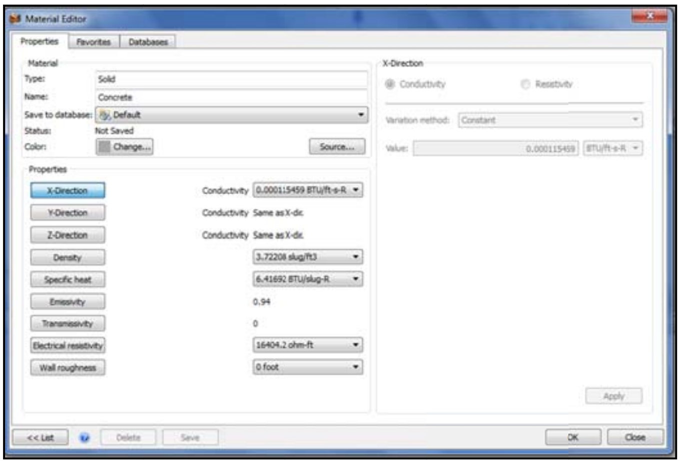
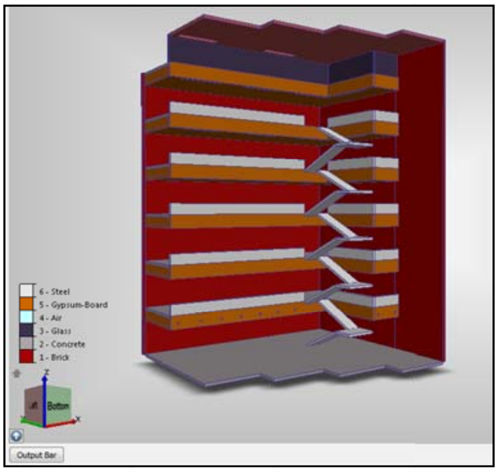


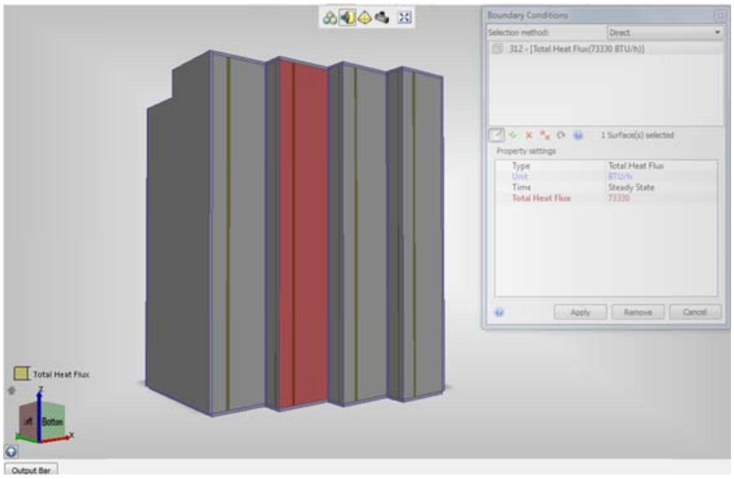


Boundary Conditions

Boundary conditions are the values assigned to objects within the model that have some type of fluid flow or heat transfer value. All the air inlet and outlets that allow air to flow in an out of the model will need to be assigned. Similarly all the heat transfer elements within the model will need to be defined.

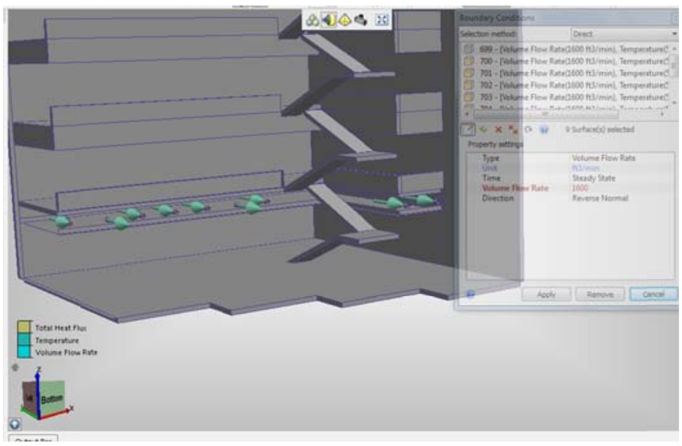
Envelope Loads

First we need to model the envelop loads in the space. With a large glass atrium there will be significant envelop loads that we need to account for in our model. A load calculation software was used to calculate the total envelope loads in the space. From the load calculation the total amount of energy from the envelope can be modeled by assigning a heat transfer boundary condition to the large glass façade. A total heat flux in BTU/hr is assigned to the envelope surfaces.



Supply Diffusers

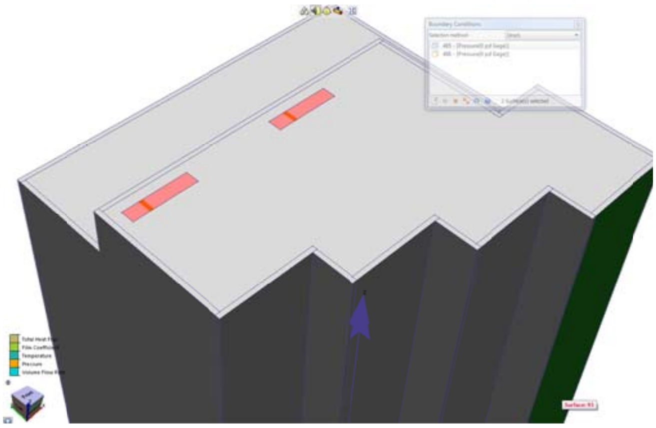
The supply diffusers need a volumetric flow rate and temperature assigned to them. To start the airflows from the load calculation will be used. The nine high throw nozzle diffusers will have a flowrate of 1,600 CFM at 55F and the linear diffusers will supply 1,120 CFM at 55F



per level. It is important when dealing with flowrate that you ensure the flow normals are correct. In this case the supply diffusers are introducing air to the space so the flow arrow should put into the space.

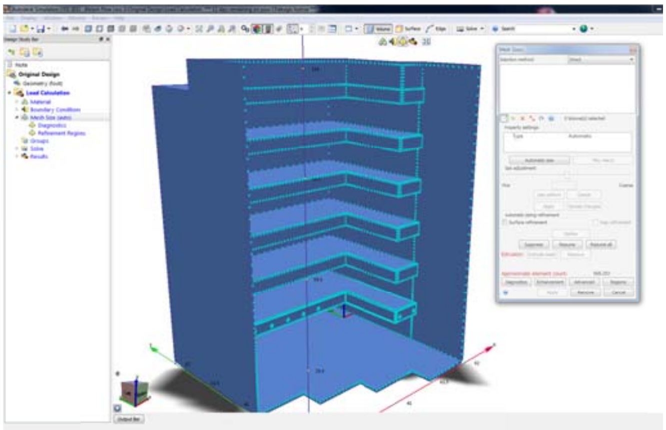
Return Grilles

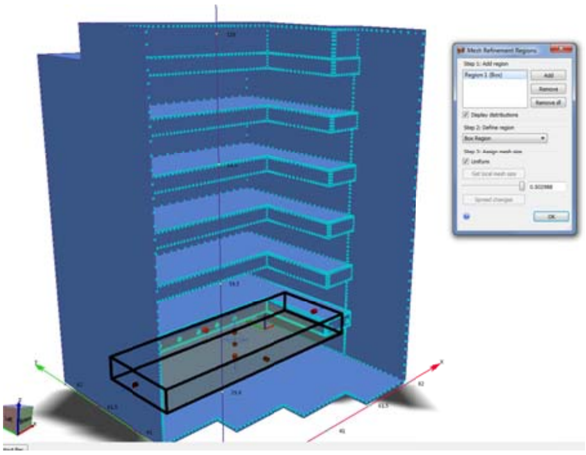
The return grilles will simply be assigned a value of atmospheric pressure. This will allow the air to push its way out of the analysis space to allow an even pressure in the space. It is important that the return air grille be set at atmospheric to ensure that the pressure values within the space calculate properly.



Mesh

Now that all the elements in the model have defined thermal and flow properties it is time to create the analysis mesh. A proper analysis mesh is the key having a successful simulation. The mesh is directly correlated to the geometry of the model and that is why a clean geometry was emphasized earlier in this handout. Luckily Simulation CFD makes creating the analysis mesh very simple so as long as you have a clean geometry the meshing process should be very straight forward. Once you are in the meshing prompt you can simply click the *Automatic Size* button.





This will automatically size and create a mesh for you. You can slide the *Size Adjustment* bar to the left or right to create a finer mesh for a more detailed calculation or a coarser mesh for preliminary studies. For the study used in this handout the default automatic sizing mesh was used. However, a refinement region was added to the area near the nozzle diffusers to ensure the airflow throws of the