



Using Simulation 360 CFD: Patient Comfort and Energy Performance in Healthcare Design

Michael Ramsay – NBBJ

AB5381

The NBBJ healthcare practice is currently using Simulation CFD 360 software to study patient comfort in healthcare design. This class will show examples of the work, and we will examine analysis of the results of this work. We will examine in-depth studies of natural ventilation, infection control, and patient comfort, and we will discuss methods and results. We will investigate other uses of Simulation CFD 360 software in the study of energy-saving concepts as they relate to healthcare design and the design of other sustainable design solutions.

Learning Objectives

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

- Set up a Basic Simulation CFD 360 simulation
- Understand energy concepts being studied in healthcare design
- Understand new patient comfort concepts being studied in healthcare design
- Investigate the possible uses of Simulation 360 CFD software to enable design research

About the Speaker

Michael is an associate and digital practice leader in the healthcare practice of NBBJ. His primary focus is on BIM and delivery supporting the healthcare practice. Michael's research focus is on simulation and development of digital tools with an emphasis on client comfort, client experience and sustainability.



mramsay@nbbj.com

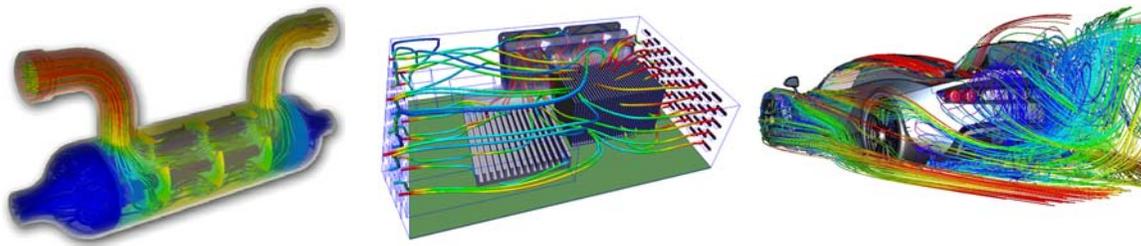
Table of Contents:

Introduction.....2
Set Up a Basic Simulation.....3-28
 Planning.....3
 Modeling.....4-8
 Import.....8-9
 Materials.....10-11
 BREAK!12
 Boundary Conditions.....13-16
 Meshing.....17-19
 Solve.....20-21
 Download.....22
 Analysis.....22-28
Case Studies.....29
 Patient Room 1.....29-37
 Patient Room 2.....38-42
 ICU Infection Control.....43-44
 Displacement Ventilation.....45-48
 Natural Ventilation.....49-52

Introduction:

“Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows.”
- Wikipedia

CFD has traditionally been used to solve mechanical and product design challenges. The car, the mechanical assembly, the plane in a wind tunnel are all familiar uses of CFD. The most common architectural example is the data center. Is there an opportunity to inform the architectural design process at an early stage through the use of CFD? The healthcare practice in NBBJs Seattle office has used CFD methodologies to support ongoing design work and to research ideas around patient comfort, infection control and sustainability.

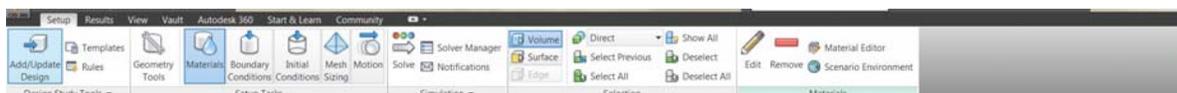


This handout describes the process used in setting up an architectural CFD simulation and will cover the design research in a more limited manner than the lecture. There are different ways to model the simulation geometry and work through the simulation process. This document will show one method.

Before Getting Started:

The Software: There are many CFD software solutions available to the end user. All of the following work was done using Autodesk's Simulation CFD 360 software. Different software may have different suggested workflows and should be investigated independently. Each part of the process will be described in depth in the following sections of this document.

- 1.) *Model your design in some authoring software (Revit, MAX, AutoCAD, other)*
- 2.) *Import the model into the Simulation CFD 360 software*
- 3.) *Apply materials to the model*
- 4.) *Apply boundary conditions to the model (BCs)*
- 5.) *Mesh the model.*
- 6.) *Setup the solve options*
- 7.) *Submit the simulation to the solver*
- 8.) *Download the results*
- 9.) *Analyze*



Set up a Basic Simulation 360 CFD simulation

Planning:

Concept: Definitive vs. Comparative

One of the hardest things to understand is the fact that simulation gives no definitive absolutes. The sheer number of possible variables to consider means that most simulations will never address each and every one. Simulation's strength is in comparative, data driven, results. Will *this* solution perform better than *that* solution? It is important to plan the simulation before starting most of the work. The list below is a good starting point for planning the simulation.

Scope of the study:

- What is being studied, What is the desired measurable outcome?

Inputs / Outputs:

- What are the inputs and outputs for the study? These may be static loads, input temperatures, output velocities. Typically these are one of the most important drivers of the simulation so consider them carefully.

Simulation Structure:

- Will this work be done in one simulation or through a series of simulations?

Modeling Methodology / Modeling software:

- What really needs to be modeled to complete the simulation?
- What elements are superfluous?
- Is the model overly complicated?
- Can the model be iterated or will it have to be re-built and re-setup each time?

Results:

- How are the results being analyzed?
- Is the analysis data driven or observational?
- Do the results need to drive the next iteration of the simulation?

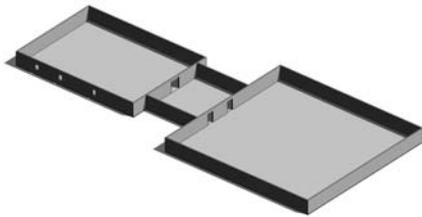
Each simulation is different and there may be other things to consider when setting up the simulation. The list above is fairly standard and addresses the basic considerations.

Modeling:

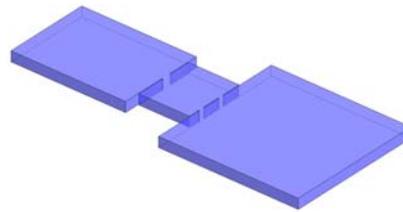
Concept: Modeling

A SHIFT IN THINKING: When using CFD simulation software it is important to understand that we are studying the movement of fluids and gases in space. Traditionally we model the bounding elements of the buildings we design. These are walls, doors, windows, floors, ceilings etc. These elements can define the space and represent design intent but they do not provide a domain for the liquids and gases to move in and interact. For this reason simply modeling the building isn't sufficient for CFD simulation. Part of the planning process suggested in the previous section is to plan the model structure. It is more important that the void spaces are modeled (Image 1)

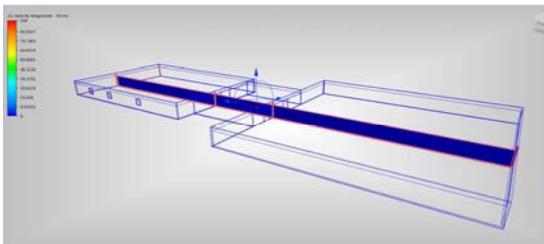
Traditional Revit Elements



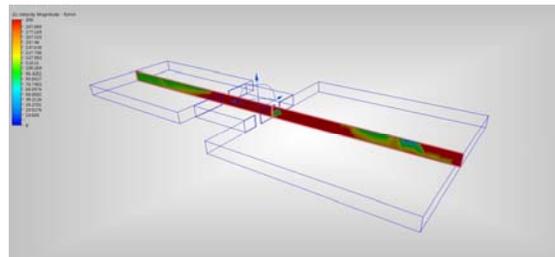
Void Space – Air domain



Traditional Revit Results (no air domain)

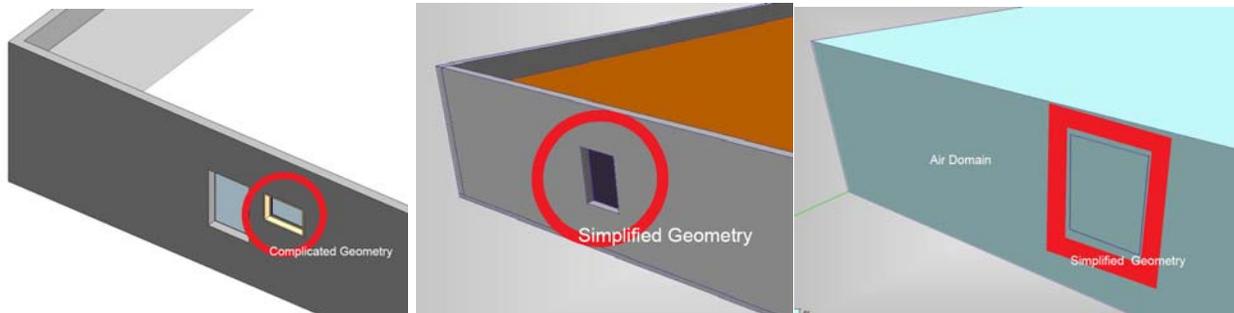


Void Space Results – Velocity.



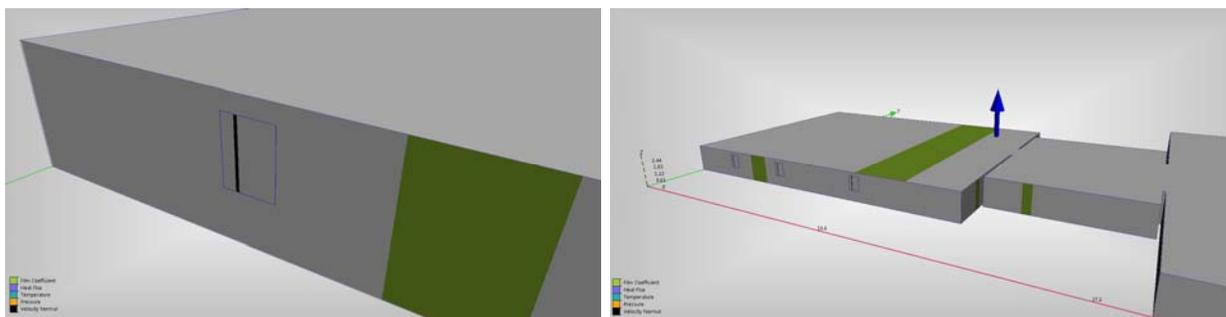
ANOTHER SHIFT: Consider that the typical multi-layered building elements such as walls and doors and windows can be frustrating to work with. While we need to model the gas / fluid domain of the simulation we can also use the surfaces of that domain to define the boundary conditions of the boundary elements. This simplifies the process but can also mean that there is some re-modeling to do when you move a design modeled in a tool like Revit to Simulation CFD 360. In the images below the complex modeling elements in Revit have been adjusted to be simple geometry, the air domain was modeled in Revit as a mass and the window families were simplified to be a basic rectangular extrusion with the extrusion directly adjacent to the air domain.

Modeling (cont):



Revit model example – The complicated window family (left) is not a good fit for CFD simulation. The simplified window (middle, glazing only) is a better option. Walls, ceilings, floors, are useful for context but will be less useful in the simulation. It's best to simplify the model to the most basic, relevant, objects (right)

The simplified geometry, the glazing and the air domain, can be used to set up the boundary conditions of the simulation without having to have the walls in place. The black line on the glazing object and the green lines on the other surfaces show that boundary conditions have been applied. The black represents an air velocity applied to the glazing object. The green lines show that a boundary condition representing the wall and the environment of the adjacent spaces is represented. The velocity boundary condition is creating 100 ft/min air flow at the glazing. The Film Coefficient boundary condition is applying temperature factor loads to the exterior walls and a temperature factor load to the interior walls adjacent to other internal spaces. Notice that all of these conditions are applied directly to the Air Domain and Glazing masses without the need for a double layered wall or other construct.

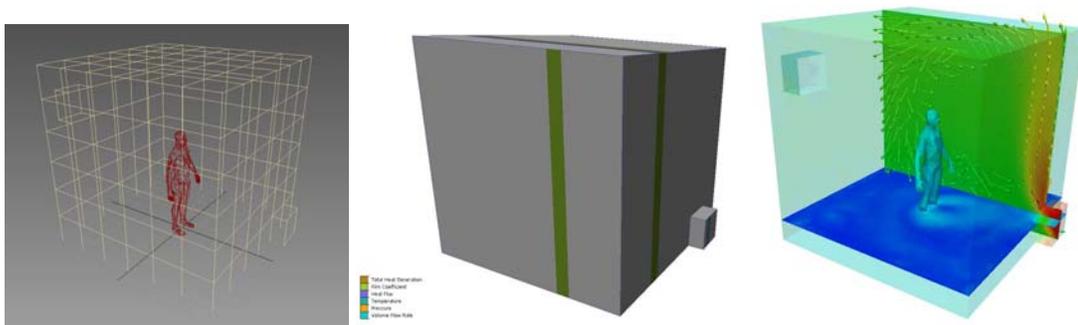


The black represents an air velocity applied to the glazing object. The green lines show that a boundary condition representing the wall and the environment of the adjacent spaces is represented. The boundary condition is applied as a factor of the total outdoor temperature taking into account the effect of the wall construction to reduce temperature differentials.

Modeling (cont):

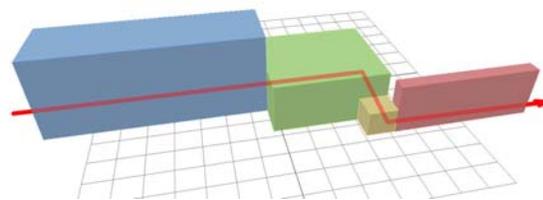
Process: Design Modeling

- **Modeling:** Basic modeling can be done in many different software. The most important thing to consider is the need to move the model from the initial modeling software to the simulation software. Some Autodesk products have a direct pipeline to the Simulation CFD 360 software. Alternatively, different file formats can be exported from the initial modeling software and imported into the Simulation CFD 360 software.



Simple solution modeled in 3D studio MAX. This model consists of a single room object and single avatar. The Single room object makes it easy to setup the simulation.

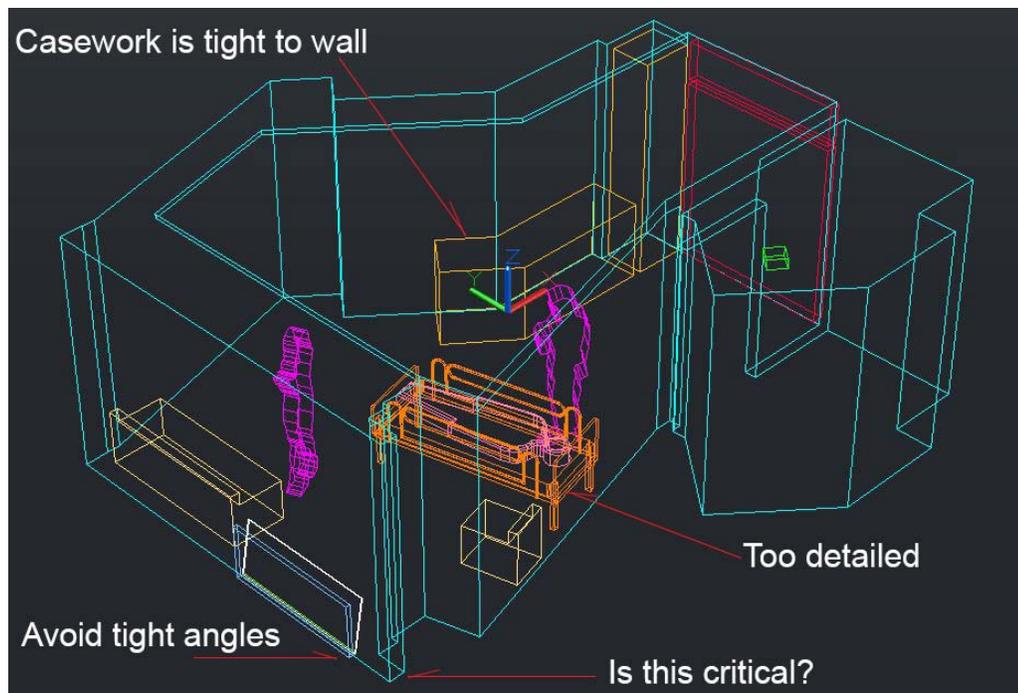
- **Concepts:** Most of the modeling concepts applied for a successful simulation are familiar concepts used regularly in the AEC industry. Key is the need for geometry to resolve as solid and not simple surfaces. Ultimately Simulation CFD 360 will utilize surfaces for the boundary conditions but the air domain should be a volumetric entity.
- **Plan the connections in the model:** In most cases it is important to have a contiguous air domain, even if that domain is more than one space. Directly adjacent rooms are simple but connections between a window, a room and a corridor may require careful consideration of how those spaces are connected.



Contiguous Masses representing connected air zones

Modeling (cont):

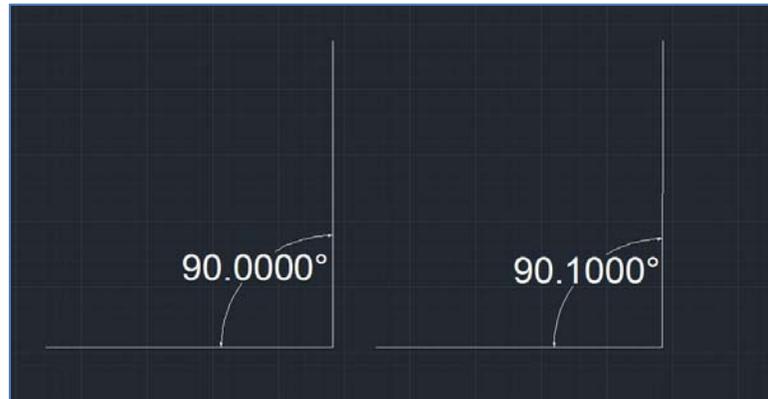
- **Structured Model:** Avoid overly complicated models. Models with every piece of furniture and equipment modeled, every geometric intricacy perfectly represented, should be avoided for an initial simulation. Set up your model to be as simple as possible initially. This will allow you to quickly setup the simulation and discover any mistakes in the modeling or the simulation setup. Consider the next step in the simulation and prepare for this but limit the detail.
- **Simplify or eliminate complex geometry:** tight corners, acute angles and small spaces between objects can have undesired affects. Ask “is this critical?” and remove any unnecessary conditions. Overly detailed content is also not useful. Simplify.
- **Avoid sliver spaces:** If there is casework against a wall make sure that it clearly touches the wall object. Small sliver spaces will increase solution time and complexity. If it’s unclear that the object is touching extend the object to penetrate the adjacent object. For example, move the casework so that it clearly protrudes from the air mass representing the testing environment. Items outside the air mass will be cut at their intersection with the air mass and those parts remaining outside the air mass will be ignored.



Avoid the issues above. Model deliberately and carefully.

Modeling (cont):

- Make sure things are straight: If you are working from a 2D background make sure the background and line work are rectilinear. Objects that are slightly angled may give the impression that they are straight but will result in 3D objects that do not exactly touch and will have to be remodeled.



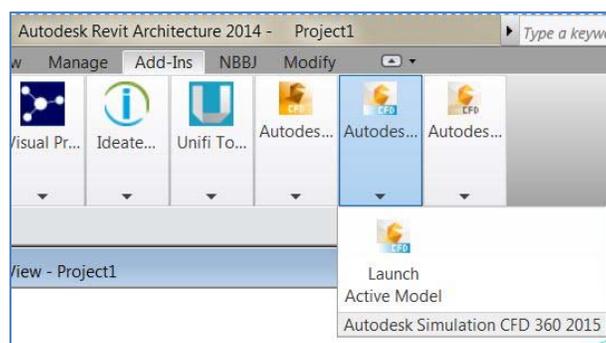
Lines that are slightly off 90D can cause tracing problems

Import

Process: Import model geometry

To import the model into the Simulation CFD 360 environment either use the available add-in in Autodesk software, or, export the model to a file format that Simulation CFD 360 can utilize.

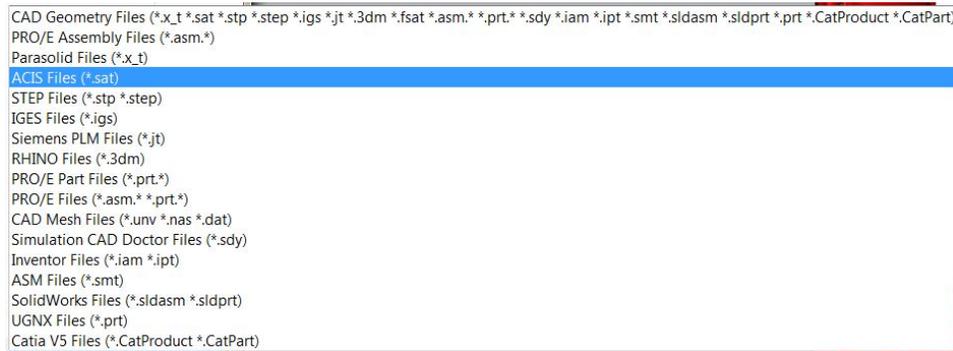
Add-ins: Installing the Simulation CFD 360 software should also install the needed add-ins for each Autodesk product. This is typically Revit, Inventor, and others. Click the “launch” button to get started.



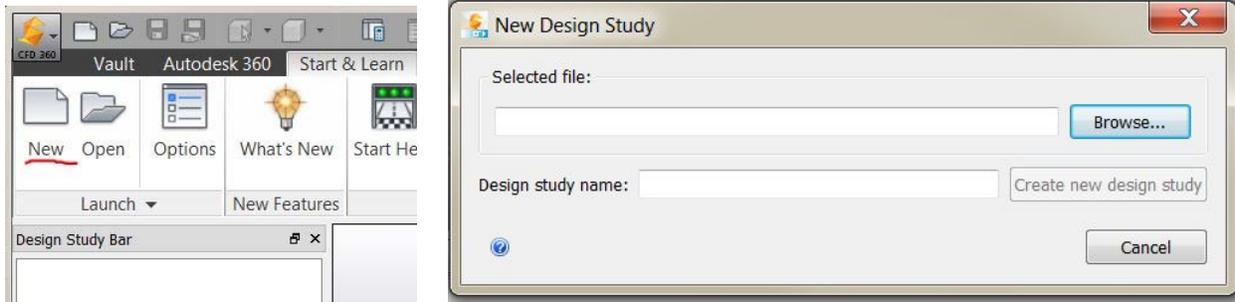
Add-in gets installed when Simulation CFD 360 is installed

Import (cont):

File Formats: Simulation CFD 360 accepts many different formats. There is no “perfect” format. Use the format that works for your project.



To bring the exported file into Simulation CFD 360 open the software and click the “New” button at the top left of the screen. This will open the New Design Study Dialog, browse to the exported file and select it. Then give the design study a name and click the “Create New Design Study” box.



Starting a new design

Caution:

- Each Simulation CFD 360 project creates an extensive file folder structure. Make sure you spend time to name and organize the different simulations clearly.
- Simulation CFD 360 projects have large file sizes. A single model can be GB worth of storage. Plan accordingly. Some complex simulations can be 100 GB+
- Work with projects stored locally or on locally attached storage. Projects stored on networks can be problematic. Remote storage is not suggested.

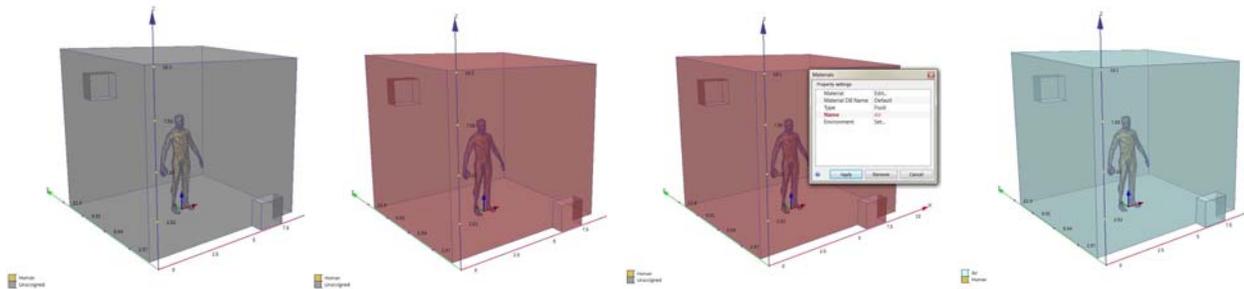
Materials:

Process: Apply materials to the model

Simulation CFD 360 comes with many predefined materials. A successful simulation can be achieved using these predefined materials. If necessary new materials can be made and applied on the fly. Materials are applied in the Simulation CFD 360 software.

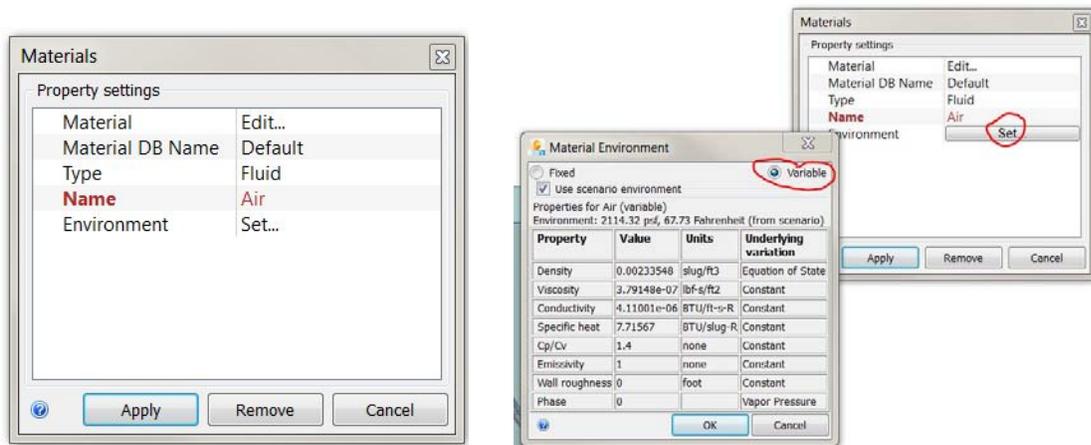
To apply materials:

Open the model and select the objects you want to apply a material to. Once selected click the edit button at the top of the screen or from the pop up right click menu. This will open the material assignment dialog box. Choose the material and click Apply.



Applying materials left to right: Select object, press “edit” choose material and apply

Materials like Air have additional settings that must be considered. Most materials and their properties are static and can simply be applied. Some materials will need to have variable properties and this is intentionally set through the materials dialog box. Air is critical to most Architectural studies and must be set to “variable” Click the “Set” button and choose “Variable”

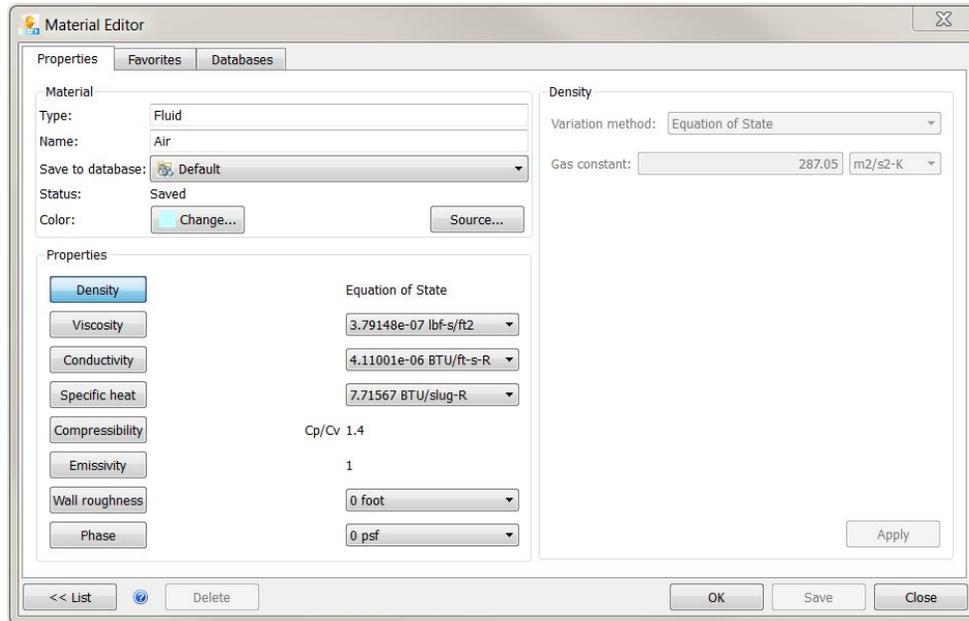


Material Dialog box

Setting Variable properties

Materials (cont):

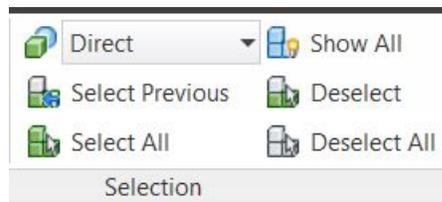
New Material can be built and saved in a user database through the material editor.



Material Editor Dialog Box

Caution(s):

- Be careful when selecting objects. Simply clicking the white space surrounding a model will not de-select the object and you may end up applying materials to the wrong objects. Make sure to use Select and De-Select accordingly.

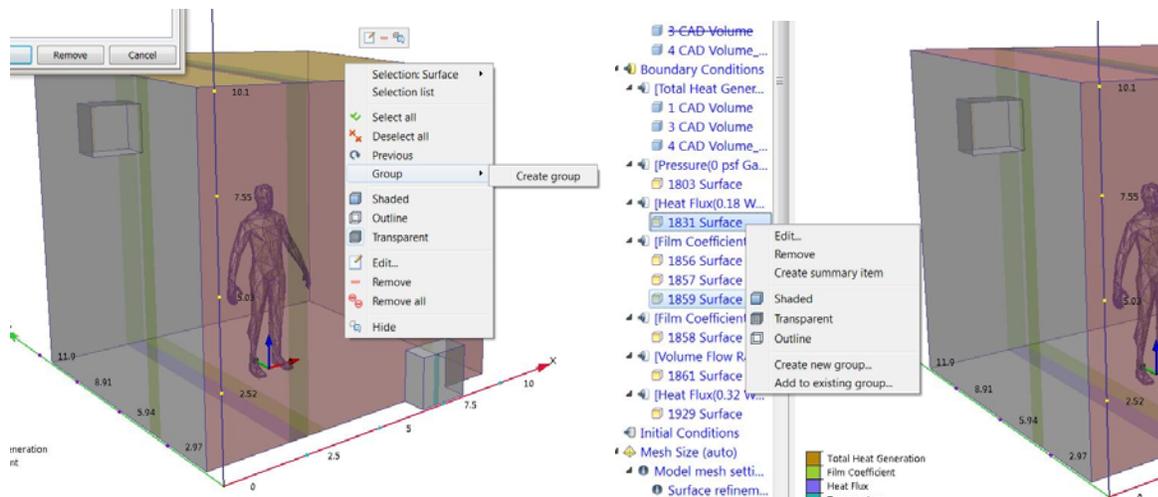


- Everything needs a material but not all materials are critical. When studying a small mechanical assembly material properties are critical, when studying architectural spaces the materials may not be as important. Temperature and Radiant interactions between a glazing object and the space may be critical. Temperature and Radiant interactions between a P-Lam counter and the space may not be important.

BREAK!**Advice: Some time saving recommendations.**

Before we jump into Boundary Conditions let's investigate some time saving processes that should be understood and applied at this stage and prior to completing the simulation setup.

- **Save:** Save often. Saving is easy and will prevent lost work down the road.
- **Scenarios:** Each design study can have multiple scenarios. A scenario is a specific test case with specific variables. Multiple scenarios are used to test different inputs and outputs per design. Do all of the setup work in a single scenario before creating new scenarios. All of the settings will be consistent and only the specific variable changes will have to be modified. Material, Boundary Condition, Meshing and Solver settings will be consistent in each scenario.
- **Grouping:** Grouping of objects, volumetric or surface, will allow for the quick selection of objects in different scenarios. Grouping will save a lot of time and effort. It's possible to group complete objects and also individual surfaces. Plan to have the objects you know you want to change scenario to scenario grouped and they will be easy to select and change. Grouping also allows for more flexibility in navigating the model as groups can be hidden and isolated with a single click. Group before cloning the scenario.



Grouping options: Select the items in the modeling environment or from the design study bar

- **Suppressed Geometry:** Do not be afraid to suppress an element if it will have no effect on the eventual simulation. Suppressed objects can have a geometric interaction in the simulation without effecting the temperature, velocity and radiant interactions.

Boundary Conditions

Concept: Boundary conditions (BC)

From the Help File:

“Boundary conditions define the inputs of the simulation model. Some conditions, like velocity and volumetric flow rate, define how a fluid enters or leaves the model. Other conditions, like film coefficient and heat flux, define the interchange of energy between the model and its surroundings.

Boundary conditions connect the simulation model with its surroundings. Without them, the simulation is not defined, and in most cases cannot proceed. Most boundary conditions can be defined as either steady-state or transient. Steady-state boundary conditions persist throughout the simulation. Transient boundary conditions vary with time, and are often used to simulate an event or a cyclical phenomena.”

I cannot describe it any better than that. Boundary conditions are the settings that change scenario to scenario. They can be temperature and flow speed inputs. They can be the representation of an external wall or window. Many complex conditions can be represented with a boundary condition or series of boundary conditions applied to a model object or surface.

Boundary Conditions

Process: Apply Boundary Conditions (BC) to the Model

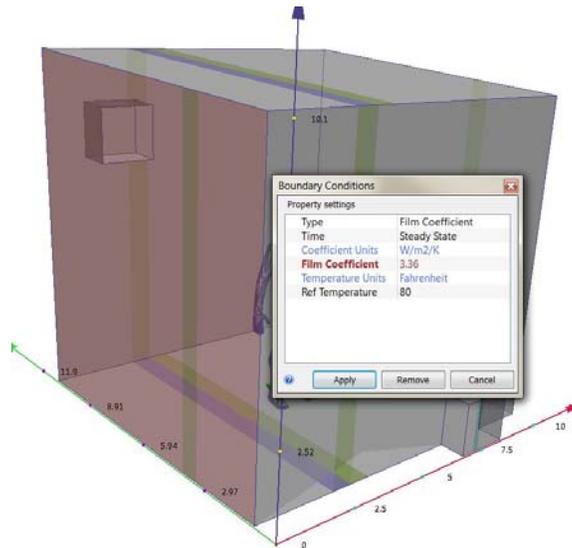
Once materials have been applied to the next step is to assign boundary conditions to the model. Unlike materials, not all objects need a boundary condition. Applying a boundary condition (BC) is similar to applying a material. Unlike a material application boundary conditions can be assigned to volumes and surfaces. Primarily the BCs will be applied to surfaces but any volumetric objects, such as a human avatar, will need to have a volumetric BC applied to it. Multiple boundary conditions can be applied to the same object. For example, temperature and flow velocity boundary conditions would be applied to the same surface representing a glazed opening.

There are many types of boundary condition and these should be reviewed and understood before beginning the simulation. The help file has decent descriptions of each and how to use them. Different studies will require different boundary conditions. The boundary conditions used for this lecture will be described below.

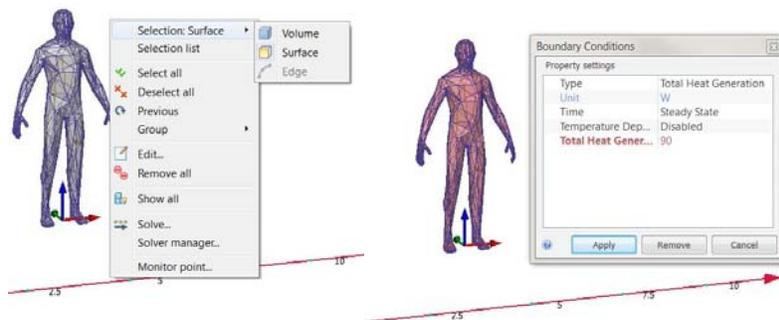
Boundary Conditions

Process: Apply Boundary Conditions (BC) to the Model

Select the boundary condition tab at the top of the screen. Next select the surface, or series of surfaces, and choose the Edit button from the top of the screen or from the right click menu. Choose the type of boundary condition and set the individual settings for the condition. Then select Apply.



To apply a volumetric BC select Boundary Condition at the top of the screen and then right click in the model environment (in void space, not over the model) and select “Volume” for the selection type. *NOTE: this is not the same as using the Volume, Surface, Edge options under the selection tab at the top of the screen. Different Surface BCs and Volumetric BCs are available when the correct selection mode is chosen. You cannot assign a volumetric BC to a surface.* Next, choose the BC and click apply.



Applying volumetric Boundary Conditions

Boundary Conditions

Concept: Boundary Condition Types

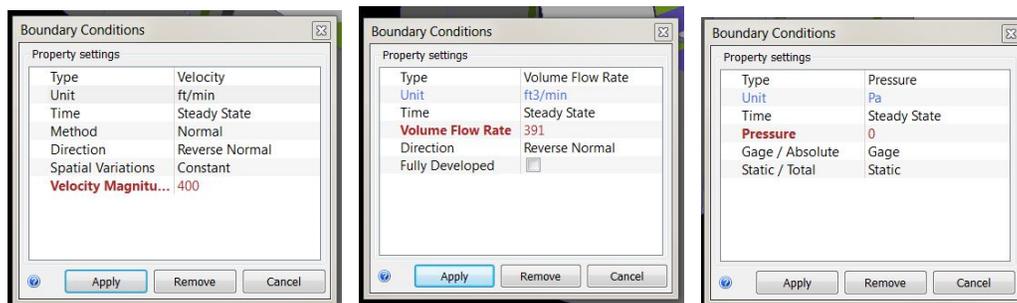
The Simulation CFD 360 help file lists the different types of boundary conditions. They are broken down into different categories such as Flow Boundary Conditions and Heat Transfer Boundary Conditions. There are more categories available and these can be investigated at your leisure. The boundary conditions below fall into either the Flow or Heat Transfer category.

Flow Boundary Conditions:

Velocity (unit/time): This BC adds flow velocity to the simulation. Used to add wind, mechanically provided air and other flow variables. This BC can be applied normal to the surface or as component vectors. It can be static or transient.

Volume Flow Rate (unit/amount/time): This is similar to velocity but handles flow totals and not necessarily flow velocity. This is good for mechanical systems where the cubic feet per minute rate might be known but not the specific velocity of each outlet.

Pressure: This is a necessary BC in all simulations involving flow. The pressure BC is typically set to zero (0) and is needed to ensure that the simulation solves correctly. Think of the simulation solver as understanding the inputs and outputs and solving for the interactions between those events. If the inputs and outputs are set to a hard velocity or Volume Flow Rate then the simulation has nothing to solve for. The 0 pressure remedies this.

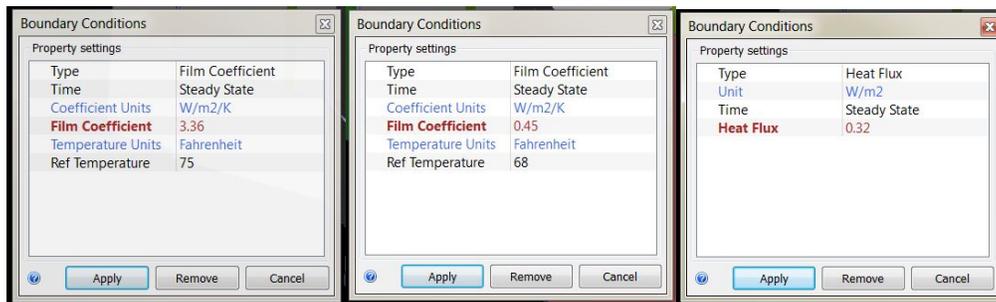


Examples of different Flow Boundary Condition types and settings

Heat Transfer Boundary Conditions:

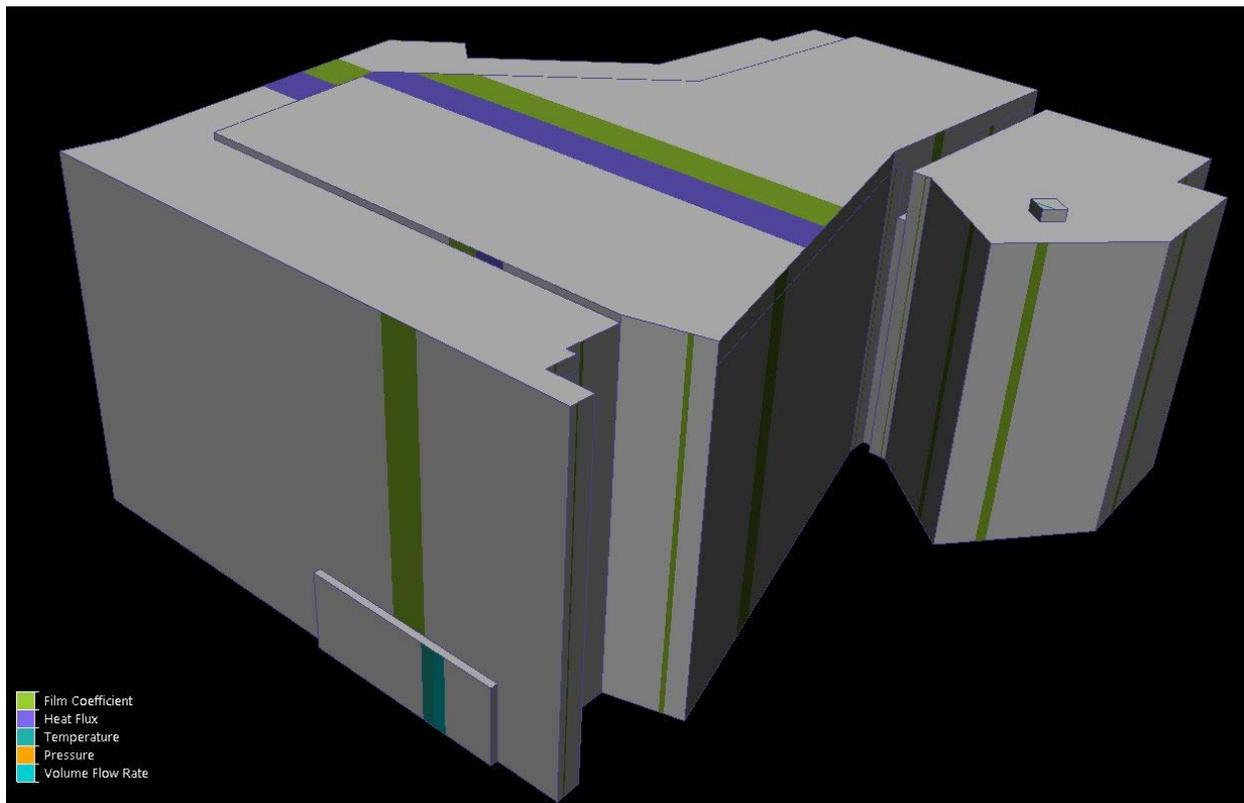
Film Coefficient: This BC is used to represent the flow of heat across a boundary. Typically these are walls, exterior or interior, represented by the surfaces that make up the air domain. Exterior wall application would be a $W/m^2/k$ factor with a reference temperature set to match the exterior temperature of the simulation. The interior walls would use a similar method but with a constant temperature representing the likely heat transfer between exterior spaces

Heat Flux: This BC allows for the addition and consideration of heat and cooling loads that might not necessarily be modeled but will have an impact on the simulation. A simulation of a large space could use the Heat Flux BC to apply a heat load factor without having to add each human avatar to the space. This not only improves model performance but also speeds up solve time. Think of the Heat Flux BC as used to represent other loads.



Heat Transfer Boundary Conditions and Settings

All of the applied Boundary Conditions can be seen in the image below. The stripes on the surfaces and volumes show the BCs applied.



Boundary Conditions Applied

Meshing

Concept: Meshing

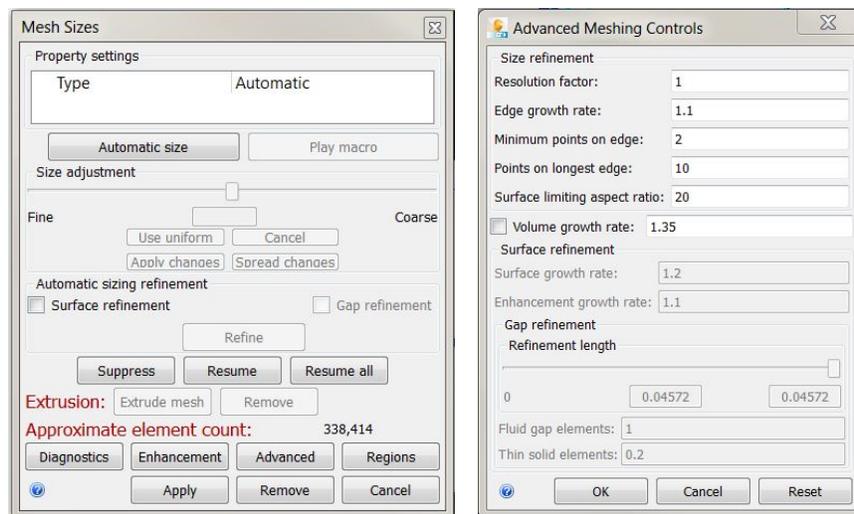
Meshing the model is a required part of building up the simulation. Entire courses could be taught about meshing. We will cover the basics in this document. Meshing the model can be as simple as clicking on the Mesh Sizing tab at the top of the screen and then pressing the Autosize button. For many simulations this will be sufficient. The density of the mesh is what is most important. Primarily the meshing density around areas of study and interest. There's no need to have a complex tight mesh on an area of the model where nothing is happening but it may be desirable to have a complex mesh where two velocities interact or at a small opening that air is passing through and it's critical to see that interaction. Many times a simulation solution will fail due to improper meshing so it is worth investigating the mesh settings.

Meshing

Process: Meshing

To mesh select the Mesh Sizing tab and either edit the mesh settings prior to autosizing the mesh or autosize the mesh and then adjust the settings and re-apply as needed.

The meshing edit button brings up the mesh settings dialog. This is where adjustments and refinements can be made. Click on the advanced button to open the specific mesh settings applied when the autosize feature is used.

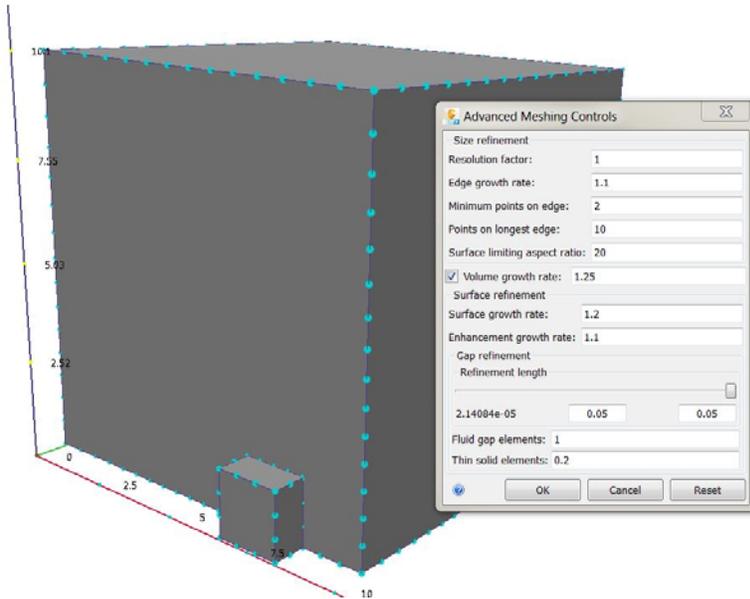


Meshing Dialog and the Mesh application settings

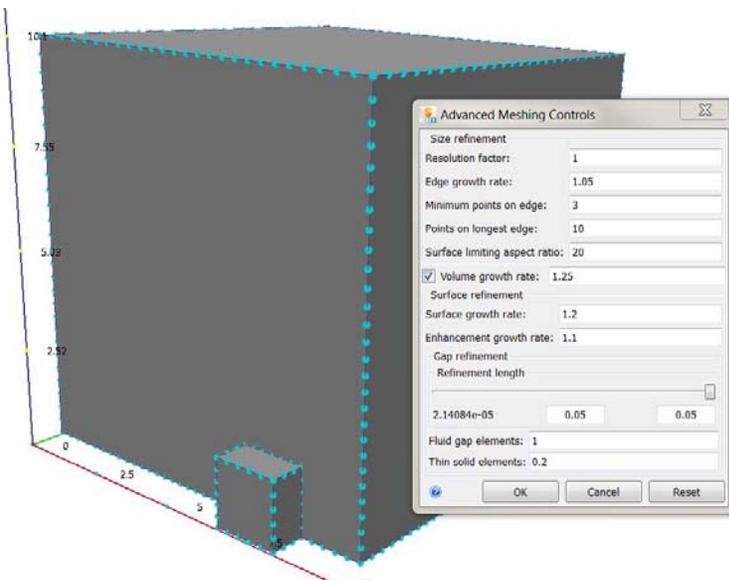
Meshing (cont)

Process: Meshing

The examples below show the effect of different mesh settings on the model. A good tight mesh will provide better results and more granularity.



Default Mesh Sizing: The default settings used when Autosize is used.

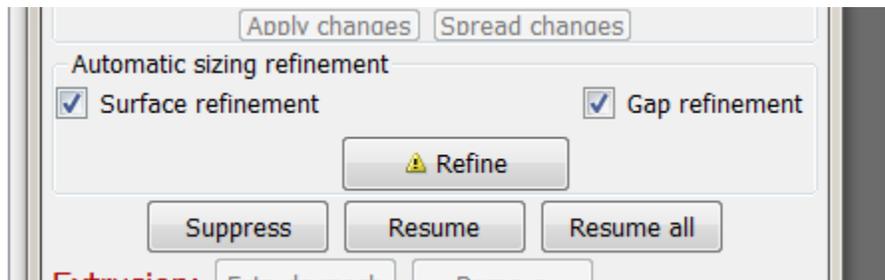


Adjusted Mesh Settings: The edge growth rate and minimum points on edge settings were changed.

Meshing (cont)

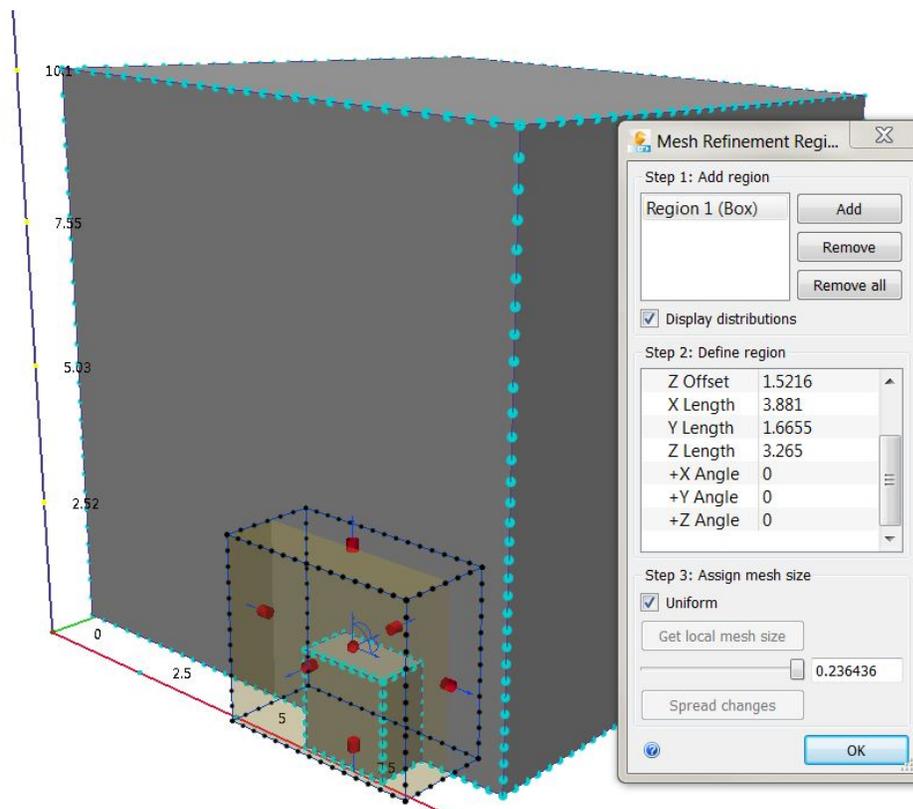
Process: Meshing

Mesh refinement can also be used to reduce unnecessary surface meshing and gap meshing.



Surface and Gap Refinement

Regions can be used to increase the mesh density at areas of interest without having to tighten the mesh overall. This is handy when working with big models and when it's important to have accurate results at a certain location in the simulation but not overall.



Mesh Refinement focuses area of interest and provides more accurate results at that location

Solve

Concept: Solution Settings

When the materials are applied, the boundary conditions setup and the meshing complete it is time to send the simulation to solve. Simulation CFD 360 uses a cloud based solver that greatly improves speed and allows for multiple simultaneous solutions. Local solvers will take longer and will not necessarily be able to solve multiple simulations simultaneously. Both options work and will provide results. The solver can be set and run for a single scenario or multiple scenarios can be run from the solver manager. It's recommended that you set your solver settings with the first scenario setup and then clone the scenario. This will ensure the same settings are used.

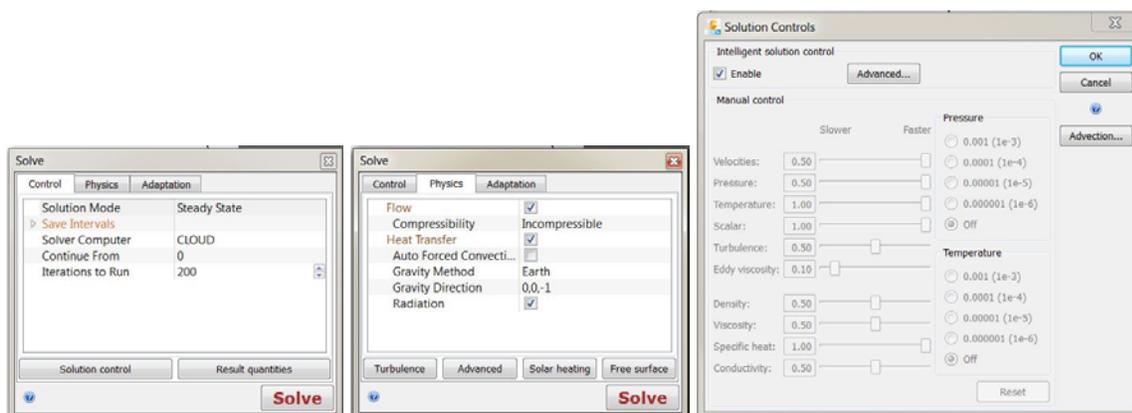
Basic Solver Control Tab: Set the basic settings, Solution mode, Solver computer, iterations etc.

Basic Solver Physics Tab: Enable Flow, Heat Transfer and Radiation (if you are solving for PMV) and set the gravity direction (look at the coordinate axis for your model to see what is the top and bottom and set gravity accordingly, models don't always import with the top up)

Basic Solver Solution Control Button: Use this button to ensure that intelligent solution control is set to on and to change the advection method.

- Advection is the numerical mechanism of transporting a quantity (velocity, temperature, etc.) through the solution domain
- These examples are using Advection 5.

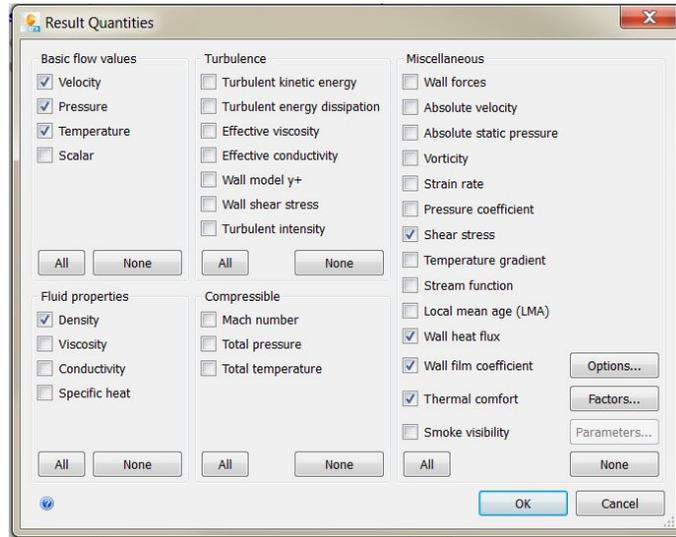
Basic Solver Result Quantities Button: Use this to choose the required result quantities. Check thermal comfort if solving for PMV.



Basic Solver setting and solution control: Control and physics tabs

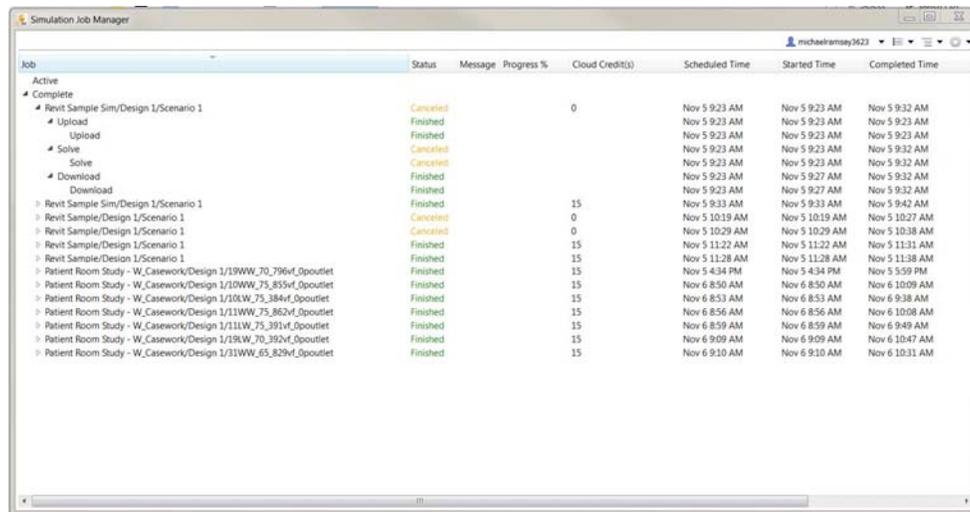
Solve (cont)

Concept: Solution Settings



Advection and Result Quantities dialogs

When the solution is sent to the solver there will be a small period of time wherein the simulation is submitted and all of the settings are applied. The Simulation Job Manager will open and show the active simulation. Times to solve vary depending on the complexity of the solution. The model does not need to remain open during the solve.

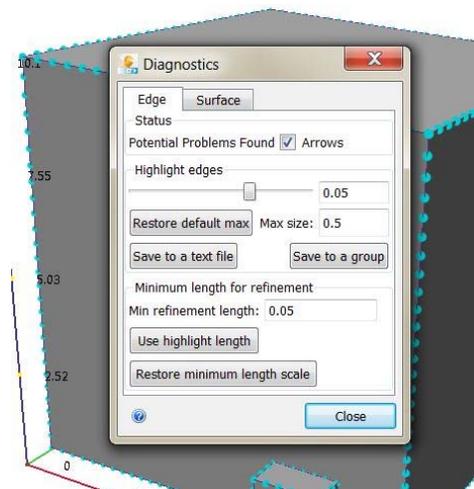


The Simulation Job Manager

Download

Process: Downloading Results

When the solution is complete the results will be automatically downloaded to the model when opened. Any errors or failures of the solution will be indicated at this point. If there are errors work through the possible solutions offered. It may be necessary to make some adjustments and re-submit. Errors are often a result of mesh errors. Use the diagnostic tools under the mesh tab to try and isolate the issue. Errors and meshing errors are different for every project.

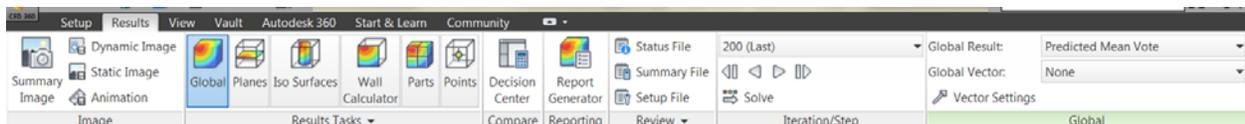


Meshing Diagnostics Dialog Box

Analysis

Concept: Analyzing results

There are many ways to review the results from the simulation solution. Planes, vectors, traces, and more can be enabled to show the results. Data can be reviewed through the use of the data output tools. Below are examples of each type of analysis. Use these to support the design variables being considered for your project.

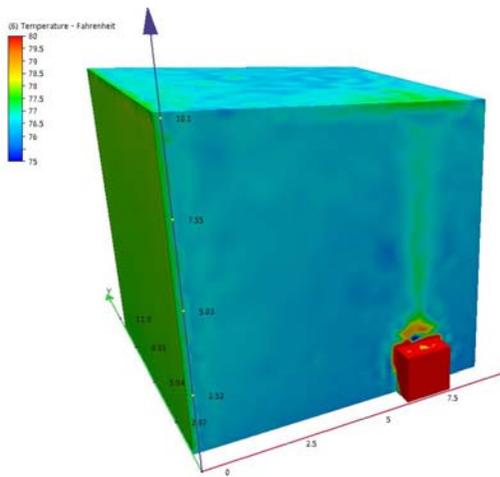


The results tab

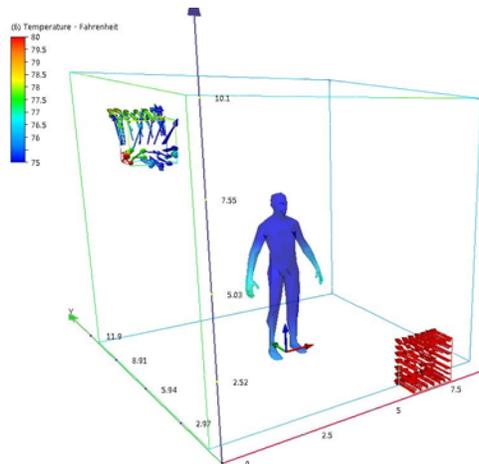
Analysis (cont)

Concept: Analyzing results

Global Results: this can be used to show results on the overall model. Predicted Mean Vote (PMV) and other comfort analysis results are applied at a global level. Vectors can be enabled if needed.

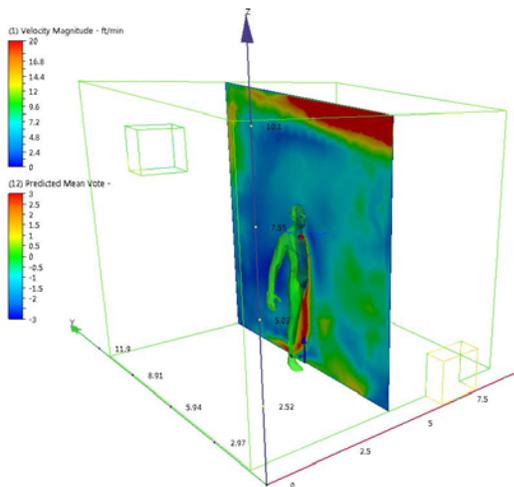


Global Temperature Results

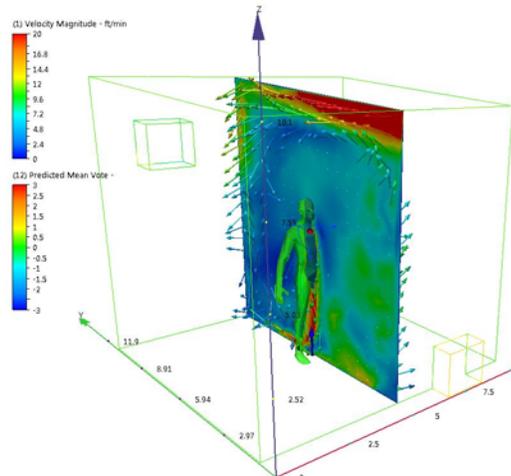


Global Results with Vectors

Planar Results: Planes are a great way to visualize the interaction in the simulation. Like all analysis methods planes can show different types of. Vectors can be enabled for the plane and set as needed.



Planar Velocity Results

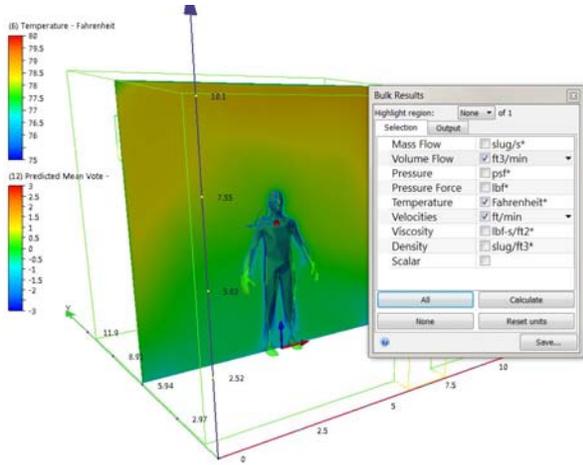


Planar Results with Vectors Enabled

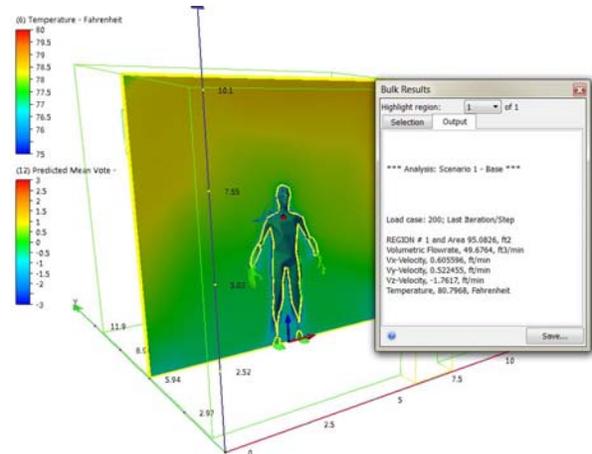
Analysis (cont)

Concept: Analyzing results

Planar Results Bulks: Bulk information per plane are very useful and show data from the solution. Use a plain to establish the area of study and enable bulk data.

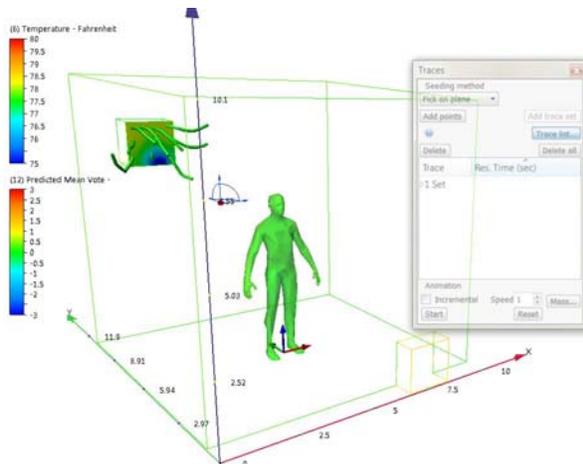


Bulk Data Results: setup

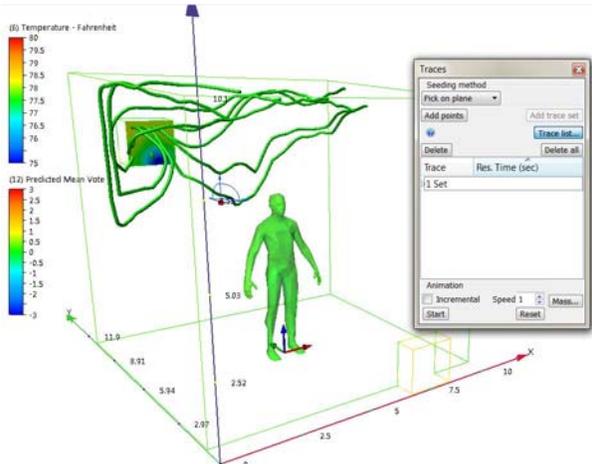


Bulk Data Results: Data

Planar Results Traces: Traces can show how inputs move throughout the simulation. Think of these as similar to a smoke test. Use the planar results mode and select traces to use. Traces are emitted from the plane.



Trace Results: Traces @ 20 steps

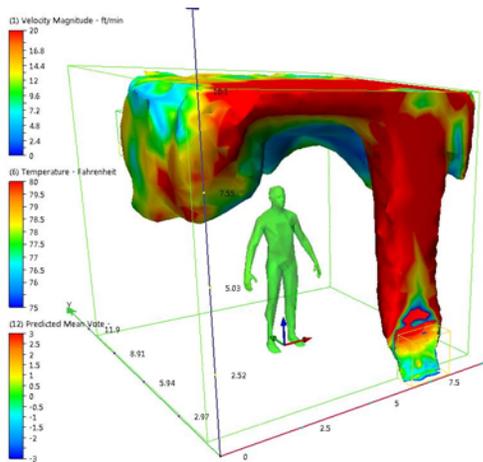


Trace Results: Traces @ 50 steps

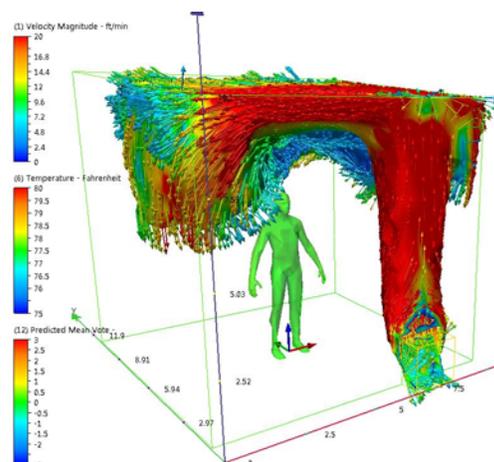
Analysis (cont)

Concept: Analyzing results

Iso Surfaces: Can be a compelling way to visualize results. Iso surfaces can be set to show a specific result, for example, it is possible to set an iso surface that is only showing 78.5 degrees F. Vectors can also be enabled.

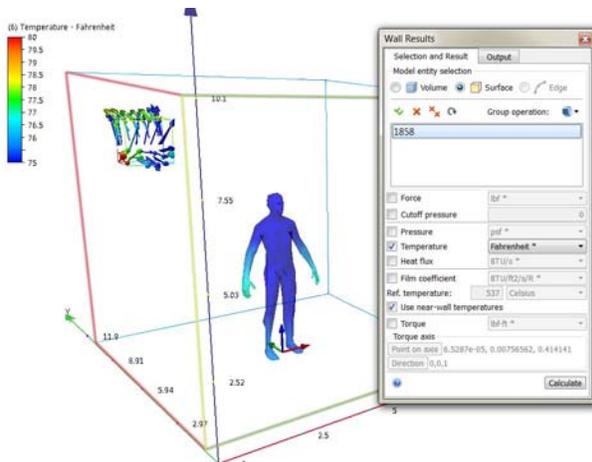


Iso Surface showing 78.5 DF, colored by velocity

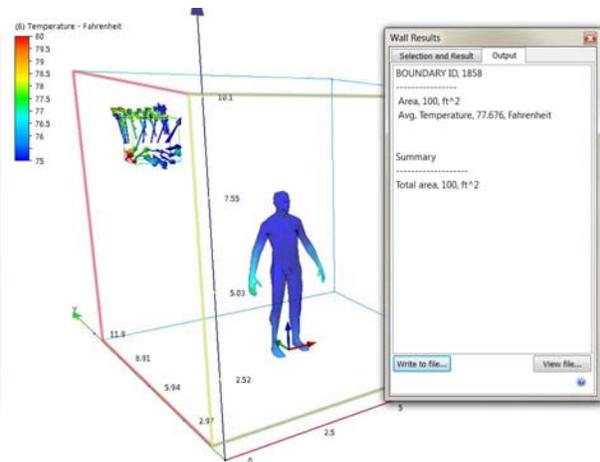


Same surface with Vectors

Wall Calculator: The wall calculator can be used to select a wall or series of walls and get specific information about that wall.



Wall Calculator Setup

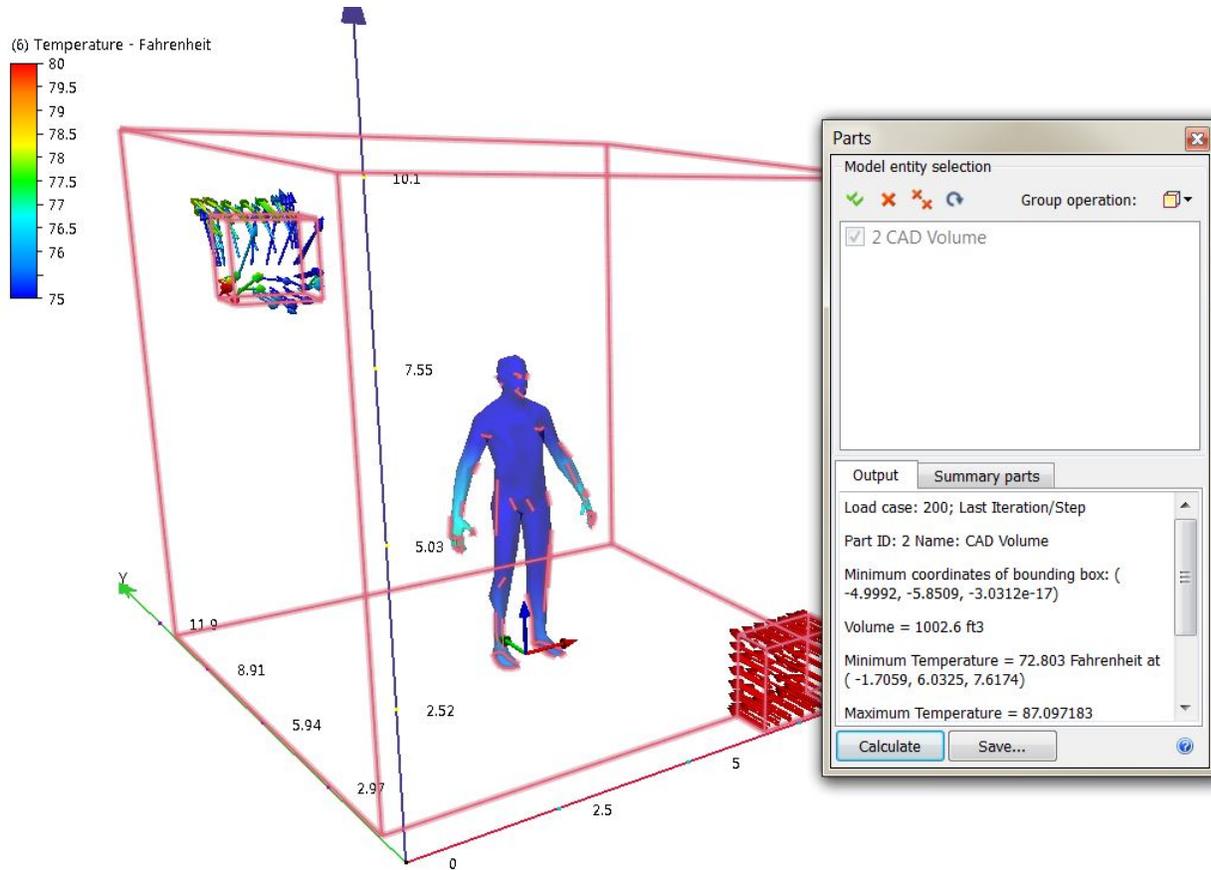


Wall Calculator Results

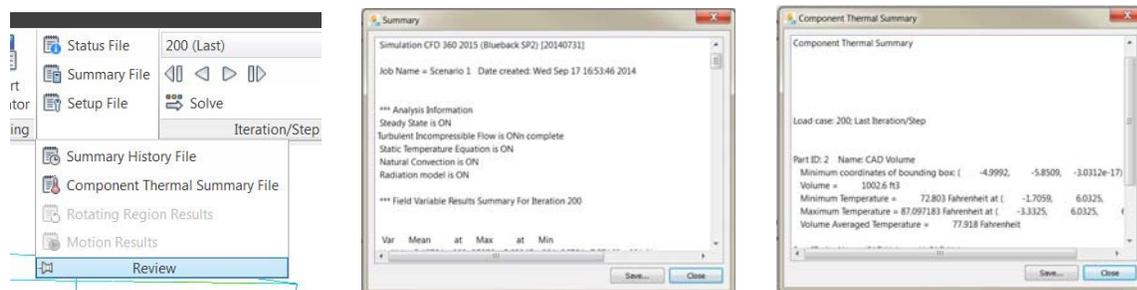
Analysis (cont)

Concept: Analyzing results

Parts Calculator: Similar to the wall calculator the parts option in results allows for the selection and query of individual objects.



There are other ways to review the results and get to the pertinent data.



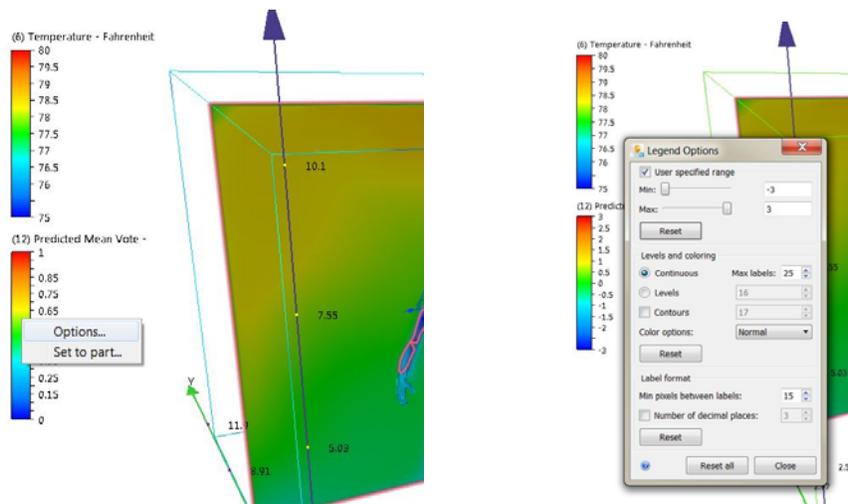
Look for other data and analysis options under the Review tab

Analysis (cont)

Concept: Analyzing results

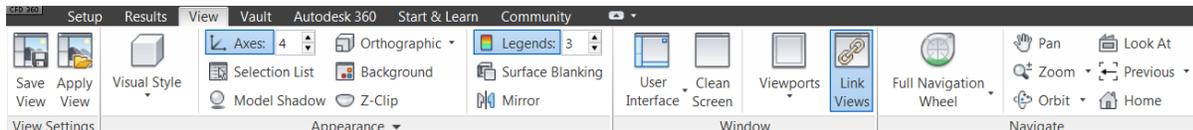
Visual Results Review

With the simulation complete and analysis types set up it's possible to navigate manually to review the results. With each analysis type used there is a supporting legend presented. The settings for this analysis types are automatically set based on the high and low results. In order to see all of the nuances in the solution the legend range and legend units may need to be adjusted. To adjust these settings right click directly over top of the legend and select Units to adjust the reporting units or Options to adjust the range.



Adjusting the legend unit and range settings

There are tools built into Simulation CFD 360 that help analyze the results. The decision center is one such tool provided to allow for comparison of the different results. Set up views in each solution and then scroll through each view in the decision center to compare. Additionally there are view settings that can be adjusted to make analysis and review easier. These are found under the view and results tab.



The view settings tab

Save a summary image to use them in the decision center. Static images can be saved to include in presentations or for manual review and dynamic images can be saved to allow for

analysis and navigation offline or for distribution to partners. Using the Save View option under the view tab it is possible to save the settings used when setting up a view. This makes it quick to set up the same view in each scenario either for comparison in the decision center or for export.

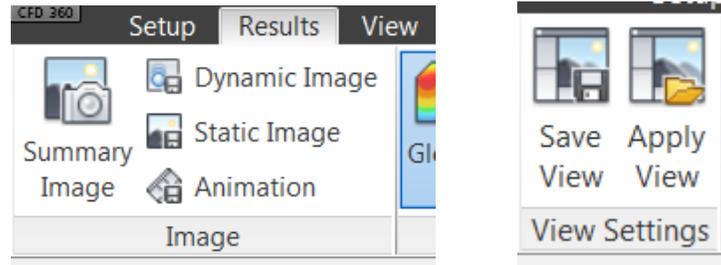
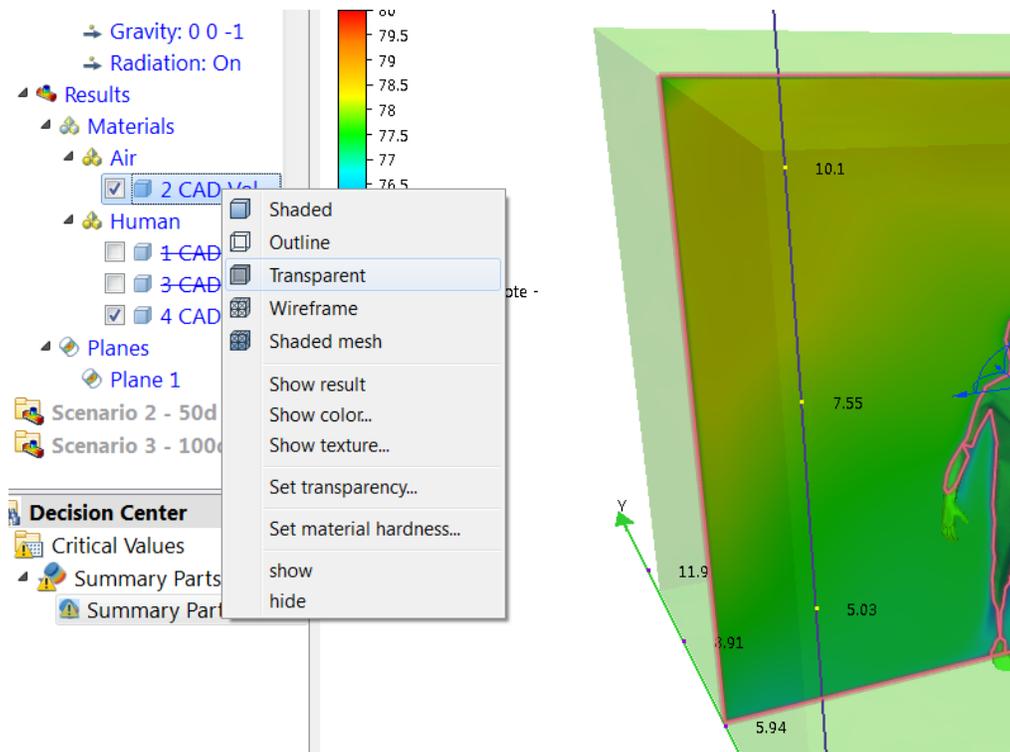


Image saving options and view setting options

Simply adjusting the view settings can make a big difference in analysis and conveying design intent and message.

When the solution is complete and the results setup it is also positive to change the way individual results show. This is accomplished by changing the object settings in the results section of each scenario. Right click overtop of the item and make a change. This might be setting wireframe objects to be transparent or solid objects to wireframe etc.



Adjust the appearance of objects per scenario

Case Studies: The best way to illustrate the use of the tool is through case studies. The guide above should be used in tandem with these studies to help you establish your own simulations. We'll look at 3 case studies, the settings, and setup of the file, the results, and our conclusions.

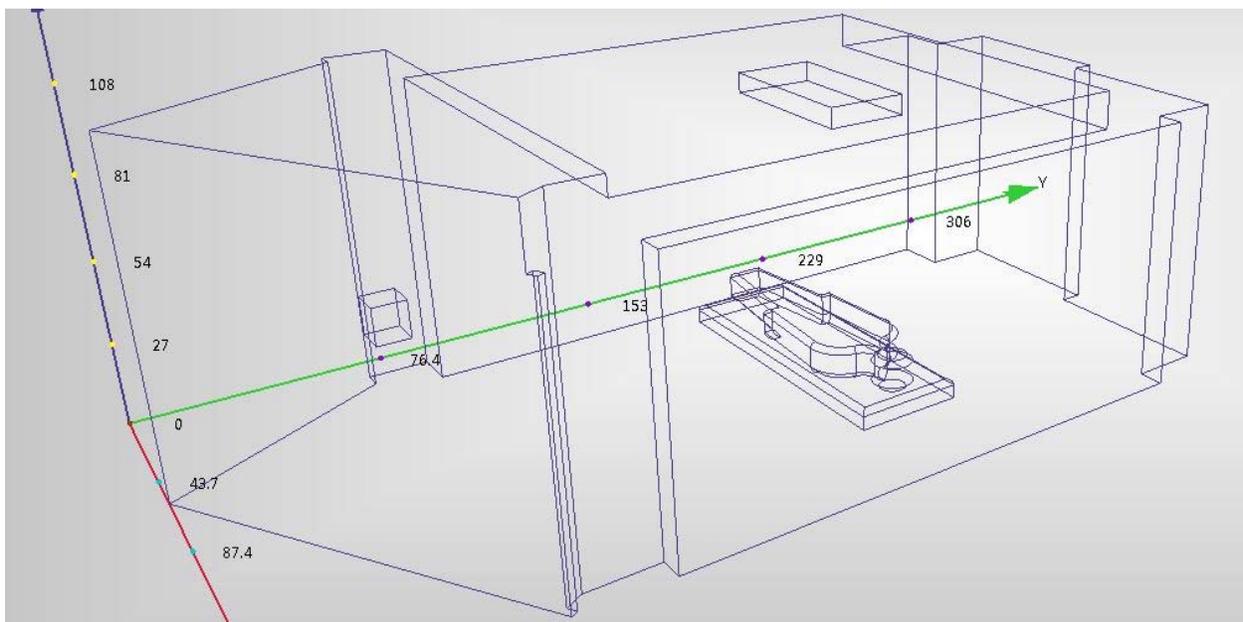
Case Study: Patient Room 1

Scenario: Due to code mandated air change requirements used in a positively pressurized airborne infection room (AIIR) it was noticed that the patients were building tent structures over their beds to stay warm.

Practical: The code mandated air changes for this room are 12. 12 air changes per hour (ACH) requires an input volume of 530 cubic feet per minute (CFM) This air was introduced through 2, two foot by four foot (2' x 4') air diffusers located directly over the bed. This configuration resulted in 530 CFM being blown directly over top of the patient at continuous velocity.

Result: This resulted in an air re-balancing effort initiated by the hospital. This re-balancing changed the performance of the building systems overall.

Study: Test different diffuser configurations and different air change rates. Different diffuser configurations may alleviate the temperature issues. Different air change rates might be supported if the argument can be made.



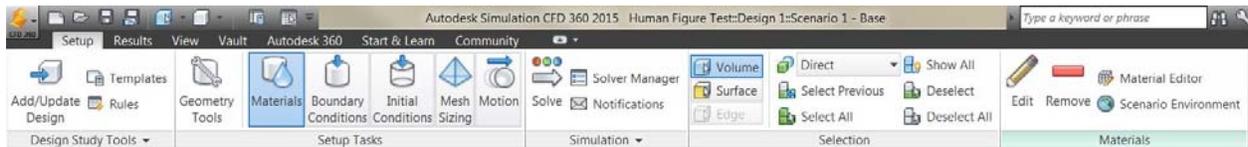
Initial model base

Setup:

Modeling:

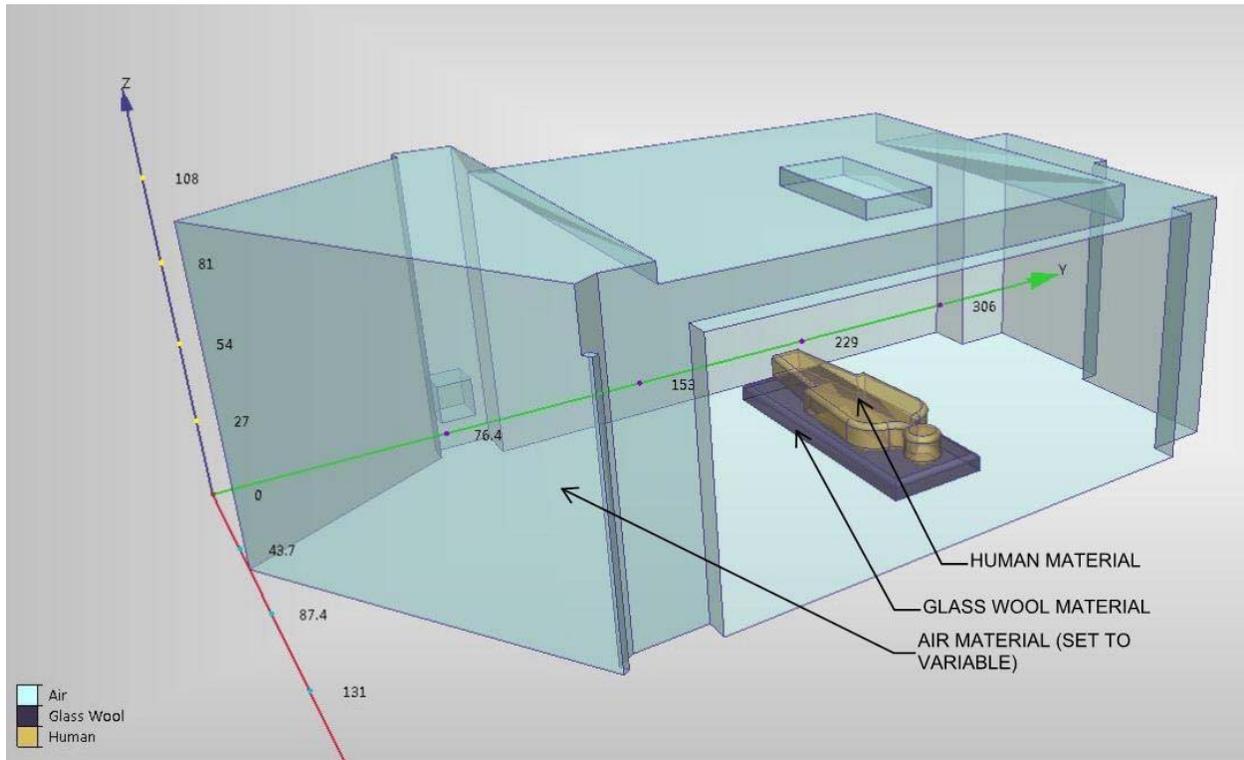
The initial modeling was complete in AutoCAD 2013. The model is comprised of an air mass representing the room volume including the diffusers, A human avatar and an object representing the patient bed. This is a basic representation of the patient room.

Materials:



Only 3 materials were used for this model

- Air – Set to Variable
- Glass Wool – Representing the patient bed
- Human – Applied to avatar



Basic model materialled

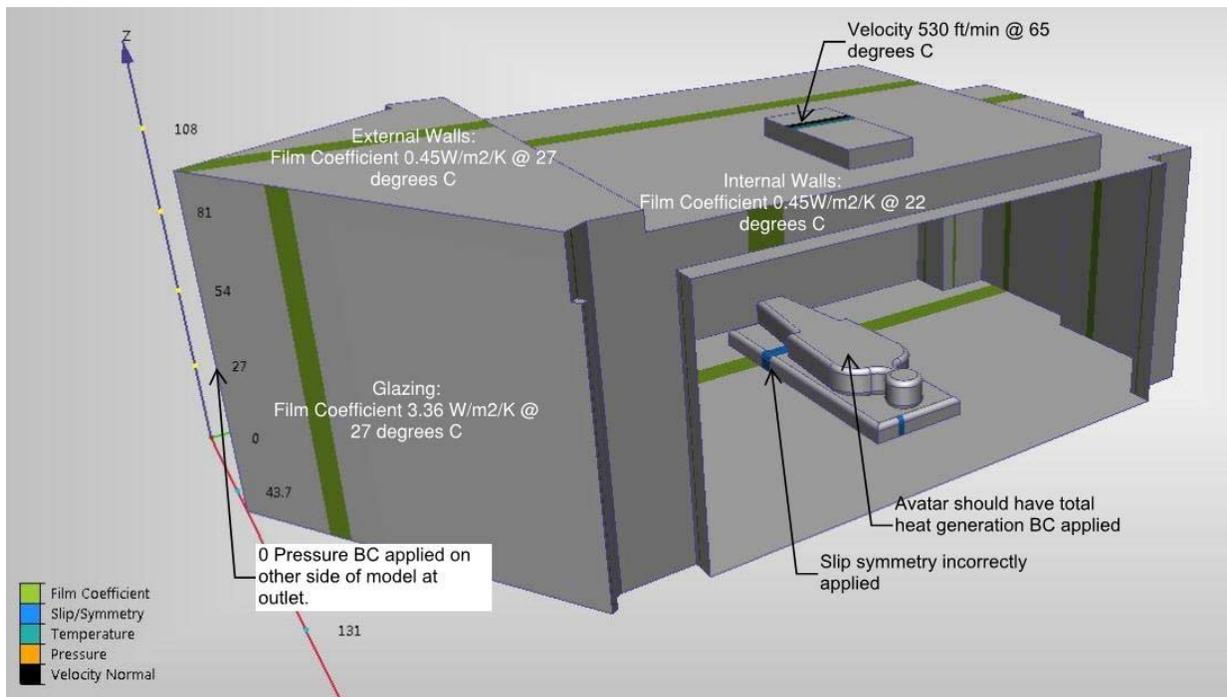
Boundary Conditions:

Five boundary conditions were applied to the model. These are flow and heat transfer boundary conditions. One boundary condition was applied in error.



Boundary conditions applied:

- External Walls: Film Coefficient = 0.45 W/m²/K @ 27 degrees Celsius.
- Internal Walls: Film Coefficient = 0.45 W/m²/K @ 22 degrees Celsius
- External Glazing: Film Coefficient = 3.36 W/m²/K @ 27 degrees Celsius
- Supply Air: Velocity (normal) = 530 feet per minute (ft/min) @ 22 degrees Celsius
- Return Air: 0 Pressure
- Patient Bed: Slip Symmetry, This BC was applied in error.
- Human Avatar: No BC applied, this is also a mistake. A volumetric BC, Total Heat Generation, should have been applied at 60 watts

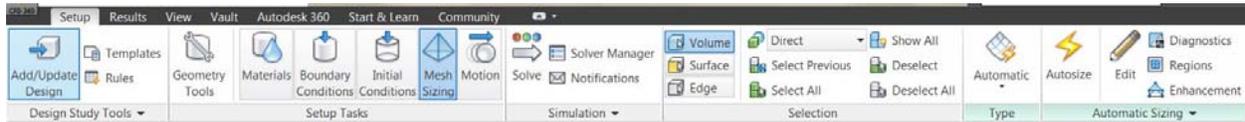


Boundary conditions applied

The slip symmetry BC was applied due to an error in knowledge. The patient avatar is missing the volumetric BC because it wasn't understood at this early stage.

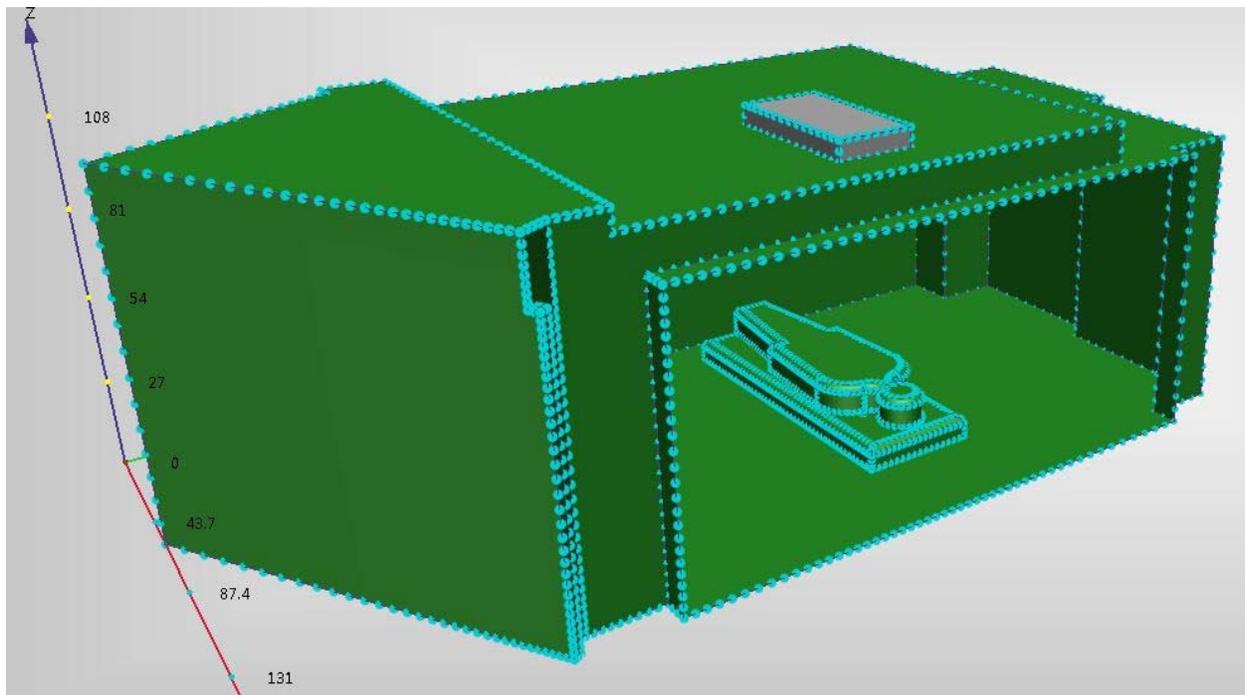
Meshing:

The initial model used the default mesh settings. Through a conversation with our training partner at an early stage in this study the mesh settings were changed. A fairly tight mesh was used to ensure accurate results.



Mesh Settings:

- Edge growth rate was changed from 1.1 to 1.05, encouraging more mesh points on edges
- The minimum points on edge was changed from 2 to 3

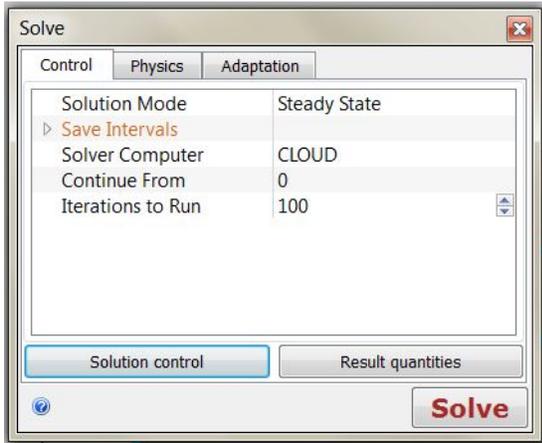


Meshing of the initial model

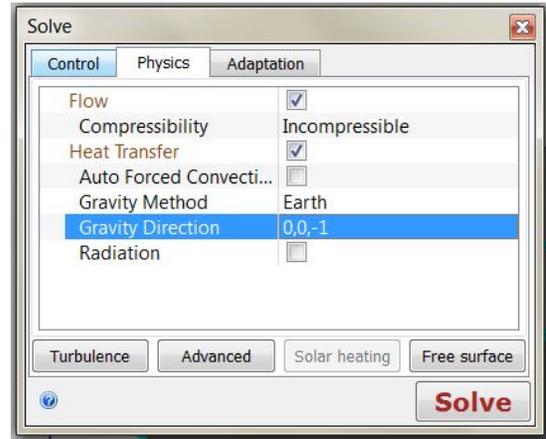
Regions could also be used to tighten the mesh at critical interactions and study points in the model. These might be at the inlets and outlets or around the patient.

Solver Settings:

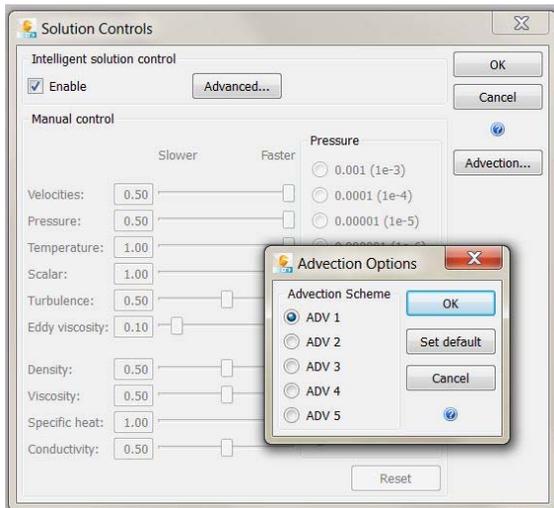
The solver is a critical part of the simulation. At this early stage in the study the solver settings being used are fairly simple. Many of the necessary changes to the settings are not set.



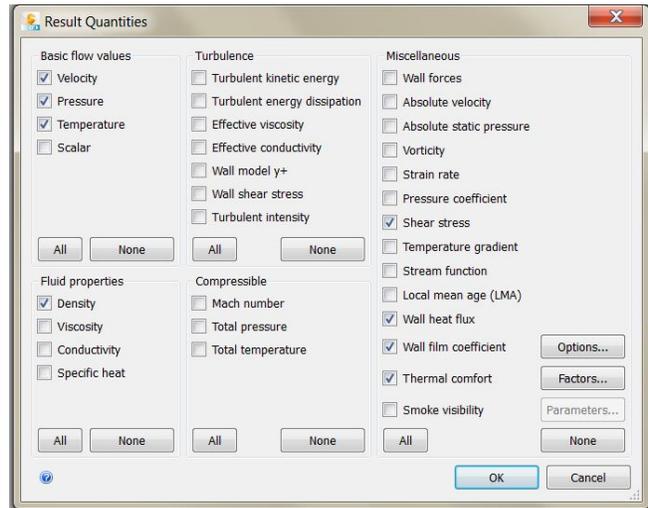
Solution Iterations set to 100, should be increased.



Radiation is not enabled.



Advection set wrong.



Thermal comfort is set but radiation is not.

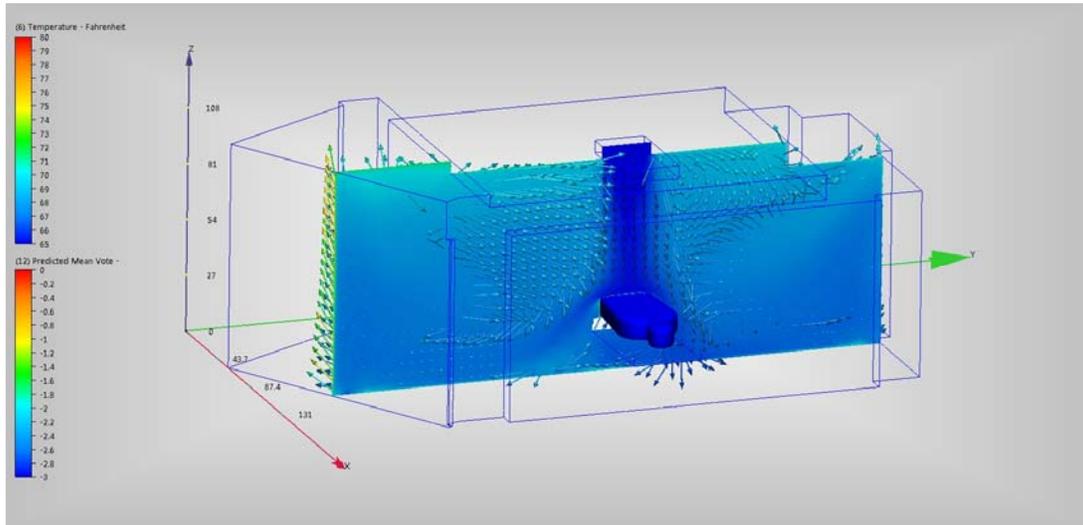
Many of the mistakes in the solution settings are due to inexperience. Being one of the first studies attempted meant that there was still much to learn.

The solution time is unknown at this point. Most likely the solution solved in an hour or two. Primarily this is due to the small, simple model and setup.

The model is only 2+ MB in size.

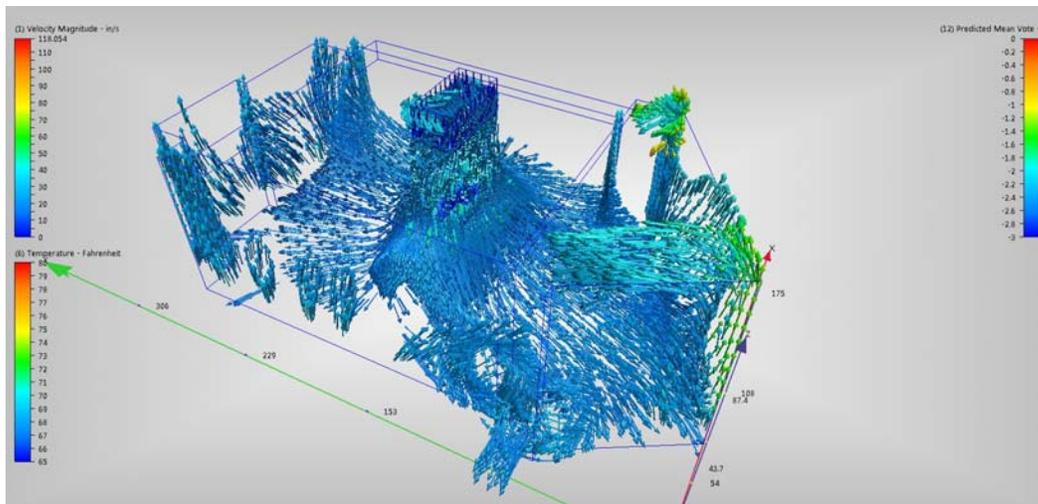
Results:

The results from this test were reviewed manually. Initial analysis used result modes like planes to understand the interactions and flow of air within the space. This initial study had the wrong diffuser configuration.

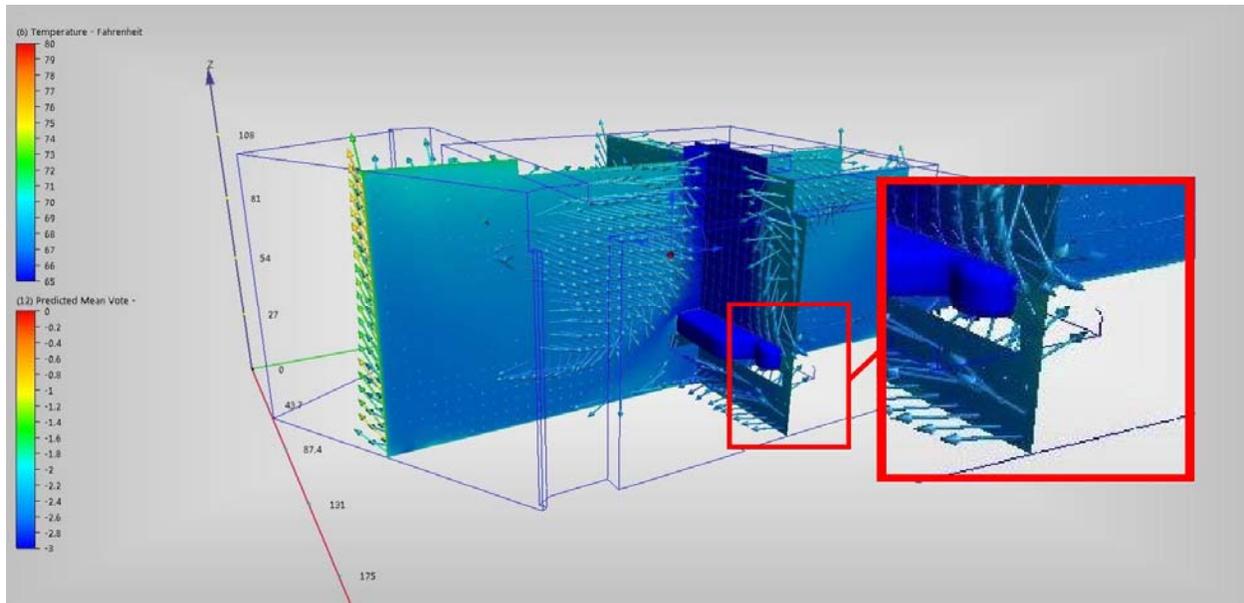


Initial results reviewed using a temperature plane with velocity vectors

Above, it is possible to see the large volume of air being blown directly down on the patient. PMV was enabled but not calculated because radiation was not enabled in the solver settings. For PMV to work radiation needs to be enabled and Thermal Comfort needs to be enabled in the result quantities dialog box (solver – result quantities) Without PMV enabled it's impossible to see the results from the simulation, however, it's possible to understand that a large volume of air being blown continuously over top of the patient creates an uncomfortable condition.



Iso surface showing temperature and velocity vectors



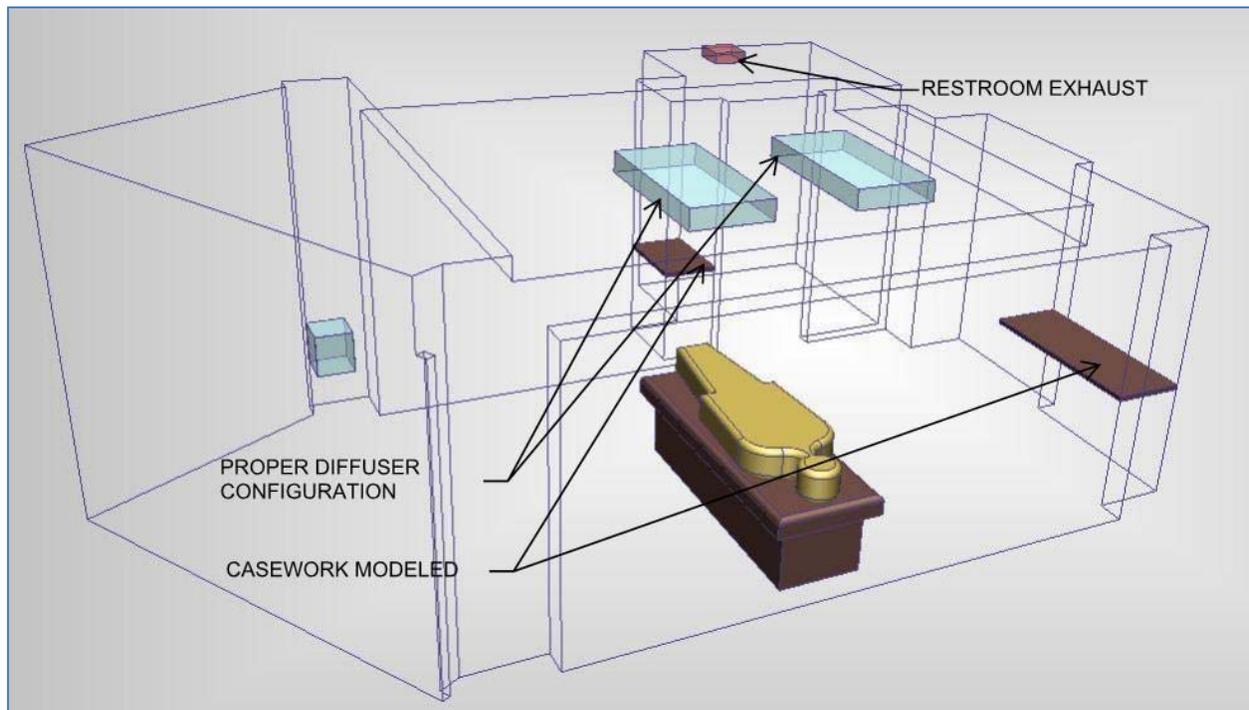
This analysis uses two planes to see the interactions over the patient. Notice the reintraintment near the patients head.

Conclusion: The simulation was a success in that it solved and provided results. It was not successful due to the mistakes in the initial setup, the lack of proper boundary conditions, the lack of a complete model and incorrect solver settings used. Add to this the fact that the overall diffuser configuration was wrong and this simulation needs to be re-worked.

Next:

These are the steps used to re-work the simulation

- Re-model: model the patient room with all of the relevant content, casework, correct diffuser configuration and patient toilet.
- Model the inlets and outlets properly. In the previous model the diffuser configuration is wrong but the model is also missing the bathroom exhaust. Although a small exhaust volume it is needed to run the solution properly.
- Apply the materials: Different materials were applied to the patient bed.
- Apply the boundary conditions: The incorrect boundary conditions were removed from the solution and the correct ones used. These include the same boundary conditions used in the initial model but also adds two separate exhaust velocities, one at the proposed return and another at the bathroom exhaust.
- The advection type was changed to advection 5 (recommended)
- The iterations to run were increased from 100 to 200

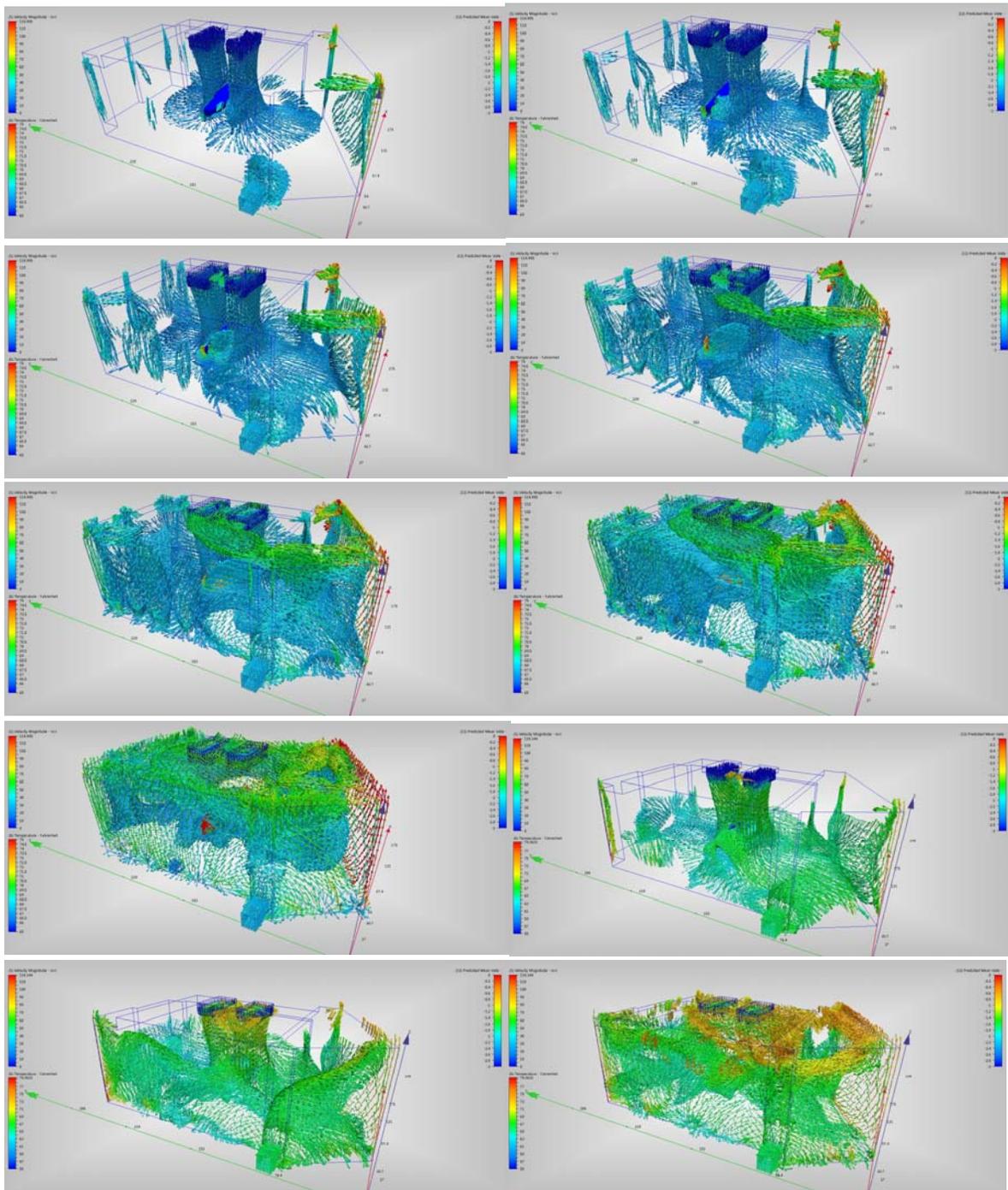


New patient room model. Two diffusers and the proper settings (mostly) applied.

These changes to the simulation should have resulted in a more accurate solution overall. Unfortunately there was still much to learn and remember at this early stage in our research and radiation was not enabled for the solution. The zero (0) pressure boundary condition is also needed for these types of solutions. Results achieved without using a zero (0) pressure boundary condition can't be trusted to be accurate. The zero (0) pressure BC was not applied in this new model setup.

Without proper PMV results and other untrustworthy results there was no way to properly measure the success of this study. The goal was to study the phenomenon occurring in the AIIR patient room, reproduce the results and study solutions. Our research team knew that visual results could still tell a story and inform the design process but was ultimately already looking to generate real data to measure from the simulations. The results matched our intuition but our intuition could be wrong. Data is unquestionable.

The following images show the secondary solution results using the new model. Although untrustworthy they are compelling. These are a series of iso surfaces set to be transparent with vectors enabled.



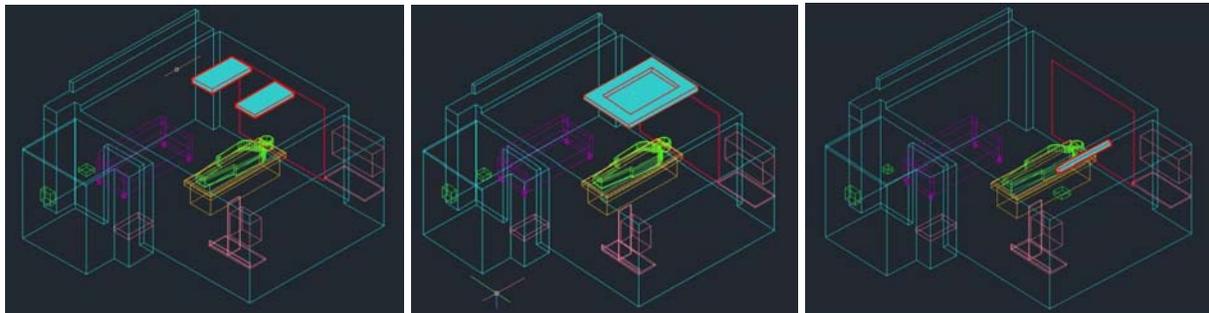
Iso surfaces showing temperature and velocity vectors shown

Case Study: Patient Room 2

Scenario: A healthcare client heard that NBBJ was using CFD to study diffuser configurations in patient rooms and requested that we assist them in studying the feasibility of building convertible patient rooms. The basic intent was to design a patient room that could be used as a traditional patient room but also still had the required air changes per hour (ACH) to be used as infection control / ICU rooms at a later date.

Practical: In order to test the feasibility of this proposed design different air change rates and different diffuser configurations were investigated. 12 air changes per hour (ACH) requires an output volume of 530 cubic feet per minute (CFM) 6 ACH is 265 CFM and 9 ACH is 398 CFM. The proposed design used 2, two foot by 4 foot rectangular diffusers. Configurations of a single linear diffuser and return and 4 linear diffusers arranged in a square above the patient bed were also studied.

Study: Test different diffuser configurations and different air change rates. Review the results to determine if a lower air change rate could be used.



Diffuser configurations tested, left to right: 2, 2' x 4" rectangular, 4 linear diffusers, 1 linear diffuser

Setup: Modeling:

The initial modeling was complete in AutoCAD 2013. The model is comprised of an air mass representing the room volume including the diffusers, A human avatar and an object representing the patient bed. The restroom exhaust is modeled as are the sink counter and toilet, family zone furniture, headwall and basic casework.

Materials:

4 materials were used for this model.

- Air – Set to Variable
- Hardwood – Representing the patient bed
- Human – Applied to avatar
- Glass – applied to casework.

Boundary conditions applied:

The boundary conditions applied were similar to the previous study but some small changes were made. Primarily in the multi-scenario setup, different air flows, and proper application of BCs to human avatar. BCs were not applied to the casework, furniture and headwall.

- External Walls: Film Coefficient = 0.45 W/m²/K @ 27 degrees Celsius.
- Internal Walls: Film Coefficient = 0.45 W/m²/K @ 22 degrees Celsius
- External Glazing: Film Coefficient = 3.36 W/m²/K @ 27 degrees Celsius
- Supply Air: Velocity (normal) = 530 CFM, 380 CFM, 265 CFM per scenario
- Supply Air: Temperature = 20 D C
- Return Air: 0 Pressure
- Human Avatar: Volumetric BC, Total Heat Generation, applied at 60 watts

Study Criteria:

Our research team took a more methodical approach to this study. Not every configuration tested all ACH. The return air location was modeled to be low in some cases like the previous study. A return diffuser was also tested in the soffit adjacent to the door.

RANGE OF STUDY CONDITIONS													
PATIENT ROOM AIR FLOW STUDY - LEVEL 6 ICU - HIGHER PROTECTION "PE" ROOMS													
SCOPE OF STUDY VARIABLES													
Diffuser Design	Baseline 2 Box Diffuser Design						4 Linear Diffuser Design						Typical 2 Linear Diffuser Design
Air Change Rates	6 ACH		9 ACH		12 ACH		6 ACH		9 ACH		12 ACH		12 ACH
Exhaust Design	1 exhaust	2 exhausts	1 exhaust	2 exhausts	1 exhaust	2 exhausts	1 exhaust	2 exhausts	1 exhaust	2 exhausts	1 exhaust	2 exhausts	High return + toilet rm
Outcomes					?		?				?		
Infection Control issues													
Patient Comfort													

Note: All solutions assume that there a closed door

Testing settings

Mesh Settings:

- The same mesh settings used in the previous study were used in this study.
- Edge growth rate was changed from 1.1 to 1.05, encouraging more mesh points on edges
- The minimum points on edge was changed from 2 to 3

Solver Settings:

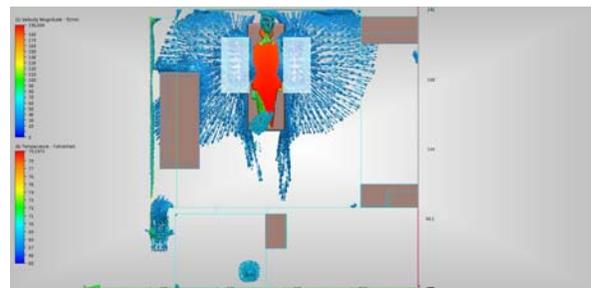
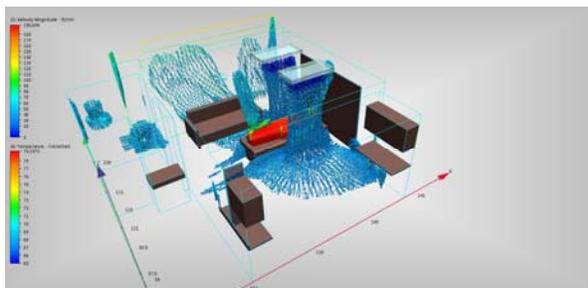
The solver settings were changed for this study. The iterations were increased from 100 – 200 and Advection 5 was used. PMV was still misunderstood process by the time of this study so Thermal Comfort was turned off in the result quantities dialog box.

The solution time was untracked and is unknown at this point. Most likely each scenario solution solved in an hour or two. The model is still relatively small at this point but growing steadily.

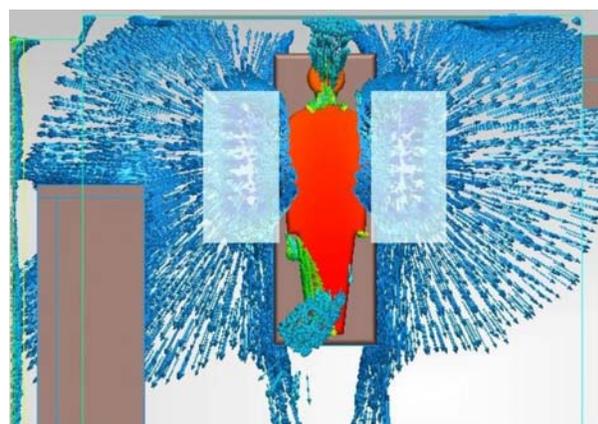
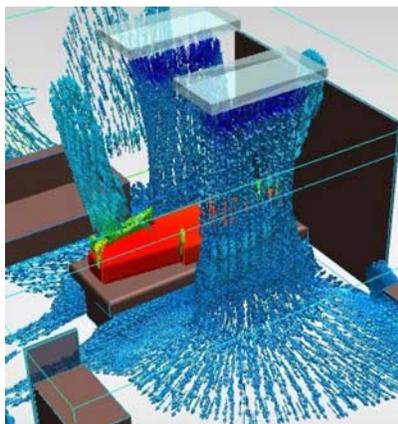
Results:

When the solutions were complete our research team applied analysis tools, planes, iso surfaces and traces, and did a visual comparison of the results. This approach didn't use the data tools available in Simulation CFD 360 but due to the careful setup and continuity from scenario to scenario the team felt that the results were as accurate as possible at this stage in our experience with the software. In addition the air flow was behaving consistently in each diffuser configuration and scenario.

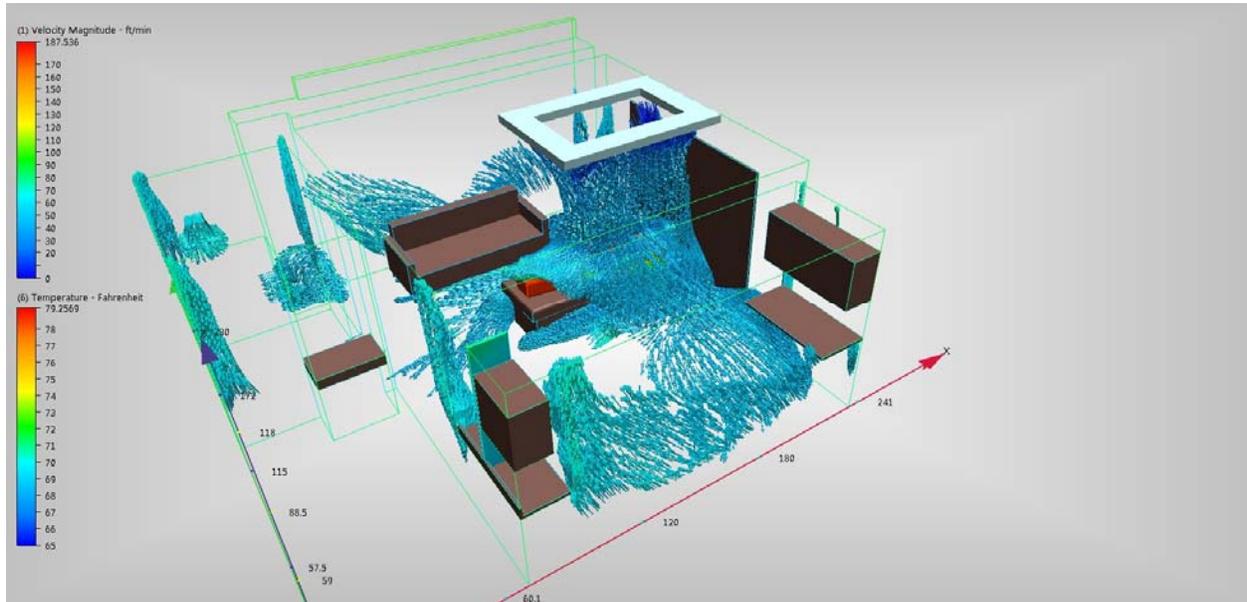
The results gave us confidence in our use of the software and of our process but the problems with PMV results, we still weren't getting any, were problematic. Without PMV it was hard to see the ultimate measure of comfort and client (patient) satisfaction. The measure used was manual observation of possible re-intrainment. Without the ability to measure comfort infection control became our only "measured" variable.



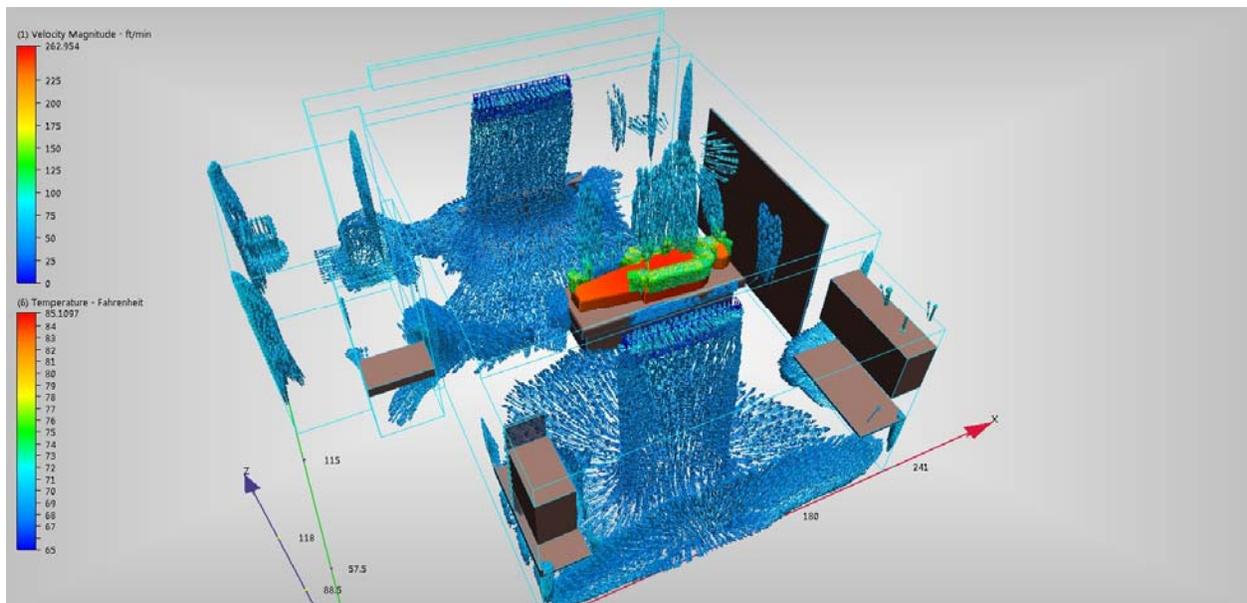
Iso Surface with Vectors showing results for original as designed configuration, 12 ACH



Results (cont)

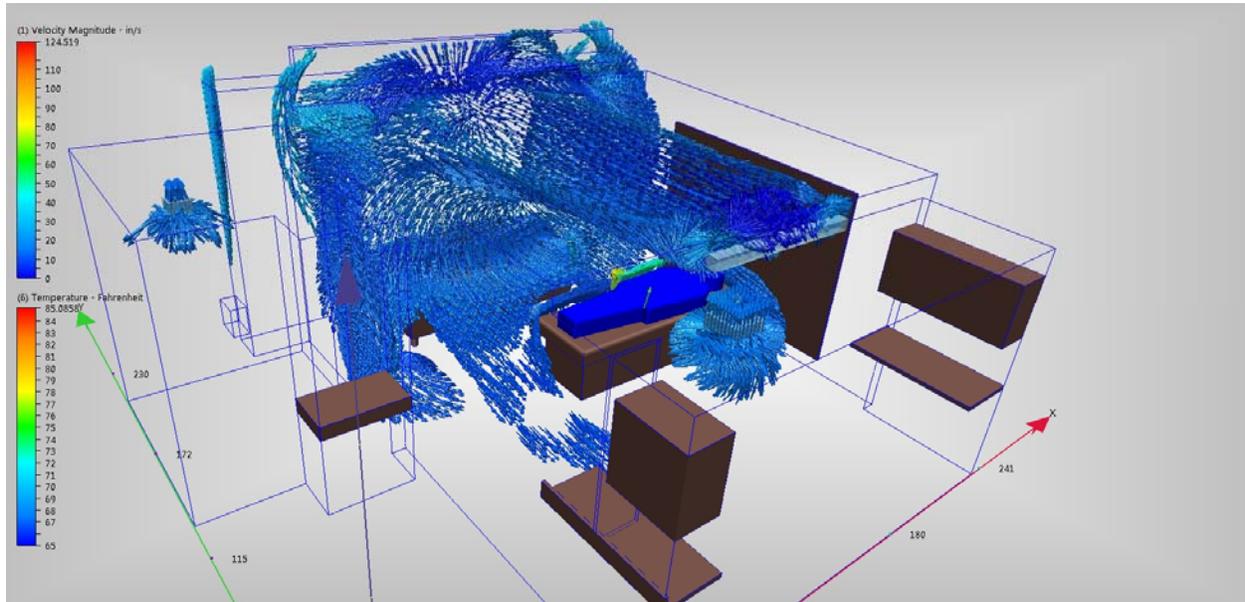


Linear diffusers arranged in rectangle over the bed



Linear diffuser vertical flow

Results (cont)



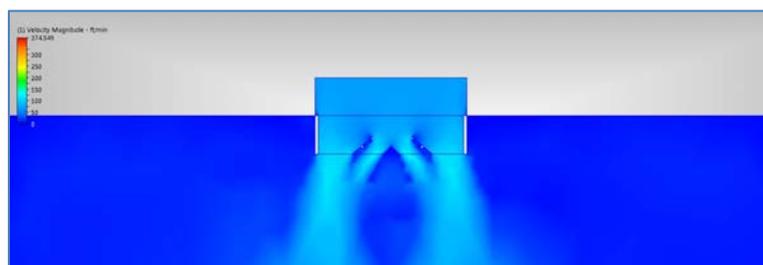
Linear diffuser horizontal flow.

Conclusion: The changes to the setup of this study were quite simple. More care was taken while planning the model and setting up the simulation and more time was taken in planning the study overall. The majority of the changes were in modeling. The modeling changes were unique to this situation; this should emphasize the need to spend adequate time planning the model.

The results were interesting and did seem to suggest one solution over the others but this was determined through observation and intuition. The need to measure data and understand what we were doing wrong with PMV was obvious.

Ultimately the client decided to build the rooms with 12 ACH but did decide to go with another diffuser configuration. This study paid for the software and our training and that seemed like the best result.

Next Steps: This is where the study ended. If we were to do a similar study we would use PMV and model the actual diffuser construction. We modeled one diffuser to test.



Case Study: ICU Infection Control

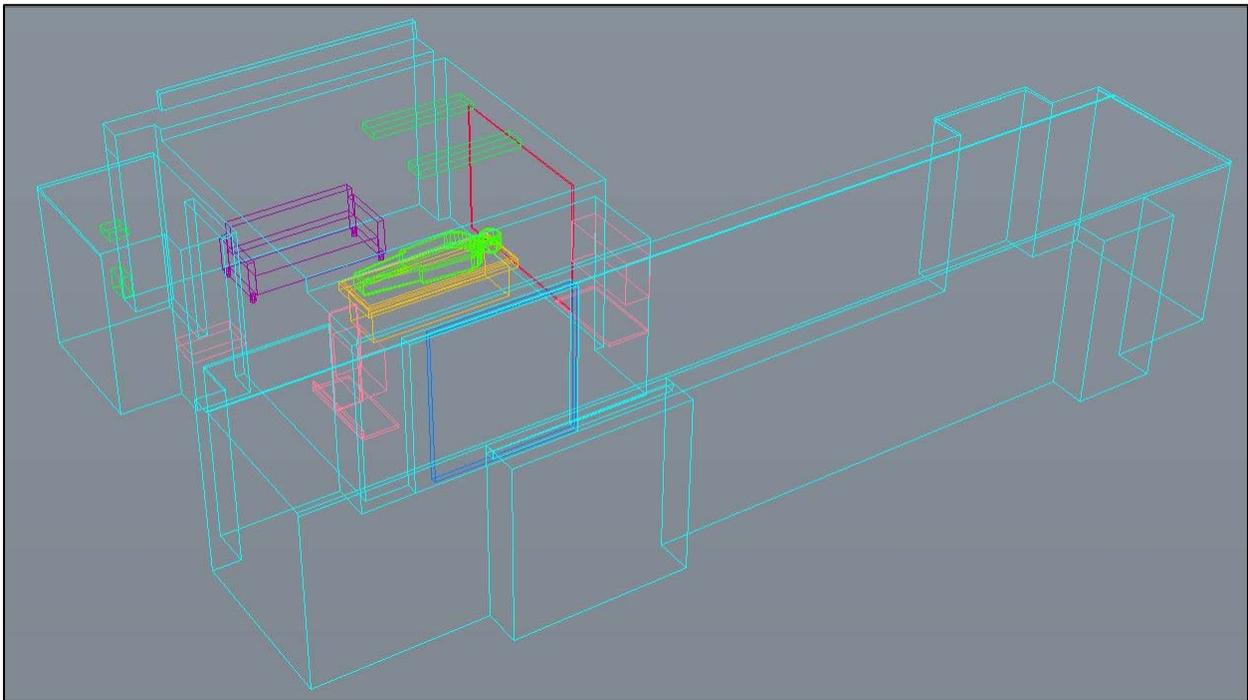
Scenario: One of the other things we studied as a result of the patient room work was the “habit” of ICU nurses propping ICU doors open by “about a foot” so they can listen for any patient distress. Although done out of care and concern it may cause air infiltration and possible infection control issues.

Practical: The corridor object had the same BCs applied as the other interior spaces in the model but the two ends of the corridor had both a volume flow rate BC and 0 pressure BC set. These were set so as to emulate basic air flow down a corridor. The room diffusers still ran at 530 CFM, the restroom exhaust was set to 0 pressure and the return diffuser was set to 390 CFM (the actual mechanical return volume)

Study: Observe the interaction of the two flows and look for air movement between the two spaces.

Setup:

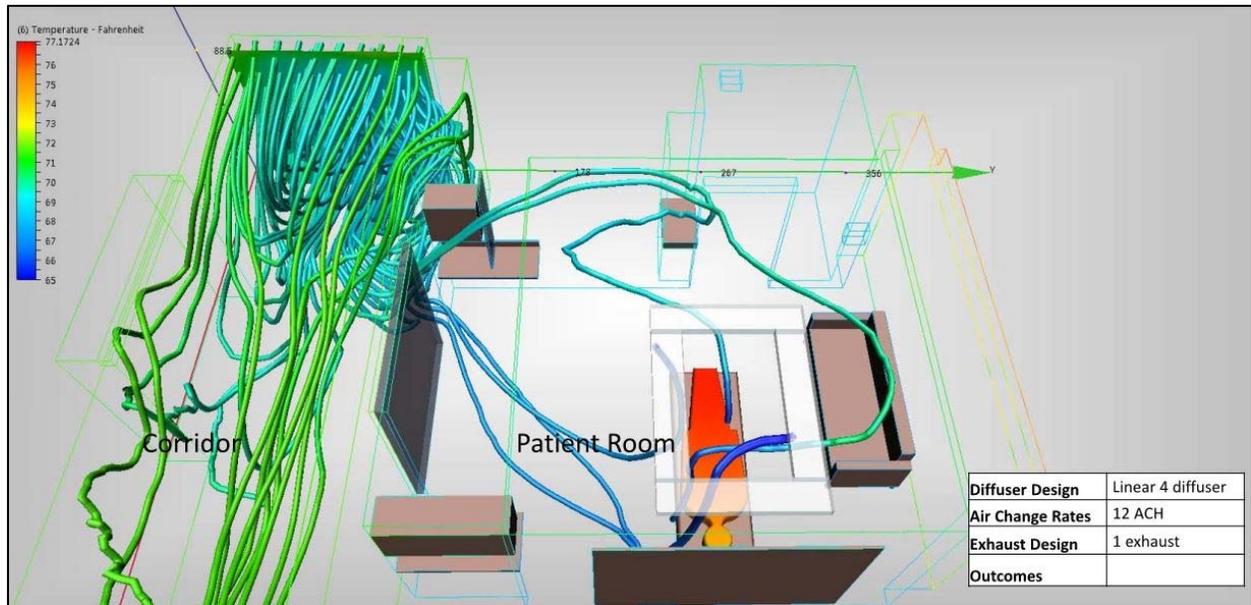
The model used for the second patient room study was altered to add the corridor and a door with a one foot wide opening the height of the door at the jamb. The each diffuser layout, with 3 different ACH rates, was used for this study. The boundary conditions described in the practical section above were also applied. All others are the same.



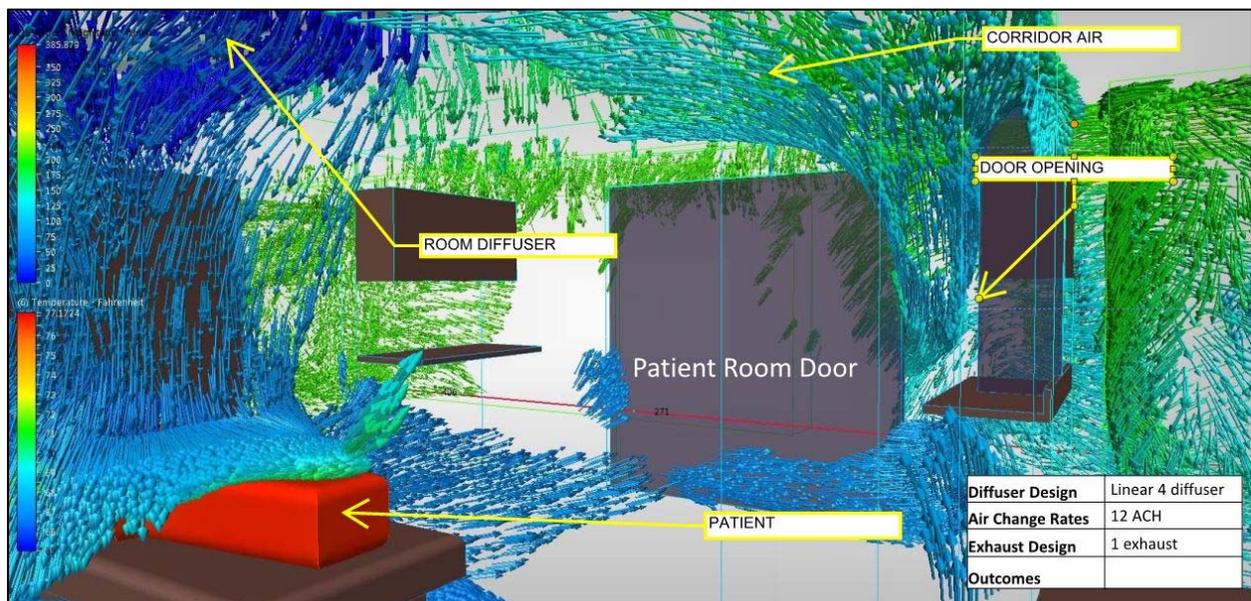
Patient room model with corridor modeled and door opening modeled.

Results:

The interactions observed proved our assumptions. In each test air from the corridor was clearly being drawn into the room. In some cases air from the corridor was mixing directly with the supply air flowing down onto the patient. Infection control experts will tell you that only a small portion of infection is airborne, however, the intent of the ICU room is to be positively pressurized to prevent any form of air intrusion.



Traces run from the Volume Flow BC in the corridor. Traces show air entering patient room and even coming near the patient. Below the same study viewed with iso surfaces and velocity vectors.



Case Study: Displacement Ventilation

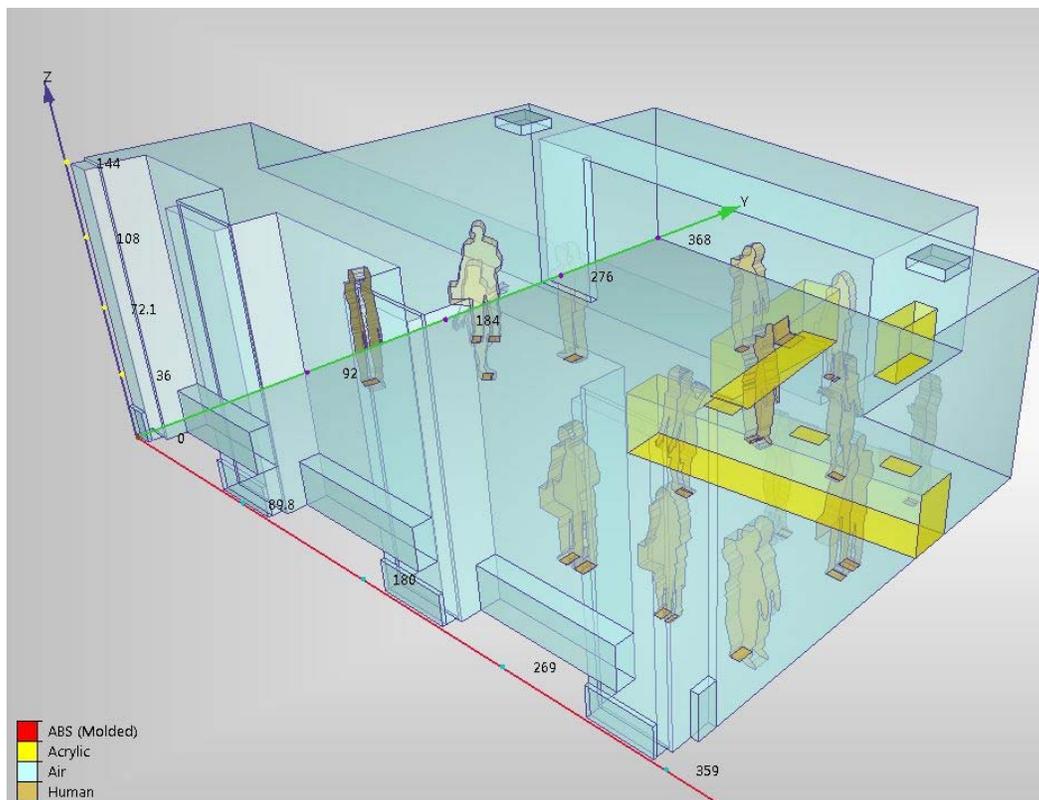
Scenario: Simulation CFD 360 can also be used to quickly study and explain concepts to clients who may not be familiar with certain concepts. Displacement ventilation was suggested as a cost saving HVAC alternative for one project. The client was reluctant to use this type of system over traditional HVAC systems. We set up a very simple displacement ventilation model to show the client that it was feasible.

Practical: A simple model of a reception area and adjacent exterior wall construction was created. Displacement ventilation was created through the use of boundary conditions. The result were observed and measured using the Simulation CFD 360 data tools.

Study: Observe the flow of air through the space. Measure the return air temperature and volume to ensure the system would work.

Setup: Modeling:

The initial modeling was complete in AutoCAD 2013. The model is comprised of an air mass representing the reception area, human avatars, casework and equipment in the form of laptops. The displacement diffuser and returns are also modeled. In addition the model was setup to be converted to a study of natural ventilation if needed. This means that the areas of operable glazing were defined in the model.



Materials:

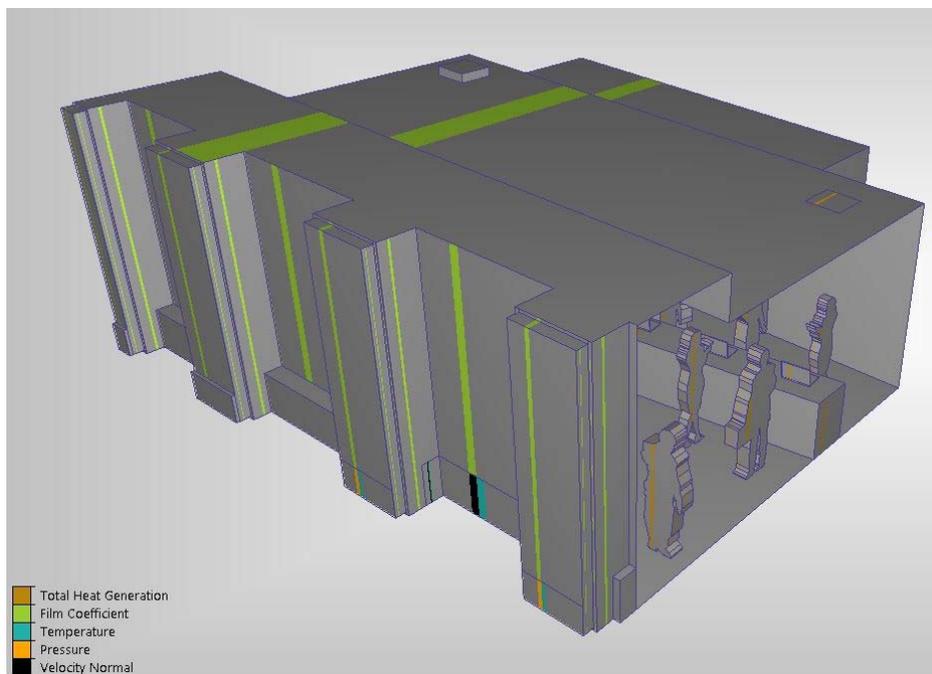
4 materials were used for this model.

- Air – Set to Variable
- ABS – Representing the Laptop
- Human – Applied to avatars
- Acrylic – applied to casework.

Boundary conditions applied:

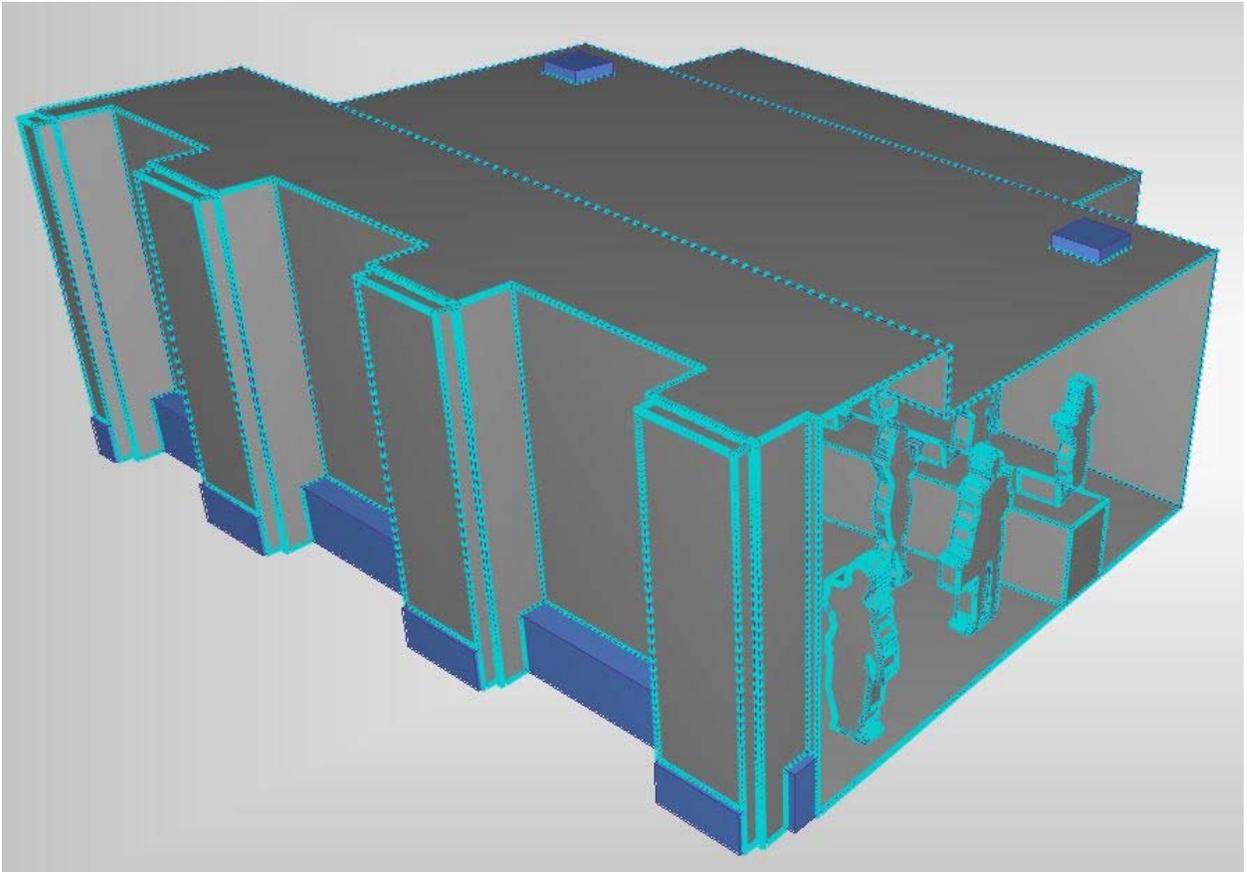
The boundary conditions applied were similar to the previous study but some small changes were made. Primarily in the multi-scenario setup, different air flows, and proper application of BCs to human avatar. BCs were not applied to the casework, furniture and headwall.

- External Walls: Film Coefficient = $0.45 \text{ W/m}^2/\text{K}$ @ 27 degrees Celsius.
- Internal Walls: Film Coefficient = $0.45 \text{ W/m}^2/\text{K}$ @ 22 degrees Celsius
- External Glazing: Film Coefficient = $3.36 \text{ W/m}^2/\text{K}$ @ 27 degrees Celsius
- Supply Air: Velocity (normal) = 50 ft/min
- Supply Air: Temperature = 20 D C
- Return Air: 0 Pressure
- Human Avatar: Volumetric BC, Total Heat Generation, applied at 60 watts
- Equipment: Volumetric BC, Total Heat Generation, applied at 150 watts
- Additionally the glazed opening was set to 0 pressure and 25 D C to represent the natural ventilation condition. (tested separately)



Mesh Settings:

- The same mesh settings used in the previous study were used in this study.
- Edge growth rate was changed from 1.1 to 1.05, encouraging more mesh points on edges
- The minimum points on edge was changed from 2 to 3
- The violet colored geometry are suppressed objects.



Mesh applied. Notice how tight the mesh is on this model.

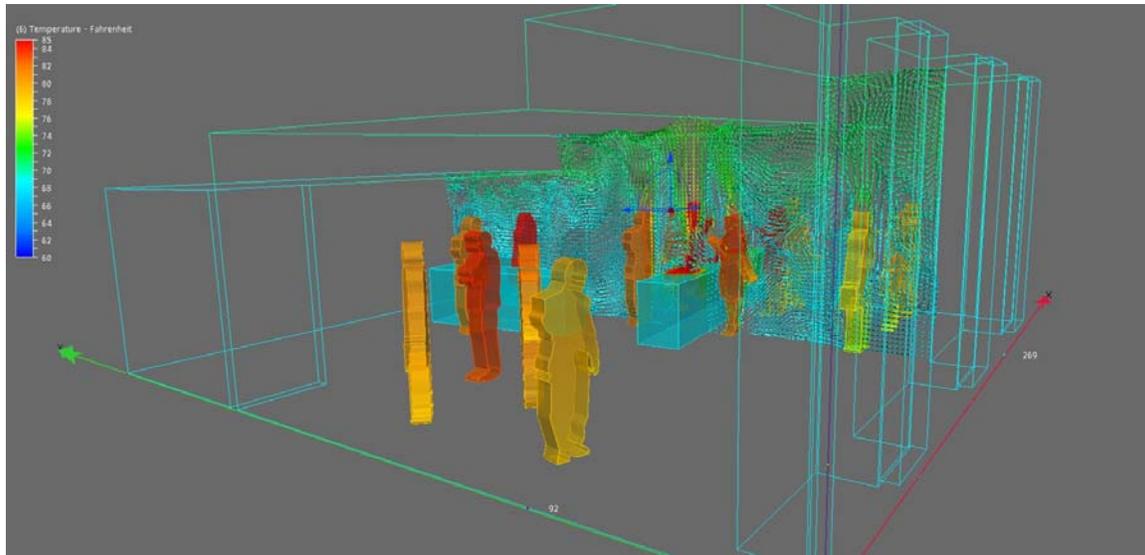
Solver Settings:

The solver settings for this study were set the same way as the previous study. The iterations were set to 200 and Advection 5 was used. PMV was not used so Thermal Comfort was turned off in the result quantities dialog box.

The solution time was untracked and is unknown at this point. Most likely each scenario solution solved in an hour or two. The model is still relatively small at this point

Results:

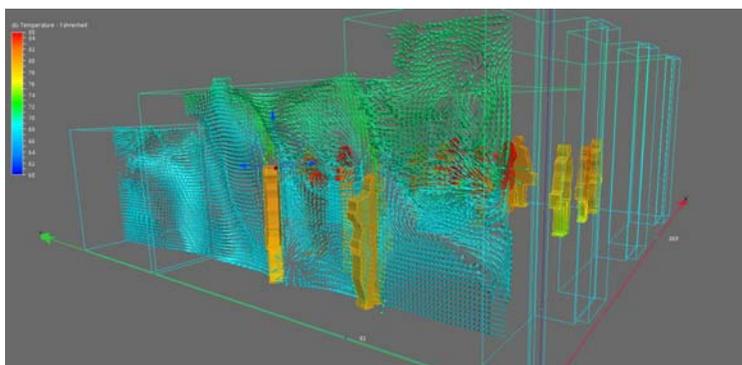
This was a quick study and the results were very simple. We reviewed the air flow analysis through observation and measure return air temp and velocity. We did this using a plane set to the location of the diffuser and running the bulk data for that plane. The temperature and volume output showed that the concept was sound.



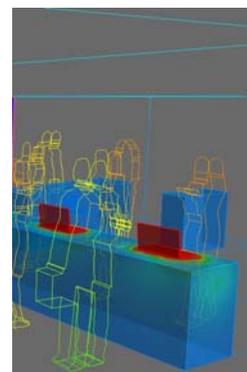
In the image above it is possible to see the human avatars colored by temperature (this is not PMV) and the plumes of heat rising from the top of the human avatars and laptops where the analysis plane intersects with those elements. This shows the system works.

Conclusion:

The client was not convinced and decided to use another method. While this was disappointing this study did allow our research team to further their efforts around CFD simulation. The natural ventilation study did not happen.



The heat plumes above each avatar are visible.



Laptops showing heat plume on casework

Case Study: Natural Ventilation

Scenario: Prior to 1992 AIA guidelines required an operable window in all patient rooms. This was primarily required for smoke evacuation. In newer hospitals the pressurization requirements for patient rooms make it a near impossibility to maintain pressure and provide an operable window. A multi-modal system could be used to mediate both these needs. Turning the HVAC system down or off while opening a window could be one possible operating mode. The positive benefits of natural ventilation, least of which is energy use reduction, can be realized.

Practical: The secondary study patient room model was used for this study. The version with the linear diffuser and soffit mounted return was altered to add an operable window and some of the objects in the room were moved to make room for the window.

Study: Test different air change rates in conjunction with different flow velocities including zero (0) flow. The temperature and volume at the return air diffuser was measured via planes and bulk analysis

Setup: Modeling:

The modeling for this study was based on the second patient room diffuser study. The model with the linear diffuser option was modified to add an operable window. The operable window was modeled as a 18" h x 24" w opening.

Unfortunately the study model was lost due to a network failure. Images of the study are available but all of the specific settings are unknown. Most of the setup is the same as the previous studies.

Materials:

The materials used in this study were typical to most patient room studies.

- Air
- Glass
- Hardwood
- Fabric
- Human

Boundary conditions applied:

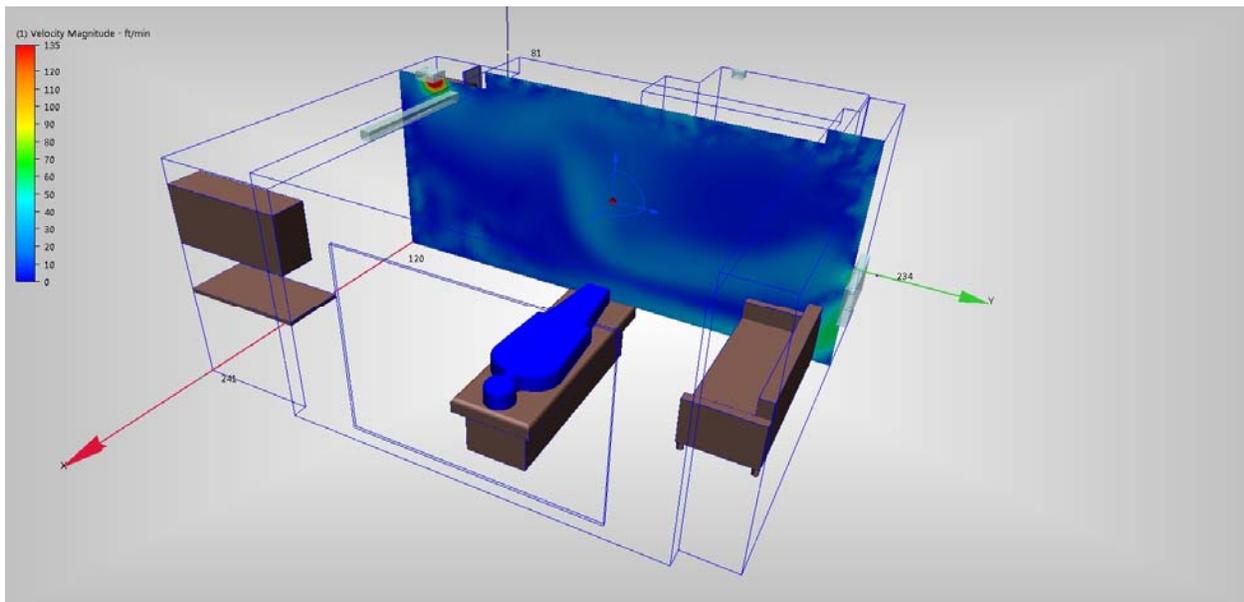
The boundary conditions were also applied in a similar manner to past studies. The operable window used a zero (0) pressure BC and variable temperature from 20 - 25 D C. The diffuser volume provided 12, 9, 6, 2 and 0 (off) ACH per scenario. The return diffuser was both powered (390 CFM) and off based on scenarios.

Mesh Settings:

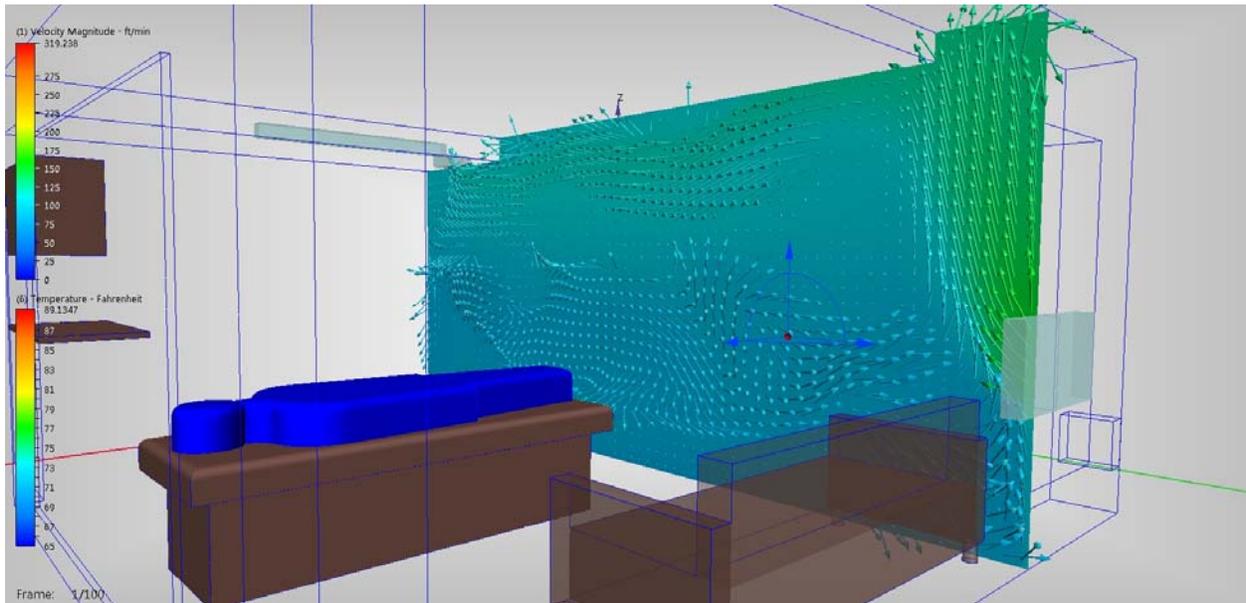
The mesh settings were applied as in the previous studies. PMV was still not enabled.

Results:

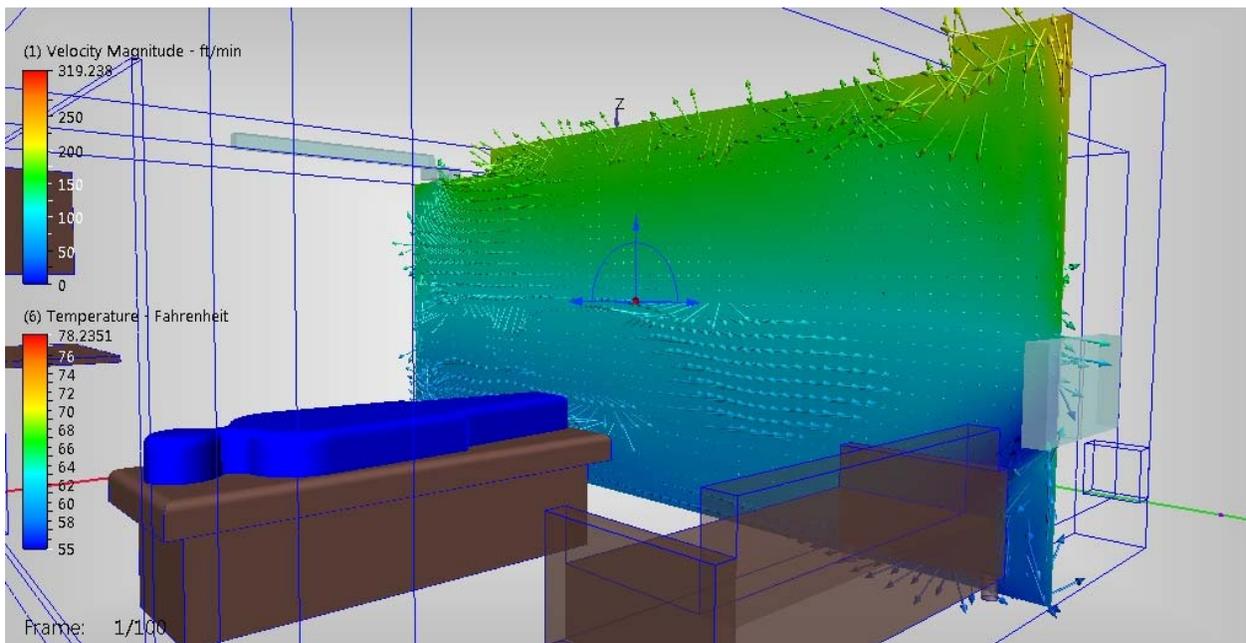
The results were varied but some obvious interactions were noted. In scenarios with a powered supply air above 6 ACH the air moved out of the window to the exterior. In other solutions the air moved into the room and provided air to the patient. The temperature and volume measured at the return suggested that natural ventilation was possible. In the image below the supply air is 6 ACH, the operable window is set to zero (0) pressure with a 20 D C temperature. It's clear that air is moving into the space and making it to the return diffuser. The measured result at the return was lost when the model was lost but it was measured at a 5 D C increase in temperature and approximately 2 ACH



When the supply air was set to 2 ACH and the operable window set to 25 D C the air movement into the room was more obvious and also less turbulent. The image below shows this scenario. Notice the air coming into the patient room is at a higher initial temperature that eventually cools as that air flow mixes with the cooler supply air. The next image shows the same scenario with a temperature at the operable window of 20 D C. The air mixes more homogenously than in the previous image. Both scenarios are using 2 ACH supply air.



Operable window at zero (0) pressure and 25 degrees Celsius. The supply air is 2 ACH at 20 D C



Operable window at zero (0) pressure and 20 degrees Celsius. The supply air is 2 ACH at 20 D C

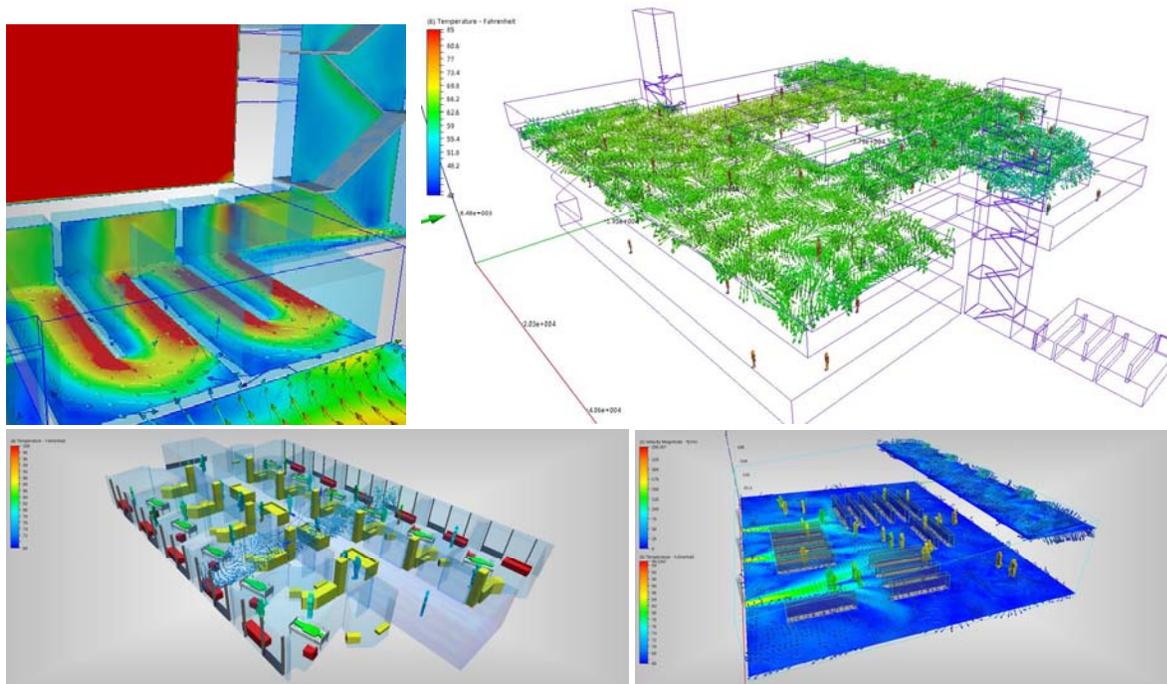
Conclusion: The study was successful in showing that natural ventilation was possible. A multi-modal HVAC system would be required. Without PMV enabled it is impossible to understand the true impact of natural ventilation in this study. This study made manifest the need to better understand and use PMV and to expand on the natural ventilation study. Our research team was becoming more competent with every study we setup but improvements were continual.

Case Studies: Conclusion

Many more case studies than are described in this document were created and tested by our research team. These studies will be shown in the lecture and supporting materials can be requested if needed.

The Natural ventilation study progressed and a full patient wing of a bed tower was analyzed and our research team finally understood and utilized PMV and other comfort measures. This enabled the team to measure the results of each simulation through, visual observations, PMV and PPD results and the data tools available in Simulation CFD 360.

In addition to natural ventilation the team has used the CFD software to study thermal labyrinth, displacement ventilation in court rooms, atriums, wind flow around buildings and more healthcare related design issues. These will be shown during the lecture.



M