CWE Introduction (July 2018) Version 1.0.0

The CWE program is designed to facilitate the creation and execution of CFD wind simulations around buildings. Users to the program can provide geometries of structures, which the CWE program will use to create a mesh and simulation, to be run on TACC, via DesignSafe, using OpenFOAM.

This framework is centered around a template model. Currently there are two templates, 2D simulation of a rigid building geometry, using an unstructured mesh, and 3D simulation a rigid building geometry using an unstructured mesh. We hope to add more templates in the future, according to user interest.

The CWE program is currently in active development. We appreciate any constructive criticism given by our users and thank everyone for their patience as we discover and fix bugs.

Installation & Registration

Registration Step 1: Since this program runs via the DesignSafe platform, *all users of the CWE must have a DesignSafe account*. This account can easily be obtained via the DesignSafe website and is linked with TACC's account system.

Registration Step 2: Due to logistical limitations, early users of CWE will need to provide their DesignSafe username to <u>psempoli@alumni.nd.edu</u> for a second registration step. (We anticipate that this requirement will go away in a few months.) (DO NOT email the password. Just the username.)

Installation: The CWE program is contained in a self-extracting archive. Double-clicking the archive should begin the install process. Follow the on-screen prompts to proceed. The windows installer is called "SetupSimCenterCWE_Win_1_0_0 .exe". The Mac installer is called "SetupSimCenterCWE_Mac_1_0_0 .app". The linux installer is called "SetupSimCenterCWE_Linux_1_0_0".

SPECIAL NOTE FOR LINUX USERS:

You might have to set the run permission of the installer to allow running after downloading it. Once installed, run the script: "RunSimCenterCWE.sh" You might have to change the permissions of that script too. (We are assuming people using linux know how to change file permissions.) There is a bug in ubuntu 18.04 that interferes with SSL and QT. We are working to resolve this. Older versions of ubuntu should work.

SPECIAL NOTE FOR MAC USERS:

You will need to Control-Click on the installer to run it, since the package is not signed thru the Apple store.

SPECIAL NOTE FOR WINDOWS USERS:

1) The Windows version requires install of the Windows c++ runtime packages. Pretty much every Windows user should already have them. (We have yet to encounter a user who did not.) But, if the program complains that you do not, you can install that from: <u>https://support.microsoft.com/en-us/help/2977003/the-latest-supported-visual-c-downloads</u>

2) The CWE Windows version requires SSL libraries. Most Windows users will already have them. If you don't, on starting the program for first time, CWE will offer to retrieve the OpenSSL libraries

automatically. This process should work. If it does not, you can get the needed package from: <u>https://indy.fulgan.com/SSL/</u> or many other places. Simply copy the 2 dll files to the same folder as the CWE executable. Either way, restart the CWE program after getting the SSL libraries.

Logging In to CWE

Run the CWE program. Use your SimCenter username and password (probably also your TACC password) to login.

Getting Help

Once logged in, you can get help for using the CWE program under the "help" tab.

Creating a CFD case

Click on the Simulation Cases tab. Click the blue *Create a New Case* button. Select (1) a case template (2D Building Slice is a good first case), (2) A folder to contain the new case folder, (3) a new case name. Click "Create New Simulation"

Assigning Case Parameters

Once you have selected, or created a new case, you can click on the "Parameters" tab. This will allow setting various parameters for both the mesh and simulation, in the various tabs, based on the provided geometry. For your first case, we recommend using the provided default geometry, of "10x10x10 cube" to get the hang of the CWE program, before uploading a geometry of your own. Click the "Save all Parameters" button to save your parameters.

Running CFD Tasks

Click the "Mesh Parameters" sub-tab in the "paremeters" tab. If you have not already run the mesh generation, click "run". You can view task progress in the *Running Jobs* tab. Once the mesh is done, you can run the simulation in the same way.

Viewing Results

*C*lick the "results" tab. Select a result to view or download and click the view or download icon.

Using Custom Geometry Files

Once you are familiar with the basic CWE workflow, you can create and upload your own custom geometries for simulation. For creating geometry files, we recommend FreeCAD (its free). *Export your geometry for simulation as an <u>Alias Mesh .obj</u> file.* Once you have your geometry file, click on the *Files* tab. Use the upload and download buttons to move files to and from Design Safe. If you right click on a remote file, there are additional options. Once uploaded, in the parameters tab, a building geometry can be selected under "Mesh Parameters", "Building Geometry", "Uploaded File".

Known Issues

In situations where an Internet connection is unstable, parameter updates can fail ungracefully. If one is dropped from the parameter screen during a parameter update, restarting the program should allow one to continue working.