Assignment: Sensitivity of Wind Loads to Mesh Resolution

A. Objective

After completing this assignment, students will understand the sensitivity of wind loads calculated using a CFD simulation to mesh refinement. The inputs used for this assignment are the same as those used in **Tutorial 2**, except for slight modifications to the computational grid, and the rate results are sampled. The goal is to understand how mesh refinement impacts the accuracy of pressure coefficient calculation.

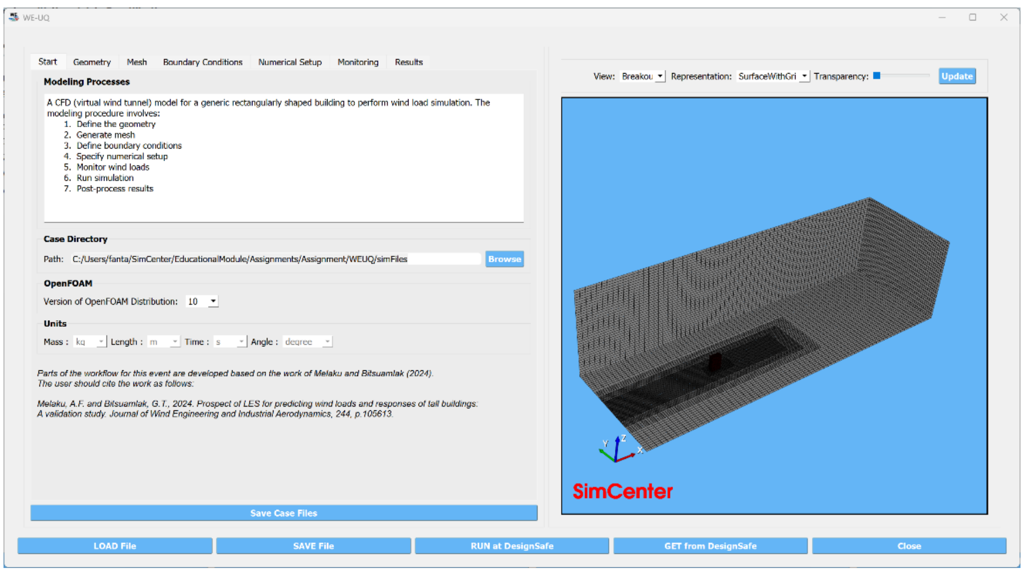
B. Steps to Follow

1. **Copy the model setup from Tutorial 2:**

   - **Step-1.1**: Please copy "`Tutorial2/WEUQ`" directory into a new location, e.g., "`Assignment/WEUQ`".

   - **Step-1.2:** Launch the WE-UQ tool and open **Tools-> "CFD - Isolated Building Wind Load Simulation"** from the menu bar.

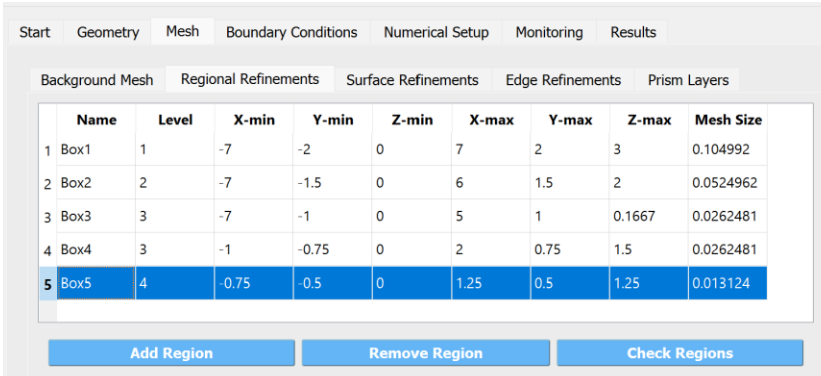
   - **Step-1.3**: Now, in the newly opened window, click "**LOAD File**" button and load "`Assignment/WEUQ/scInput.json`" file to the workflow. If the path in the "**Start**" tab does not automatically point to the new case directory, please change the "**Case Directory**" path to the new folder you set up, as shown in the following figure.



   - **Step-1.4**: Save the new JSON file by clicking the "**SAVE File**" button and locating it to the "`Assignment/WEUQ/scInput.json` " path.

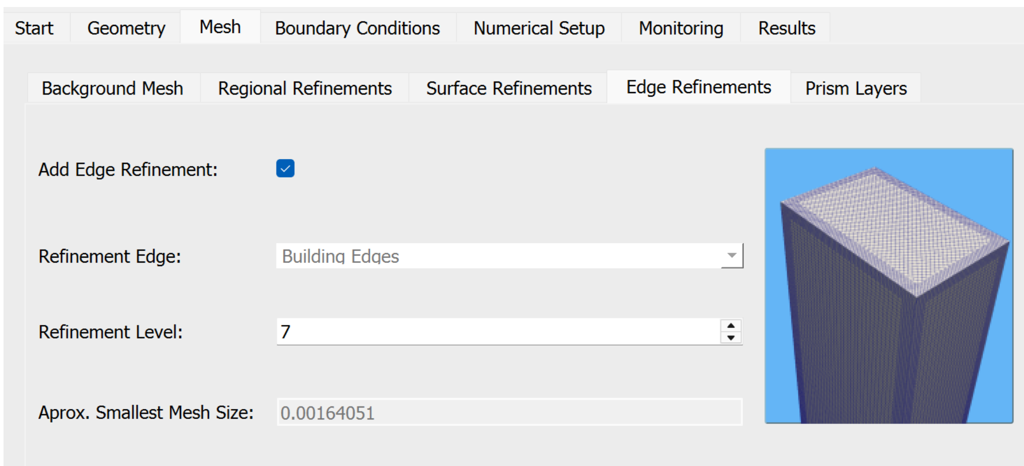
2. **Create a refined mesh near the building vicinity:**

   - **Step-2.1**: Navigate to the "**Mesh**" tab in the workflow and add a new refinement region near the study building. To do so, select the "**Regional Refinement**" sub-tab and click "**Add Region**". This automatically updates the table with the newly added region ("Box 5"). Edit the row for "Box 5," as seen below. The columns should be ("*Name*" = Box5, "*X-min*" = -0.75, "*Y-min*" = -0.5, "*X-max*" = 1.25, "*Y-max*" = 0.5, "*Z-max*" = 1.25, leave "*Mesh Size*" empty).

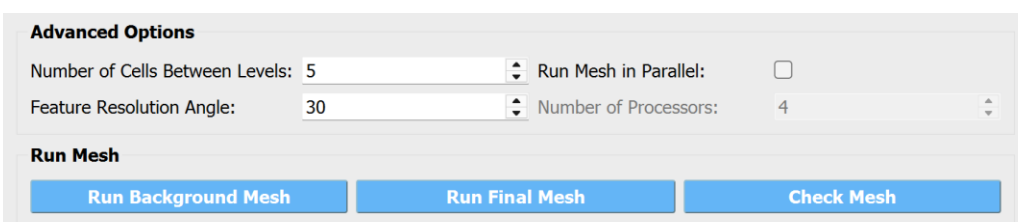


	Name	Level	X-min	Y-min	Z-min	X-max	Y-max	Z-max	Mesh Size
1	Box1	1	-7	-2	0	7	2	3	0.104992
2	Box2	2	-7	-1.5	0	6	1.5	2	0.0524962
3	Box3	3	-7	-1	0	5	1	0.1667	0.0262481
4	Box4	3	-1	-0.75	0	2	0.75	1.5	0.0262481
5	Box5	4	-0.75	-0.5	0	1.25	0.5	1.25	0.013124

- **Step-2.2**: In the same main tab(“**Mesh**”), switch to the “**Edge Refinement**” sub-tab and change the “**Refinement Level**” to 7, as shown in the figure below.

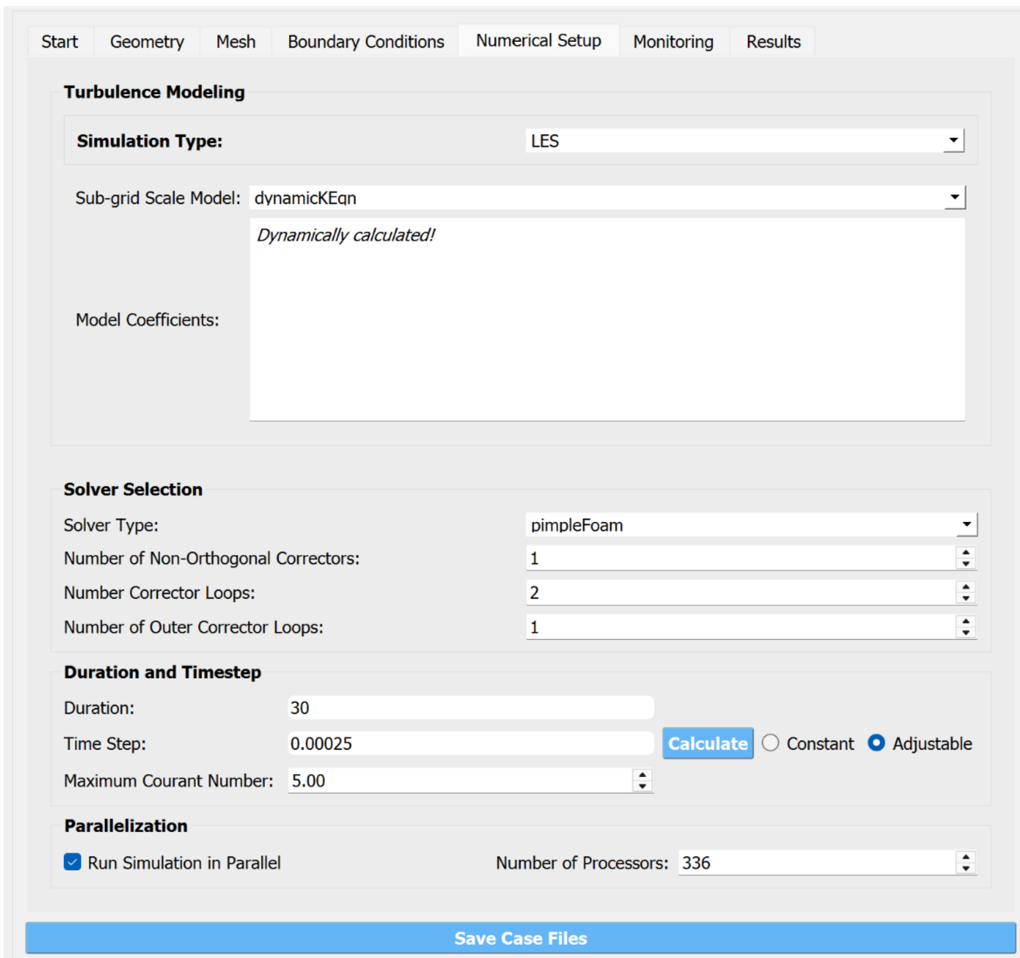


- **Step-2.3**: In the same main tab (“**Mesh**”), generate your mesh by clicking the “**Run Final Mesh**” button and wait until the grid generation is completed. Depending on your machine, this will take a few minutes (3 - 10 min).



3. **Update the numerical settings:**

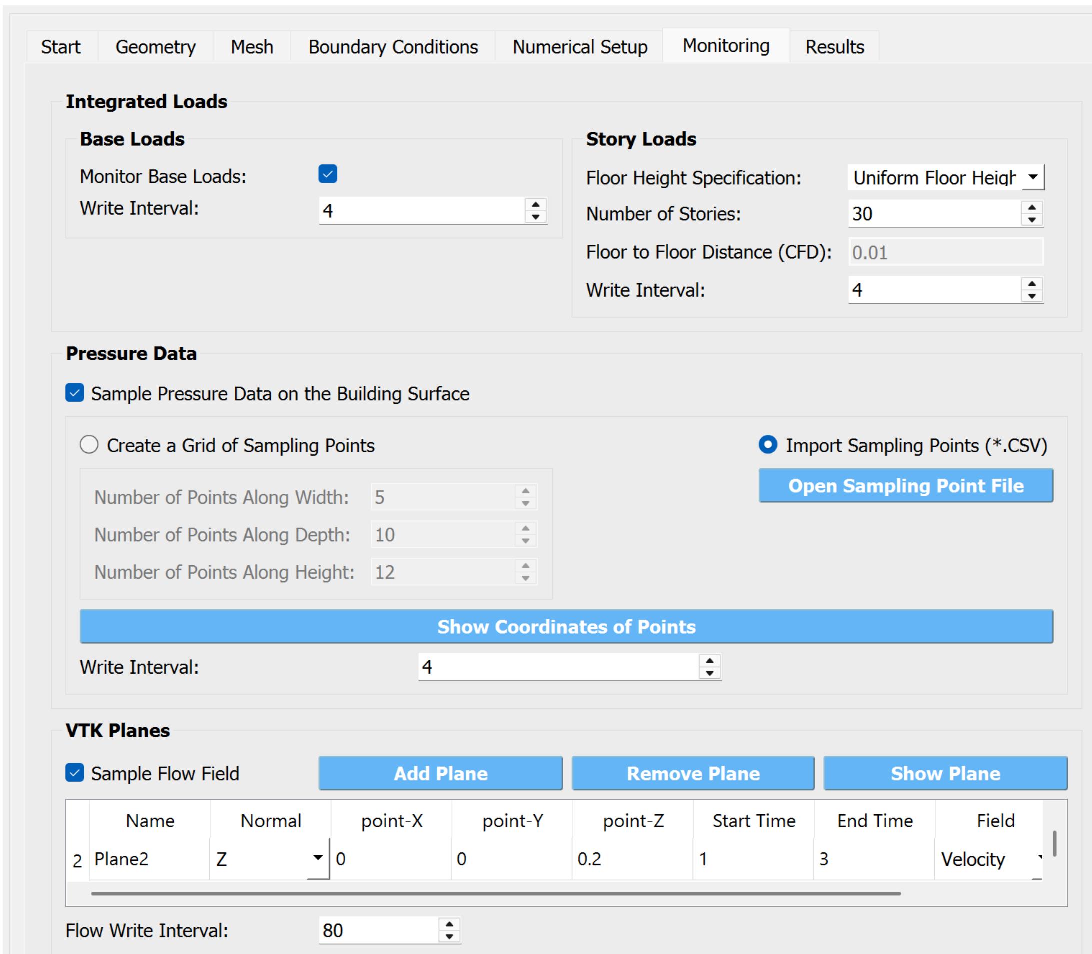
   - **Step-3.1**: Since we refined the mesh, we need to use a smaller time step. You can estimate the new time step by clicking "**Calculate**" in the "**Numerical Setup**" Tab. Then, approximate it to ***0.00025*** s (for workability), as shown in the following figure.
   - **Setp-3.2**: In the same tab, change the "**Duration**" of the simulation to **30** seconds.
   - **Step-3.3**: Also, change the number of processors to 336 cores because our mesh count increases almost by two-fold (see the figure below).



4. **Change sampling rate(intervals):**

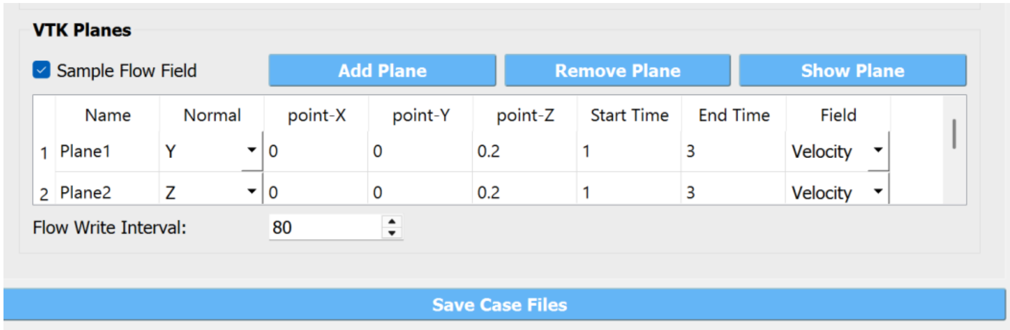
   - **Step-4.1**: Now switch to the "**Monitoring**" tab. There are three groups here, "**Base Load**", "**Story Load**" and "**Pressure Data**" each with its own writing/sampling intervals. Change the "**Write Interval**" for the "**Base Loads**", "**Story Loads**", and "**Pressure Data**" to **4**. This means the results will be monitored every **4** time steps, which results in *4 × 0.00025 s= 0.001s* time interval, the same as the sampling rate used in the experimental data (1000Hz or 0.001s).

- **Step-4.2**: Similarly, change the "**Flow Write Interval**" field for the "**VTK Planes**" to **80**, as seen below.
- **Step-4.3**: Now, save all your updated input data to the JSON file by clicking the "**SAVE File**" button and pointing to "`Assignment/WEUQ/scInput.json`" path.



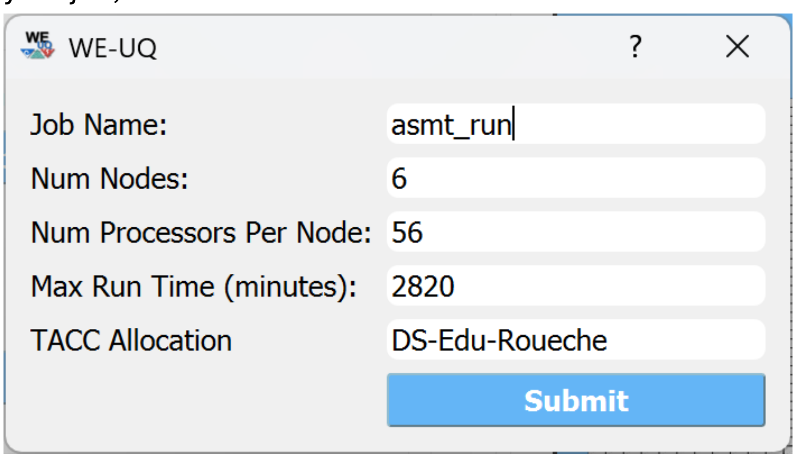
5. **Submit the simulation and monitor:**

- **Step-5.1**: Before submitting the simulation, click the "**Save Case Files**" button to update the OpenFOAM dictionary files in your local machine.



- **Step-5.2**: Login to DesignSafe, putting your "**Username**" and "**password**" in the main WE-UQ window.

- **Step-5.3**: Now, click the "**RUN at DesignSafe**" button to submit your simulation. Specify "**Job Name:**" = "*asmt_run*", "**Num Nodes:**" = *6*, "**Num Processors Per Node:**" = *56*, "**Max Run Time (minutes):**" = *2820* minutes and "**TACC Allocation:**" "*DS-Edu-Roueche*" as shown in the figure below. The **2820-minute** maximum run time translates to 47 hours, which is one hour shorter than the 48-hour max limit at TACC. Then, track the progress of your job, as demonstrated in **Tutorial 2**.



C. Deliverables

The deliverables should be presented in short report form, with each question attempted adequately. Please put all the required plots in the report.

1. **Simulation input file**
    - Provide the **JSON** file used in WE-UQ workflow for this assignment. This JSON file should be the one saved after you completed editing all the required fields in the GUI.
2. **Report integrated (base) loads from the simulation**
    - Report the four statistical moments of the base loads, i.e., the mean and standard deviation, skewness, and kurtosis in a table. The table should present statistics of the forces and moments in x-, y-, and z-directions. Comment on the mean value of x- and y-direction forces. Why does the force in the y-direction have nearly zero mean?
3. **Comparison of mean and RMS pressure coefficient (Cp) distribution**
    - Modifying the Python script used in **Tutorial 2**, plot the mean and RMS (standard deviation) of the Cp distributions on the building surface for both experiment and CFD data.
    - Perform a comparison of the mean and RMS values from CFD and experiment using scatter plots shown in **Tutorial 2**. Also, compute the mean absolute error for both mean and RMS values and comment on the results. Explain how fine mesh simulation improved the accuracy (compared to the corresponding values in **Tutorial 2**).
4. **Peak integral load calculation [Optional]**
    - Plot the time series for base forces in the x- and y-direction from the CFD simulation. Calculate Lieblin Blue fitted peak values (for both time series) and write your observation. Use the `lieblin_blue` function in the Python script used in **Tutorial 2**.