 Professional>. License not active.
```

...an unexpected second instance of PSCAD was being licensed. This would occur for licensing that is only programmed to allow a single instance of PSCAD, either because the license is a legacy lock-based license with one PSCAD seat, or it is a legacy lockless trial license.

**Solution 4**

Ensure only one instance of PSCAD is running.

**Notes**

1. This limitation is applicable when trying to license a second instance with either a lockless trial license or with lock-based licensing with only one PSCAD seat.

2. This limitation is not applicable when trying to license a second instance with Certificate Licensing; a second instance of PSCAD using Certificate Licensing may be run.

**Cause 5**

On some systems, PSCAD v4.4.1 is able to ping the server, but cannot obtain a license. This issue was resolved in later versions of PSCAD.

**Solution 5**

Run the latest License Manager version and update your PSCAD software (send your request to our Sales Desk, sales@pscad.com. Ensure to notify them of your PSCAD license number).
5.2 Receiving an "Error: Product mismatch" Message

Problem
When attempting to install a lockless trial license for PSCAD v4.2.1, the following message displays:

Cause
The lockless trial licensing requires the Windows ProductID value from the registry.
PSCAD 4.2.1 only looks at the 32-bit registry section for the ProductID value. However, 64-bit versions of Windows store the Windows ProductID in the 64-bit section of the registry, which is not accessible to PSCAD 4.2.1, and therefore the trial installation fails.

Affected Software
PSCAD 4.2.1 running on any 64-bit version of Windows, such as Windows Vista (x64) or Windows 7 (x64).

Note
PSCAD X4 is not affected by this issue.

Solution
The ProductID registry key value should be copied from the 64-bit section of the registry to the 32-bit section of the registry.
The recommended method to perform this is through the Fortran Medic tool, as follows:

- Download and run the latest Fortran Medic tool as per Appendix A.2.
- When the tool has completed its review of your setup, browse down to "Machine Info", right-click on the error message, and select the option to repair the error.

Machine Info

Conflicts
- You may encounter errors when installing lockless trial licenses.
The other method for copying this registry key is to perform this manually (not recommended). Here are the steps:

**Note**
Your organization’s IT policy might prohibit this solution, as it involves modification to the Windows Registry Editor. Your IT staff may need to assist with this solution.

- From the Windows Start menu, type in regedit.exe, as shown below:

- Select regedit.exe from the Windows menu:

- The Windows Registry Editor will display.

- In the Registry Editor, browse to:
  
  HKEY_LOCAL_MACHINE\Software\Microsoft\Windows NT\CurrentVersion.

- Double-left-click on the ProductID key, and Edit String will display:
- Copy all of the text from the Value Data field, then select Cancel. Paste this data somewhere temporary (e.g. in a text file).

- Browse to `HKEY_LOCAL_MACHINE\Software\Wow6432Node\Microsoft\Windows NT\CurrentVersion`.
- In the right pane, right-click, select New, then select String Value:

  ![Registry Editor](image)

  **Note**
  If the following error displays, this type of action has been blocked by IT policy. Please obtain assistance from your IT staff.

  ![Error Creating Value](image)

- This will create a new registry key called New Value #1 (or something similar).
- Right-click on the new registry key, select Rename, then rename this to `ProductID`. 
- Double-left-click on the new ProductID key, and Edit String will display. In the Value data field, paste in the text that you copied in Step 6, then select OK.

- Close the Windows Registry Editor.

[case 27877]
5.3 Receiving a “Windows OS mismatch” Error Message

Problem
When trying to activate a trial for PSCAD v4.2.1, the following error message displays:

![Error Message](image)

Cause
This error indicates that something on the machine was updated after the PSCAD trial request file was generated.

*Note*
For a lockless trial, no updates must be performed to the machine.

Solution
Regenerate the trial request file, and submit to the sales desk (sales@pscad.com).
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 L)

Solution 3
Update your software to v4.5.2 or later (contact our Sales Desk at sales@pscad.com).
Problem
The Component Wizard pane is not displaying correctly:
Note
The correct layout for the Component Wizard is as follows:

![Component Wizard](image)

Enter the number of ports to add to each side when Add Ports is clicked.

- Left: [ ]
- Top: [ ]
- Right: [ ]
- Bottom: [ ]

Add Ports Remove All

Finish Reset Help

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:
  
    ![Conflicts Message]

  - 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:

![Unable to connect to MyCentre server.]

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:

![Security Alert](image)

Revocation information for the security certificate for this site is not available. Do you want to proceed?

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:

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:

  ![Help system screenshot]

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

  ![Help system section screenshot]
6.12 Receiving a “Server Busy” Error Message

Problem

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

![Server Busy Message](image)

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 Application Options dialog, Matlab is not detected, and may therefore not be selected:

![Application Options Dialog](image)

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:

![Matlab Installation Details](image)

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:

![Security Alert dialog box]

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:

![Certificate dialog box]

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:

![Image](image.png)

- 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:

![Error Message](image)

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:

```
1. Right-click on the shortcut, and select Properties
```

```
2. Add a SPACE, followed by:
   "/LocalAppData:C:\PSCAD\LocalAppData"
```

```
3. Select OK
```

- 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]
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
- Run PSCAD on a machine on which Cygwin is not installed.
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 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:

![Error Message]

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@pscad.com:
- 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

Problem 1

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

![Error Message](image)

**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:

C

ause 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.

olution 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)

Uninstall GFortran, and use an Intel FORTRAN compiler instead.
Solution (2)
If the offending compiler is not required on this computer, simply uninstall it.

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:\QNX650_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.

Solution (4)
Use two separate computers, one for running the offending compiler and one for running PSCAD.
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_’
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@pscad.com).
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”:

  ![Image](workspace.png)

  ![Image](project_settings.png)

  ![Image](library_settings.png)
• 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.o:(.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.

Problem 3

The following compiling error is displayed:

```
C:\<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.
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@pscad.com), along with all Build messages, and a snapshot 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 programmed 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 permissions 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 Unexpectedly, with no Runtime Error

Problem
When compiling a case, PSCAD stops unexpectedly, with no runtime error.
Running the PSCAD case manually outside of the PSCAD application results in the following error message:

![Error Message]

The program can't start because EtranSvr.dll is missing from your computer. Try reinstalling the program to fix this problem.

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:

1. Right-click on the workspace, and select "Workspace Options."
2. Select the "Build" page from the drop-down menu.
3. Increase the number of "Wireless Radio Links"
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”):

![Workspace Settings Dialog Box](image)

b. When the Workspace Settings dialog box displays, perform the following:
   
   - Select the “Runtime” page from the drop-down menu.
   
   ![Runtime Page](image)

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

     ![Communication Port Base Value](image)

   - Select “Ok” to apply the change:

     ![Ok Button](image)
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:

![Error Message](image)

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:

<table>
<thead>
<tr>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LINK : fatal error LNK1104: cannot open file 'dfor.lib'</td>
</tr>
<tr>
<td>NMAKE : fatal error U1077: 'C:\Program Files (x86)\Microsoft Visual Studio *\VC\Bin\linkexe' : return code '0x450'</td>
</tr>
<tr>
<td>Unable to generate a simulation executable for namespace ‘*’</td>
</tr>
</tbody>
</table>

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:

![Error Message]

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:

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”.

![Windows Defender dialog box showing Settings and Exclusions options]

![Windows Defender dialog box showing Add an exclusion option]

![Windows Defender dialog box showing Exclude a folder option]
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”:

  ![Exclusions](image)

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

    ![Processes](image)

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

    ![Add exclusion](image)

  - 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 displays:

When “OK” is selected above, the following dialog displays:

When “Yes” is selected above, the following dialog displays:

The socket buffer was unable to send data using the current device. There may be an unexpected disconnect blocking the send.
WinSock Error = 10054
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_VARIMP_BRANCH( (IBRCH(3)+10),SS(3),1,0,0,R_B,0.0)
```

Error: Syntax error in argument at (1)

The correct text should look something like this:

```
CALL E_VARIMP_BRANCH( (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 address. For more information on this, please refer to the following link:


Solution 1
- Decrease the size of your network, or
- Switch to a PSCAD 64-bit application, available as of PSCAD v4.5. To update your software, please forward your license number to our Sales Desk ([sales@pscad.com](mailto: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:

![Webroot SecureAnywhere](image)

Webroot SecureAnywhere
Webroot SecureAnywhere Endpoint Protection v8.0.876 - No Infections Found

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
The following error messages display when compiling PSCAD cases with Intel Fortran and Microsoft® Visual Studio 2015:

Compiling ‘station.f’ into object code.
Compiling ‘main.f’ into object code.
Linking objects and libraries into binary ‘*.exe’
C:\*.o: fatal error LNK1112...
NMAKE : fatal error U1077: ‘C:\Program Files (x86)\Microsoft Visual Studio 14.0\VC\BIN\x86_amd64\link.exe’ : Error code 0x458
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: One is used for earlier versions (VS2010, VS2012, VS2013), and the other for VS2015. If PSCAD is configured for the wrong VS version, an error like the one 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; earlier 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;
- Intel Fortran compiler v15 and newer;
- Visual Studio 2015 and newer
Solution (1.a) – Automatic Changes using the Fortran Medic (Windows 64-bit)

The Fortran Medic tool can make changes to your 64-bit environment to use Visual Studio 2015 and newer:

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

**Note**
Windows Administrator privileges are required.

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:

  ![Installed PSCAD versions](image)

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

  ![Configuration dialog](image)

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

  ![Confirmation dialog](image)

- 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.

Solution (1.b) – Automatic Changes using the Fortran Medic (Windows 32-bit)

The Fortran Medic tool will make changes to your 32-bit environment to use Visual Studio 2015 and newer:

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

**Notes**
Windows Administrator privileges are required.

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” 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.

Solution (2) – Manual Changes

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) above, instead.

Notes
Windows Administrator privileges are required

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

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”:
     ![Image]
   - If using v4.6.2 or newer, locate the following three files, and delete the “main.obj” file:
     ![Image]
   - Then, for v4.6.1 and newer, make a copy of the “main_vs2015.obj” file, and rename it as “main.obj”:
     ![Image]

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

- Open a Windows file browser to the following folder:
  
  `<PSCAD Share Folder>\emtdc\if15\windows`

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

  ![Image of emtdc_vs2015.lib and emtdc.lib]

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

  ![Image of emtdc_vs2015.lib and emtdc_vs2010.lib]

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

  ![Image of emtdc_vs2015 - Copy.lib and emtdc.lib]

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

- Open a Windows file browser to the following folder:
  
  `<PSCAD Share Folder>\emtdc\if15_x86\windows`

- Make the same changes to the emtdc library files as performed in Step 3 above.
7.37 Receiving a “The simulation process has stopped unexpectedly” Message

Problem

The following dialog displays:

![Microsoft Visual Studio C++ Runtime Library]

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 appears 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@pscad.com). 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\n
    - Save the file you receive from Support to the above path.

```
# Resolving PSCAD Issues

## 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

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:

* .exe – System Error
The program can’t start because VCRUNTIME140.dll is missing from your computer. Try reinstalling the program to fix this problem.

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:

![Visual Studio [VS2015]](image1)

**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 PATH EXT 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 PATH EXT environment variable.

Solution
Use the Fortran Medic tool to easily fix the PATH EXT 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:

   Creating 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:

- 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:

- 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
PSCAD v4.6.2

Cause

The ‘*’ characters related to Visual Studio 2017 cause the compiling error.

Solution (1)

Update your software - 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:

![Error Message]

Make File Error: The dependency ‘Battery_model’ requires a link library:
C:\...\battery.obj’ that does not exist with the given path.

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:

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

d. Save the variable.
7.57  Receiving a “Cannot determine the location of the VS Common Tools folder” Error Message

Problem
When compiling a case using the Intel Fortran compiler, 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.

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.
Resolving PSCAD Issues

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:

![](image)

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”):

![](image)

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

![](image)

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.
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:

  ![Screenshot of Medic utility](image)

  - 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:


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.

```
PATH (Machine) Environment Variable
...
- C:\Program Files (x86)\<Some folder containing the & symbol>\<some installed program>\`
```

Continued...
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]:30193
  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](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](mailto: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.\n- C:\...\<filename>.gf46>call "C:\Program Files (x86)\GFortran\4.6\bin\gf46vars.bat"\n- \The system cannot find the path specified.\n- Compiling 'Station.f' into object code.\n- 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 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 required prerequisite for the Intel Fortran compiler is missing, 'ws2_32.lib'.

Solution

Install the Windows 10 SDK kit:


Log out then log back to your machine before attempting to compile.
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:

![Warning Message]

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:

![EMTDC Runtime Error Message]

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:

![Server Busy Error Message]

This action cannot be completed because the other program is busy. Choose ‘Switch To’ to activate the busy program and correct the problem.

Solution
Contact support@pscad.com for the solution. Ensure to provide your PSCAD license number.

System
PSCAD v4.6.1.
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:

![PSCAD Error Message]

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

![Image of Region and Language settings dialog]

- Click on Additional Settings (Note that we tested this with Dutch settings).

![Image of Additional settings highlighted in Region and Language settings dialog]
- 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:

![Error Message](image)

Problem (1)
Anti-virus software is blocking the execution of files with double extensions (e.g. *.mak.bat).
Specifically, McAfee released an update that detected and blocked these types of files in June 2018.

Solution (1.a)
Turn off anti-virus software.

Solution (1.b)
Somehow configure the anti-virus software to not block the PSCAD files.

Solution (1.c)
Switch to a different anti-virus software. For example, one customer switched to Microsoft Security Essentials.

Solution (1.d)
Confine all of PSCAD project files to a particular folder, (e.g. C:\PSCAD), and configuring 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.

Problem (2)
User was trying to run a project from a remote drive.

Solution (2)
Move the project to a local drive.
9. Issues with MyCentre

Content moved to new manual “Resolving MyCentre Issues”
See Section 1.2
10. Issues with Diagnostic Tools

10.1 Receiving a “GetInfo32.exe – System Error” Message (Get Info Tool)

Problem (1)
When trying to run the Get Info tool, the following error is displayed, and the Get Info tool will not launch:

![GetInfo32.exe - System Error](image)

The program can’t start because mfc100.dll is missing from your computer. Try reinstalling the program to fix this problem.

Note
If the error specifies a different missing file, “mfc140.dll”, then refer to Problem (2), below.

Cause (1)
The Get Info tool is unable to launch because of missing pre-requisite software:

*Microsoft Visual C++ 2010 Redistributables (x86)*

Solution (1)
Install the missing software. For your convenience, this may be installed by using another tool, the Fortran Medic:

- Download and run the latest Medic as per Appendix A.2, Steps (a) and (b).
- Use the Medic to install the missing software as shown:

![Fortran Medic Utility](image)

- The Get Info tool should now be able to launch.
Problem (2)

When trying to run the Get Info tool, the following error is displayed, and the Get Info tool will not launch:

![Get Info Error](image)

Note
If the error specifies a different missing file, “mfc100.dll”, then refer to Problem (1), above.

Cause (2)

The Get Info is unable to launch because of missing pre-requisite software:

- Microsoft Visual C++ 2015 Redistributables (x86)
- Microsoft Visual C++ 2015 Redistributables (x64)

Solution (2)

Install the missing software. For your convenience, this may be installed by using another tool, the Fortran Medic:

- Download and run the latest Medic tool as per Appendix A.2, Steps (a) and (b).
- Use the Medic to install the missing software as shown:

![Fortran Medic Utility](image)

- The Get Info tool should now be able to launch.
10.2 Receiving an “Unable to proceed” error message when using the License Update Utility

Problem
When using the License Update Utility, the following error displays:

![Screenshot of License Update Utility error message]

Cause
The License Update Utility detects that an older PSCAD tool that is no longer used, called “Upgrade License”, is running in the background. The License Update Utility cannot proceed with this other program running.

Applicability
This error may occur when using the License Update Utility that was released with License Manager v1.42 and earlier.

Solution
- The above error message will not display in the License Update Utility that was released with License Manager v1.43 or later. Update your tool to v1.43 or later (this may be done either by installing the standalone tool or installing the License Manager v1.43 or later. For this software, please contact our support desk (support@pscad.com).

- or

- Display your Windows Task Manager, halt the Upgrade program, then proceed to use the same License Update Utility version.
10.3 Receiving an “Unable to update the lock. Unable to initialize interface due to: Comm error.” Error

Problem
When using the License Update Utility, the following error displays:

Unable to update the lock. Unable to initialize interface due to: Comm error

Cause
There may be some sort of group policy that requires that the user have Windows Administrator privileges when any applications are attempting to communicate with the USB drivers.

Solution
Close the License Update Utility, re-launch the License Update Utility with Windows Administrator privileges.
10.4 Receiving an “Unable to update the lock” Error Message

Problem
When using the Fix Lock Utility to downgrade or deactivate a USB lock, the following messages display in the utility:

Will attempt to update lock ***.***
...
Sentinel USB driver version: <unknown>
Unable to update the lock. Unable to load ‘sx32w.dll’. BadImageFormat: Not compiled for x86.
Error: An attempt was made to load a program with an incorrect format. (Exception from HRESULT...??)
Version: 7.1.0.14
Length: ***

Cause
Sentinel Drivers for communicating with the USB lock are not installed.

Solution
Option 1: Install Sentinel Drivers, then retry running the utility.

Option 2: Run the Fix Lock utility on a machine on which Sentinel Drivers are already installed (e.g. on a PSCAD machine or PSCAD license host machine).
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 compatibility, 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.

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@pscad.com, 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@pscad.com.

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:

<table>
<thead>
<tr>
<th>Error</th>
<th>Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. PATH (Machine) Environment Variable</td>
<td>Right-click on the error and select the option to remove it. This environment variable will be removed.</td>
</tr>
<tr>
<td>Conflicts</td>
<td></td>
</tr>
<tr>
<td>Intel MKL Segments</td>
<td></td>
</tr>
<tr>
<td>C:\Program Files (x86)\Inte...\bin</td>
<td></td>
</tr>
<tr>
<td>2. Intel Fortran (xx.x.xxx)</td>
<td>Right-click on the error and select the option to update the .exe file.</td>
</tr>
<tr>
<td>Conflicts</td>
<td></td>
</tr>
<tr>
<td>Missing expected file</td>
<td></td>
</tr>
<tr>
<td>File: C:\Program Files (x86)\Intel...\ifort.exe</td>
<td></td>
</tr>
<tr>
<td>PSCAD V4 will not recognize this compiler</td>
<td></td>
</tr>
<tr>
<td>3. Installed PSCAD versions</td>
<td>Right-click on the message and select the option to delete this file.</td>
</tr>
<tr>
<td>PSCAD X4 Release...</td>
<td></td>
</tr>
<tr>
<td>Workspace file: C:...user_profile.xml</td>
<td></td>
</tr>
<tr>
<td>File exists but appears to be empty</td>
<td></td>
</tr>
<tr>
<td>4. Installed PSCAD versions</td>
<td>Right-click on the message and select the option to restore the start page.</td>
</tr>
<tr>
<td>PSCAD X4 Release...</td>
<td></td>
</tr>
<tr>
<td>Conflicts</td>
<td></td>
</tr>
<tr>
<td>PSCAD will not display the latest Start Page.</td>
<td></td>
</tr>
<tr>
<td>5. Visual Studio [VS2005]</td>
<td>You are running an older version of the Fortran Medic, and the following is occurring:</td>
</tr>
<tr>
<td>Installation Folders:</td>
<td></td>
</tr>
<tr>
<td>VS folder: C:\Program Files (x86)\Microsoft Visual Studio 8\ Folder</td>
<td></td>
</tr>
<tr>
<td>exists</td>
<td></td>
</tr>
<tr>
<td>VC folder:</td>
<td></td>
</tr>
<tr>
<td>Visual Studio VS2005 is not installed. Folder not specified.</td>
<td></td>
</tr>
<tr>
<td>6. Firewalls and anti-virus programs</td>
<td>See manual posted here:</td>
</tr>
<tr>
<td></td>
<td>Also, refer to Appendix F for information on the creation of executable files during a simulation.</td>
</tr>
<tr>
<td>Error</td>
<td>Solution</td>
</tr>
<tr>
<td>-------</td>
<td>----------</td>
</tr>
</tbody>
</table>
| **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.** Licensing  
  Use only FIPS compliant algorithms  
  Value: 1 (Enabled)  
  Conflicts  
This computer is not configured to support a required algorithm  
Or  
Licensing  
Use only FIPS compliant algorithms  
Status: <Registry value not found>  
Conflicts  
This computer is not configured to support a required algorithm  
Or  
Licensing  
Use only FIPS compliant algorithms  
Value: 5  
Status: <Registry value is invalid>  
Conflicts  
This computer is not configured to support a required algorithm | See Section 5.1, Solution #3 for the solution. |
<table>
<thead>
<tr>
<th>Error</th>
<th>Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>10. “Installed PSCAD versions”…&lt;br&gt;“PSCAD X4 Release…”&lt;br&gt;ZLib2.dll appears to be from a different version of PSCAD</td>
<td>Right-clicking on the message displays a dialog with the solution steps:&lt;br&gt;&lt;br&gt;<strong>Notice</strong>&lt;br&gt;To fix this issue, you need to:&lt;br&gt;- install or repair PSCAD 4.x.y which will re-install this file. See Section 2.8 for additional information.</td>
</tr>
<tr>
<td>11. Intel Fortran (*)&lt;br&gt;Conflicts&lt;br&gt;INTEL_LICENSE_FILE – Duplicate path segments may cause this compiler to fail</td>
<td>Right-clicking on the message displays a dialog with the solution steps:&lt;br&gt;&lt;br&gt;<strong>Confirm Environment Changes</strong>&lt;br&gt;To fix this issue, the FortranMedic must replace an environment variable value.&lt;br&gt;Name: INTEL_LICENSE_FILE&lt;br&gt;Type: Machine&lt;br&gt;Required value: C:\Program Files (x86)\Common Files\Intel\Licens*es&lt;br&gt;Select OK to replace the path segment&lt;br&gt;Select Cancel to leave the environment value unmodified&lt;br&gt;Select “OK” to proceed with fixing the variable. See Section 7.27 for additional information.</td>
</tr>
<tr>
<td>Error</td>
<td>Solution</td>
</tr>
<tr>
<td>----------------------------------------------------------------------</td>
<td>--------------------------------------------------------------------------</td>
</tr>
<tr>
<td>12. “PATH (Machine) Environment Variable”</td>
<td>Right-clicking on the error displays the following dialog:</td>
</tr>
<tr>
<td>Conflicts</td>
<td>Click “OK” to hide the warning.</td>
</tr>
<tr>
<td>The PATH variable is excessively long and you may lose your shortcuts to Notepad and other system programs.</td>
<td>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: <a href="https://hvdc.ca/knowledge-base/read/article/217/update-client-myupdater-issues/v">https://hvdc.ca/knowledge-base/read/article/217/update-client-myupdater-issues/v</a>:</td>
</tr>
<tr>
<td>13. Content moved to new manual “Certificate Licensing Issues”</td>
<td>Resolve as per Section 7.24, Problem #2.</td>
</tr>
<tr>
<td>(see Section 1.2)</td>
<td></td>
</tr>
<tr>
<td>14. PATH (Machine) Environment Variable</td>
<td>Resolve as per Section 7.28.</td>
</tr>
<tr>
<td>Conflicts</td>
<td></td>
</tr>
<tr>
<td>Segments containing a ‘&amp;’ character</td>
<td></td>
</tr>
<tr>
<td>“C:\Program Files (x86)***”</td>
<td></td>
</tr>
<tr>
<td>15. Folder Info</td>
<td>Resolve as per Section 7.28.</td>
</tr>
<tr>
<td>Short path conversion: Not working</td>
<td></td>
</tr>
<tr>
<td>16. Detecting Intel Compilers</td>
<td>Resolve as per Section 7.30.</td>
</tr>
<tr>
<td>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:</td>
</tr>
<tr>
<td></td>
<td>Cases compiled with Intel Fortran 15.0.221 should compile and link, but MAY fail to run due to Visual Studio 2015 being installed. To resolve this, you need to: 1. Use Intel Fortran 16.0 or better, or 2. Rename your Visual Studio 2015 installation folder, or, 3. Uninstall Visual Studio 2015.</td>
</tr>
<tr>
<td>Error</td>
<td>Solution</td>
</tr>
<tr>
<td>-------</td>
<td>----------</td>
</tr>
<tr>
<td><strong>17. Visual Studio [•]</strong>&lt;br&gt;Required Environment and Registry Values&lt;br&gt;VS90COMNTOOLS (environment) does not exist&lt;br&gt;Compiling might not work. The Medic tool can detect and in some cases repair this missing variable for the following Visual Studio releases:</td>
<td><strong>Error Solution</strong>&lt;br&gt;Compiling might not work. The Medic tool can detect and in some cases repair this missing variable for the following Visual Studio releases:</td>
</tr>
<tr>
<td></td>
<td>Visual Studio edition / version</td>
</tr>
<tr>
<td></td>
<td>2008 (commercial)</td>
</tr>
<tr>
<td></td>
<td>2008 Premier Partner (Shell Edition)</td>
</tr>
<tr>
<td></td>
<td>2010 (commercial)</td>
</tr>
<tr>
<td></td>
<td>2010 Premier Partner (Shell Edition)</td>
</tr>
<tr>
<td></td>
<td>2012 (commercial)</td>
</tr>
<tr>
<td></td>
<td>2013 (commercial)</td>
</tr>
<tr>
<td></td>
<td>2013 Premier Partner (Shell Edition)</td>
</tr>
<tr>
<td></td>
<td>2015 (commercial)</td>
</tr>
<tr>
<td><strong>18. MyUpdater</strong>&lt;br&gt;Products file&lt;br&gt;C:\Users[Your Windows Username]\AppData\Local\Manitoba HVDC Research Centre\UpdateClient\Products.xml&lt;br&gt;File does not exist&lt;br&gt;Or&lt;br&gt;Products file&lt;br&gt;C:\Users[Your Windows Username]\AppData\Local\Manitoba HVDC Research Centre\UpdateClient\Products.xml&lt;br&gt;File exists&lt;br&gt;File does not contain a MyCentre identity&lt;br&gt;See Section 7.56 for further information.</td>
<td>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.&lt;br&gt;This message is intended to assist users experiencing an issue with logging in to MyUpdater due to their computer or network restrictions.&lt;br&gt;For further assistance on this a MyUpdater login issue, please contact our Support Desk (<a href="mailto:support@pscad.com">support@pscad.com</a>).</td>
</tr>
<tr>
<td>Error</td>
<td>Solution</td>
</tr>
<tr>
<td>-------</td>
<td>----------</td>
</tr>
</tbody>
</table>
| 19. * Environment Variable | Multiple unnecessary segments should be removed because they might cause performance issues on your computer. Right-click on each duplicated segment, and select “OK” when prompted: |}
| Listing: [ length = * characters ] | 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: |
|  — Conflicts | |}
| Duplicated segments | |}
| ▶ C:\*** | |}
| 20. Conflicts | For example, a program called “Single Sign On Engine” is suspected of halting PSCAD simulations. See Section 7.37 for details. |}
| Identity and access management software can affect operation of PSCAD | |}
| Double-clicking on this error displays the following dialog box: | |}
| ![Warning dialog box](image1.png) | |}
| Identity and access management software can prevent: | |}
| - PSCAD from compiling and linking simulation cases | |}
| - PSCAD from launching and running simulations | |}
| ![Notice dialog box](image2.png) | |}
| Successfully removed duplicates | |}

---

Page 207
21. **Git/mingw may cause PSCAD to fail when compiling with Intel FORTRAN**  
   Double-clicking on this error displays the following dialog box:  
   ![Warning Dialog](image)  
   Git and Mingw appear to be installed, and may cause PSCAD to fail when compiling cases with Intel Fortran.  
   To resolve this issue, you need to:  
   1. rename the Git folder,  
   2. remove Git and Mingw, or  
   3. Install PSCAD and Intel FORTRAN on another machine.  
   See Section 7.38 for more information.

22. **Lock-based licensing might not work due to Arabic regional settings**  
   Double-clicking on this error displays the following dialog box:  
   ![Warning Dialog](image)  
   Lock-based licensing might not work due to Arabic regional settings.  
   After activating a license, you may encounter the following error message:  
   Your temporary license has expired  
   To resolve this issue:  
   1. Configure your system to use the English language and the Gregorian calendar, or  
   2. Upgrade to PSCAD 4.6.1 (or better)  
   (See Section 3.9 for more information)
<table>
<thead>
<tr>
<th>Error</th>
<th>Solution</th>
</tr>
</thead>
</table>
| **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: |
<p>| Refer to the Fortran Medic warning for options. |</p>
<table>
<thead>
<tr>
<th>Error</th>
<th>Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>25.</strong> Content moved to new manual “Certificate Licensing Issues” (see Section 1.2)</td>
<td></td>
</tr>
</tbody>
</table>
| **26.** Machine Info Conflicts  
You may encounter errors when installing lockless trial licenses. | This issue is only relevant if you will be installing a PSCAD v4.2.1 lockless trial license on a Windows 64-bit edition. Refer to Section 5.2 for details. |
| **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.** DeviceGuard/CredentialGuard is enabled and may block lock-based licensing | If you right-click on the message, the following dialog displays:  
Select "OK" to close the warning. Refer to Section 3.17 for details. |
<table>
<thead>
<tr>
<th>Error</th>
<th>Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>29. User Info</strong>&lt;br&gt;Account type: Domain account&lt;br&gt;Profile type: mandatory&lt;br&gt;</td>
<td>Mandatory profiles are not supported</td>
</tr>
<tr>
<td><strong>30. Machine Info</strong>&lt;br&gt;Echo.vbe&lt;br&gt;C:\Windows\echo.vbe</td>
<td></td>
</tr>
<tr>
<td><strong>31. Automation Library</strong>&lt;br&gt;Install folder: C:\Program Files (x86)\PSCAD\Automation&lt;br&gt;Publish date: unknown</td>
<td></td>
</tr>
<tr>
<td><strong>32. PATH (Machine) Environment Variable</strong>&lt;br&gt;Conflicts&lt;br&gt;The PATH environment variable is missing the \Windows\System32 path segment.&lt;br&gt;The PATH environment variable is missing the \Windows path segment.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Error</td>
</tr>
<tr>
<td>---</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
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)  
do not exist  
... | These errors may be disregarded. A PSCAD project may be compiled if these errors are present.                                                                                                                                                                                                                                          |
| 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  
... | Notice  
Matlab R2018 is not supported by any version of PSCAD  
To resolve this issue, you need to install and use Matlab R2017 or earlier.
<table>
<thead>
<tr>
<th>Error</th>
<th>Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>36. Identical numerical group and decimal separators.</td>
<td><img src="image" alt="Warning" /> The same character is used as a numerical group separator and as a decimal separator. This may cause issues when compiling and running PSCAD cases. 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.</td>
</tr>
<tr>
<td>Error</td>
<td>Solution</td>
</tr>
<tr>
<td>-------</td>
<td>----------</td>
</tr>
<tr>
<td><strong>37.</strong> Content moved to new manual “Certificate Licensing Issues” (see Section 1.2)</td>
<td></td>
</tr>
<tr>
<td><strong>38.</strong> Content moved to new manual “Certificate Licensing Issues” (see Section 1.2)</td>
<td></td>
</tr>
</tbody>
</table>
| **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. |
| **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 | Warning  
msys appears to be installed and can cause compiles using GFortran 4.2.1 to fail.  
To resolve this issue, you need to:  
1. use this utility to remove msys from the PATH, or  
2. rename the msys installation folder when using GFortran 4.2.1, or  
3. uninstall the msys application.  
(Or)  
GFortran ***  
...  
**Conflicts**  
GFortran 4.2.1 compiles might fail - msys appears to be installed and is in the PATH |
<table>
<thead>
<tr>
<th>Note</th>
<th>Warning</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>This issue is applicable also to GFortran 4.6.2</td>
<td><strong>msys appears to be installed and is in the PATH, which can cause compiles using GFortran 4.2.1 to fail.</strong> To resolve this issue, you need to: 1. use this utility to remove msys from the PATH, or 2. rename the msys installation folder when using GFortran 4.2.1, or 3. uninstall the msys application.</td>
<td><strong>Refer to Section 7.64 for further details.</strong></td>
</tr>
</tbody>
</table>

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.

   ```
   Warning

   Your IPv4 localhost IP address is not configured, or is not configured as expected, which can cause PSCAD to encounter errors when using legacy lock-based licensing, or when launching EMTDC runs.

   To resolve this issue, you need to:
   1. ensure that the IPv4 localhost IP is configured to be ‘127.0.0.1’
   ```

   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.

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 enabled, 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:

   ```
   Confirm Registry Action

   A required network protocol, TLS 1.0 for clients, appears to be disabled on this machine.

   As a result:
   - the MyUpdater will not be able to display your authorized products
   - PSCAD Free, PSCAD v4.5.4 and better, will not be able to use certificate licensing

   To resolve this issue, this utility needs to create/update the following registry value:
   - Folder: HKEY_LOCAL_MACHINE (Default registry)
   - Key: SYSTEM\CurrentControlSet\Control\SecurityProviders\SCHANNEL\Protocols\TLS 1.0\Client
   - Name: Enabled
   - Value: 1

   Select OK to enable the TLS 1.0 protocol for clients on this machine
   Select Cancel to leave the registry value unmodified
   ```
44. **Folder Info**

<table>
<thead>
<tr>
<th>Conflicts</th>
</tr>
</thead>
<tbody>
<tr>
<td>Windows is not generating short (8.3) pathnames</td>
</tr>
<tr>
<td>8.3 filename creating is disabled on all volumes</td>
</tr>
</tbody>
</table>

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.
# A.6 Functions Listed in the Medic

The following table lists the functions within the Medic:

<table>
<thead>
<tr>
<th>Function</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>1. PSCAD Configuration...</strong>&lt;br&gt;Installed PSCAD versions...&lt;br&gt;PSCAD X4 Release (4.6.X)&lt;br&gt;Configure EMTDC for Visual Studio 2015 and later</td>
<td>Refer to Section 7.36 for instructions on this configuration.&lt;br&gt;Note&lt;br&gt;Applicable to PSCAD v4.6.1 and later.</td>
</tr>
<tr>
<td><strong>2. PSCAD Configuration...</strong>&lt;br&gt;Installed PSCAD versions...&lt;br&gt;PSCAD X4 Release (4.6.X)&lt;br&gt;Configure EMTDC for Visual Studio 2010, 2013&lt;br&gt;Under a PSCAD installation heading (v4.6.1 and later)</td>
<td>Right-click on this function, and the following dialog displays:&lt;br&gt;If you proceed with this action, PSCAD will be configured to use Visual Studio 2010, 2012, or 2013.&lt;br&gt;Warnings&lt;br&gt;1. If you proceed with this action, PSCAD will no longer be configured for using Visual Studio 2015.&lt;br&gt;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.&lt;br&gt;Notes&lt;br&gt;1. This function does not install Visual Studio 2010, 2012, or 2013, it merely configures PSCAD to be able to use these versions.&lt;br&gt;2. Configuration with Visual Studio 2015 may be restored as per Section 7.36.</td>
</tr>
<tr>
<td><strong>3. Automation Library</strong>&lt;br&gt;Install folder: C:\Program Files (x86)\PSCAD\Automation&lt;br&gt;Publish date: YR.MO.DAY...</td>
<td>The BuildTime.txt file has been properly detected by the Fortran Medic.&lt;br&gt;(see Appendix A.5 Item 31 to see the text when this file is not detected)</td>
</tr>
<tr>
<td>Function</td>
<td>Action</td>
</tr>
<tr>
<td>----------</td>
<td>--------</td>
</tr>
<tr>
<td>4. <strong>Content moved to new manual “Certificate Licensing Issues” (see Section 1.2)</strong>&lt;br&gt;...</td>
<td>If the Medic detects that PSCAD 4.6.3 is installed, the medic will display a conflict that a newer version of the master library and EMTDC are available. (see left). Right-clicking on the conflict will display the notice shown below. The newer versions of the master library and EMTDC are available as a hot fix as of February 8, 2019. The hot fix may be requested from <a href="mailto:sales@pscad.com">sales@pscad.com</a>. Ensure to provide your PSCAD license number in your request. For details on the hot fix, please contact <a href="mailto:support@pscad.com">support@pscad.com</a>.</td>
</tr>
<tr>
<td>5. <strong>PSCAD X4 Release (4.6.3...)</strong>&lt;br&gt;...&lt;br&gt;<strong>Conflicts</strong>&lt;br&gt;► A newer version of the master library and EMTDC is available</td>
<td>If the Medic detects that PSCAD 4.6.3 is installed, the medic will display a conflict that a newer version of the master library and EMTDC are available (see left). Right-clicking on the conflict will display the notice shown below. The newer versions of the master library and EMTDC are available as a hot fix as of February 8, 2019. The hot fix may be requested from <a href="mailto:sales@pscad.com">sales@pscad.com</a>. Ensure to provide your PSCAD license number in your request. For details on the hot fix, please contact <a href="mailto:support@pscad.com">support@pscad.com</a>.</td>
</tr>
<tr>
<td>6. <strong>PSCAD X4 Release (4.6.3...)</strong>&lt;br&gt;...&lt;br&gt;► Dockable pane settings</td>
<td>The dockable pane default settings may be restored using the Medic. Close all instances of PSCAD, then in the Medic, right-click on “Dockable pane settings”, and select the option to restore the pane settings.</td>
</tr>
</tbody>
</table>
Appendix B  Using the Get Info Utility

B.1  Overview

The Get Info utility is used to detect system parameters for troubleshooting licensing issues. Some of the more important parameters include software and license installation, operating system, and user level.

This information is gathered and displayed in the utility, as well as saved to a file which may then be forwarded to our Support Desk to assist with troubleshooting.

Section 3.4 lists some of the common issues that are identified using this utility.

B.2  Running the Get Info Utility

The Get Info utility may be run as follows

- Ensure the PSCAD USB lock is plugged in (if applicable).
- Download “GetInfo32.zip’ from the following link:
  http://updater.pscad.com/utilities/GetInfo32.zip
- In the downloaded folder, open “GetInfo32.exe”.
- When prompted whether to test shortcuts, select “No”.
- The Get Info utility will gather information about your system and display it.

B.3  Using the Utility to Detect Licensing

The Get Info utility may be used to detect the license number associated with a USB lock and license database file installed on a machine. Proceed as follows:

- Run the utility as per Section B.2.
- Ensure the USB lock is securely plugged in (if applicable).
- Once the utility has finished running, scroll down to “Lock Info (direct access)”. If detected, the licensing information related to the license database file and/or USB lock will be displayed.
Appendix C  
Lock-based Legacy Licensing - License Manager Requirements

The following are requirements when using the Standalone License Manager:

- On the license host server, Ping Echo and Ping Reply must be enabled on the Firewall.
- The protocol for communication between the server and clients is UDP/IP on port 2053 and 2054.
- The license database file must be installed on the server.
- The Sentinel USB lock must be plugged in on the server.
- PSCAD client machines must be able to send out ping requests and receive ping replies.
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 BATCHnnnn.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 BATCHnnnn.bat file created by GFortran typically looked like the following:

```bash
@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.
The files and folders currently used by PSCAD are as follows:

<table>
<thead>
<tr>
<th>Item</th>
<th>Type</th>
<th>Install time Permissions</th>
<th>Runtime Permissions</th>
<th>Path</th>
<th>State (Released versions)</th>
<th>State (Future)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Licensing log file</td>
<td>File</td>
<td>Write</td>
<td></td>
<td>C:\ProgramData\Manitoba HVDC Research Centre\LicenseManager\PscadLmgr.txt</td>
<td>Fixed</td>
<td>May possibly be reconfigured</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>C:\Users\Public\Documents\Manitoba HVDC Research Centre\LicenseManager\PscadLmgr.txt</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>C:\Users\USERID\AppData\Local\Manitoba HVDC Research Centre\LicenseManager\PscadLmgr.txt</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Certificate license (if used)</td>
<td>Folder</td>
<td>Read/Write</td>
<td></td>
<td>For PSCAD v4.5 and v4.6: C:\Users\Public\Documents\Manitoba HVDC Research Centre\Licensing\Licenses</td>
<td>Fixed</td>
<td>Fixed</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>For PSCAD v5: C:\Users&lt;user_id&gt;\AppData\Local\Manitoba Hydro International\Licensing\Licenses</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Legacy License (if self-licensing or License Manager is used)</td>
<td>File</td>
<td>Write</td>
<td>Read/Write</td>
<td>C:\Users\Public\Documents\Manitoba HVDC Research Centre\LicenseManager\lmgr-hvdc</td>
<td>Fixed</td>
<td>Fixed</td>
</tr>
<tr>
<td>User settings</td>
<td>File</td>
<td>Write</td>
<td>Read/Write</td>
<td>C:\Users\USERID\AppData\Local\Manitoba HVDC Research Centre\PSCAD\</td>
<td>Fixed</td>
<td>May possibly be reconfigured</td>
</tr>
<tr>
<td>Examples</td>
<td>Folder</td>
<td>Write</td>
<td>Read/Write</td>
<td>C:\Users\Public\Documents\PSCAD\4.6.2</td>
<td>Moveable by user</td>
<td>Moveable by the user</td>
</tr>
</tbody>
</table>
Appendix G  Testing Connectivity for Certificate Licensing

Content moved to new manual “Resolving Certificate Licensing Issues”
See Section 1.2
# DOCUMENT TRACKING

<table>
<thead>
<tr>
<th>Rev.</th>
<th>Description</th>
<th>Date</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Initial</td>
<td>17/Jul/2014</td>
</tr>
<tr>
<td>1</td>
<td>Added new sections (7.3, 8.1, 8.2, A.5); Updated sections (6.1, 10.1, A.4); Updated the document title</td>
<td>29 Jul/2014</td>
</tr>
<tr>
<td>2</td>
<td>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)</td>
<td>08/Aug/2014</td>
</tr>
<tr>
<td>3</td>
<td>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)</td>
<td>17/Oct/2014</td>
</tr>
<tr>
<td>4</td>
<td>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)</td>
<td>28/Nov/2014</td>
</tr>
<tr>
<td>5</td>
<td>Deleted Section 6.6 (duplicated); Added new Sections 6.6, 6.7, 7.17, 7.18, 7.19</td>
<td>28/Jan/2015</td>
</tr>
<tr>
<td>6</td>
<td>Update to Section 3.4, Section 9.2, Appendix A.5; Added new Section 7.20</td>
<td>20/Apr/2015</td>
</tr>
<tr>
<td>7</td>
<td>Added new Sections 2.4, 4.3, 9.5</td>
<td>20/May/2015</td>
</tr>
<tr>
<td>8</td>
<td>Update to Sections 4.1, 7.8; Added New Sections 2.5, 2.6, 7.21, 7.22</td>
<td>28/May/2015</td>
</tr>
<tr>
<td>9</td>
<td>Update to Section 7.8</td>
<td>07/Jul/2015</td>
</tr>
<tr>
<td>10</td>
<td>Added new Section 2.7, populated Section 3.3</td>
<td>28/Jul/2015</td>
</tr>
<tr>
<td>11</td>
<td>Added new Sections 5.1, 3.5, and 6.8</td>
<td>15/Sep/2015</td>
</tr>
<tr>
<td>12</td>
<td>Added new Cause #3 within Section 5.1; Added new Section 3.6; Update to Appendix A.5</td>
<td>22/Sep/2015</td>
</tr>
<tr>
<td>13</td>
<td>Added new Cause #3 within Section 7.3, Added new Section 3.7 and new Appendix C</td>
<td>24/Sep/2015</td>
</tr>
<tr>
<td>14</td>
<td>Added new Sections 3.8, 7.23, 7.24, and 7.25; Update to Section 3.6</td>
<td>02/Nov/2015</td>
</tr>
<tr>
<td>15</td>
<td>Added new Section 2.8; Update to Sections 2.7, 3.4 and Appendix C</td>
<td>13/Nov/2015</td>
</tr>
<tr>
<td>Rev.</td>
<td>Description</td>
<td>Date</td>
</tr>
<tr>
<td>------</td>
<td>-----------------------------------------------------------------------------</td>
<td>------------</td>
</tr>
<tr>
<td>16</td>
<td>Added new Item 10 to Appendix A.5; Added new Section 7.26 and new Appendix D; Updated to Sections 2.8 and 3.8</td>
<td>12/Jan/2016</td>
</tr>
<tr>
<td>17</td>
<td>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</td>
<td>25/Jan/2016</td>
</tr>
<tr>
<td>18</td>
<td>Update to Section 1.1; New Section 7.27; New Item 11 in Appendix A.5</td>
<td>04/Feb/2016</td>
</tr>
<tr>
<td>19</td>
<td>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</td>
<td>29/Feb/2016</td>
</tr>
<tr>
<td>20</td>
<td>Added new Cause 2 to Section 3.1; Added new Problem 2 to Section 7.15; Added new Section 7.29;</td>
<td>31/Mar/2016</td>
</tr>
<tr>
<td>21</td>
<td>Added new Sections 3.9, 3.10, 5.2, 7.30 and 7.31</td>
<td>5/Apr/2016</td>
</tr>
<tr>
<td>22</td>
<td>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)</td>
<td>27/Apr/2016</td>
</tr>
<tr>
<td>23</td>
<td>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</td>
<td>18/May/2016</td>
</tr>
<tr>
<td>24</td>
<td>Added new Sections 2.10, 4.5, 6.11, 7.34, 7.35; Added new Item 19 in Appendix A.5</td>
<td>23/June/2016</td>
</tr>
<tr>
<td>25</td>
<td>Added new Section 7.36; Updated Section 3.9</td>
<td>04/Nov/2016</td>
</tr>
<tr>
<td>26</td>
<td>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</td>
<td>30/Nov/2016</td>
</tr>
<tr>
<td>27</td>
<td>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</td>
<td>14/Dec/2016</td>
</tr>
<tr>
<td>Rev.</td>
<td>Description</td>
<td>Date</td>
</tr>
<tr>
<td>------</td>
<td>-----------------------------------------------------------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>28</td>
<td>Update to Section 7.8 and Appendix D</td>
<td>26/Jan/2017</td>
</tr>
<tr>
<td></td>
<td>Added Section 4.7, 4.8, Problem #2 to Section 7.27, Section 8.4, and Appendix E</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Added Item 23 to Appendix A.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Added new Appendix A.6, Items 1 and 2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Updated weblinks</td>
<td></td>
</tr>
<tr>
<td>29</td>
<td>Added Sections 6.15, 6.16, 7.39, 8.5, and Appendix A.5 Item 24</td>
<td>14/Mar/2017</td>
</tr>
<tr>
<td></td>
<td>Added Solution 3 to Section 7.26, Added Solution 1 to Section 7.36</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Updated Appendix A.6 Items 1 and 2</td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>Added Sections 3.12, 4.9, 8.6</td>
<td>03/Apr/2017</td>
</tr>
<tr>
<td></td>
<td>Update to Sections 6.12, 6.16 and 7.5</td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>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</td>
<td>31/May/2017</td>
</tr>
<tr>
<td>32</td>
<td>Added Sections 4.10 and 4.11</td>
<td>04/Jul/2017</td>
</tr>
<tr>
<td></td>
<td>Added Item 25 in Appendix A.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Added a second solution to Solution 1 in Section 7.10</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td>Added Sections 3.15, 6.17, 6.18, 7.41, and 7.42</td>
<td>03/Aug/2017</td>
</tr>
<tr>
<td></td>
<td>Added Solution 2 to Problem 1, Section 7.27</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Corrections to Solution 2 of Section 7.36</td>
<td></td>
</tr>
<tr>
<td>34</td>
<td>Update to Section 7.2 Solution 1, Update to Section 7.30</td>
<td>31/Aug/2017</td>
</tr>
<tr>
<td></td>
<td>Update to Section 7.36 “Cause”</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Added Sections 7.43 and 7.44</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>Added Sections 5.3, 7.45 and 8.7, and Appendix F</td>
<td>07/Sep/2017</td>
</tr>
<tr>
<td></td>
<td>Added Solution 3 to Section 6.6</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Added Solution 26 to Appendix A.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Update to Sections 7.39, 8.5 and 8.6</td>
<td></td>
</tr>
<tr>
<td>36</td>
<td>Added Sections 3.16, 3.17, 7.46, 7.47, 7.48, and 7.49</td>
<td>21/Sep/2017</td>
</tr>
<tr>
<td></td>
<td>Update to Section 11.1 and Appendix F</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Added Items 27 and 28 in Appendix A.5</td>
<td></td>
</tr>
<tr>
<td>37</td>
<td>Update to Section 6.9 (Solution 3), Section 9.5, and Appendix A.1</td>
<td>08/Nov/2017</td>
</tr>
<tr>
<td></td>
<td>Deleted Section 10 “Issues with Intel Fortran Licensing” (material moved to manual “Setting up the Intel Fortran Compiler”)</td>
<td></td>
</tr>
<tr>
<td>38</td>
<td>Added Sections 2.12, 4.12 (Cause 1), and 4.13</td>
<td>28/Nov/2017</td>
</tr>
<tr>
<td></td>
<td>Added Item 29, Appendix A.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Moved Solution 3 (Section 5.1) to Solution 2 (Section 4.12)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Renumbered Solution 4 to Solution 3 in Section 5.1</td>
<td></td>
</tr>
<tr>
<td>39</td>
<td>Added Sections 3.18, 4.14, 7.50, 7.51 and 7.52</td>
<td>03/Jan/2018</td>
</tr>
<tr>
<td></td>
<td>Added Item 30, Appendix A.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Update to Section 3.17</td>
<td></td>
</tr>
<tr>
<td>40</td>
<td>Added Sections 7.53 and 11.1</td>
<td>04/Jan/2018</td>
</tr>
<tr>
<td></td>
<td>Update to Section 3.18</td>
<td></td>
</tr>
<tr>
<td>41</td>
<td>Added Appendix A.5 Item 31, and Appendix A.6 Item 3</td>
<td>07/Feb/2018</td>
</tr>
<tr>
<td></td>
<td>Added Sections 2.13, 6.19, 7.54, and 7.55</td>
<td></td>
</tr>
<tr>
<td>Rev.</td>
<td>Description</td>
<td>Date</td>
</tr>
<tr>
<td>------</td>
<td>-------------</td>
<td>------------</td>
</tr>
</tbody>
</table>
| 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 |
<table>
<thead>
<tr>
<th>Rev.</th>
<th>Description</th>
<th>Date</th>
</tr>
</thead>
<tbody>
<tr>
<td>60</td>
<td>Update to Section 7.31 title; Added Section 7.68; Correction to Section 7.67</td>
<td>22/May/2019</td>
</tr>
<tr>
<td>61</td>
<td>Added Section 10.3; Added Solution 2 to Section 7.65; Added Item 42 to Appendix A.6</td>
<td>06/Jun/2019</td>
</tr>
<tr>
<td>62</td>
<td>Update to Item 42, Appendix A.5; Added Sections 3.20, 7.69 and 7.70</td>
<td>20/Aug/2019</td>
</tr>
<tr>
<td>63</td>
<td>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</td>
<td>28/Oct/2019</td>
</tr>
<tr>
<td>64</td>
<td>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</td>
<td>25/Nov/2019</td>
</tr>
</tbody>
</table>