AUTODESK UNIVERSITY 2013

CFD + Energy: Integrated Optimization

Jacob Knowles – Bard, Rao, + Athanas Consulting Engineers, LLC
Michael Gilroy – Bard, Rao, + Athanas Consulting Engineers, LLC

SM2329

Computational fluid dynamics (CFD) is a critical simulation tool for sustainability. By integrating CFD into the broader energy-simulation workflow, we can make more informed decisions and illustrate these issues for our clients. The key is to understand how CFD results translate into real-world metrics in terms of thermal comfort and energy and, ultimately how the value of each strategy compares to other high-performance investment opportunities. This case-study class focuses on one simple example: using CFD to compare finned-tube perimeter heat to radiant ceiling panels in a typical private office. Our class outlines not only the specific steps of the CFD modeling process, but also how this process fits into the overall energy-performance optimization trajectory for the project.

Learning Objectives

At the end of this class, you will be able to:

- Identify the key parameters that influence thermal comfort and pose potential solutions
- List the steps involved in performing a CFD model
- Use results from CFD modeling to make informed decisions
- Integrate CFD modeling into the design, optimization, and energy-simulation process for your projects

About the Speakers:

As Director of Sustainable Design at BR+A, Jacob Knowles heads the building simulation and sustainability consulting team. His work focuses on the integration of strategies that reduce environmental-footprint, while supporting occupant wellbeing and maximizing return on investment. Over the past decade, he has championed the sustainability agenda for major research, healthcare and other commercial and institutional projects. Project highlights include achieving LEED Platinum, meeting the 2030 Challenge, and receiving a MA DOER High Performance Buildings Grant. Jacob has presented these success stories and other topics at speaking engagements such as Labs21, NESEA BuildingEnergy, Healthcare Design, Build Boston (ABX), and Tradeline.

jwk@brplusa.com

Michael Gilroy is an HVAC engineer at BR+A with three years of experience in the design of Heating, Ventilating, and Air Conditioning (HVAC) systems, energy modeling and analysis, systems life cycle cost analysis, feasibility studies, and Building Information Modeling (BIM). Michael's project experience has included healthcare, research, and academic projects for clients nationwide. Michael used his collegiate background in the study of HVAC engineering and CFD to assist in the development of BR+A's in-house CFD modeling services. Michael plans to continue to optimize synergies between HVAC engineering, BIM, and CFD modeling.

mpg@brplusa.com

Introduction

This course outlines a case study of an office renovation project which employed the use of a computational fluid dynamics tool with BIM to model and evaluate HVAC design options. The CFD model was used to analyze airflow and temperature distribution within the sample zone. Additionally, the CFD model results were extrapolated for use in an energy model to model the effects of the perceived temperature of the room the potential impact on the central mechanical system. The simulation provided insight into potential occupant thermal comfort and building energy-consumption as engineers compared perimeter systems including finned tube, radiant panel and chilled ceiling panel.

In this course, we intend to demonstrate the desire and need for consulting engineers to push for more energy efficient measures without sacrificing occupant thermal comfort or negatively impacting the project budget. Often tradeoffs occur when trying to balance all three components. CFD modeling is becoming a more useful tool to demonstrate to owners and clients the performance of MEP system variations. However, room level MEP systems are only a portion of the overall energy conservation measures. This course aims to deliver a comprehensive look into a real application of CFD, in combination with comprehensive energy savings measures, to deliver the most appropriate MEP system. Ultimately, the CFD analysis led to a recommendation to incorporate finned tube perimeter heat (not radiant panels), and demonstrated the effectiveness of active chilled beams.

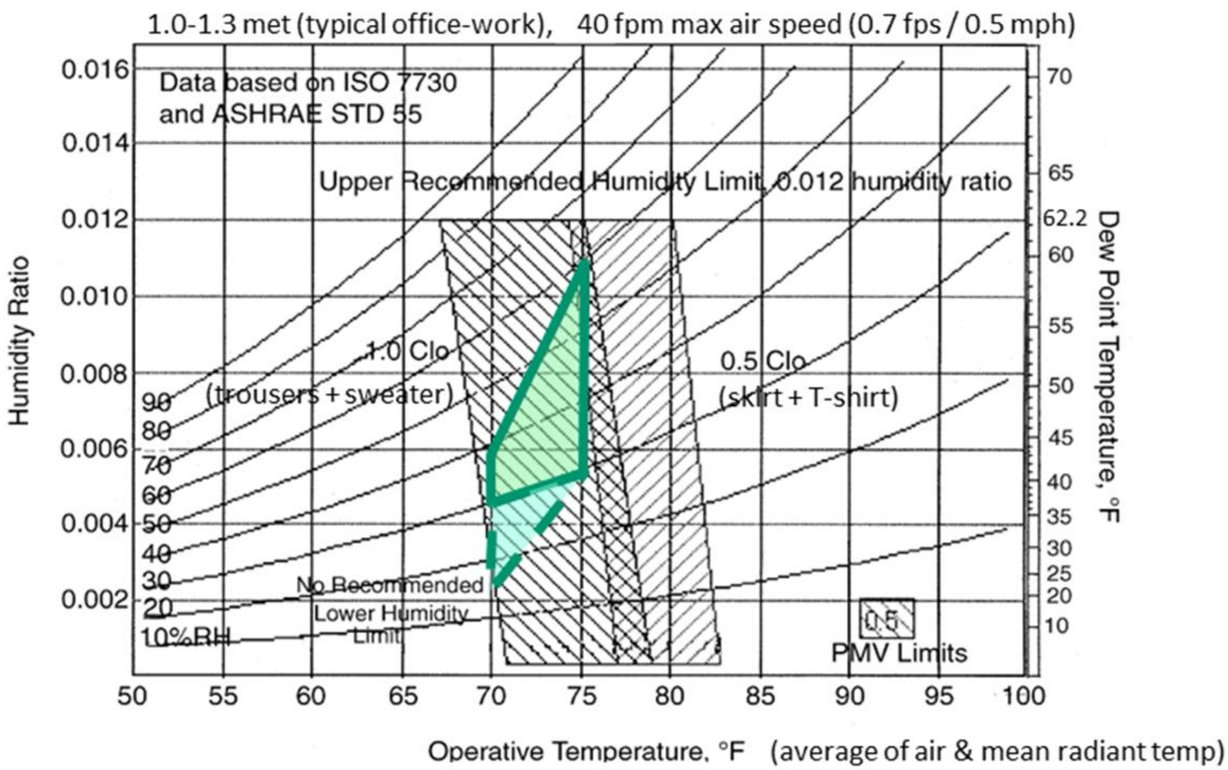
.

Identify the key parameters that influence thermal comfort and pose potential solutions

Thermal comfort is a condition of mind which expresses satisfaction with the perceived thermal environment. The primary factors that impact thermal comfort are:

- Metabolic Rate ("met")
- Clothing Insulation ("clo")
- Air Temperature
- Radiant Temperature
- Air Speed
- Humidity

The nationally-recognized standard defining thermal comfort in buildings is ASHRAE Standard 55. The diagonally-hatched quadrilaterals in the psychrometric chart below show the typical temperature range considered acceptable for thermal comfort (one represents thermal comfort at 1.0 Clo and the other represents thermal comfort at 0.5 Clo). The colored zone indicates typical engineering design temperature and humidity range.



When it comes to building systems, the variables that a designer can typically influence are: air temperature, radiant temperature, air speed and humidity. For example: to achieve thermal comfort, one can utilize warmer surface temperatures in a space to make a space feel comfortable, while allowing a lower air-temperature setpoint.

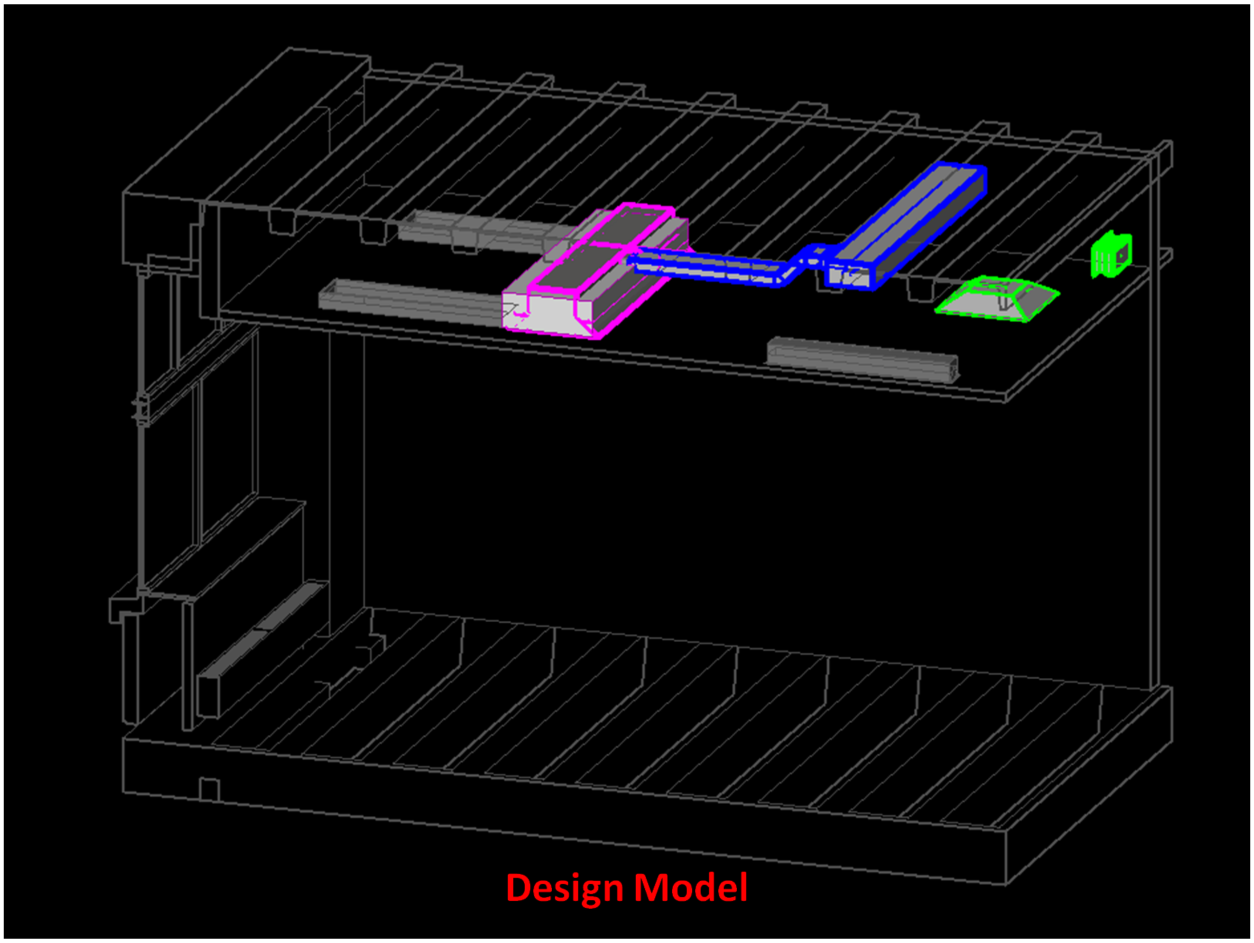
List the steps involved in performing a CFD model

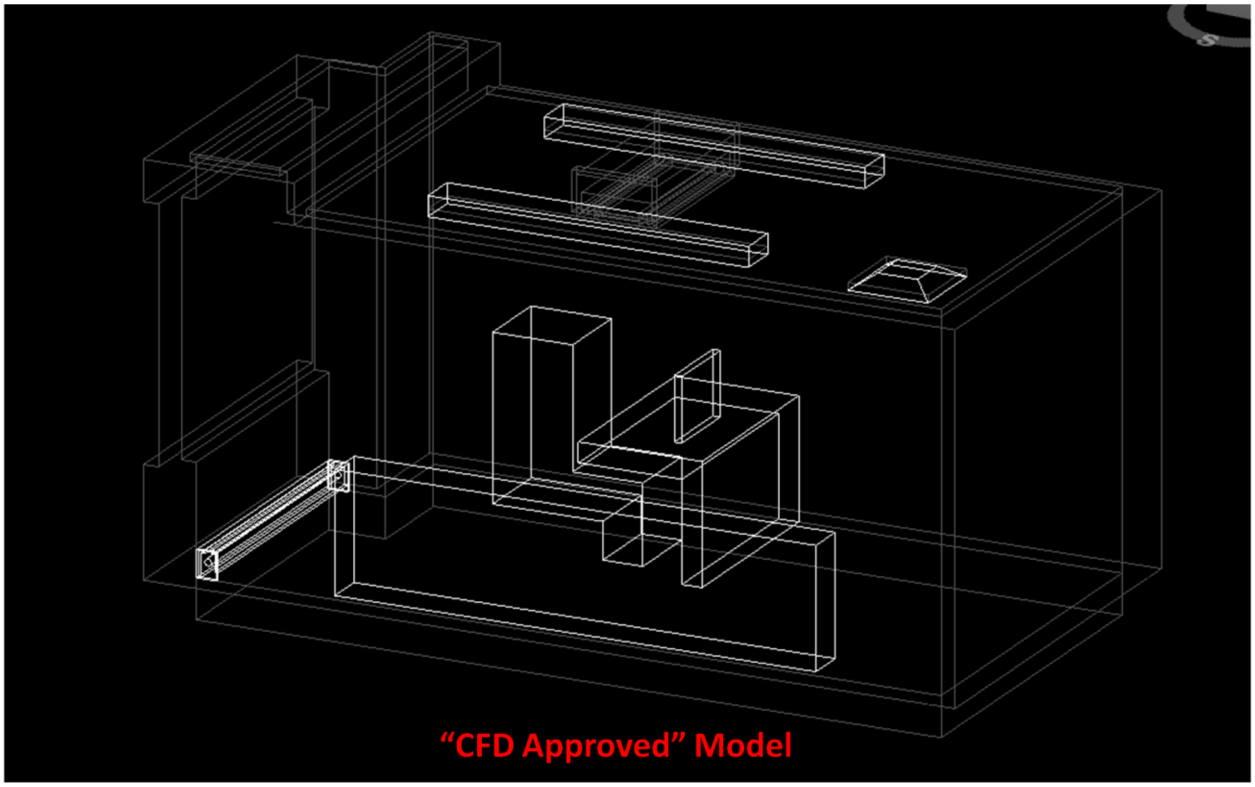
Create Geometry

It is important to understand how a typical Revit Architecture or Revit MEP model differs from a model used for CFD analysis. Seen in the adjacent image is a three dimensional view of the typical office space analyzed, as found in the mechanical engineer's Revit MEP model (with the architect's Revit Architecture model linked in). Note the location of the active chilled beam (magenta), return grille (green) and finned tube element (gray, along the base of the perimeter wall, below the window).

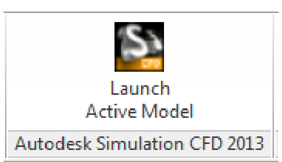
From Revit, this view could be exported for CFD analysis, but it is far too complex to be useful. The detail level of the curtain wall and mullions is too great for CFD analysis. Also, it does not appear the exterior wall completely connects with the floor, and the ceiling plenum is open to the "atmosphere" surrounding the office space. This lack of continuity will also not create a useful CFD geometry. Therefore, it is not best practice to directly take the design model into CFD analysis software.

The most efficient means to set up a correct geometry is to recreate the model with minimal detail and "air tight" construction. We will see later in this handout how this level of detail affects the meshing of the model. Rebuilding the model allows complete control over what elements are modeled and their level of detail. This step may require a considerable amount of time, but is recommended for a successful CFD analysis.





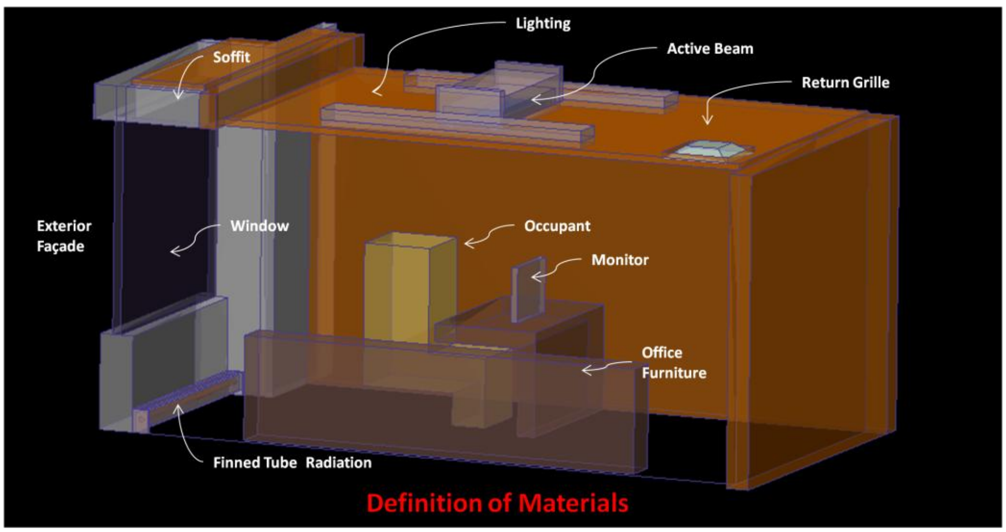
The above image represents the "CFD Approved" model. The geometry is consistent, yet much simpler than the design model. It was created within Revit MEP using design documents as a guideline. A finned tube element, chilled beam, and return grille were all modeled to match the design model. The window and mullions were greatly simplified. The ceiling plenum also does not exist because we do not need to include heat transfer occurring in the ceiling plenum as it has no effect on occupant thermal comfort in the room. Also, a simplified human and furniture were added for a two reasons: to create obstacles that obstruct airflow and to provide heat generation sources crucial to thermal balance of the room.

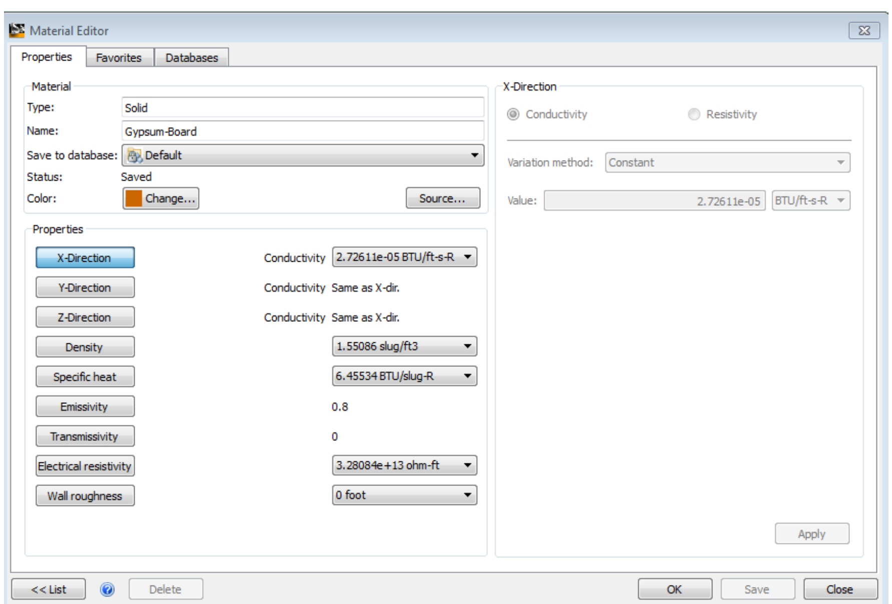


The convenience of recreating the model within Revit MEP or Revit Architecture is the ease of the export to Simulation CFD. An add-in to Revit can be installed which exports any three dimensional view into a Simulation CFD accepted file format. Once exported and launched into Simulation CFD, it is important to ensure the model is airtight. If modeled properly, Simulation CFD will create an additional volume not created in Revit. The volume created represents the air in the office space that we intend to analyze. Also, if changes are required to the geometry, the launch can be repeated. In Simulation CFD's Design Study Manager, you can select whether to update an existing CFD file (and maintain material properties, boundary conditions, etc.), add a new geometry to an existing file, or start anew with a new file. The choice is yours!

Define Materials

Once the model geometry has been successfully created, the next step is the assignment of materials to the model components. Fortunately, the Simulation CFD software comes preloaded with a database of typical materials. In the model below, the internal air volume was modeled using the default “air” material, the cooling coil of the Active Chilled beam as a heat exchanger, and the remaining model components as various types of solids. It is important to note that when using CFD to visualize temperature distribution, the scenario environment of the air must be changed from the default of “Fixed” to “Variable.” Within each material, the opportunity exists to create custom materials by editing properties such as density, viscosity, conductivity, specific heat, emissivity, etc. For the purposes of this simulation, most materials used the default values. The active chilled beam required edits to its flow and heat transfer characteristics. Before creating Simulation CFD geometry (step 1) it is important to be aware of the types of materials available and identify if you require a unique approach.



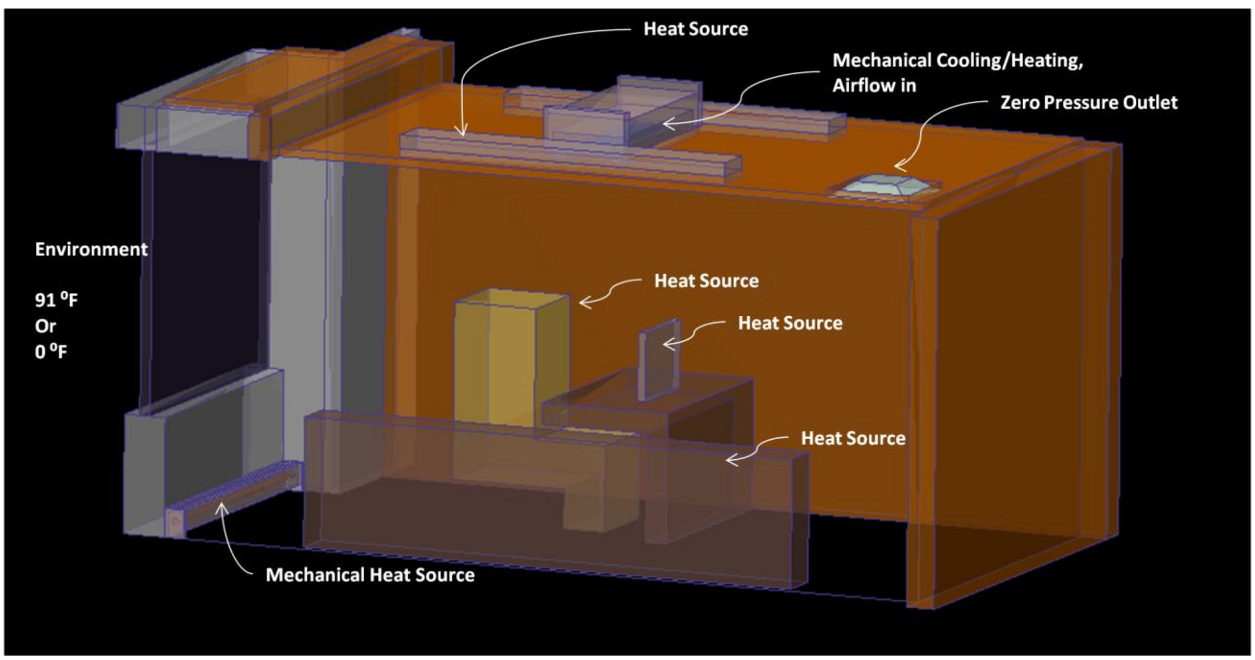


Material Editor window within Simulation CFD

Add Boundary Conditions

Once materials have been selected and added to all components in the model, boundary conditions can be applied to the appropriate components. In this model, the impact of the exterior environment is very critical. The models discussed used the Solar Heating portion of Simulation CFD. An exterior volume was created within the Simulation CFD software to model the exterior environment with solar impacts specific to our project location. To allow the solar energy to properly enter our office zone, we adjusted the emissivity and transmissivity properties of the perimeter wall elements. As you will see in the results, most heat transfer occurred at the perimeter window. Additional heat sources were added to represent heat gains from occupants, lighting, miscellaneous plug loads, and mechanical heat sources where applicable. The Active Chilled Beam was modeled with a primary air inlet and a heat exchanger that drew air from the space over a coil and into a mixing zone with the primary air. Appropriate airflows and capacities were added per calculated design data. Finally, a zero pressure (or open to the atmosphere) outlet, was added to allow the simulation to naturally converge.

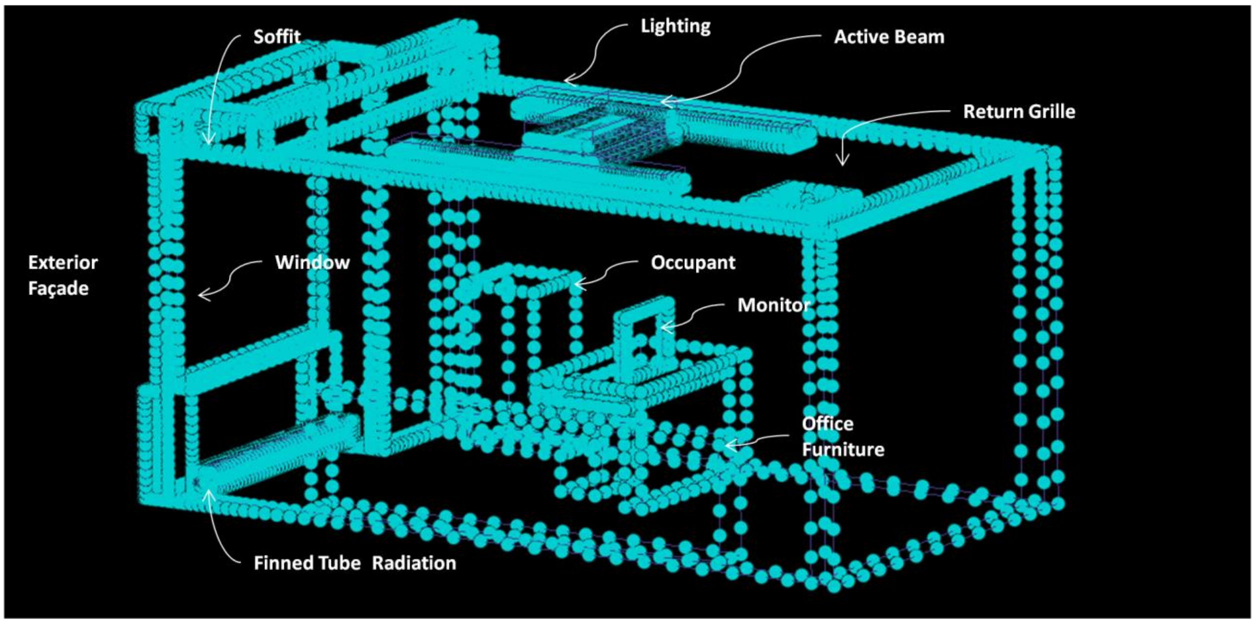
A user must resist adding an outlet airflow equivalent to the inlet airflow as this will over-specify the problem and not allow the simulation to properly solve, producing inaccurate results. During winter modeling, the exterior environment's temperature was adjusted and the solar effect was minimized to test the mechanical heating devices in the worst case scenario.



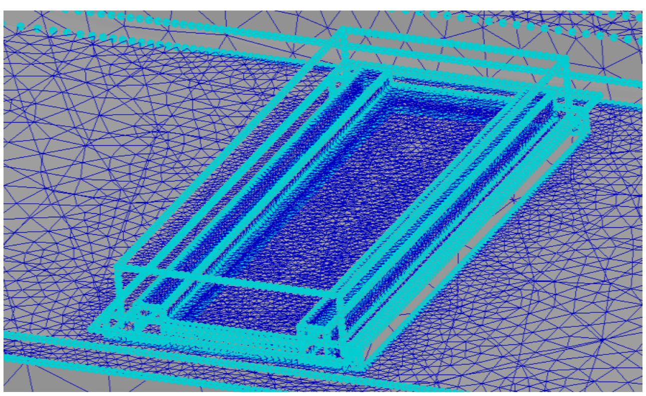
Boundary Conditions

Generate Mesh

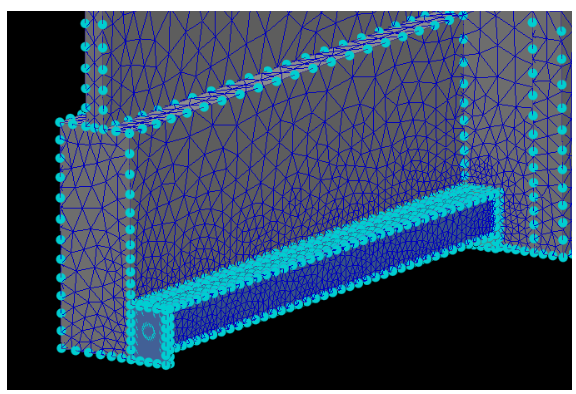
The generation of mesh is the next, and perhaps most crucial step in producing an accurate CFD analysis. Meshing the model divides the model's elements into millions of minute cells. Within each mesh cell, heat transfer and fluid flow differential equations are continuously calculated and passed on to neighboring mesh cells. The process continues until the model's elements converge on a solution. To promote convergence in models, it is important to increase the intensity of mesh near boundary conditions. In the image below, the blue circles outline the mesh grid. Notice that the mesh near the Active Chilled Beam, the finned tube element, the return grille, and the lighting is extremely dense. Conversely, the interior wall opposite the window has a more coarse mesh distribution. Increasing mesh is possible by selecting an item while in the mesh tab and reducing the size distribution or creating a region where mesh should be denser.



Generation of Mesh



Mesh local to the Active Chilled Beam



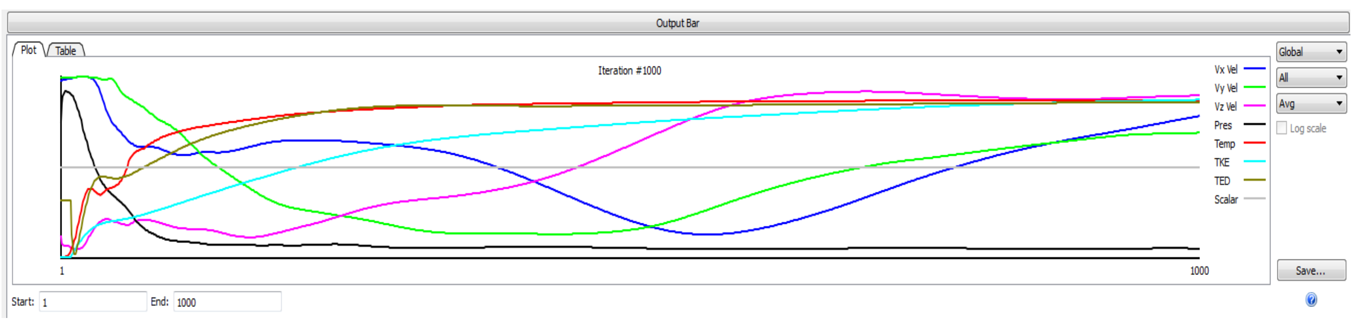
Mesh local to the finned tube element

Solve

Setting the mesh accurately is the final step required before starting the CFD simulation. Within the solve window, there are different possibilities for simulation settings that the CFD modeler must carefully apply to each simulation. For our simulations we applied the following options. (Every model requires a reevaluation of the solve criteria)

- Steady State solution (snapshot in time, other option is transient)
- K-epsilon turbulence model
- Solved via the CLOUD
- Thermal Comfort result quantities added
- 1000 iterations specified
- Advection Scheme 5 (recommended for AEC simulations)
- Flow and Heat Transfer calculated
- Radiation calculated
- Gravity set to –z direction for buoyancy effect
- Solar Heating initiated, set to Boston MA 9/21 at 3:00 PM

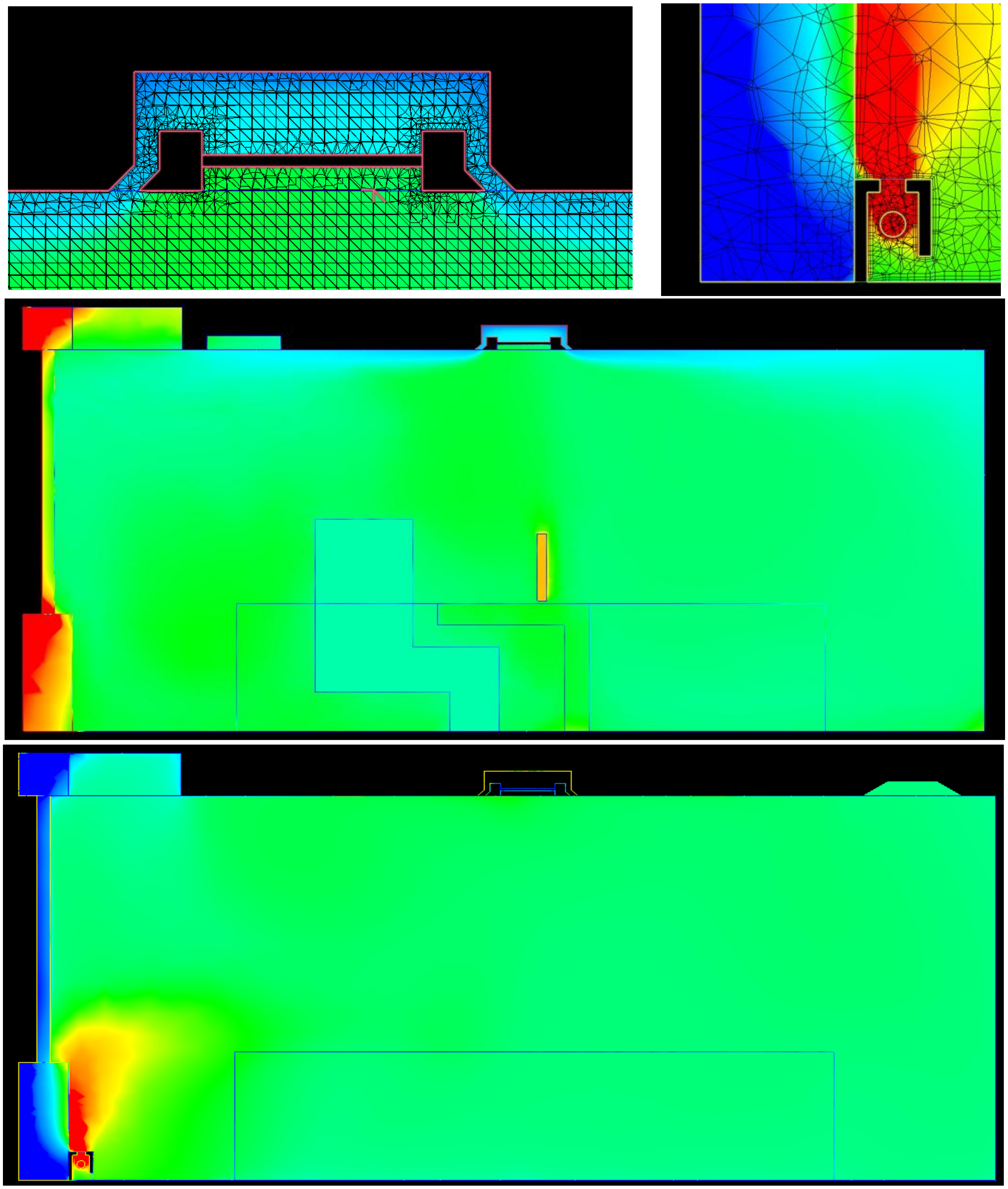
After starting the simulation, the output bar plots the variables the simulation is attempting to solve. Among these variables are velocity (x, y, and z directions), pressure, temperature, turbulent kinetic energy, and turbulent energy dissipation. When the majority of the variables' plots level, it can be concluded that the model has converged and the results can be analyzed. The size of the mesh created in earlier steps directly correlates to how long the solve step will take. The four situations that were analyzed ranged from 2.7-3.9 million mesh elements and each required 9-15 hours to simulate.



Analyze Results

After a simulation has completed with convergent results, we analyzed the areas of importance for accuracy of our modeling and data input. We first started at the chilled beam element and noted that the room return air was entering the heat exchanger, cooling down, and mixing with the primary air as intended. The mixed air relieved into the room space via the chilled beam slots. The supply air adhered to the ceiling and mimicked the Coanda effect, also as anticipated.

In simulations during heating mode, we assessed the performance of the finned tube element or the radiant panel. In the image on the following page, an image of the finned tube element is shown adjacent to the cold exterior wall, inducing room air through the bottom opening, heating of the induced air at the finned tube element, and warm air rising through the top opening of the enclosure. Once it was verified that the crucial heat transfer elements were performing as intended, the entire room could be analyzed.

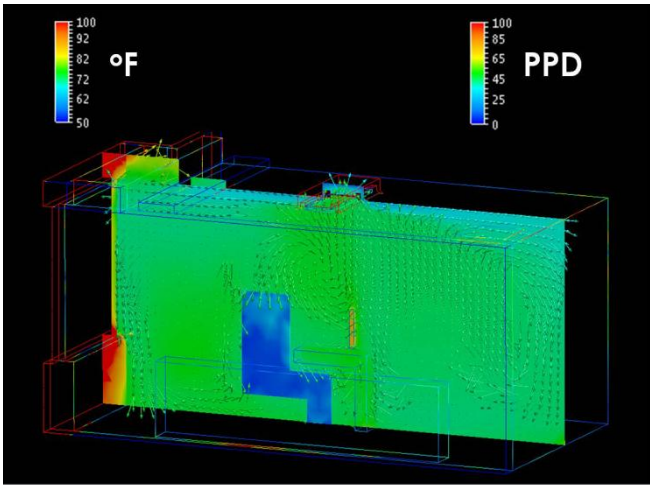


Above right: Image of Active Chilled Beam cross section with mesh, temperature distribution
Above left: Image of finned tube element cross section with mesh, temperature distribution
Middle: Image of room cross section with Active Chilled Beam in cooling mode
Bottom: Image of room cross section with finned tube element in heating mode

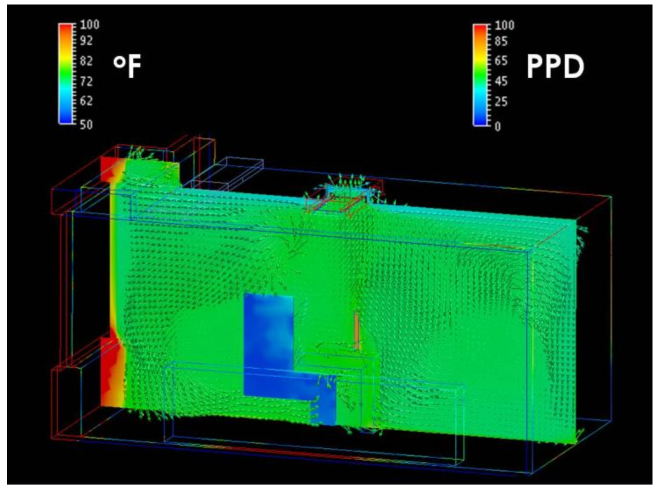
Repeat

The last step involves repeating the previously discussed steps as necessary to address unanticipated shortcomings in the model results. Rarely will a CFD model be completed in a single iteration of this process. Often, the simulation results will direct the modeler to areas that require improvement or adjustment.

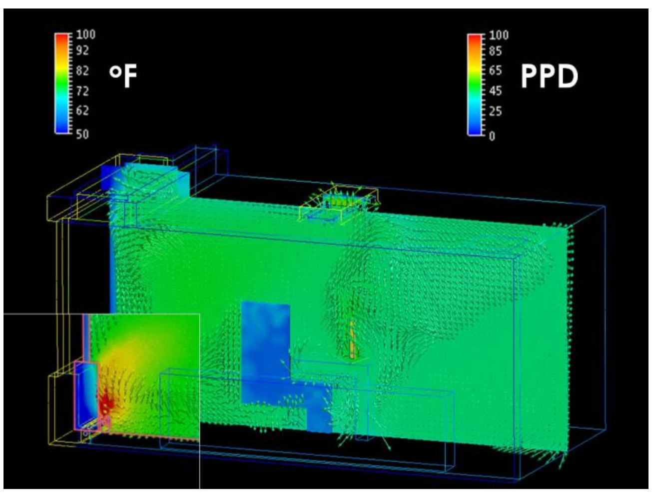
Use results from CFD modeling to make informed decisions



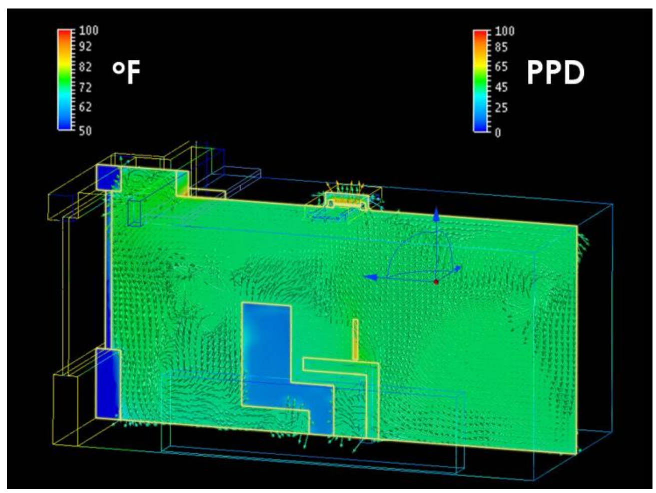
Active Chilled Beam only (Cooling mode)



Active Chilled Beam, Chilled Radiant Panel (Cooling mode)



Active Chilled Beam, Finned Tube Radiation (Heating mode)



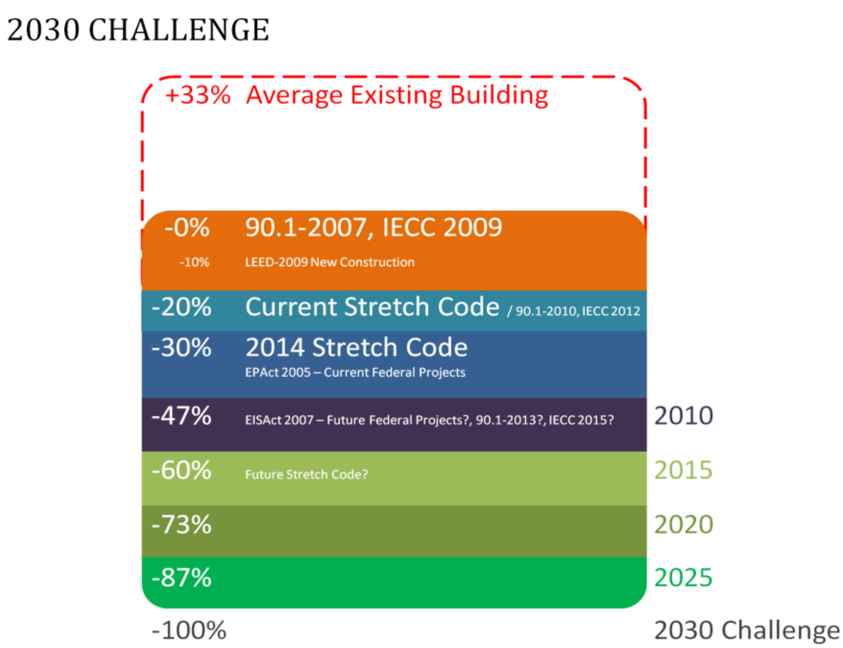
Active Chilled Beam, Radiant Panel (Heating mode)

The images on the previous page were among results reviewed to study the effects of the different systems on temperature distribution within the room and the likelihood of an occupant being thermally comfortable in each environment. Simulation CFD produced distribution plots of temperature, velocity, and thermal comfort criteria (PPD, PMV) on planes, isosurfaces, and points that were used to pinpoint simulation results.

It was concluded that the active chilled beam with chilled radiant panel was most thermally effective for cooling mode and the active chilled beam with finned tube radiation was most thermally effective for heating mode. However, the cost of implementing such systems needs to be evaluated by another means. In this case, Simulation CFD was used to communicate the better performing system. Ultimately, the decision is up to the engineer, architect, and client to determine the appropriate system.

Integrate CFD modeling into the design, optimization, and energy-simulation process for your projects

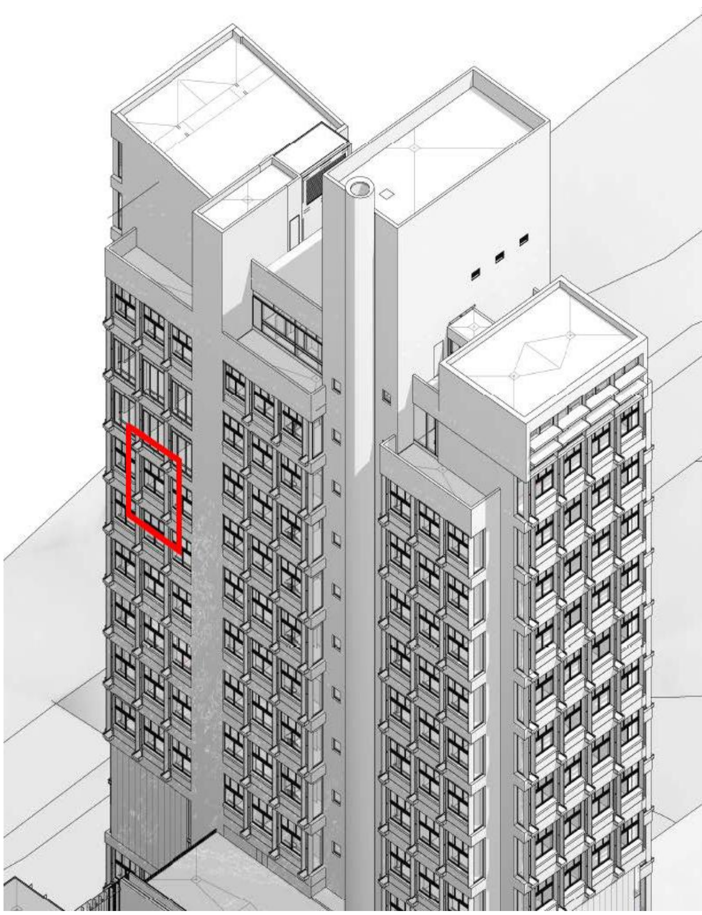
The application of CFD that we have discussed involves the analysis of a typical office space within an existing building planned to undergo a complete renovation. The existing building currently has outdated and efficient mechanical systems that attempt to combat a single pane glazing assembly. Similar to the graphic below, the existing building likely consumed far more energy than required by current ASHRAE standards. Looking into the future, energy standards for buildings will only become more stringent. Therefore, the use of energy simulation tools to accurately predict energy usage in a given building with a given system is highly critical.

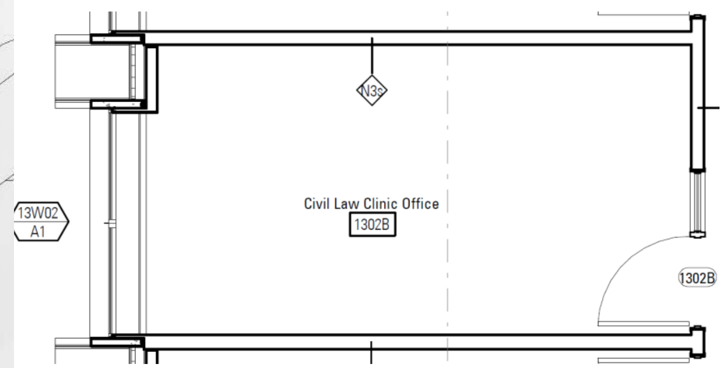


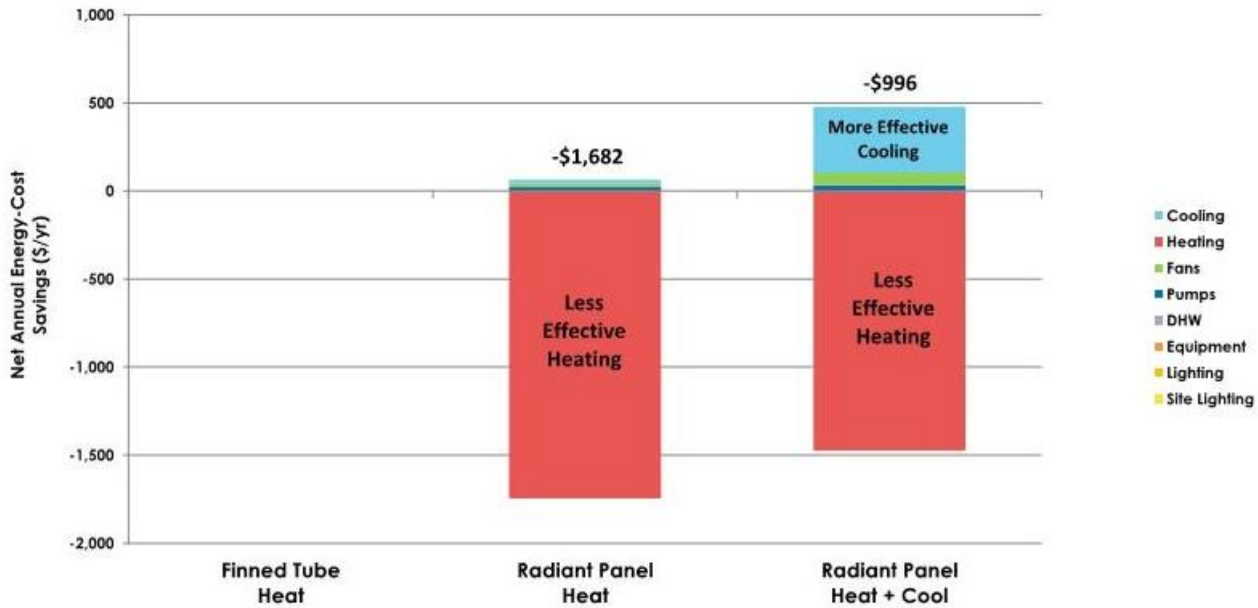
The existing structure, to remain untouched, was comprised of a low floor to floor height of 12'-6" feet with 9'-0" ceilings. Also, the layout of the floor plates in the "Tower" were fairly typical, hosting similarly sized perimeter offices and interior support spaces. Given those project conditions, the design team arrived at a chilled beam based air distribution system with the desire to limit the amount of airflow and thus the size of ductwork required in the ceiling plenum. Also, due to the repeated perimeter offices, a sample office zone could be analyzed with a high level of detail within a CFD modeling program and extrapolated to similar office zones. As mentioned before, a sample office space was analyzed with at the following conditions:

- Chilled Beam Ventilation and Cooling (Cooling mode)
- Chilled Beam Ventilation and Cooling, Radiant Cooling at Perimeter (Cooling mode)
- Chilled Beam Ventilation and Perimeter Finned Tube Radiation (Heating mode)
- Chilled Beam Ventilation and Radiant Heating at Perimeter (Heating mode)

Design conditions of 91^{o}F (cooling mode) and 0^{o}F (heating mode) were utilized.







Net annual energy-cost savings per year for the tower perimeter heating and cooling (negative values indicate worse overall performance)

Based on the results of the energy analysis, finned-tube perimeter heating was recommended in the Tower. Finned tube reduces the demand on the air handling units by compensating for envelope heat loss during unoccupied times in the winter, rather than requiring the air handing unit to cycle on to meet unoccupied temperature set points. Through the CFD analysis, it has been confirmed that radiant ceiling panels prove to be slightly less effective providing thermal comfort in heating mode. Corresponding adjustments in space temperature are assumed, resulting in increased energy-consumption.

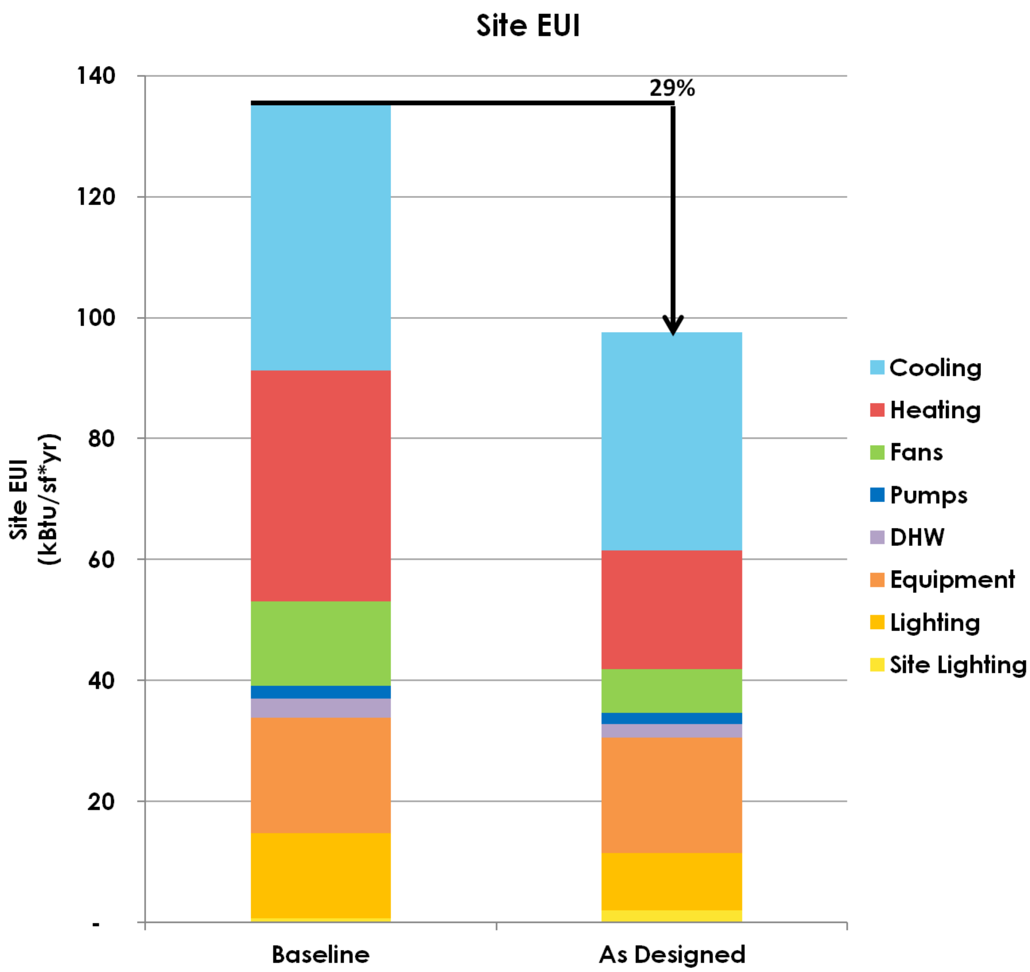
Another permutation incorporates a switchover of the radiant panels to radiant cooling mode, providing perimeter cooling when solar gains are high. Although this reduces overall cooling demand by providing improved radiant symmetry, the savings does not overcome the negative impact of the less effective heating mode. In addition, connecting the radiant panels to the chilled-water loop would likely result in a significant cost increase. Therefore, the design including finned-tube perimeter heating is recommended.

	Baseboard	Radiant Panel
Cooling	-	-0.25 ºF
Heating	-	+1 ºF

Thermostat credit (green) and penalty (red) associated with use of radiant panel.

Conclusion

For the project as a whole, we developed a comprehensive approach to Energy Conservation, incorporating a broad array of strategies. The energy model indicates **29%** savings in energy-cost compared to the ASHRAE 90.1-2007 baseline and is achieving **10** LEED points for energy performance (weighted score, based on the proportion of renovation vs. new construction). The analysis also shows **31%** savings in site-energy, exceeding the 20% minimum requirement for the Massachusetts Stretch Energy Code.



But, high-performance buildings are more than just energy-efficient. The value of providing thermal comfort to the building occupants, to support their health, wellbeing and productivity far exceeds the energy-cost savings. The enhancements to the perimeter zones in the tower, including window replacement, enhanced insulation and air-sealing, chilled beams and perimeter finned tube heat resulted in a high-performance cost-effective solution that will maximize thermal comfort, resulting in an immeasurable benefit to the client.