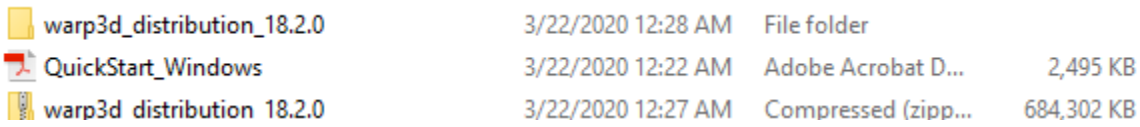
PART I: DOWNLOAD AND RUN WARP3D ON AN EXAMPLE PROBLEM

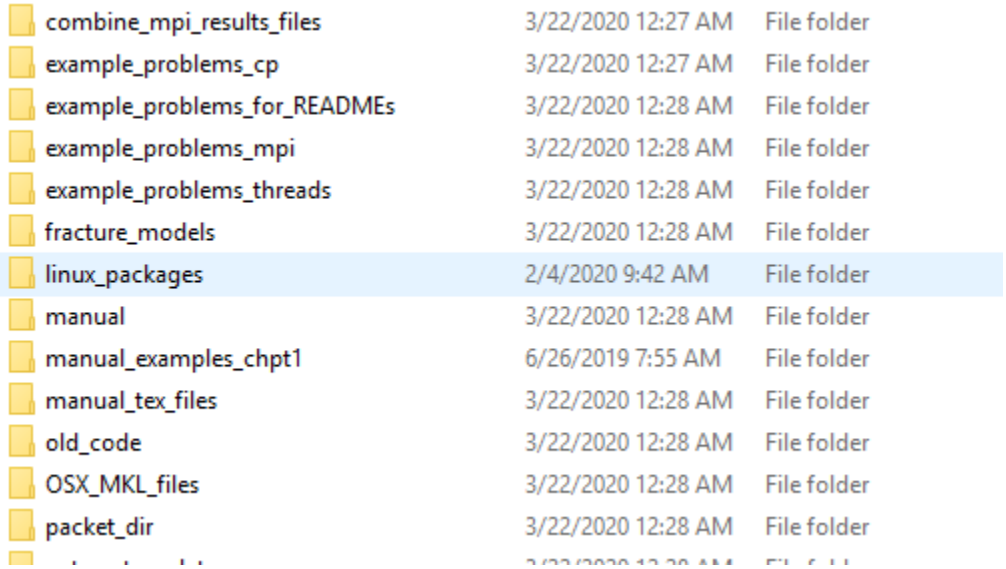
http://www.warp3d.net/downloads - There's a quick-start guide (QuickStart_Windows.pdf) in the link QuickStart: Windows Users

Download **warp3d_distribution_18.2.0.zip** (3/22/2020, 12:27 AM) – This is about 684 MB

- Extract the folder in the zip file into your preferred directory
- Know the path where you installed this folder – For me it looks like:



- Inside the first warp3d_distribution_18.2.0 folder are the following files



- The HOME of warp3d is the location of the above files which is (for me, as an example): C:\Sameer\ME531_TA\WARP3D\warp3d_distribution_18.2.0

Now following slide 5 in QuickStart_Windows.pdf

- Open command prompt window and enter the commands as given in the PDF
- I got the error for missing libiomp5md.dll (a needed library file if your PC doesn't have it) – This file is provided with the installation and a rename command is given for that ("move…")
- Then run the example problem test_18a.inp
- **All commands I entered from start to finish (ran the input script successfully) are given below:**

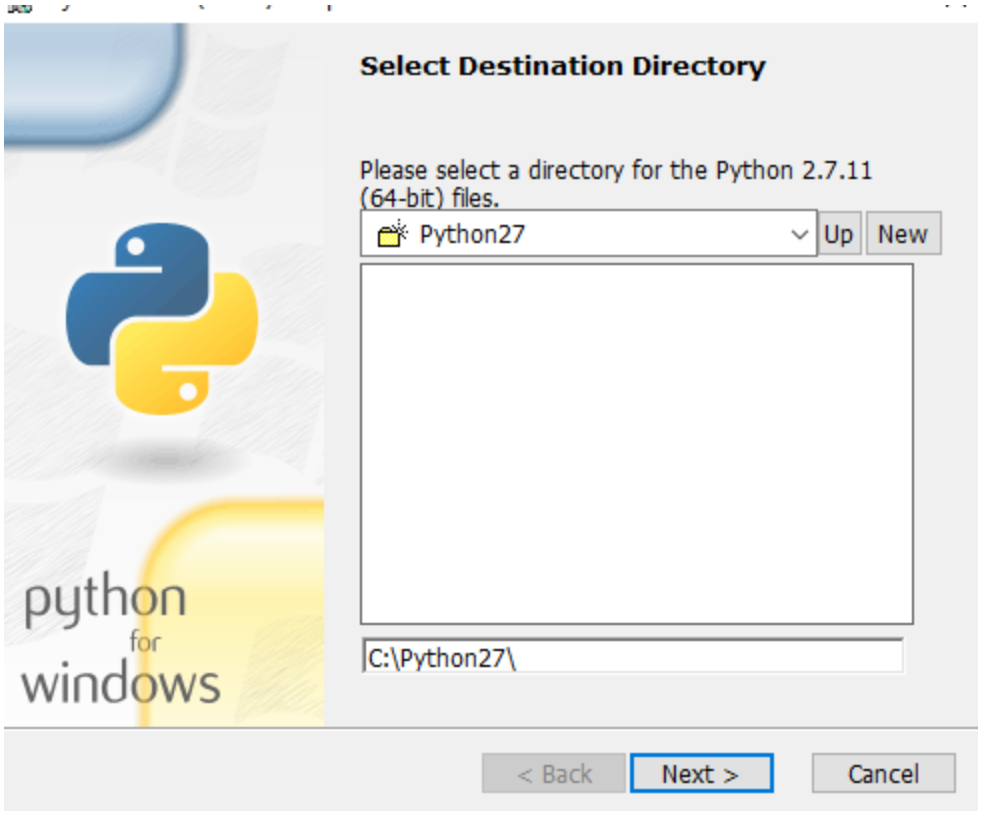
```
C:\Users\am14>set WARP3D_HOME="C:\Sameer\ME531_TA\WARP3D\warp3d_distribution_18.2.0"

C:\Users\am14>set warp="%WARP3D_HOME%\run_windows\warp3d.exe"

C:\Users\am14>%warp%

C:\Users\am14>cd %WARP3D_HOME%\run_windows

C:\Sameer\ME531_TA\WARP3D\warp3d_distribution_18.2.0\run_windows>move save_libiomp5md.dll libiomp5md.dll
        1 file(s) moved.

C:\Sameer\ME531_TA\WARP3D\warp3d_distribution_18.2.0\run_windows>cd %WARP3D_HOME%\example_problems_for_READMEs

C:\Sameer\ME531_TA\WARP3D\warp3d_distribution_18.2.0\example_problems_for_READMEs>set OMP_NUM_THREADS=4

C:\Sameer\ME531_TA\WARP3D\warp3d_distribution_18.2.0\example_problems_for_READMEs>set MKL_NUM_THREADS=4

C:\Sameer\ME531_TA\WARP3D\warp3d_distribution_18.2.0\example_problems_for_READMEs>%warp% < test_18a.inp > outa
```

- If you open the "outa" file with Notepad or another text editor you'll see that WARP3D was executed for the example problem
- You can close the command prompt window

PART II: DOWNLOAD AND INSTALL PYTHON 2.7.13 (WITH NUMPY AND SCIPY LIBRARIES)

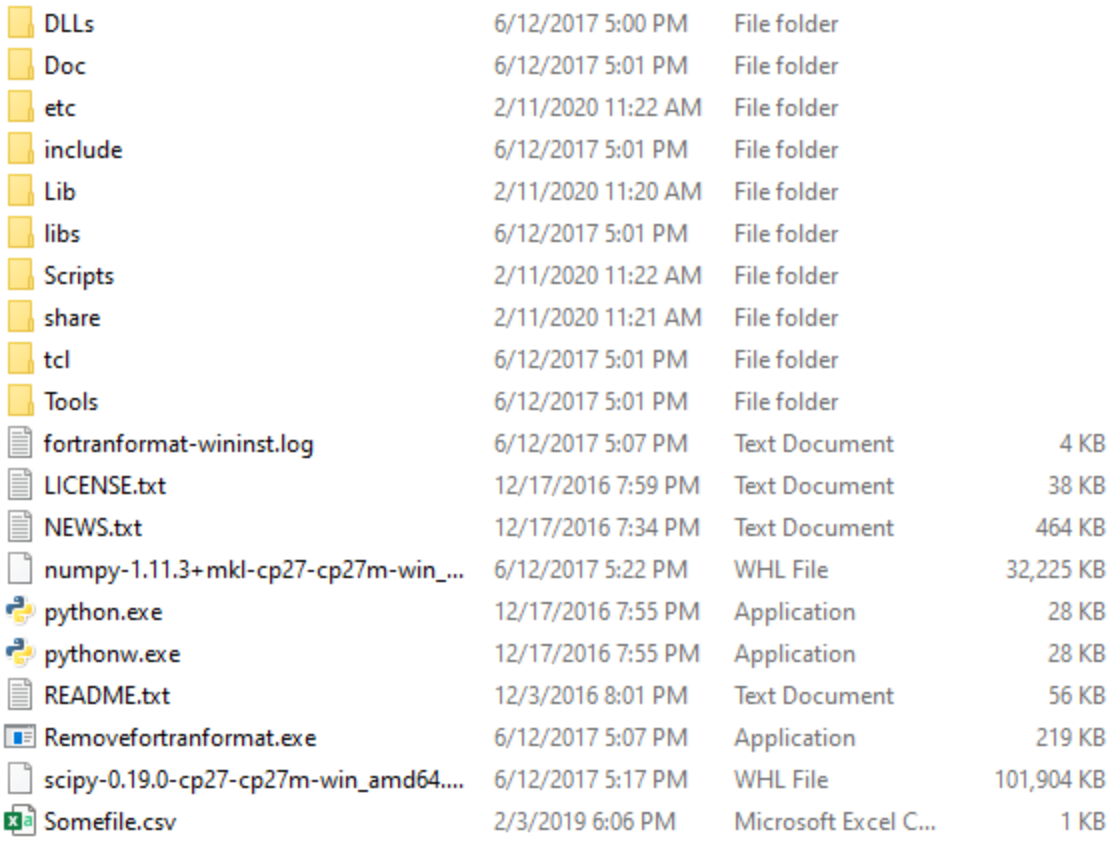
At this stage, WARP3D is running successfully. You need to set up some software to visualize your results

- You'll need Python 2.7.X (In this tutorial we work with Python 2.7.13)
- https://www.python.org/downloads/release/python-2713/ - Download the Windows x86-64 MSI Installer – It should have a filename like *python-2.7.13.amd64.msi*
- Run the Installer, it will ask you to specify where you want to install the folder



- I set the directory of installation as C:\Python\Python27

So my folder of installation looks like:



DLLs	6/12/2017 5:00 PM	File folder	
Doc	6/12/2017 5:01 PM	File folder	
etc	2/11/2020 11:22 AM	File folder	
include	6/12/2017 5:01 PM	File folder	
Lib	2/11/2020 11:20 AM	File folder	
libs	6/12/2017 5:01 PM	File folder	
Scripts	2/11/2020 11:22 AM	File folder	
share	2/11/2020 11:21 AM	File folder	
tcl	6/12/2017 5:01 PM	File folder	
Tools	6/12/2017 5:01 PM	File folder	
fortranformat-wininst.log	6/12/2017 5:07 PM	Text Document	4 KB
LICENSE.txt	12/17/2016 7:59 PM	Text Document	38 KB
NEWS.txt	12/17/2016 7:34 PM	Text Document	464 KB
numpy-1.11.3+mkl-cp27-cp27m-win_...	6/12/2017 5:22 PM	WHL File	32,225 KB
python.exe	12/17/2016 7:55 PM	Application	28 KB
pythonw.exe	12/17/2016 7:55 PM	Application	28 KB
README.txt	12/3/2016 8:01 PM	Text Document	56 KB
Removefortranformat.exe	6/12/2017 5:07 PM	Application	219 KB
scipy-0.19.0-cp27-cp27m-win_amd64....	6/12/2017 5:17 PM	WHL File	101,904 KB
Somefile.csv	2/3/2019 6:06 PM	Microsoft Excel C...	1 KB

- Now download the numpy and scipy files from the UofI Box link that was sent to you and move them to the same folder (you can see that I have them above)
- Now install numpy and scipy – The following commands should do it

(What are the commands doing? – The first two commands set environment variables which locate the *python* executable and the *pip* executable. The *pip* executable is used to install the packages)

```
C:\Python>cd Python27

C:\Python\Python27>set python27="C:\Python\Python27\python.exe"

C:\Python\Python27>set pip27="C:\Python\Python27\Scripts\pip.exe"

C:\Python\Python27>%pip27% install numpy-1.11.3+mkl-cp27-cp27m-win_amd64.whl
Requirement already satisfied: numpy==1.11.3+mkl from file:///C:/Python/Python27/numpy-1.11.3%2Bmkl-cp27-cp27m-win_amd64
.whl in c:\python\python27\lib\site-packages
You are using pip version 9.0.1, however version 20.0.2 is available.
You should consider upgrading via the 'python -m pip install --upgrade pip' command.

C:\Python\Python27>%pip27% install scipy-0.19.0-cp27-cp27m-win_amd64.whl
Requirement already satisfied: scipy==0.19.0 from file:///C:/Python/Python27/scipy-0.19.0-cp27-cp27m-win_amd64.whl in c:
\python\python27\lib\site-packages
Requirement already satisfied: numpy>=1.8.2 in c:\python\python27\lib\site-packages (from scipy==0.19.0)
You are using pip version 9.0.1, however version 20.0.2 is available.
You should consider upgrading via the 'python -m pip install --upgrade pip' command.

C:\Python\Python27>
```

(The output messages that you would see will be different – I had installed them before so when I re-ran the same command it says that "Requirement already satisfied")

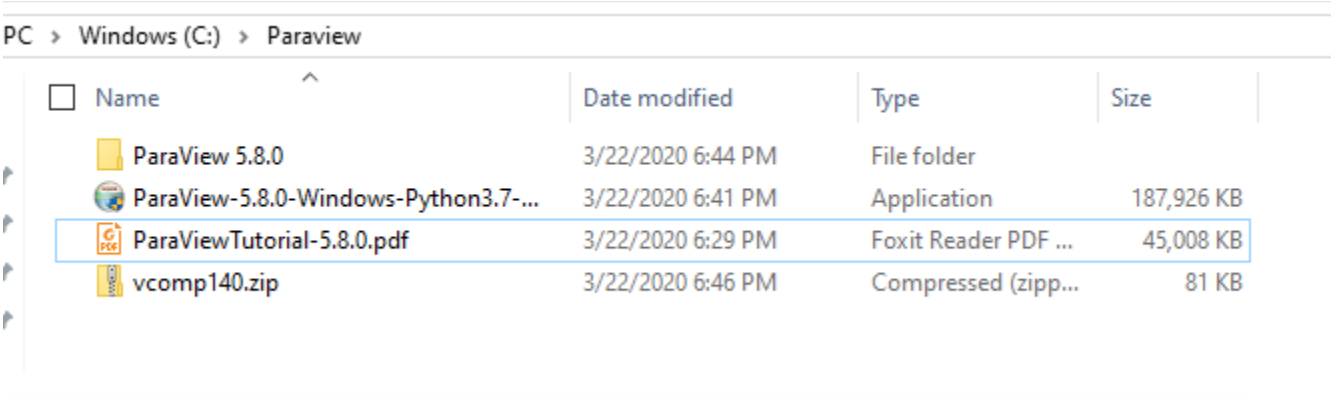
Now you have **python installed, with the numpy and scipy libraries**

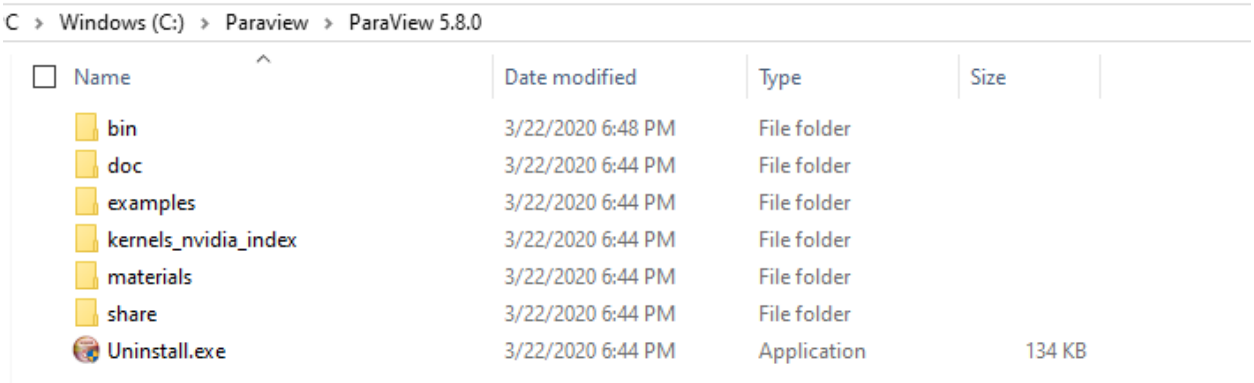
- You can close the command-prompt

PART III: DOWNLOAD AND INSTALL PARAVIEW

- Download the windows installer from https://www.paraview.org/download/ - ParaView-5.8.0-Windows-Python3.7-msvc2015-64bit.exe
- Another Useful file is the Paraview tutorial - ParaViewTutorial-5.8.0.pdf
- Move the installer to any preferred folder and double-click – Follow the instructions to install Paraview on your system

It looks like this for me:





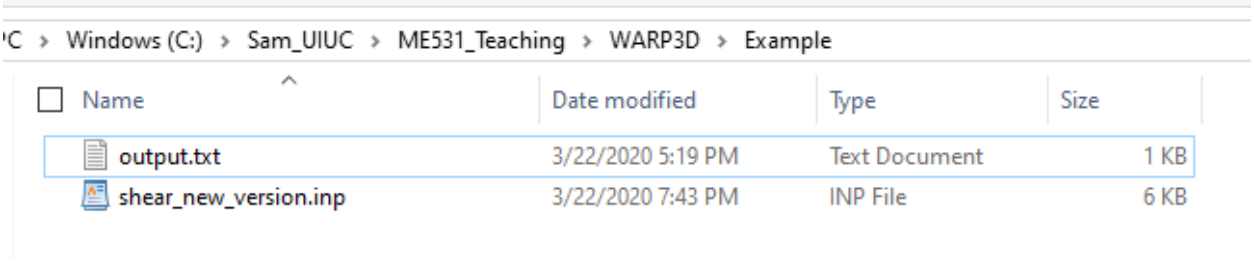
- Paraview was installed in C:\Paraview\ParaView 5.8.0\ in my system
- To open Paraview, go to the folder \ParaView 5.8.0\bin\
- Scroll down to find the executable – paraview.exe
- Double-click to open
- If Paraview opens, then you have it up and running, else there is likely an issue
- **An issue:** In my system, there was an error which popped up, saying that it needs a library file VCOMP140.dll
- I downloaded this file from https://www.dll-files.com/vcomp140.dll.html – And just FYI, this site is hosted by a third party, not Windows or Paraview
- Download this file and put it inside the bin folder

- Now open Paraview again, it should work

PART IV: RUN EXAMPLE SCRIPT AND VISUALIZE OUTPUT

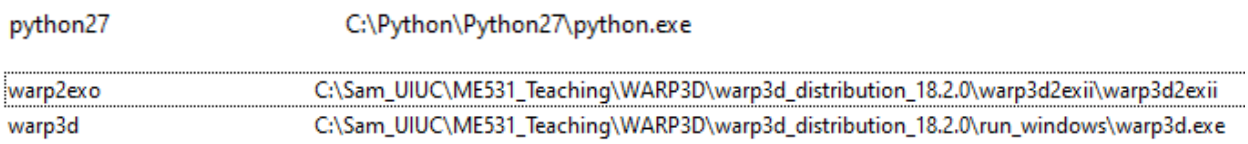
- An example script is provided in the UofI box link: shear_new_version.inp
- It has some *output* commands delegated to the file output.txt, so it should be in the same folder as this input script.
- Move both these files into a folder, just for convenience. When you run the input script, it will output lot of output files (like the CFG file in LAMMPS) inside this folder

For instance, I created an Example folder:



Create the following Environment Variables for your system:

- Right-Click on My computer
- Go to Properties
- Go to Advanced System Settings
- Go to Environment Variables
- In the System Variables
- Create "New Variables" – Enter a name and then Browse for the specific Executable file
  In this way make the following 3 variables



- Open command_prompt, use "cd" to go to the Example problem where you have the input script
- Then execute WARP3D using the following command

```
C:\Sam_UIUC\ME531_Teaching\WARP3D\Example>%warp3d% < shear_new_version.inp
```

(Several files will be generated in your folder – Need to convert it into a format which Paraview can understand)

Run the following commands to convert the output data into a "Exodus" File (.exo) which Paraview can read:

```
C:\Sam_UIUC\ME531_Teaching\WARP3D\Example>%python27% %warp2exo%


*** warp3d2exII version 2.0 (Python) ***

... Reads .text or .str flat results files. See Section 2.12 of WARP3D manual

=> Filename for Exodus II output: shear.exo
=> Path to flat file for model description: single_e_model_flat.text
=> Path to directory with results files: ./
=> Input time file with physical times for each step (y/n)? n
=> Write a large model Exodus II file [recommend no] (y/n)? y
```

This will create the “shear.exo” file which Paraview can read, to visualize the output