

See discussions, stats, and author profiles for this publication at: <https://www.researchgate.net/publication/313924098>

# Linking ABAQUS 2017 and Intel Parallel Studio XE2016 (Visual Fortran) in Windows 10 x64

Technical Report · February 2017

DOI: 10.13140/RG.2.2.14625.40806

---

CITATIONS

0

READS

30,095

1 author:



Petri Tanska

University of Eastern Finland

42 PUBLICATIONS 262 CITATIONS

SEE PROFILE

Some of the authors of this publication are also working on these related projects:



Adaptation of articular cartilage and subchondral bone to abnormal loading in knee joints [View project](#)

## Linking ABAQUS 2017 and Intel Parallel Studio XE2016 (Visual Fortran) in Windows 10 x64

Petri Tanska, Ph.D.

Department of Applied Physics, University of Eastern Finland, Kuopio, Finland.

Contact: Petri.Tanska(-at-)uef.fi

This brief manual describes how to get FORTRAN linking and user subroutines working in Abaqus2017. These instructions have been tested with the following software combination (Microsoft community Visual Studio 2013 + ABAQUS 2017 + Intel Parallel Studio XE 2016 update 4). These instructions may work with other versions but you will need check that Intel Parallel Studio can be integrated to MS Visual Studio and that Intel Parallel Studio does not override MS MPI version coming with Abaqus installation. **Please note that this guide assumes that you have access to administrator rights in Windows 10.** Contact me if you have additional suggestions and I will try keep this up to date.

### Install the following software using the following order

1. Microsoft community Visual Studio 2013
  - free, available at <https://www.visualstudio.com/en-us/news/releasenotes/vs2013-community-vs>
2. Intel parallel studio XE 2016 update 4 (includes Intel Visual Fortran and C++)
  - Cluster Edition is free for students (requires registration), available at <https://software.intel.com/en-us/qualify-for-free-software/student>
  - Note that Cluster Edition includes lot of extra software, for these purposes only C++ and FORTRAN compilers are needed (however, in this guide the default installation option is assumed)
  - Ensure that Parallel Studio is linked to Visual Studio 2013 during installation (default, but double check this if you use custom installation option)
3. Abaqus 2017 solvers and CAE environment

Abaqus installation will check FORTRAN and user subroutine linking if it can find the path for FORTRAN compiler. If you are following this guide the path is not set up yet.

### Setting the environmental variable path for Visual Studio and FORTRAN compiler

1. Navigate to the folder containing the visual studio .bat file (vcvars64.bat). By default it is C:\Program Files (x86)\Microsoft Visual Studio 12.0\VC\bin\amd64
2. Navigate to the folder containing the FORTRAN compiler .bat file (ifortvars.bat). By default it is C:\Program Files (x86)\IntelSWTools\compilers\_and\_libraries\_2016.4.246\windows\bin
3. Go to Control Panel -> System -> Advanced System Information -> Advanced tab -> Environment Variables (requires administrator rights)

- From the System Variables section, locate variable PATH and press edit button. In the edit Environment Variable window, click edit and add paths containing vcvars64.bat and ifortvars.bat. Now you have set up system environmental variables for Visual Studio and FORTRAN compiler.

### Configure Abaqus CAE and Abaqus command line to load Fortran compiler by default.

1. Navigate to location C:\ProgramData\Microsoft\Windows\Start Menu\Programs\Dassault Systemes SIMULIA Abaqus CAE 2017 or use search to find Abaqus command and Abaqus CAE shortcuts.
2. Right click Abaqus Command shortcut -> properties -> target:
  - a. Target should read now: C:\WINDOWS\system32\cmd.exe /k
  - b. Edit target to read:

```
"C:\Program Files
(x86)\IntelSWTools\compilers_and_libraries_2016.4.246\windows\bin\ifortvars.bat"
intel64 vs2013 & C:\WINDOWS\system32\cmd.exe /k
```

(note spaces, quotes, &-mark, and intel64 and vs2013 commands)

- c. Optional: you can define default working directory for Abaqus command prompt by adding the desired path to target after command "/k". For example, adding the following command

```
"C:\Program Files
(x86)\IntelSWTools\compilers_and_libraries_2016.4.246\windows\bin\ifortvars.bat"
intel64 vs2013 & C:\WINDOWS\system32\cmd.exe /k C:\users\your_user_name
```

Changes the default directory to C:\users\your\_user\_name

3. Right click Abaqus CAE shortcut -> properties -> target:
  - a. Target should read now:  
C:\SIMULIA\CAE\2017\win\_b64\resources\install\cae\launcher.bat cae || pause
  - b. Edit target to read:

```
"C:\Program Files
(x86)\IntelSWTools\compilers_and_libraries_2016.4.246\windows\bin\ifortvars.bat"
intel64 vs2013 & C:\SIMULIA\CAE\2017\win_b64\resources\install\cae\launcher.bat
cae || pause
```

(again, note spaces, quotes, &-mark, and intel64 and vs2013 commands)

Now everything is set up. However, the installation requires administrator rights meaning that the link between FORTRAN and Abaqus does not work correctly if you do not have administrator privileges on your user account.

*To fix this (administrator rights required): 1) run Abaqus CAE or Abaqus command prompt in "run as administrator" mode or 2) set Abaqus CAE or Abaqus command prompt shortcuts to run always in Administrator mode from shortcut properties-> shortcut tab -> advanced -> tick "run as administrator".*

### Verify that Abaqus is able to locate Fortran compiler.

In the Abaqus command prompt type (without quotes): "abaqus info=system"

This should now find FORTRAN compiler and c++ linker indicated as red markings in the figure 1 (note that these should show up even though you do not run abaqus command in “run administrator mode”). You should get output close to the figure 1.

```

Administrator: Intel Compiler 16.0 Update 4 Intel(R) 64 Visual Studio 2013
Intel(R) MPI Library 5.1 Update 3 for Windows* Target: Build Environment for Intel(R) 64 applications
Copyright (C) 2007-2015 Intel Corporation. All rights reserved.
Copyright (C) 2005-2016 Intel Corporation. All rights reserved.
Intel(R) Compiler 16.0 Update 4 (package 246)

C:\WINDOWS\system32>abaqus info=system
Abaqus 2017
Abaqus 2017
Abaqus Site ID:
Abaqus is located in the directory C:\SIMULIA\CAE\2017 C:\SIMULIA\CAE\2017\win_b64 C:\SIMULIA\CAE\2017\win_b64\code C:\SIMULIA\CAE\2017\win_b64\code\bin C:\SIMULIA\CAE\2017\win_b64\code\bin\SPMExternal C:\SIMULIA\CAE\2017\win_b64\code\resources C:\SIMULIA\CAE\2017\win_b64\SPM C:\Program Files\Dassault Systemes\SimulationServices\VR2017\win_b64
Abaqus solver stack is located in the directory C:\Program Files\Dassault Systemes\SimulationServices\VR2017\win_b64
Sequence Information:
C:\Program Files\Dassault Systemes\SimulationServices\VR2017\win_b64: 2016_09_28-00:54:59 126836
C:\SIMULIA\CAE\2017\win_b64: 2016_09_28-00:54:59 126836
The Abaqus information files are located in the directory C:\SIMULIA\CAE\2017\win_b64\SPM\Info

Running system configuration checks for Windows/64-bit.
Please wait until all necessary information has been gathered...

Current system configuration is:
Date: Thu Feb 23 13:07:15 2017
Hostname:
Username:
System Host ID:
DAS Target ID:
IP Address:
Processor: Intel(R) Core(TM) i5-6200U CPU @ 2.30GHz
Number of CPUs: 4
Physical Memory: 16288 MB
Virtual Memory: 14314 Available / 20641 Total Bytes
OS Version: Windows 10 Education (Build 15033)
++ compiler: Microsoft Visual C++ 12.0.31101.0
Linker Version: Microsoft (R) Incremental Linker Version 12.00.31101.0
Fortran Compiler: Intel Fortran Compiler 16.0
MPI:
Browser: Firefox 51.0.1.0

Graphics Details:
Device Name: Intel(R) HD Graphics 520
Video Processor: Intel(R) HD Graphics Family
Adapter RAM: 1824 MB
Driver Date: 09/22/2016
Driver Version: 20.19.15.4663
State Flags: PRIMARY_DEVICE
Horizontal Res: 1536 pixels in 518 mm -> 2,965251 pix/mm
Vertical Res: 960 pixels in 324 mm -> 2,962963 pix/mm
Display Settings: 1520 x 1200 @ 60 - 32-bit - 60 Hz

Device Name: Intel(R) HD Graphics 520
Video Processor: Intel(R) HD Graphics Family
Adapter RAM: 1824 MB
  
```

**Figure 1:** Typical output for the command "abaqus info=system". Output indicates that FORTRAN compiler and linker are found.

**Ensure that Abaqus is linked correctly and subroutines can be compiled correctly.**

Run subroutine verification in the Abaqus command prompt by typing (without quotes):  
 “abaqus verify –user\_std” (Abaqus Standard subroutines) or  
 “abaqus verify –user\_exp” (Abaqus Explicit subroutines) or  
 “abaqus verify –all” (All Abaqus verifications)

If the output shows like in Figure 2 below you have successfully linked Abaqus and FORTRAN compiler. Enjoy.

```

Administrator: Intel Compiler 16.0 Update 4 Intel(R) 64 Visual Studio 2013
C:\Users\tanska>abaqus verify -user_std
Abaqus Product Install Verification...
Fri Feb 24 10:17:24 2017
Making C:\Users\tanska\verify. All verification files will reside in this
directory.
Abaqus/Standard with user subroutines
...PASS
Continuing...
Clean-up...
Verification directory C:\Users\tanska\verify has been deleted.
C:\Users\tanska>
  
```

**Figure 2:** Successful verification for the command "abaqus verify –user\_std”.