ANSYS AIM HIGH SCHOOL PROJECT GUIDE

EARTHQUAKE RESISTANT BUILDING

SCHOOL

Peters Township High School (McMurray, Pennsylvania)

INSTRUCTOR

Christopher Allen

CONTACT

allenc@pt-sd.org

PROJECT OBJECTIVE

Design a structural steel system for a building that can survive a 4.0 earthquake for 10 seconds.

PROJECT OVERVIEW

This project follows the iterative design method (prototyping, testing, analyzing, and refining the design) to achieve minimum requirements with a primary goal of a project-based approach to architectural and structural design. This portion of the overall project is completed in a digital format using ANSYS AIM to test structural steel systems in the event of a 4.0 earthquake. Results of the structural analysis are utilized when building a physically scaled model of the design.

PROJECT STRATEGY

This project relies upon the scaffolding method in education culminating with a project-based challenge. Prior knowledge includes parametric modeling techniques, structural engineering concepts, architectural concepts, technologies in construction, and material science. This project is presented to students in three phases: research, design and simulation, physical scaled model test.

Research Phase: As an introduction to this project students are exposed to skyscraper building design concepts and technologies past, present, and future. The next step in this phase is to discuss concepts relating to natural disasters and how they affect buildings, focusing specifically on earthquake. Finally, students research steel structure systems and how they are used in the construction of a building.

Design and Simulation Phase: Students begin preliminary designs in the form of rough sketches and moving on to technical drawings. This leads them into the parametric modeling phase where a teams of two students produce individual 3D models of a steel structure system for a four-floored building. These models are each tested in ANSYS AIM to determine which of the two designs would perform better during a 4.0 earthquake.

Physical Scaled Model Phase: Based on the results from the Design and Simulation Phase, the teams will build a 40:1 scaled model of the steel structure system they perform the best during the simulation. The scaled model will be tested on an earthquake simulation shake table and video will be recorded for analysis and evaluation. Students will used the video to determine the failures in their design and compare with the results from the ANSYS AIM simulation test.

EARTHQUAKE RESISTANT BUILDING
REQUIREMENTS AND GRADING RUBRIC

DIRECTIONS

Section 1: Design and develop a 3D model of a steel structure system for a building with the given *Building Requirements* below.

Section 2: Export your building design as a *STEP File* and use ANSYS AIM to test the *Vertical and Horizontal Loads* with the parameters below.

SECTION 1: 3D MODEL

Building Requirements (20 pts.): Minus 5 points from final grade for every infraction		
Base/Foundation Dimensions		
Length	**Width**	**Height**
160 in.	160 in.	20 in.
Overall Tower Dimensions		
Lengths	**Width**	**Height**
160-200 in.	160-200 in.	520 in.
Individual Floor Dimensions		
Length	**Width**	**Height (including 1 beam)**
160-200 in.	160-200 in.	120 in.
General Building Requirements		
Required Number of Floor	**Steel Beam Dimensions**	**Live Load Per Floor**
4	5 in.	30 psi.

SECTION 2: SIMULATION

Vertical Load Simulation Results (60 pts.)

Equivalent Stress (40 pts.): If the maximum value of equivalent stress induced in the material is more than strength of the material the design will fail. Points are awarded based on the performance of your design in the simulation.

Mass	**Low (32,000 & Below)**			**Mid (32,001-38,000)**			**High (38,001 & Above)**		
Stress	Below Yellow	Yellow	Above Yellow	Below Yellow	Yellow	Above Yellow	Below Yellow	Yellow	Above Yellow
Value	40	37	35	33	30	27	25	23	20

Displacement Magnitude (20 pts.): Distance, as measured directly between the start point and the end point with no directional consideration.

Below Yellow	Yellow (based on the number of contact points)	Above Yellow
20	6-19	5

Horizontal (Seismic) Load Simulation Results (60 pts.)									
Equivalent Stress (40 pts.): If the maximum value of equivalent stress induced in the material is more than strength of the material the design will fail. Points are awarded based on the performance of your design in the simulation.									
Mass	**Low (32,000 & Below)**			**Mid (32,001-38,000)**			**High (38,001 & Above)**		
Stress	Below Yellow	Yellow	Above Yellow	Below Yellow	Yellow	Above Yellow	Below Yellow	Yellow	Above Yellow
Value	40	37	35	33	30	27	25	23	20

Displacement Magnitude (20 pts.): Distance, as measured directly between the start point and the end point with no directional consideration.		
Below Yellow	Yellow (based on the number of contact points)	Above Yellow
5	6-19	20

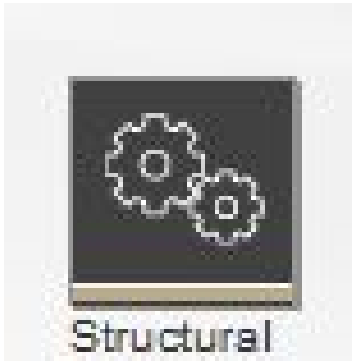
EARTHQUAKE RESISTANT BUILDING
SIMULATION PROCESS GUIDE

---

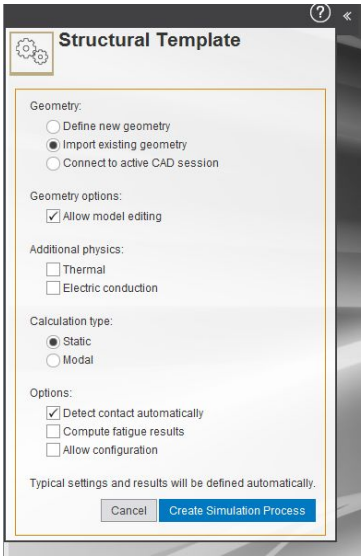
1. ANSYS AIM and File Setup

a. When AIM opens you will see the following screen

b. In the **Study Panel** select **<u>Structural</u>** for the “Simulation Process Template”



c. On the next panel, “**Structural Template”,** everything for the project file should be set and look like the following image.

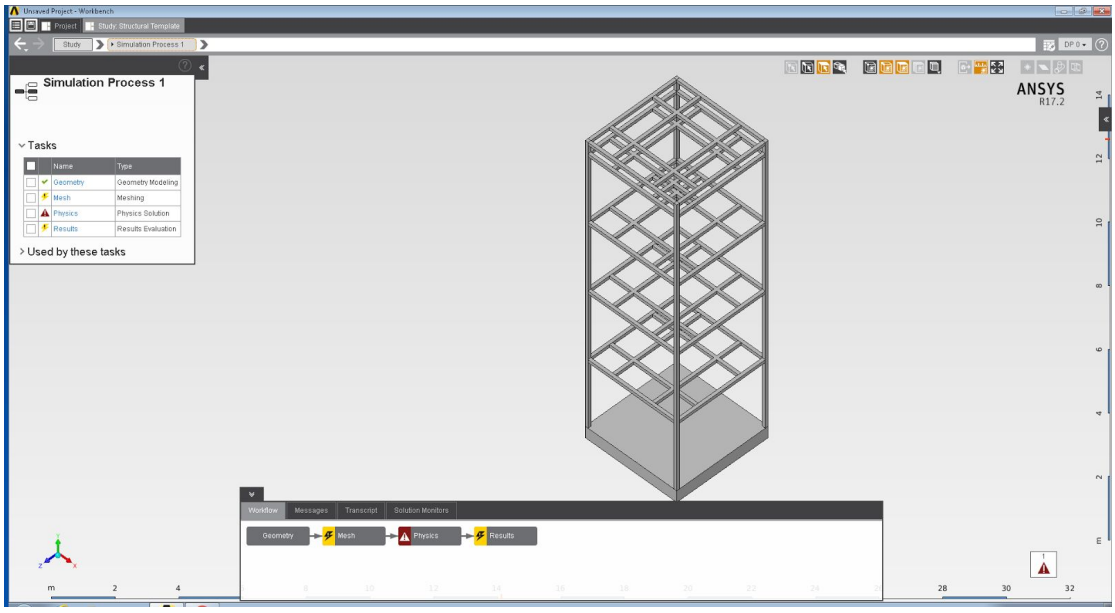


d. Click **<u>Create Simulation Process</u>** and a window will open to **Import Existing Geometry,** which is where you will select your **<u>STEP File (.STP)</u>**

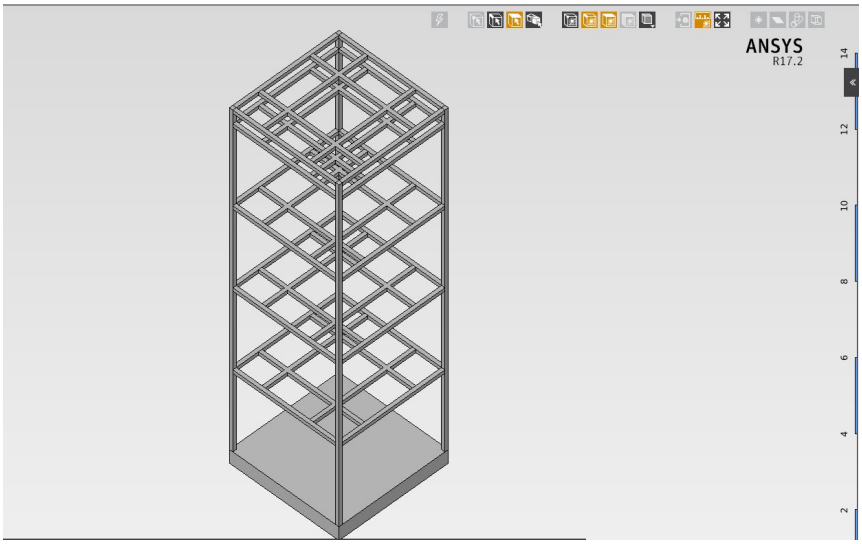
---

2. Prepare and Edit Geometry

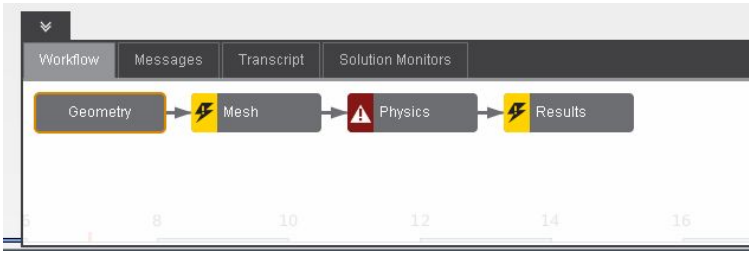
a. Once your geometry has imported you will be looking at a new window with 3 important areas we will be using to set up our simulation



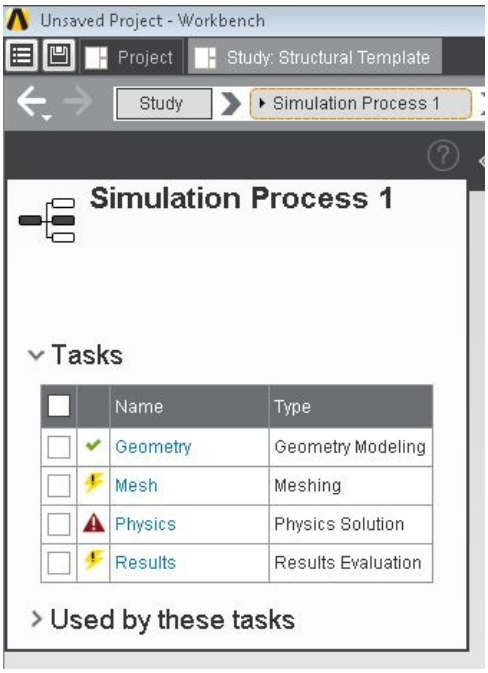
i. Area 1 - Geometry View



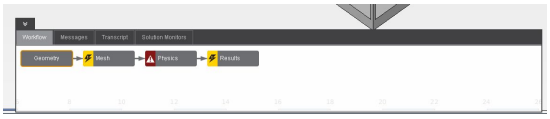
ii. Area 2 - Workflow Panel



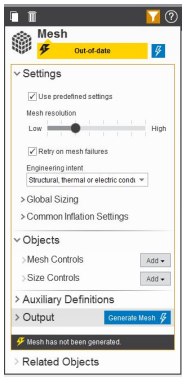
iii. Area 3 - Simulation Process Panel



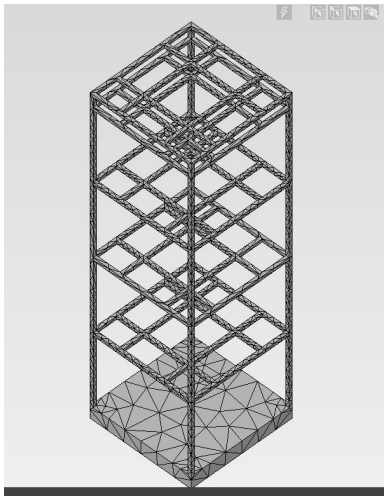
3. **Preparing the Mesh**
    a. In the **Workflow** area select **Mesh**.



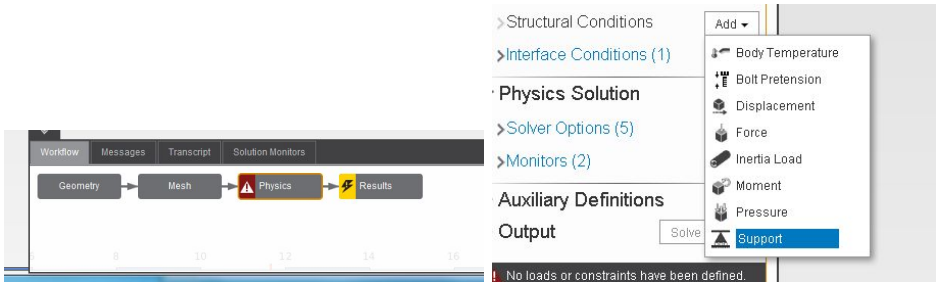
    b. In the **Mesh Panel** do not make any changes and click **Generate Mesh**.



    c. The building will now look like it has a net all over it which is the **Mesh.** The mesh represents the discretization of the physical domain into a mathematical model. The mesh enables AIM to assemble a system of equations, which are used to compute the physical response of the domain.

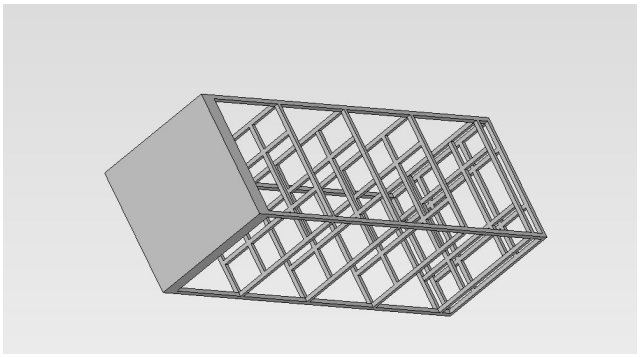


4. **Define the Support Structural Condition**
    a. In the **Physics Process Panel** find the **Structural Conditions** item. Click on the **Add** button and select **Support**.



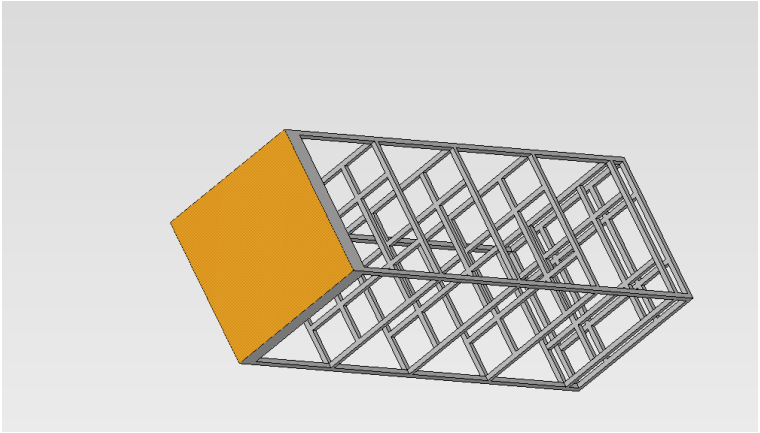
b. The **Support 1 Panel** will open in the top left of the screen and under the **Location** box we will need to add the area of our model where the support of the ground will be located. In our case it will be at the bottom of our **Foundation.**

c. Next, navigate to the **bottom** of your **Foundation** by **clicking on the building and dragging your mouse until you can see the bottom.**

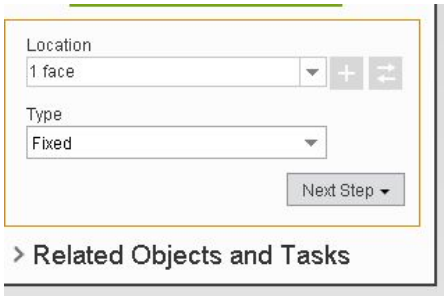


d. To actually select the bottom of the foundation you need to use the **Selection Buttons** located in menu on the **top-right** of the screen. Make sure that the **<u>Face Selection</u>** button is selected which is indicated by the it being shown as **orange**.

e. Click the **<u>Bottom Face</u>** of the **Foundation** to select this face, which will turn **orange**.



f. Finally, with the face selected go **Support 1 Panel** and in the **Location** box click the **Plus Button** to add the face to the **Support 1 Condition**.

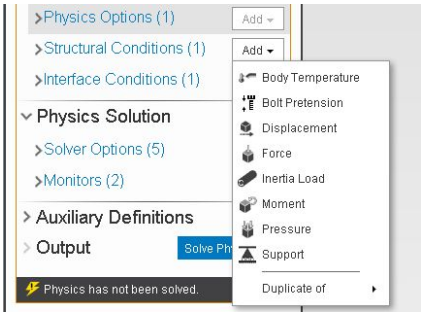


---

5. Define the Pressure Structural Condition

a. Return to the **Physics Process** by selecting it in the **Workflow Panel**.

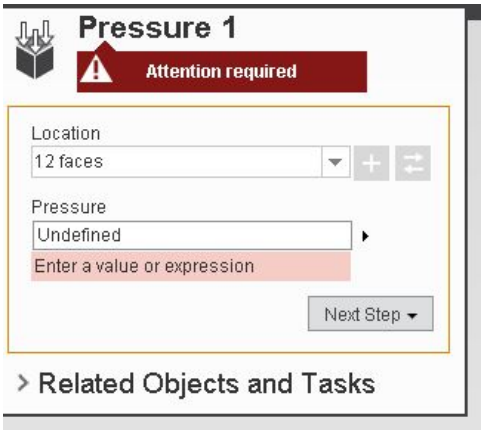
b. Add another **Structural Condition** using the **Add** button located next to the **Structural Conditions** line and select **Pressure**.



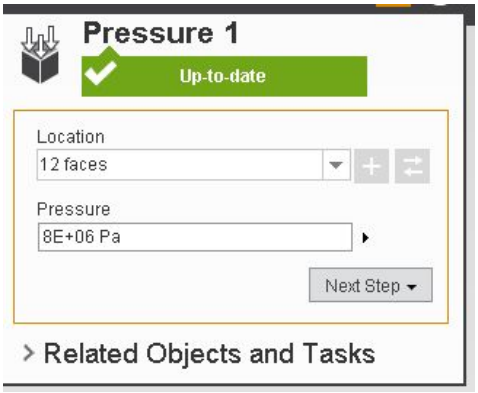
c. The **Pressure 1 Conditions Panel** will appear in the top-left of the screen and just like with the **Support 1 Condition** we will need to add a location where the pressure will be applied to our structure.

d. Using the same method of navigating around the building, select the **top faces of each floor.**

e. Add these to the **Pressure Condition** by clicking the **Plus Button** in the **location** line of the **Pressure 1 Panel**.



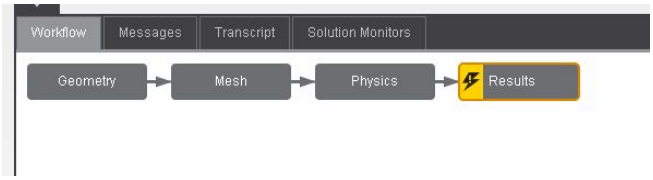
f. Next we need to define the amount of pressure to be applied to the locations that you selected. Enter the amount of **206842.719 [Pa]**



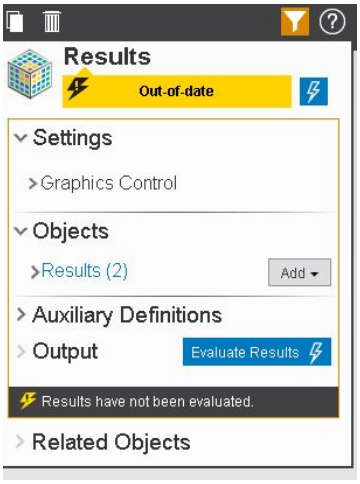
g. Finally, click the **Physics** button located in the **Workflow Panel** to return to the **Physics Process Panel**. To complete the Physics portion of this simulation click the **Solve Physics** button located at the bottom of the this panel.

---

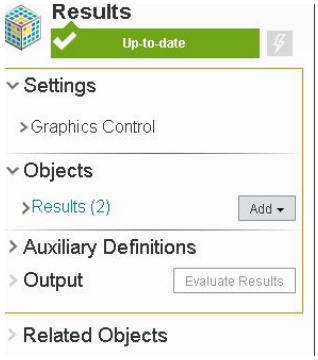
6. **Run the Simulation and Evaluate Results**
   a. Click Evaluate in **Workflow Panel**



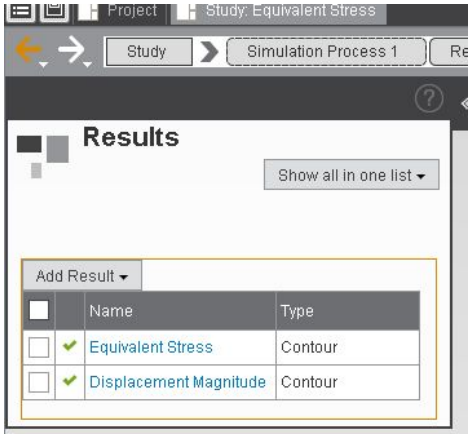
   b. In the **Evaluate Process Panel** click on the **Evaluate Results** button to run the simulation.



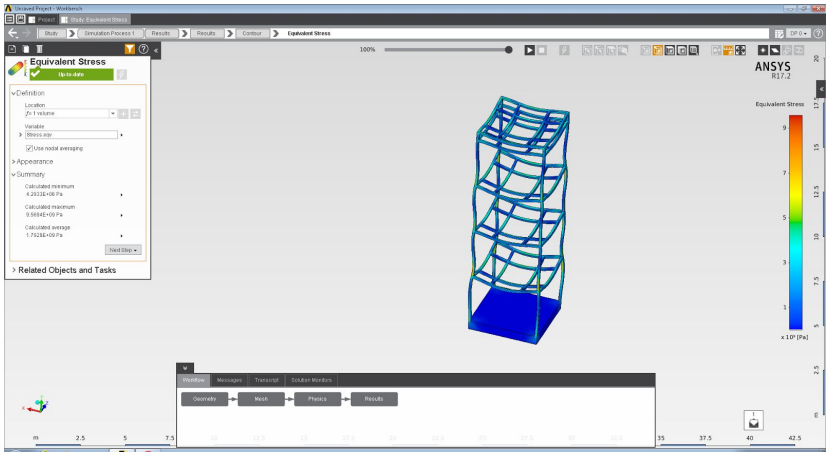
   c. The **Results Process Panel** will have then show an “Up to Date” symbol in green.



   d. Now we can see the results by click on the word **Results** in this panel. Doing this will bring up another panel with two items, **Equivalent Stress** and **Displacement Magnitude. Clicking the arrows located in the top-left** will allow you to navigate between the two results.



e. Selecting one of these will bring up the results with the option to play an animation of the simulation. To play the animation click on the **play button** located above your building.

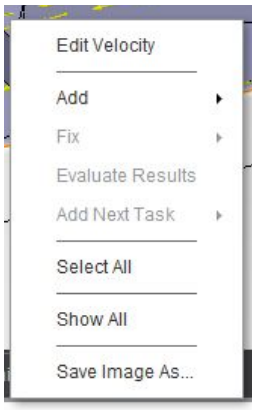


---

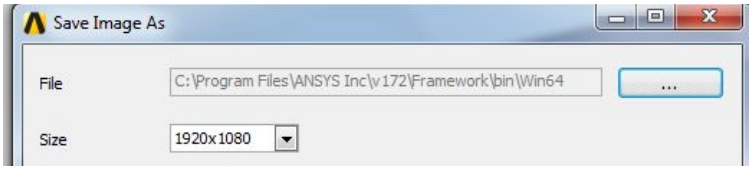
7. Document Results

a. Navigate the camera to a view that shows your entire design.

b. **Right-Click** on your design and then click **"Save Image As…"**

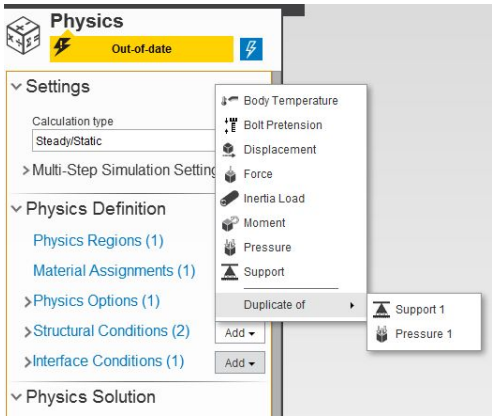


c. In the **"Save Image As"** box that opens click on the button to the right of the **"File"** box that has **3 dots** and select your **H: Drive** to save each picture



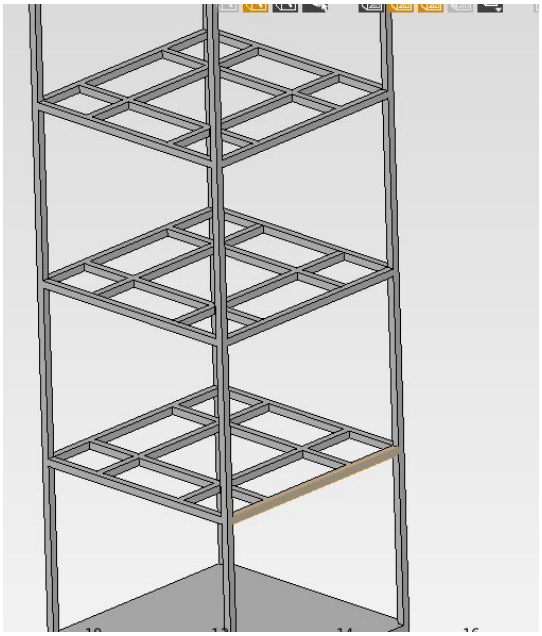
---

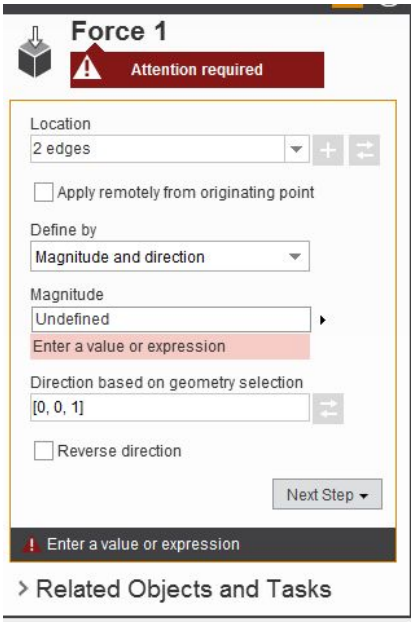
8. **Add Seismic (Lateral) Forces to Simulation**
    a. Add another **Structural Condition** using the **Add** button located next to the **Structural Conditions** line and select **Force**.



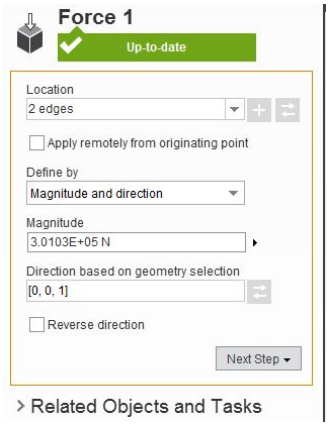
    b. In the **Selection Menu** we need to use the **Line Selector** tool, shown below in **orange**.

    c. Select the lines along the **first floor beams or foundation** (depending on your design) holding **CTRL** to select both. Then in the **Force 1** menu click the **+** button in the **Location** field to add those lines.

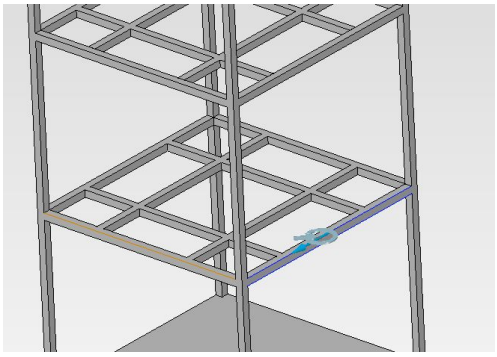




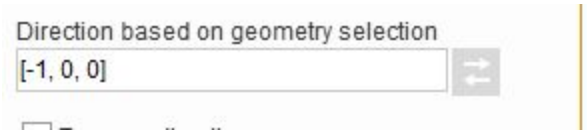
d. In the **Magnitude** field add a value of **<u>301033.4[N]</u>**

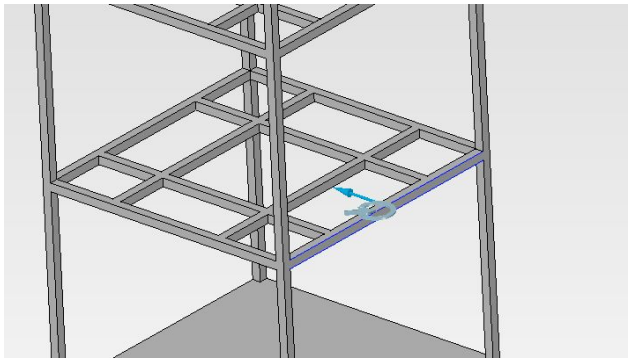


e. Click in the **Direction Based on Geometry Selection** section and **select an edge that is perpendicular** to the edges you selected for the **Location**



f. Back in the **Force 1** menu click on the **<u>Replace</u>** button in the **Direction Based on Geometry Selection which** which will change the direction of the arrow to facing directly at the beam selected earlier.





g. Repeat these steps to add **Force Structural Conditions** in the same exact way for each floor. Use the following amounts below as the **magnitude** in Newtons.
   i. Floor 2 = 301033.4[N]
   ii. Floor 3 = 560547 [N]
   iii. Floor 4 = 820060.8 [N]
   iv. Roof = 1123691[N]

h. After adding all of the new **Structural Conditions** go back to the **Physics Process Panel** and click **Solve Physics**.

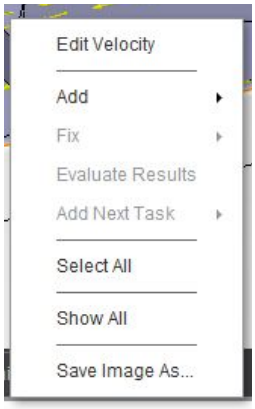
i. Finally, go to the **Results Process Panel** and click **Evaluate Results**

---

9. Document Results

a. Navigate the camera to a view that shows your entire design.

b. **Right-Click** on your design and then click **"Save Image As…"**



c. In the **"Save Image As"** box that opens click on the button to the right of the **"File"** box that has **3 dots** and select your **H: Drive** to save each picture