TUDUBLIN

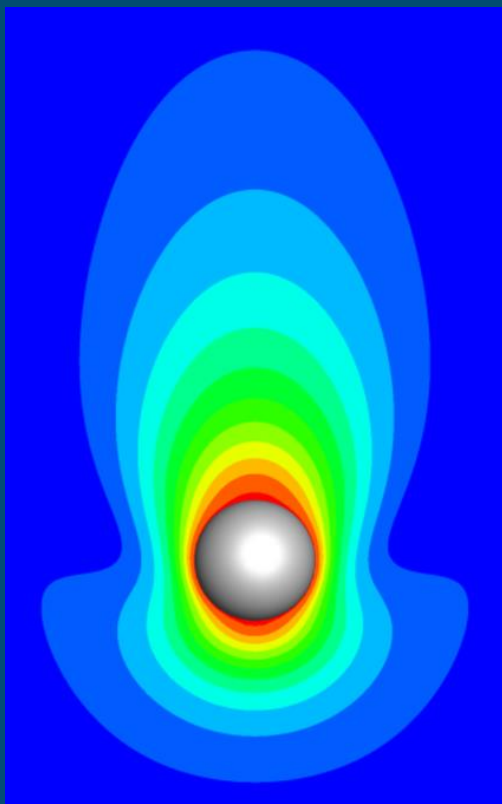*CFD Research Group Guide Series*

Running the Palabos Lattice Boltzmann CFD Package on Windows Guide

Author: Gerald Gallagher

Rev. 3

OLLSCOIL TEICNEOLAÍOCHTA
BHAILE ÁTHA CLIATH

CFD
RESEARCH GROUP

TU DUBLIN

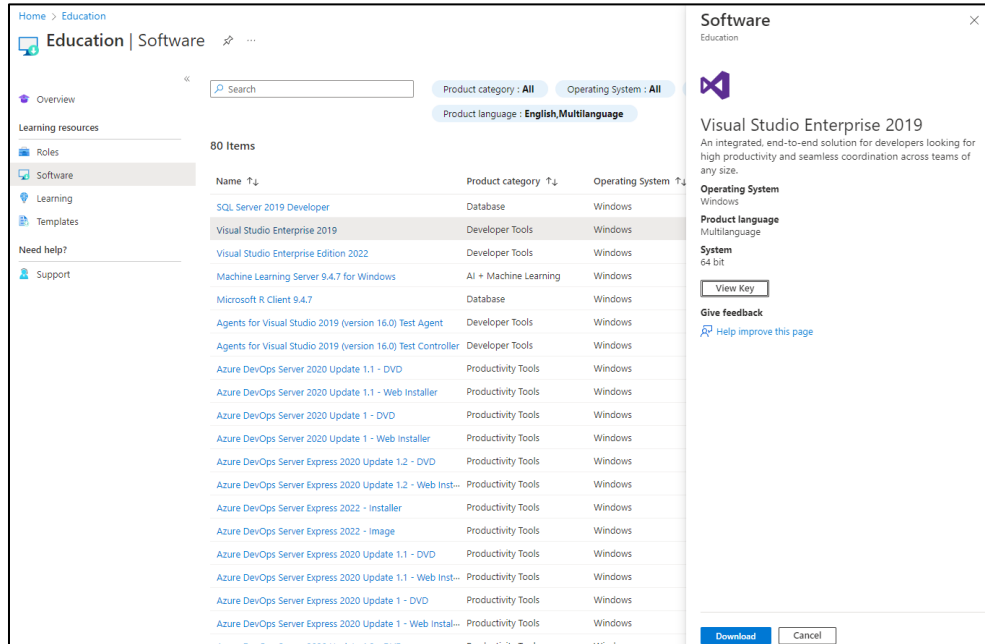TECHNOLOGICAL
UNIVERSITY DUBLIN

This guide contains information on how to set up and run the Palabos lattice Boltzmann CFD software package on Windows.

1. First, download the Palabos files by going to https://gitlab.com/unigespc/palabos and downloading a .zip file as shown below (it should download as palabos-master):
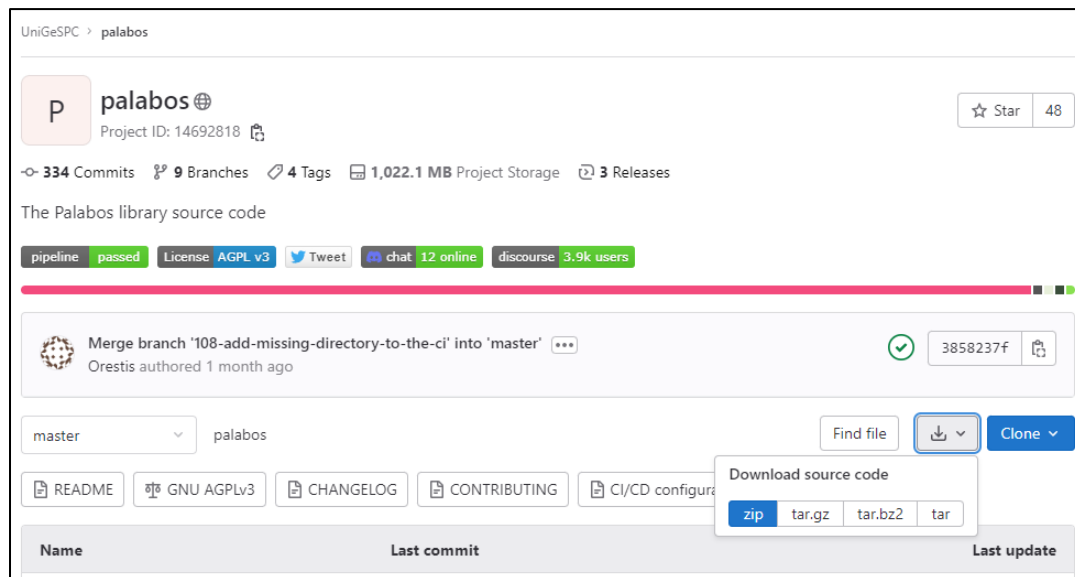
2. Unzip the folder to a destination of your choice

3. Go to https://azureforeducation.microsoft.com/devtools and sign in with your student ID and password

4. Click on "Software" on the left hand side and search for "Visual Studio Enterprise 2019". It might already be visible. Select it and then select "Download" in blue on the bottom right of the screen:

Running the Palabos Lattice Boltzmann CFD Package on Windows Guide

DUBLIN

OLLSCOIL TEICNEOLAÍOCHTA
BHAILE ÁTHA CLIATH

TECHNOLOGICAL
UNIVERSITY DUBLIN

CFD RESEARCH GROUP

Document: 1/1

Rev. 3

5. This will download a Visual Studio installation file (.exe) to your Downloads folder. Double click the file to start the installation process.

6. Make sure that the following is selected and click "Install" in the bottom right corner:
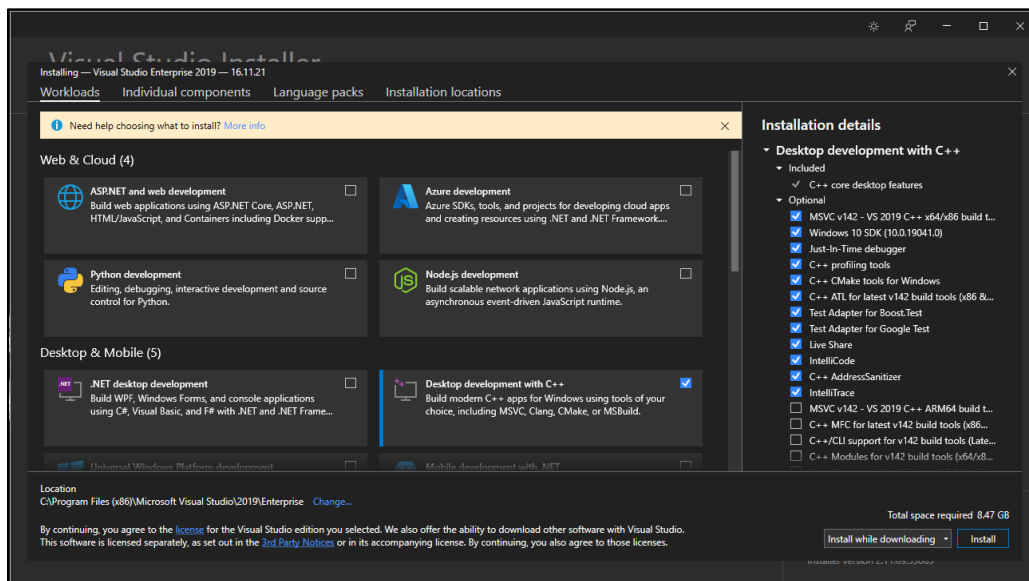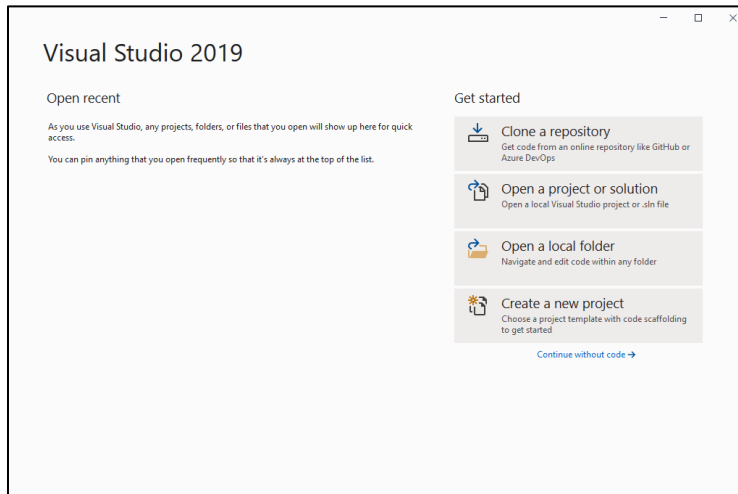
7. The installation should continue as shown below:
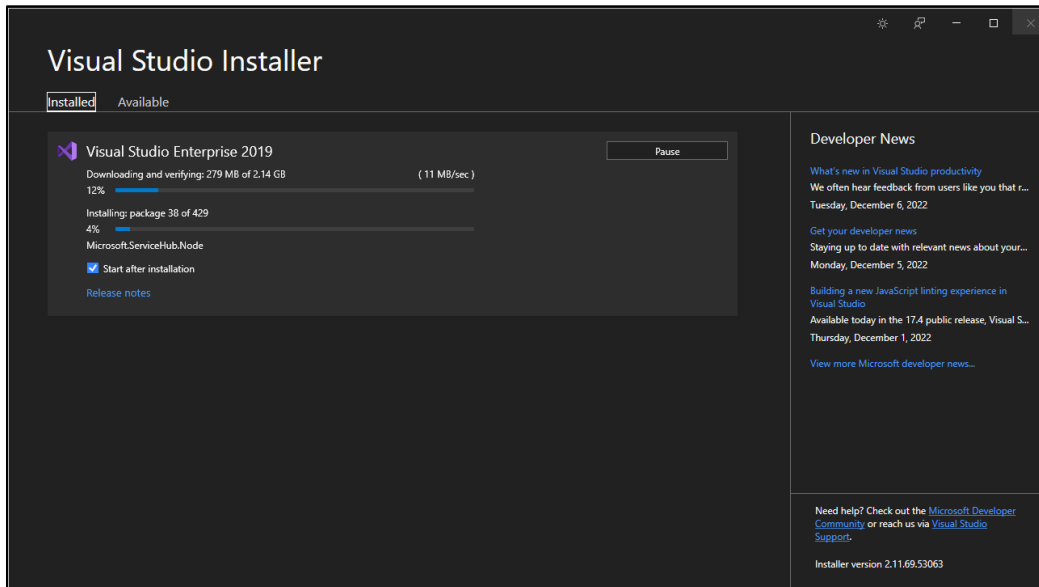
8. Skip creating a Visual Studio account (unless desired), make sure to select "Visual C++" under "Development Settings" and click "Start Visual Studio":
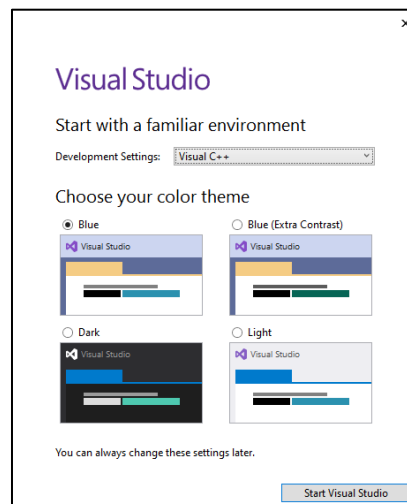

9. Click "Open a local folder" and select the downloaded "palabos-master" folder:

10. There will likely be error messages which is mentioned on the Palabos Gitlab website (https://gitlab.com/unigespc/palabos):

"You may get error messages in the "Output messages" window if MPI and/or HDF5 are not installed on your system. In these cases, deactivate MPI and/or HDF5 by selecting "Manage Configurations" in the "Configuration" button in the toolbar. In the section "CMake variables and cache", you should be able to un-click ENABLE_MPI and/or BUILD_HDF5 (you may need to wait a bit for these to appear). After this, click "Save and generate CMake cache to load variables" and proceed as indicated above."
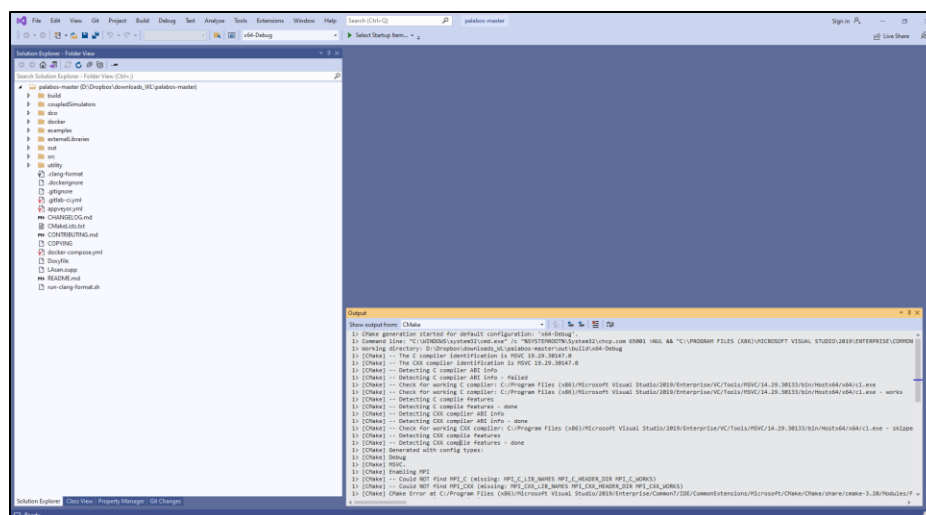
11. To remedy this, go to "Project" --> "CMake Cache (x64-Debug Only) --> "View CMakeCache.txt":
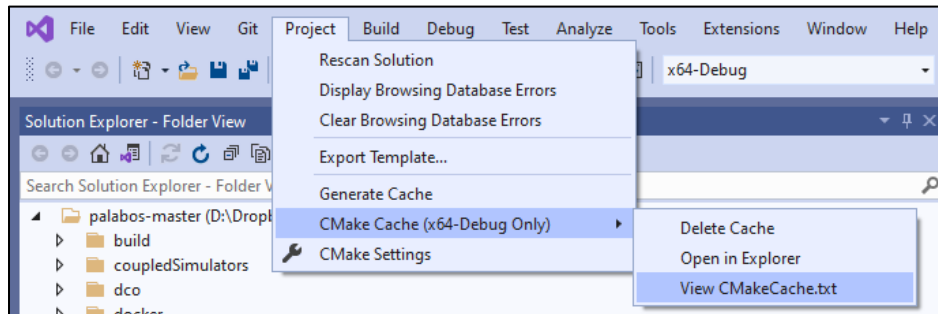
12. The file will open. First modify Line 202 as shown (ON to OFF) and save:
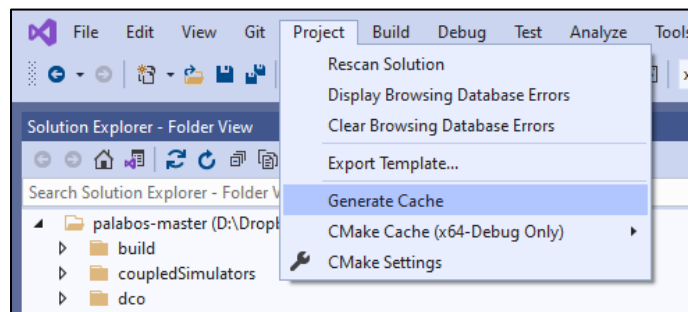
```
201    //Enable MPI
202    ENABLE_MPI:BOOL=ON
```
```
201    //Enable MPI
202    ENABLE_MPI:BOOL=OFF
```

13. Click "Generate Cache" as shown:

14. Modify Line 18 as shown (ON to OFF), save and click "Generate Cache" once more:
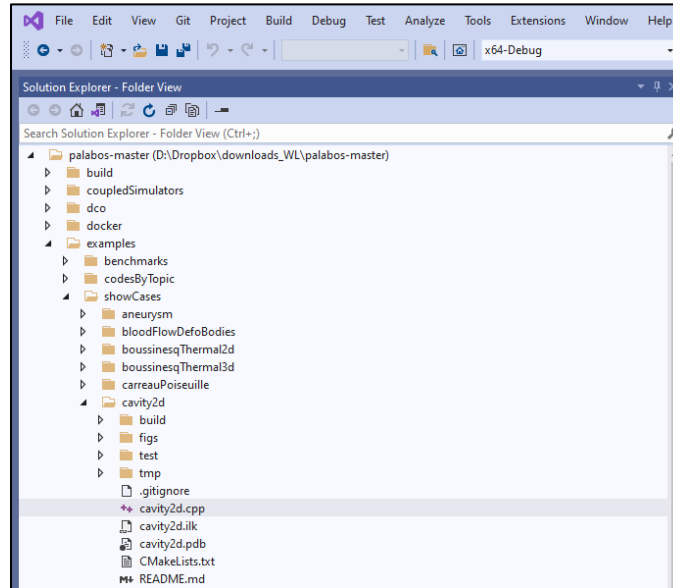
```
17    //compile with hdf5
18    BUILD_HDF5:BOOL=ON
```
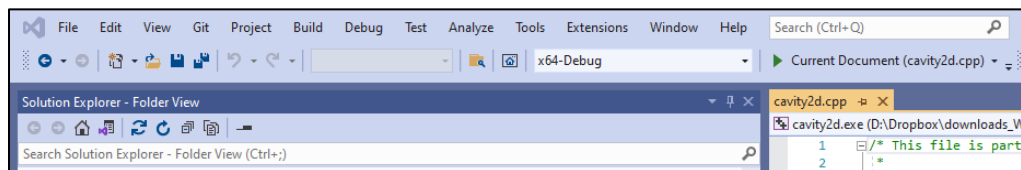```
17   //compile with hdf5
18   BUILD_HDF5:BOOL=OFF
```

Note that from now on "Generate Cache" in the menu will be replaced with "Configure palabos"
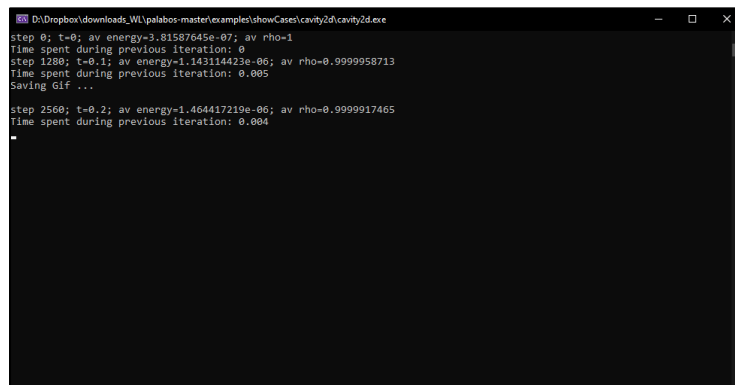
15. Find a test case in the tree like shown and double click on the .cpp file (cavity.cpp chosen here):

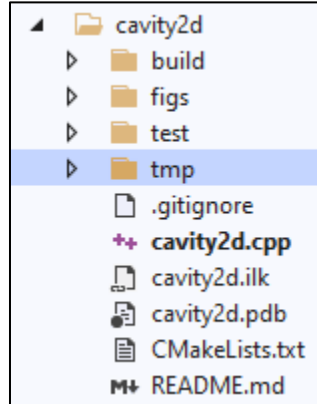16. Click on "Current Document (cavity2d.cpp)" underneath the search box.



17. A build process will begin and the simulation will run:



18. Any output files can be found in the "tmp" folder in the tree on the left:

**Note**: If the output files are not generated this way, double click the up-to-date .exe file in the cavity2d folder in Windows Explorer outside of Visual Studio to generate the files.

19. To create a new simulation, the easiest thing to do is to copy an existing simulation folder (cavity2d chosen in this case) and rename as desired.

20. Rename the .cpp file to be the same as the folder name e.g. mylbsimulation.cpp

21. Following that, open CMakelists.txt in the "palabos-master" main folder and add your simulation name as shown (example used is mylbsimulation on Line 204), then save:
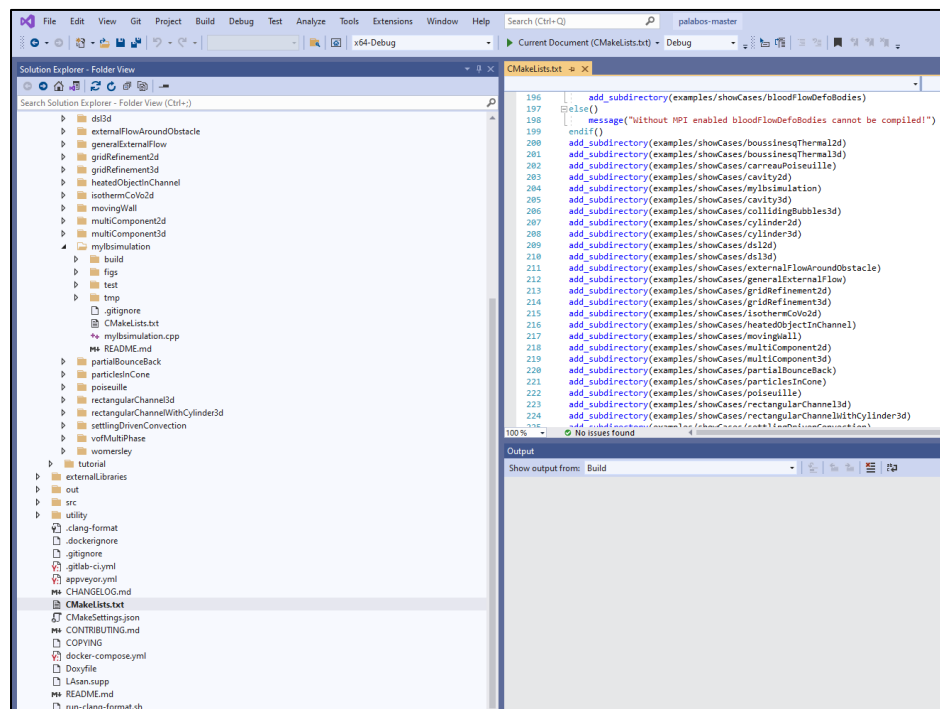
22. CMake should automatically refresh. Following that, modify set(EXECUTABLE_NAME "…") in the CMakeLists.txt file within the simulation folder to include the correct name (mylbsimulation used as an example again):
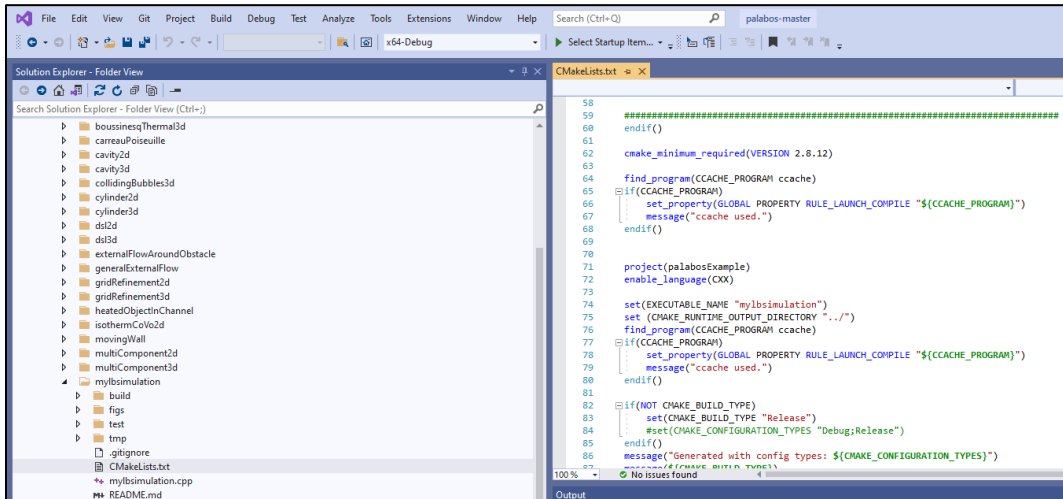


23. Open your new .cpp file, and click "Build" --> "Build Current Document":
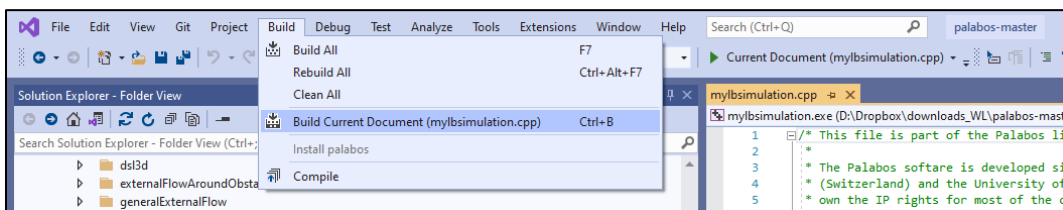


24. After that, you can run the simulation as done previously (click "Current Document (mylbsimulation.cpp) underneath the search box)