AUTODESK UNIVERSITY

SIM20859

Crank Up Your Volumes: Revit-Enabled CFD

Michael Potts, P.E., LEED AP BD+C, CxA
Mechanical Engineer / Facilities Modeling Lead
M+W U.S., Inc.
Michael.Potts@mwgroup.net

Marcus Zavala
Director of VDC
Marcus.Zavala@mwgroup.net

Learning Objectives

- Develop a Revit CFD model using industry best practices.
- Learn what Revit information translates to Autodesk CFD and how to leverage it.
- See examples of Revit design geometry that can cause issues with CFD.
- Leverage Revit data and parameters to manage the CFD project.

Description

The interoperability between Autodesk Revit and Autodesk CFD can be challenging in a number of areas. Most Revit design models typically contain a high level of detail for construction, gaps in geometry, clashes, and other unresolved conflicts that can create issues when imported to the CFD environment. In this session, participants will learn how to develop and maintain a Revit model for geometry creation suitable for Autodesk CFD analysis using industry best-practices, and review examples of the type of Revit geometry that can be repurposed and what should be replaced. The session will also present methods of leveraging Revit's powerful data, visualization, coordination, and documentation functionality to enhance CFD development, design coordination, data management, and reporting.

This class will take you through the setup and CFD analysis of a simplified and idealized 10,000 SF ISO 7 Cleanroom using with M+W Products SILENT model Fan Filter Units (FFU's) and M+W Raised Metal Floor Systems Perforated Tiles.
http://www.products.mwgroup.net/

Your AU Expert

Michael Potts is Mechanical Engineer, Mechanical Facilities Modeling Lead and CFD Lead for M+W U.S., Inc. specializing in the design, construction, commissioning, and troubleshooting of high tech, clean manufacturing facilities. He is a graduate of the University of Central Oklahoma, holding a B.S Engineering Physics. Michael is also a Certified Commissioning Authority (CxA), LEED Accredited Professional and a Mechanical Contractor.

Develop A Revit CFD Model Using Industry Best Practices

Overview

The application of Computational Fluid Dynamics (CFD) in the facility design, engineering and construction industry can place the associated CFD models, analysis and input / output data within the category of Virtual Design & Construction (VDC) building information models (BIM). Executing facility CFD modeling and analysis under VDC and BIM practices and standards and guidelines can greatly improve data and information coordination and alignment across multiple software platforms and project teams.

Design and construction BIM's often contain a higher level of detail/development (LOD) than is required or desired for a CFD analysis. These BIM's may also contain geometric clashes, gaps, and other imperfections that may be within acceptable tolerance for the design or construction project, but can cause issues in the CFD analysis environment. This may drive the CFD engineer to create an independent geometry model with simplified and idealized geometry suitable for the CFD software. There are several 3D modeling applications that are compatible and interoperable with Autodesk CFD. This document will focus on use of Autodesk Revit with Autodesk CFD.

Review Project BIM Execution Plan

A standard design and construction industry practice is to develop Project BIM Execution Plans and specifications. It is useful for the facility CFD engineer to have an understanding of these key project documents.

These documents contain useful information such as:

- Project Information
- Project Location and Site Information
- Project Staff Contacts
- Project Organizational Structure
- Staff Roles and Responsibilities
- BIM Usage Matrix
- Level of Development Matrix
- Collaboration Procedures and Workflows
- Software/Technology Utilized
- File Naming Conventions & Legend

More information can be found at:
BIM Execution Plans: http://bim.psu.edu/
Level of Development Specifications: http://bimforum.org/lod/

Determine CFD Analysis Inputs

One of the first, most critical tasks when starting up a CFD project is collecting and organizing the input parameters for the facility CFD model.

Here are a few examples of project documents and resources that may contain CFD input data:

- Project Design/Engineering Basis of Design
- Project Engineering Narratives
- Project Engineering Calculations
- Equipment/Systems Sequence of Operations
- Facility Operation Setpoints and Trend Reports
- Industry Standards: for this cleanroom example, ISO 14644 & FS209E

Class ISO 146144-1 (Federal Standard 209E)	**Average Airflow Velocity m/s (ft/min)**	**Air Changes Per Hour**	**Ceiling Coverage**
ISO 8 (Class 100,000)	0.005 – 0.041 (1 – 8)	5 – 48	5 – 15%
ISO 7 (Class 10,000)	0.051 – 0.076 (10 -15)	60 – 90	15 – 20%
ISO 6 (Class 1,000)	0.127 – 0.203 (25 – 40)	150 – 240	25 – 40%
ISO 5 (Class 100)	0.203 – 0.406 (40 – 80)	240 – 480	35 – 70%
ISO 4 (Class 10)	0.254 – 0.457 (50 – 90)	300 – 540	50 – 90%
ISO 3 (Class 1)	0.305 – 0.457 (60 – 90)	360 – 540	60 – 100%
ISO 1 – 2	0.305 – 0.508 (60 – 100)	360 – 600	80 – 100%

Developing a CFD inputs worksheet that aggregates this information can help organize and communicate input parameters.

Item	Quantity	Units	Source	Comments
Site Elevation	500	ft	Site	
Site Barometric Pressure	99,507	Pa	Site	
Cleanroom Classification	ISO 7	-	Owner	Bay/Chase Configuration
Cleanroom Area	10,000	SF	Arch	
Cleanroom Length	120	FT		
Cleanroom Width	80	FT		
Cleanroom Height	16	FT		
Cleanroom Volume	160,000	CF		
Air Change Rate Min	60	ACH	ISO	ISO 7 Range 60-90 ACH
Air Change Min Airflow	160,000	CFM		
Space Velocity Avg	15	FPM	ISO	ISO 7 Range 10-15 FPM
Space Velocity Min Airflow	150,000	CFM		
Cleanroom Temperature	70	deg F	Owner	Range 70F +/- 2F
FFU Coverage Avg	20	%	ISO	ISO 7 Range 15-20%
FFU Area	16	SF	M+W Products	48" x 48" FFU
FFU Min. Quantity	125	Ea	Calc	
FFU Design Quantity	160	Ea	Mech Eng	
FFU Flow Ea.	1000	CFM	Mech Eng	MW Silent FFU
FFU Total Flow	125,000	CFM	Calc	
Perf Tile Size	4	SF	M+W Products	
Perf Tile Free Area	30	%	M+W Products	MW RMF Perforated Tile
Perf Tile Max Velocity	200	FPM	Mech Eng	
Perf Tile Max Flow	800	CFM	Calc	
Perf Tile Min Qty	156	Ea	Calc	
Perf Coverage	6.25	%	Calc	
Equipment Quantity	16		Owner	
Coil Max Pressure Drop	0.1	in H2O	Mech Eng	
Coil Max Velocity	400	FPM	Mech Eng	
Coil K Factor	10.1	K Factor	Calc	
Coil Min Total Area	313	SF	Calc	
Coil Qty	10	EA	Mech Eng	5 coils per chase
Coil Min Area Each	31.3	SF	Calc	
Coil Selected Area Each	35.0	SF	Mech Eng	84" x 60" coils
Chase Max Velocity	300	FPM	Mech Eng	
Chase Free Area	80%	%	Mech Eng	
Chase Qty	2	Ea	Arch	
Chase Min Area, Ea	521	SF	Calc	
Chase Min Width	7	FT	Calc	

Setup The CFD Revit Cleanroom Model

To setup and configure the Revit CFD model using the methods outlined in this document, you will need to have basic knowledge of both Revit Architecture, Revit Structure and Revit Systems tools. With the Revit Onebox product, all of these tools should be available.

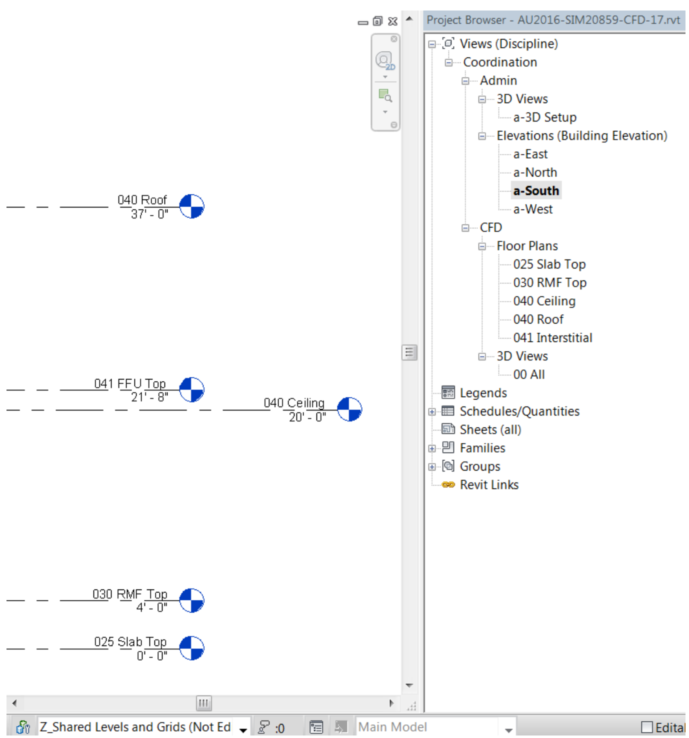
Tip: A lesson learned from experience with these types of cleanroom CFD models at this particular scale is that no element should have a dimension in any direction less than 6”. Elements that are less than 6” should be omitted, or grouped together with other smaller elements if they existing in an array or cluster (i.e. pipes). This includes RMF, walls/partitions, coils, etc. Being aware of this and configuring the geometry as such beforehand will be useful later when meshing the model in CFD. The effect of increasing the thickness of the RMF and cleanroom partitions from 2” to 6” on the CFD results is negligible. Differences in thickness can be split about the centerline of the components to further reduce risk if there is any concern. Of course it is critical that the CFD engineer be familiar with the aspects of their project requirements and 3D model, and should base model modification decisions on their judgement.

Create The Levels and Plans

- Create levels that align with the standard project building and major component levels. Generally, you can link in the project Architectural Revit model for this.

- Create additional levels for the upper a lower limits of the subcomponents.

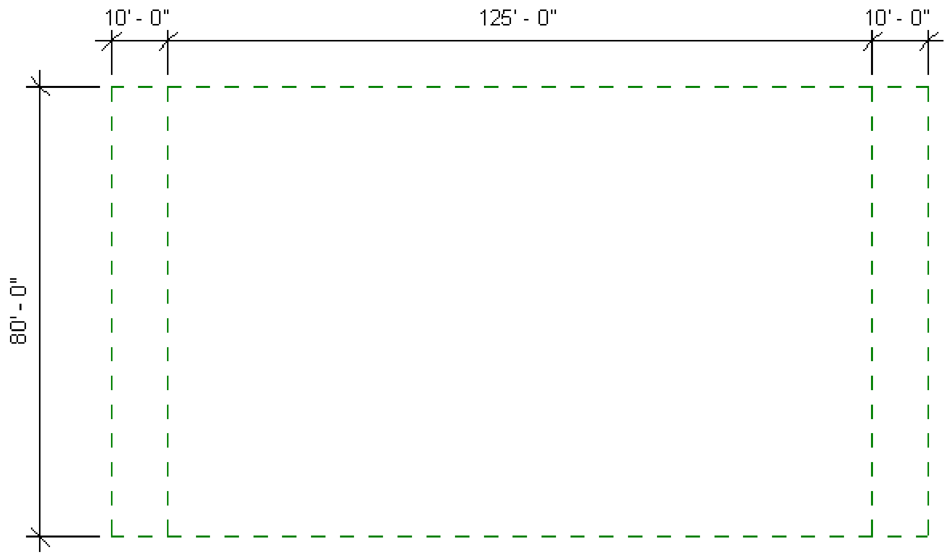
- Create floor plans of the major floor levels and set the discipline to Coordination so that Architectural and Mechanical elements can be created and edited.



CREATE LEVELS THAT ALIGN WITH PROJECT

Create The CFD Domain Boundaries

Make reference planes that will define the boundaries of the CFD domain and any subcomponents. These reference planes could align with the Architectural Revit model boundaries. But in this exercise, there is no Architectural model.



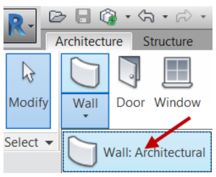
CREATE REFERENCE PLANES FOR DOMAIN BOUNDARIES

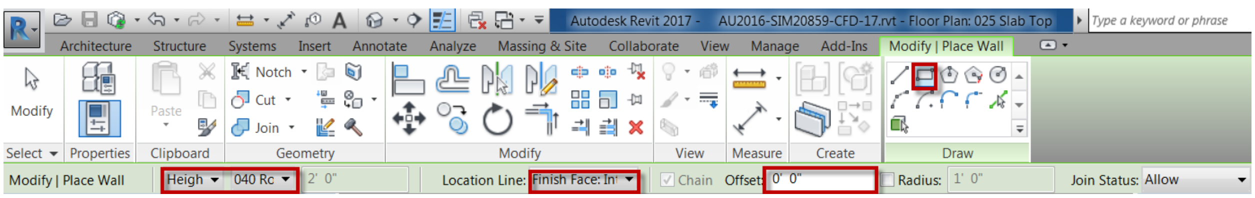
Create Generic Walls and Partitions

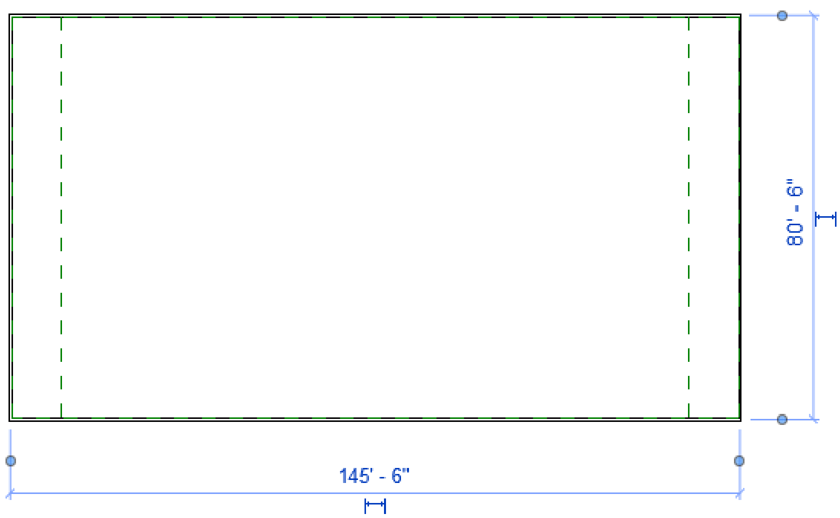
- Create the outer wall at the *025 Slab Top* level using the **Basic Wall Cleanroom Partition – 6"** family on the **CFD-WALL** workset.

- Set the **Height** constraint to **050 Roof,** set **Location Line** to **Finish Face: Interior,** and **Offset** to **0'-0"**.

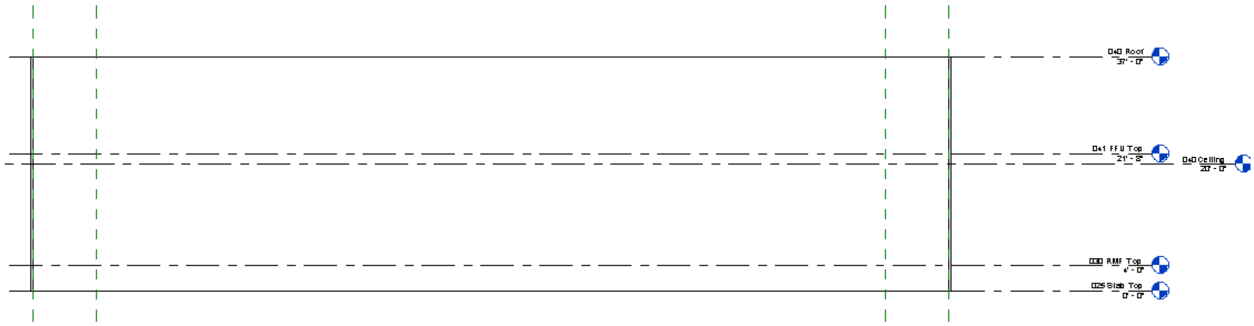
- Snap the outer sketch boundaries to the outer reference planes.







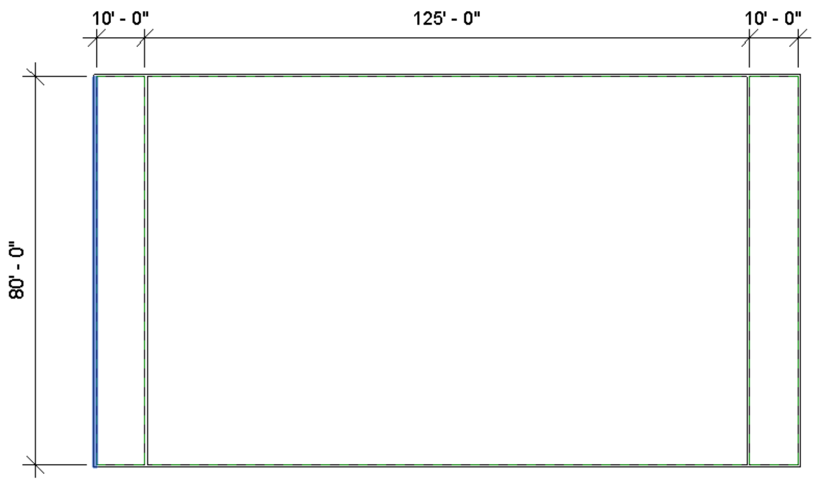
Open an elevation and ensure the wall extends from *025 Slab top* to *050 Roof.*



- Create the chase walls at the *030 RMF Top* level using the **Basic Wall Cleanroom Partition – 6"** family on the **CFD-WALL-CR** workset.

- Set the **Height** constraint to **050 Roof,** set **Location Line** to **Finish Face: Interior,** and **Offset** to **0'-0"**.

- Draw the sketch lines for each chase wall along the chase wall reference planes.

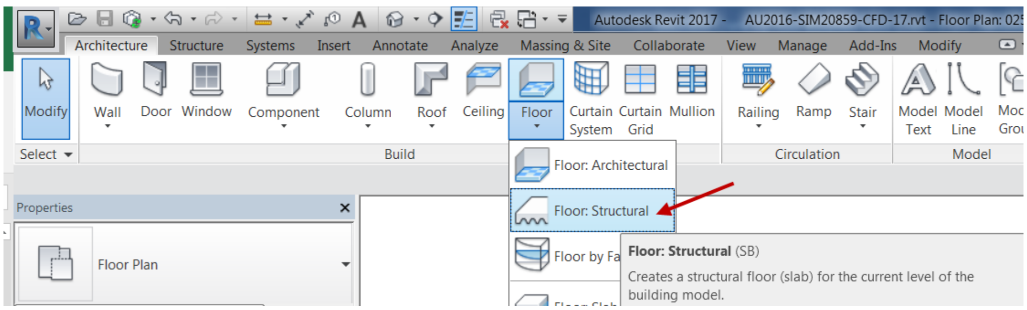


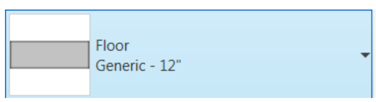
CREATE CHASE WALLS

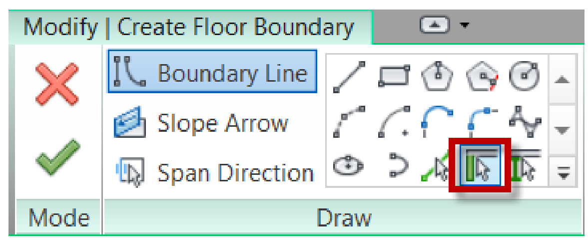
Create Generic Floors, Ceiling and Roof

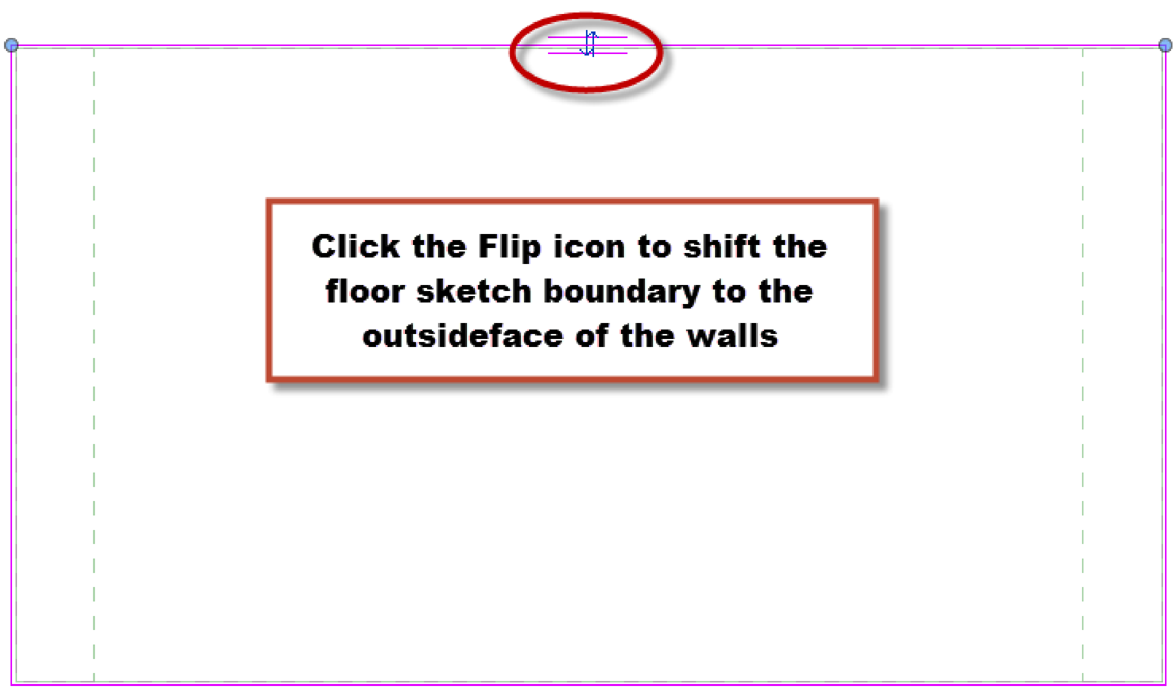
- Create the slab floor at the *025 Slab Top* level using the **Floor - Generic 12"** family on the **CFD-SLAB** workset.

- Select the Outer Walls reference planes.







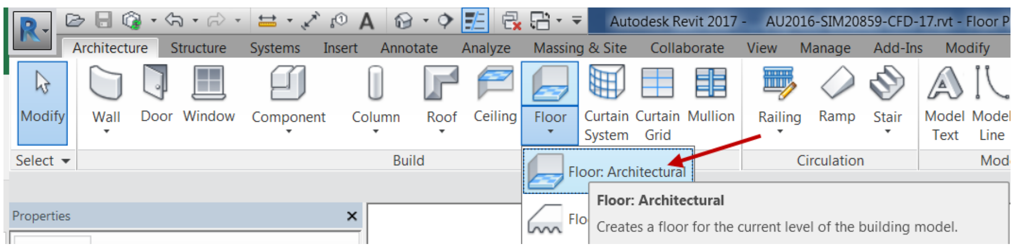


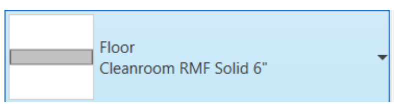
- Create the RMF generic floor at the *020 RMF Top* level using the **Floor – Cleanroom RMF Solid 6”** family on the **CFD-RMF-SOLID** workset.

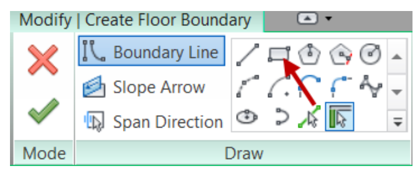
- Set the *Height Offset From Level* in *Properties* to *0’-0”*.

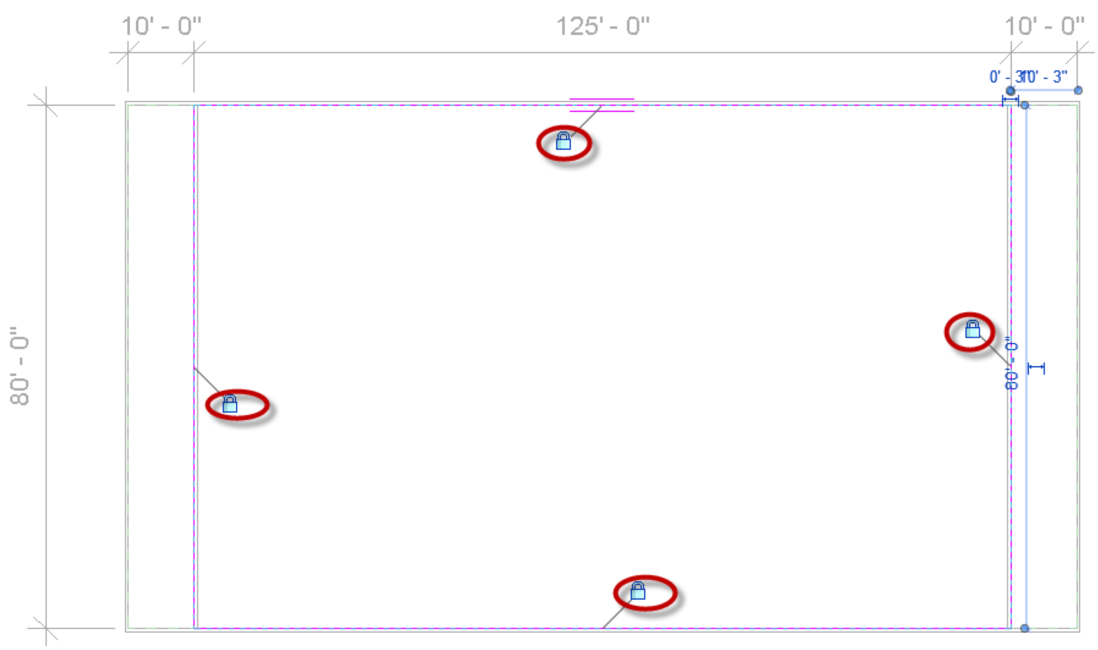
- Sketch the rectangle to the outside of the East and West Chase walls and the inside of the North and South outer walls.

- Lock the sketch lines to the walls. This can help later if adjustments need to be made.





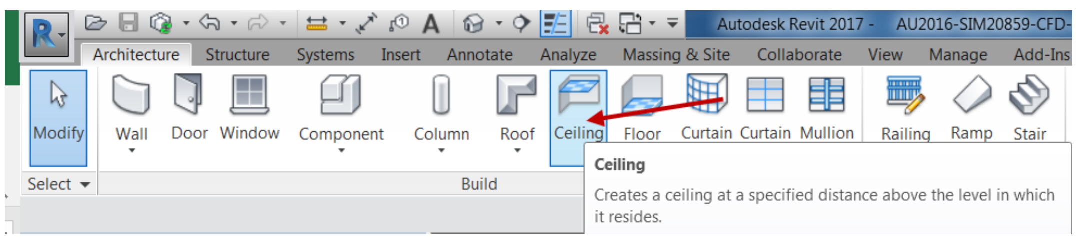


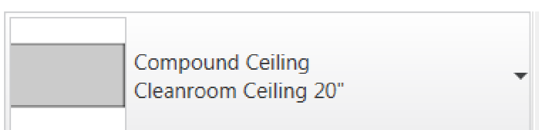


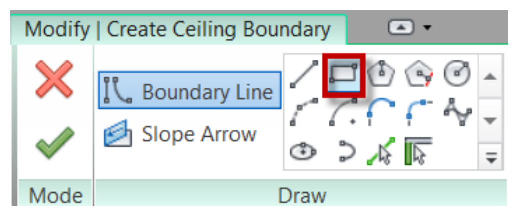
- Create the generic ceiling at the *040 Ceiling* level using the **Compound Ceiling – Cleanroom Ceiling 20”** family on the **CFD-CEILING** workset.
    - Note that the generic ceiling is the same thickness as the FFU’s are tall. This will become a suppressed volume in the CFD model later to prevent issues with FFU’s in close proximity. Set the *Height Offset From Level* in *Properties* to *0’-0”*.

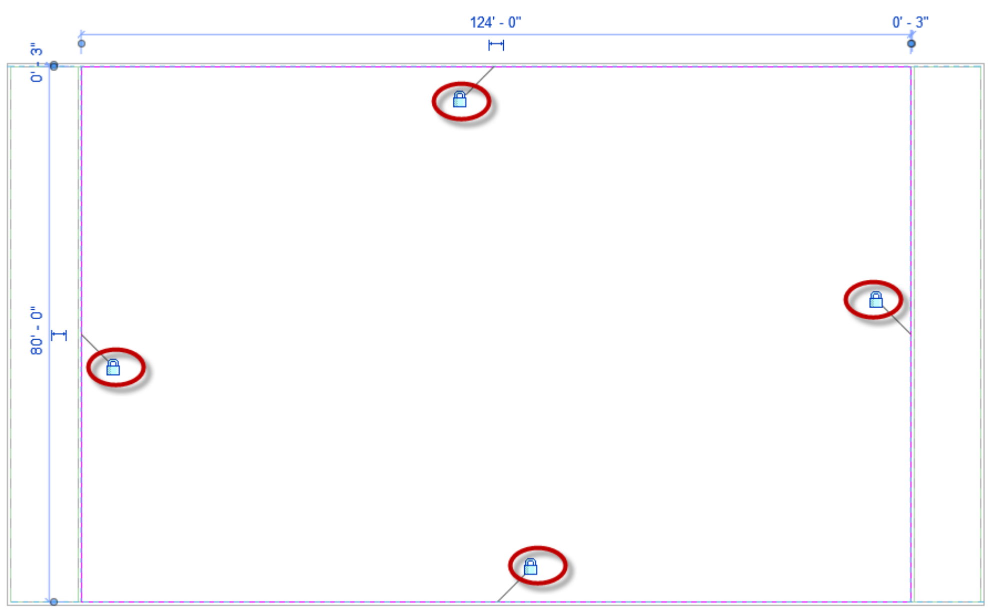
- Sketch the rectangle to the inside of the East and West Chase walls and the inside of the North and South outer walls.

- Lock the sketch lines to the walls.





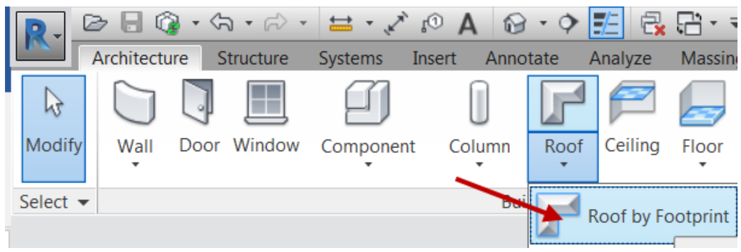


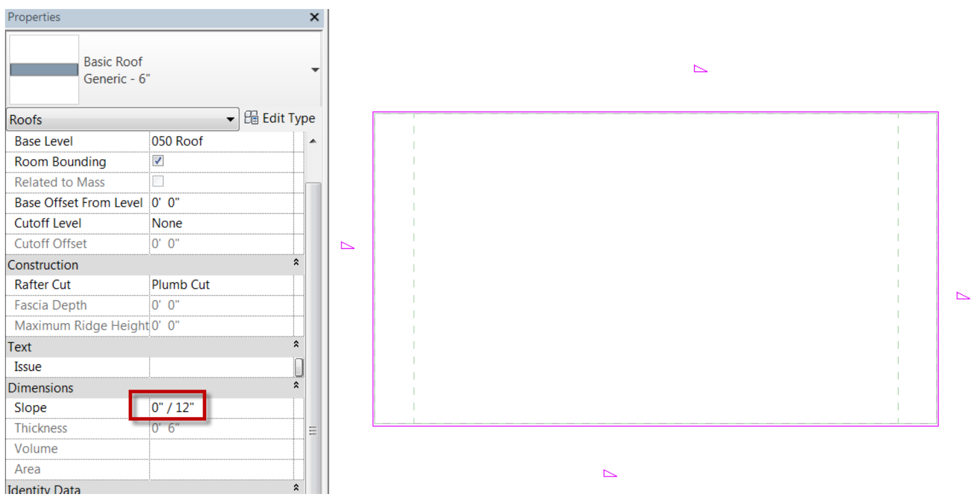


- Create the generic roof at the *050 Roof Top* level using the **Basic Roof Generic – 6"** family on the **CFD-ROOF** workset.

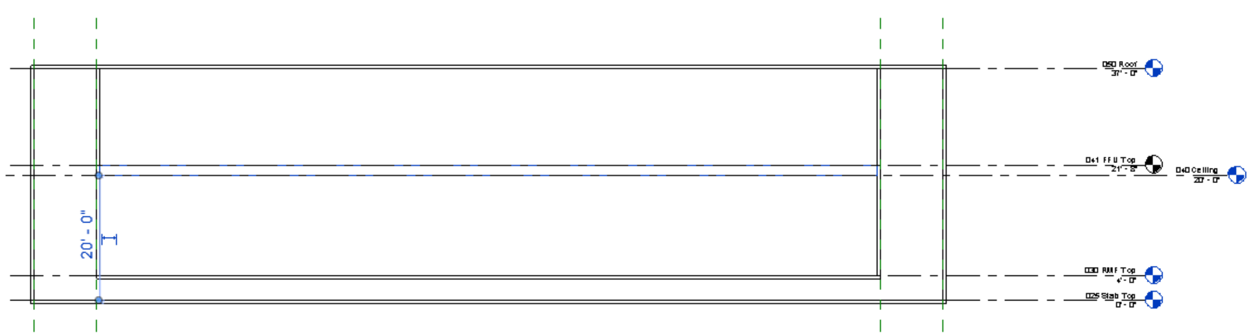
- Set the *Slope* in *Properties* to *0'-0"*.

- Sketch the rectangle to the outside of outer walls. Lock the sketch lines to the walls. This can help later if adjustments need to be made.





Open an elevation view and ensure than walls, floors and roof meet properly.



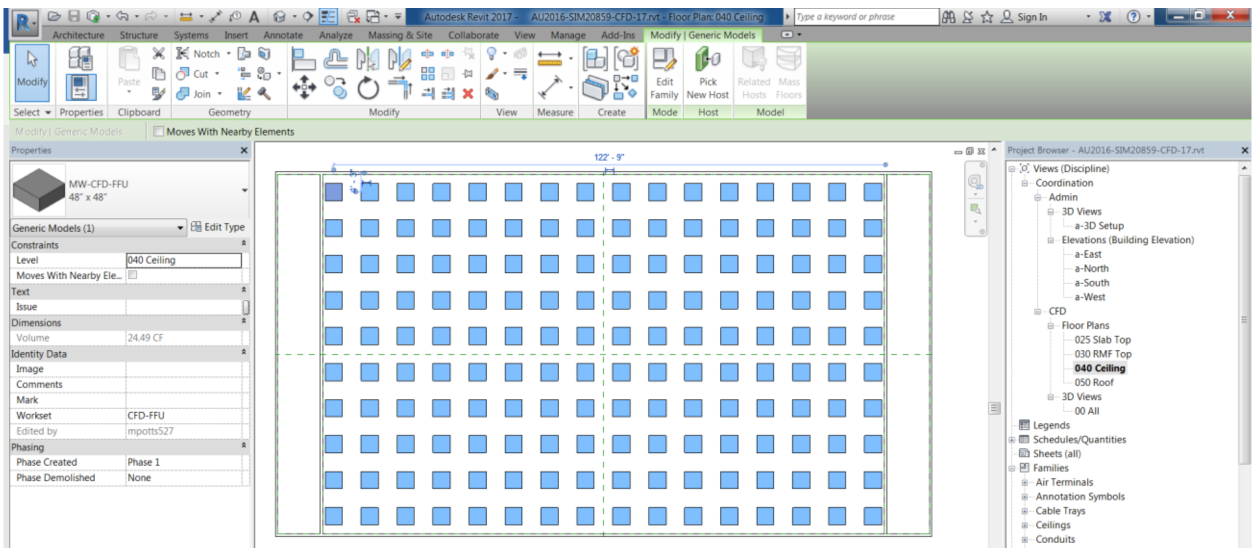
You should now have an air-tight cleanroom and outer envelope.

Create The Airside and Equipment Components

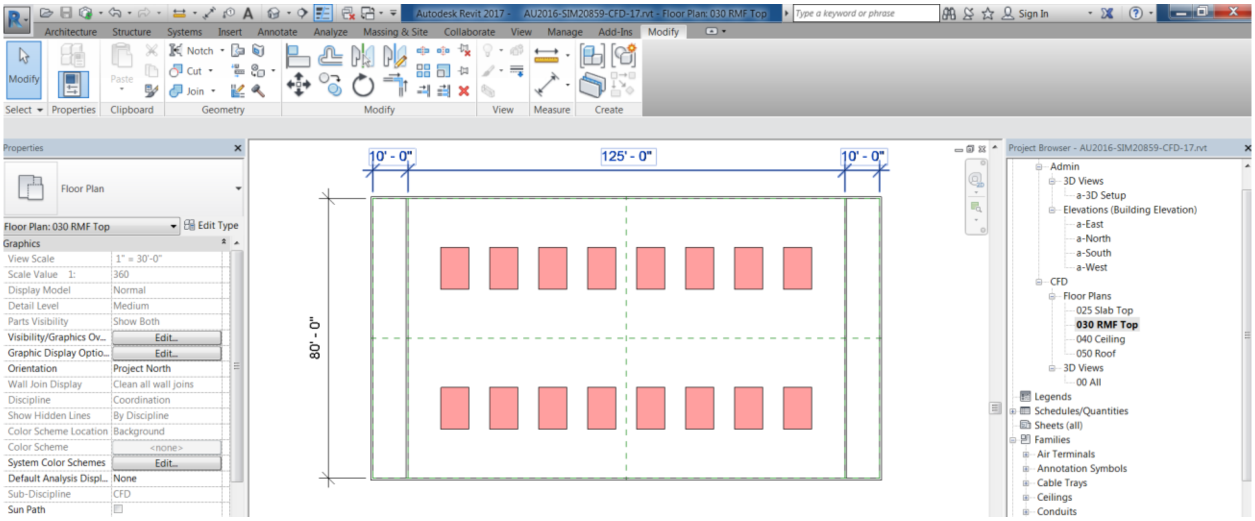
Now we need to place the recirculation air components, such as the FFU's, perforated tiles and sensible cooling coils (dry). These generic families have been custom created to be hosted in floors, ceilings and walls, and they cut openings in their hosts so that there are no geometric clashes or interferences.

From the CFD input parameters, we need to place 160 FFU's. If there is an Architectural model containing them, they could either be copied/pasted from the Architectural model and types changed to the simplified CFD FFU's, or the CFD FFU's could be placed over the corresponding FFU's in a linked Architectural background. These may also reside in a Mechanical design model or as-built plans, etc.

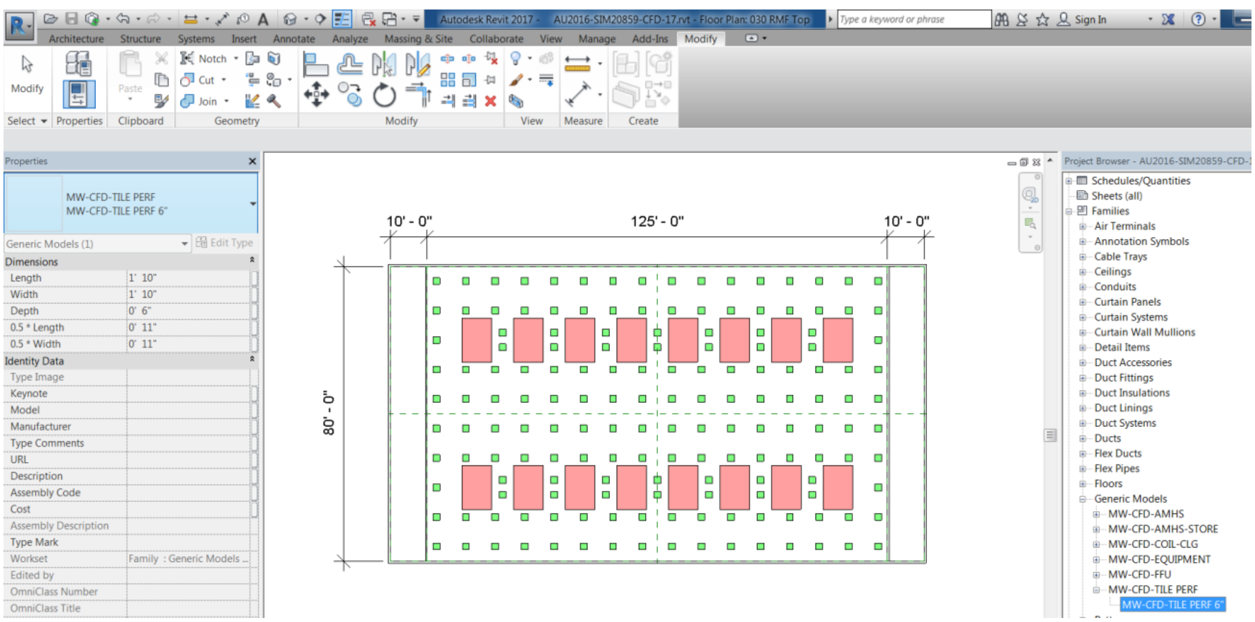
Place 160 **MW-CFD-FFU** family instances manually in the ceiling on the *040 Ceiling* level on the **CFD-FFU** workset. Assume that this cleanroom ceiling system consists of 48" x 48" grid. Feel free to use whatever placement methods you are comfortable with. Two reference planes at the centerlines of the domain can be helpful for symmetry and alignment.



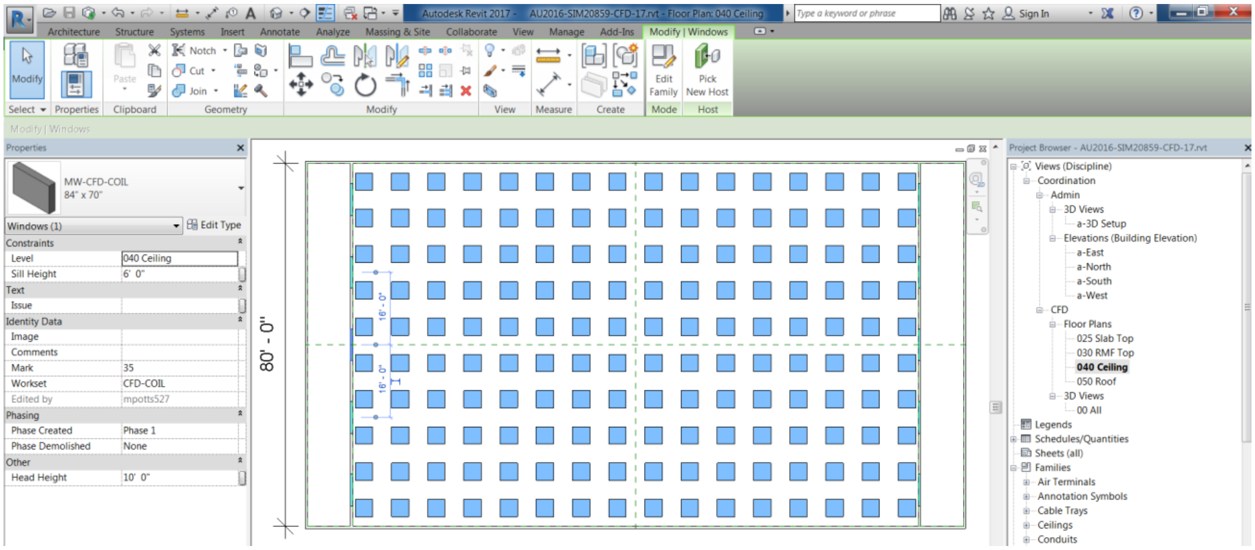
Per the CFD inputs worksheet, place the 16, **MW-CFD-EQUIPMENT** family instances manually on the *030 RMF* level on the **CFD-EQUIPMENT** workset.

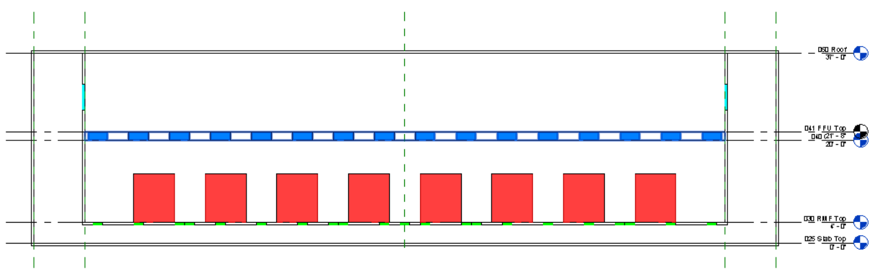


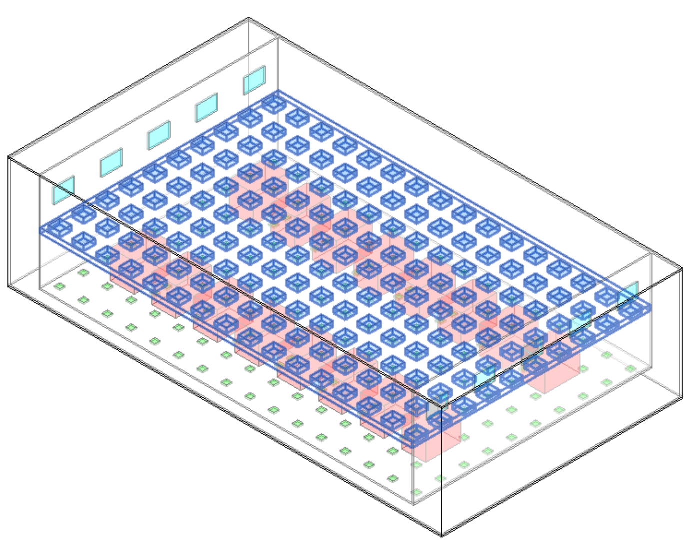
Next, per the CFD inputs worksheet, place 160, **MW-CFD-TILE-PERF 6"** family instances evenly around the tools and across the space. Assume this RMF system consists of 24" x 24" grid. Note that these tiles are the same thickness as the RMF Solid floor, which will be suppressed later in CFD.



From the CFD inputs worksheet, place 10, **MW-CFD-COIL 84”x60”** coils families, 5 coils in each chase wall on the *040 Ceiling* level above the ceiling at a *Sill height* of 6’-0” on the **CFD-COIL** workset. Note that this coil family is actually a modified *Windows* family so that it can be hosted by, and cut into walls.





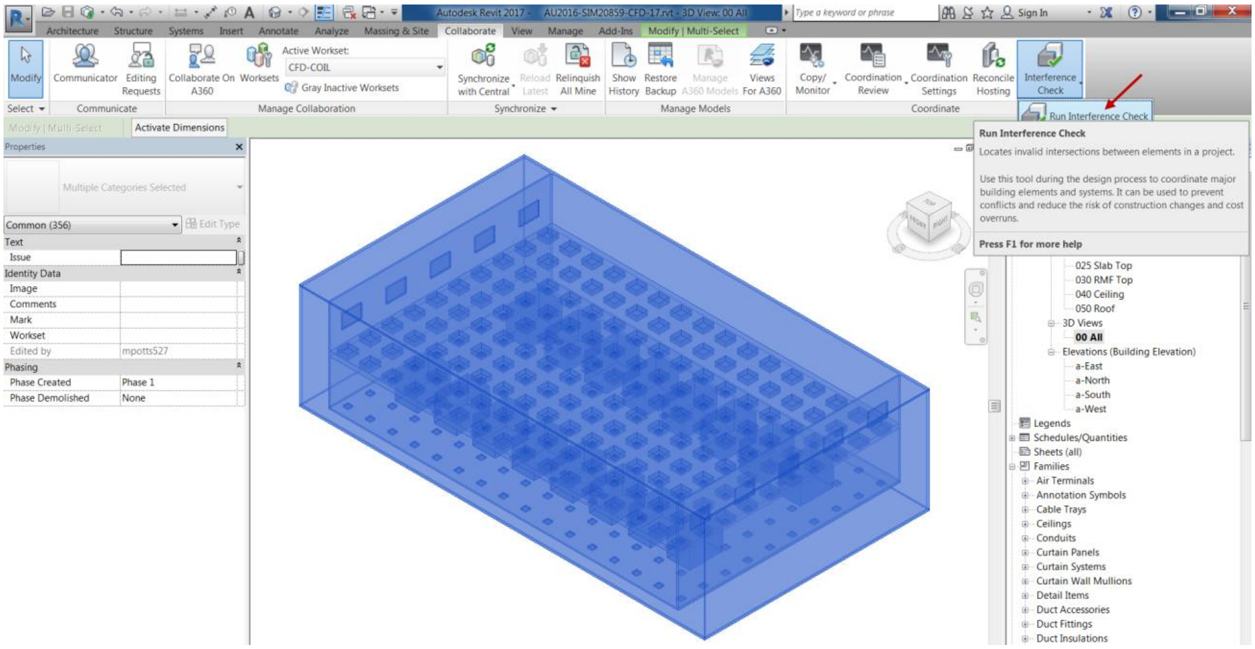


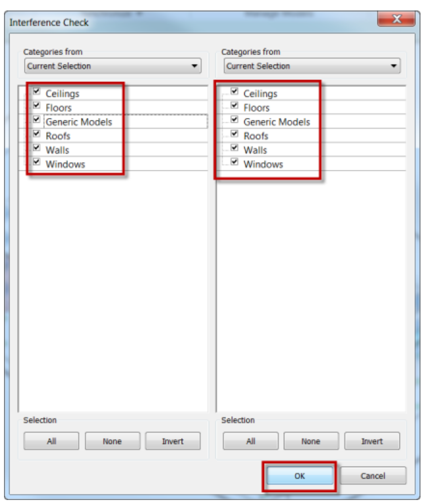
REVIT CFD MODEL WITH ALL COMPONENTS PLACED

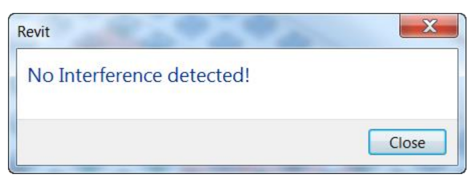
Run Revit Interference Check

At this point, we should have a good, clean idealized Revit model. As a last precaution to prevent issues in CFD, we should run Revit Interference Check.

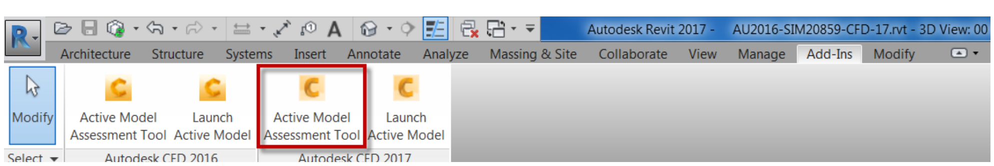
Go to a 3D view showing all model elements, select them all and go to *Collaborate>Coordinate>**Run Interference Check***. No interference should be detected.







Note that for more complex models, it may be beneficial to review the model in the CFD Active Model Assessment Tool. Go to Add-Ins>Autodesk CFD 2017>Active Model Assessment Tool.



Learn What Revit Information Translates to Autodesk CFD and how to Leverage It.

Overview

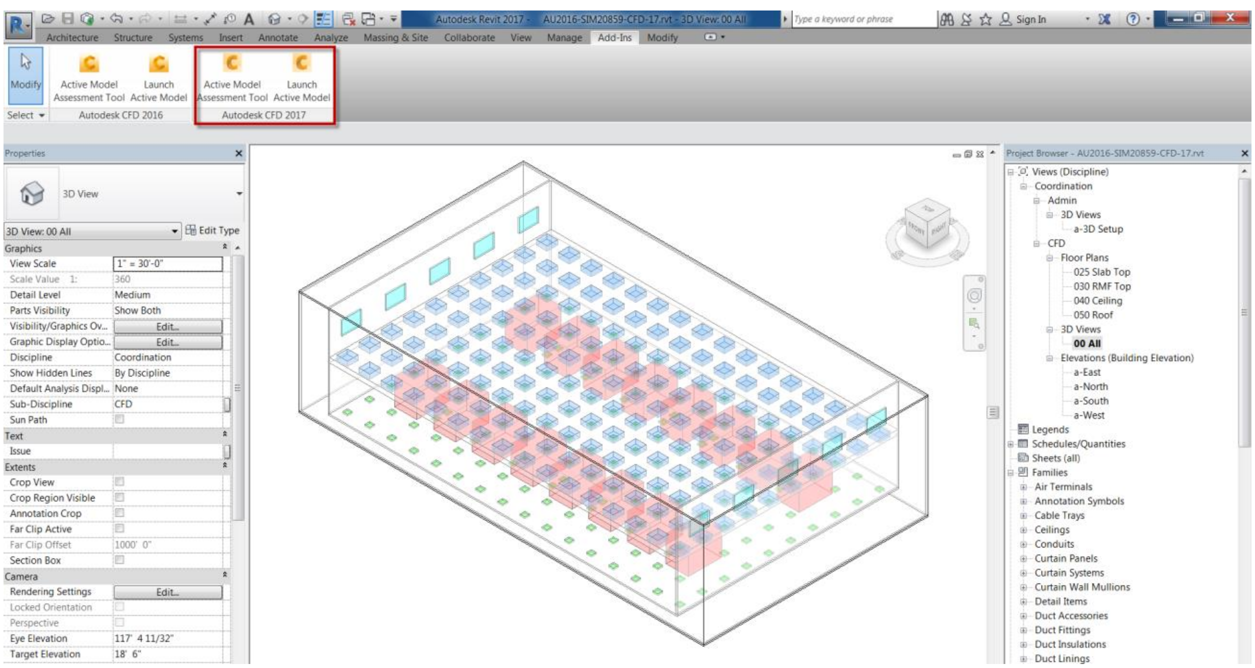
While it is common CFD industry practice to model the air volume in the 3D application, doing so in Revit seems to limit Autodesk CFD's ability to retain object associations when Revit models have geometry updates and are subsequently relaunched. This results in lost or misplaced assignments on existing objects that should not have changed. Therefore, as a lesson learned, it is recommended that if the project and Revit model is in development and not static, that Autodesk CFD should be allowed to create the air volume and it NOT be modeled in Revit. Note that this issue does not seem to exist at this time in AutoCAD-based applications.

Launch The Revit Model Into CFD

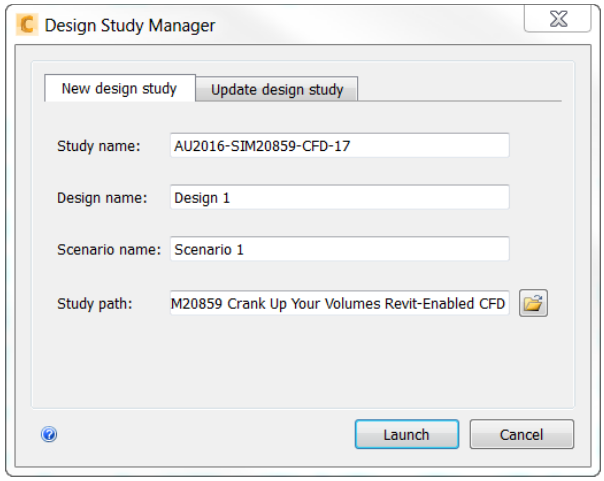
In this exercise, we will launch all of the geometry into CFD 2017. Note that only visible Revit elements will be launched. This implies that multiple 3D views can be configured in Revit to be sequentially relaunched and progressively build up the CFD from shell, to recirculation components only, to full model with all obstructions and equipment.

- Go to a 3D view of the Revit CFD model that displays all elements that are to be launched into CFD.

- Go to *Add-Ins>Autodesk CFD 2017>**Launch Active Model***.



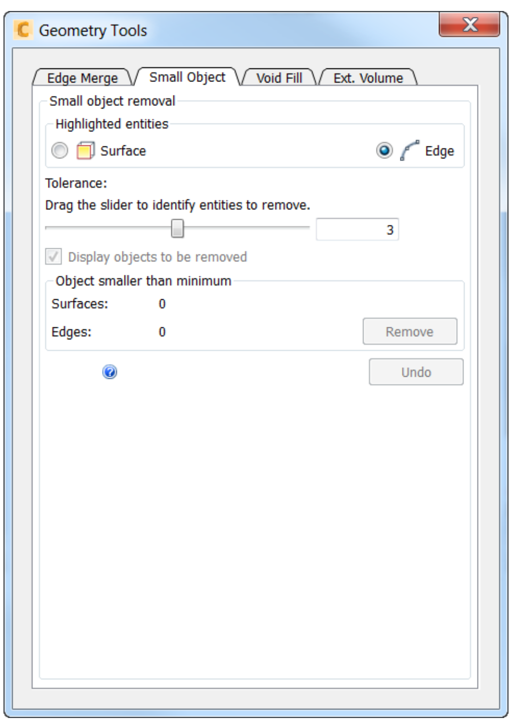
- In the **Design Study Manager** window, edit the *Study, Design and Scenario name* fields as desired, set the *Study path* and click **Launch.**

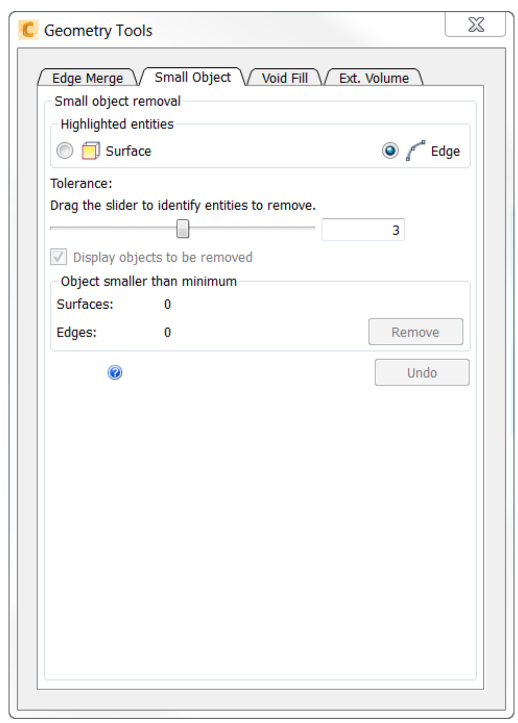


Review CFD Geometry Tools Results and Settings

When a model is launch into CFD from Revit or other Autodesk 3D modeling application, void fill volumes (to be assigned as Air later in this exercise) are created automatically and a geometry diagnostic sweep is performed. Potential model issues are reported in the **Geometry Tools** window, and can be reviewed and processed. Again, it is important to know your 3D model and element size, especially here.

Here we see the benefit of our decision to limit the minimum element dimension to 6". Note that there are no *Small Object* or *Edge Merge* issues.



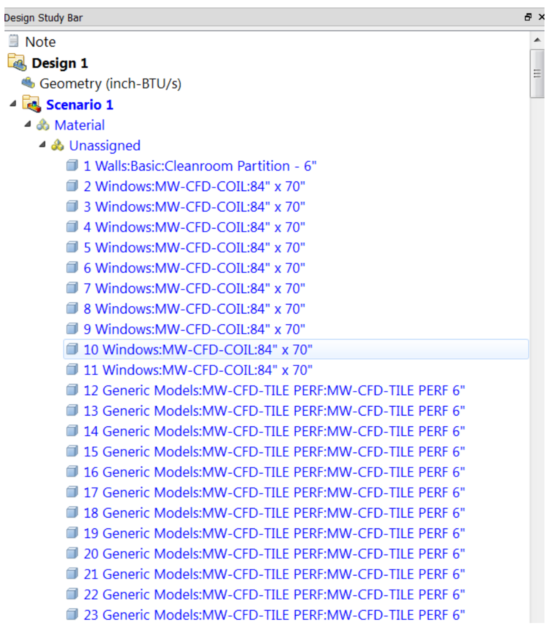


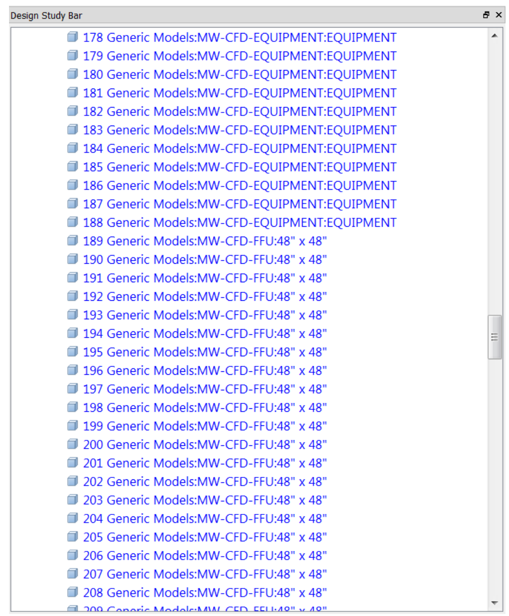
Review The Translated Revit Information

Revit Family categories, names and types translate to CFD on launch from Revit. Note that had we modeled the air volume in Revit, that volume name would prefix the Revit family info making much longer. This is yet another argument not to model the air and rely more on the air tightness of the envelope components.

With some pre-planning and configuration of the Revit family and type names, they can be leveraged in CFD to automate CFD Materials assignments. However, CFD automation is outside the scope of this document.

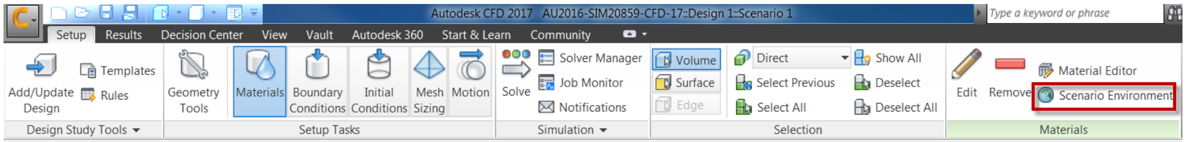
Note the Revit Family Categories, Names, and Types as the CFD object names below.

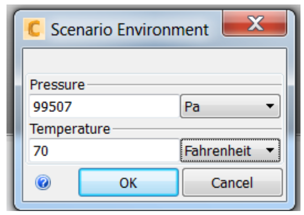




Set The CFD Scenario Environment

Enter the domain pressure and temperature from the CFD inputs worksheet into the *Setup>Materials>Scenario Environment>Pressure and Temperature* fields.





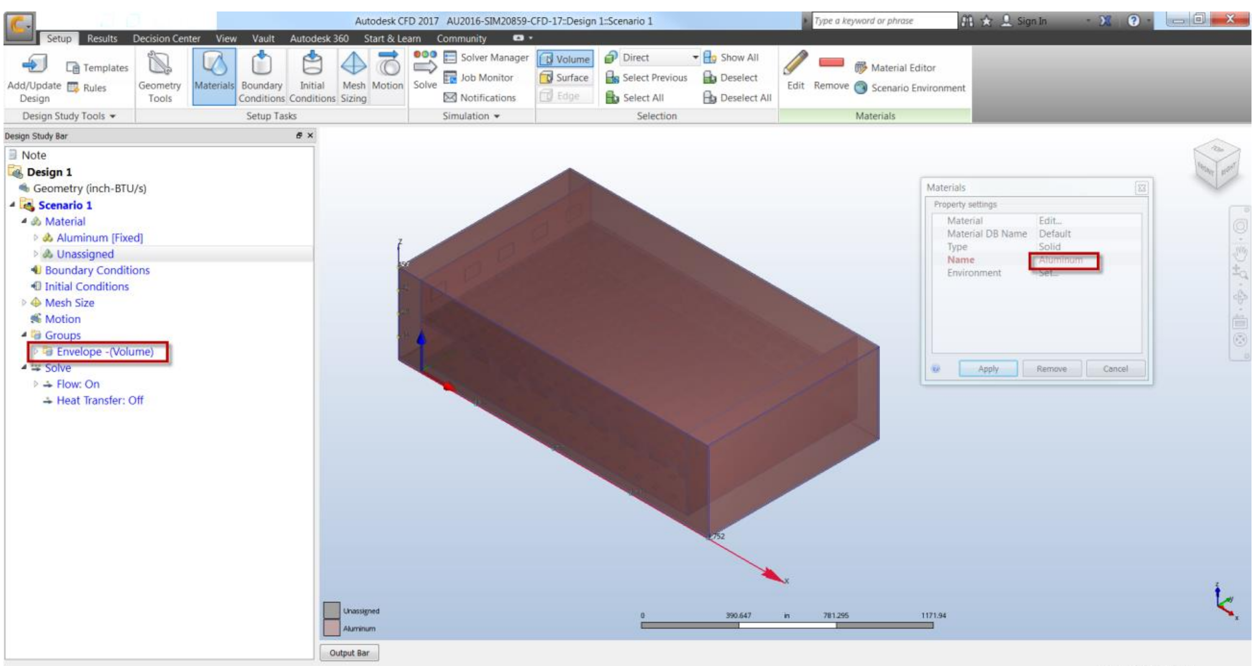
Group The CFD Objects and Make Materials Assignments

Grouping the CFD objects is beneficial in both general organization of the content, and later meshing and later reselection and reassignment in additional scenarios.

Note that at this time CFD 2017 does not support CAD Entity Grouping in Revit 2017. This functionality exists between CFD 2017 and AutoCAD-based applications.

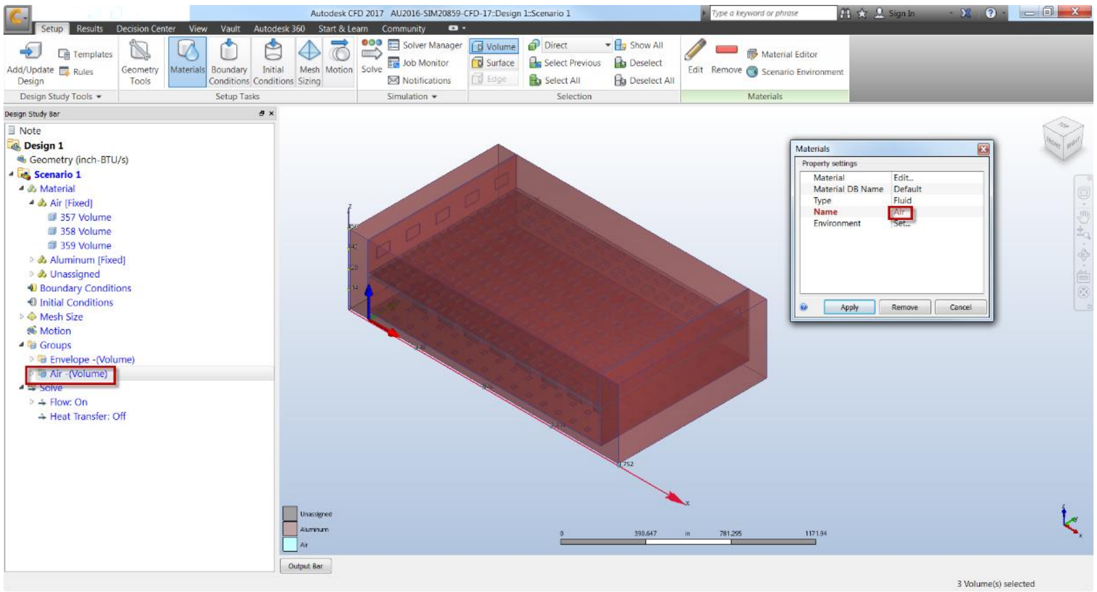
Select the envelope components, create the group **Envelope** and assign as ***Default>Solid>Aluminum***. This will be suppressed later, so the actual material is irrelevant. Use a material that will not be used by any unsuppressed objects.

It is good practice to hide objects that have been assigned as you drill down through the model objects.



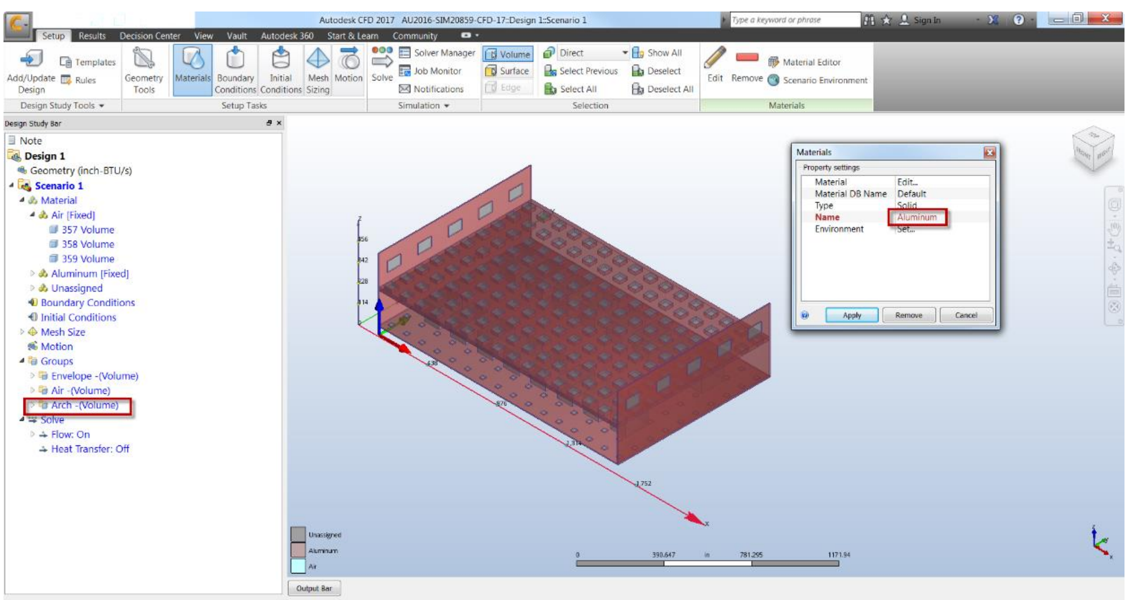
- Select the void volumes created by CFD on launch from Revit, create the group **Air** and assign them as ***Default>Fluid>Air.***

- Select the **Air** group and hide it.



- Select the internal cleanroom architectural components chase walls, ceiling and floor, create the group **Arch**

- Assign them as ***Default>Fluid>Aluminum*** (to be suppressed later)**.** Select the **Arch** group and hide it.

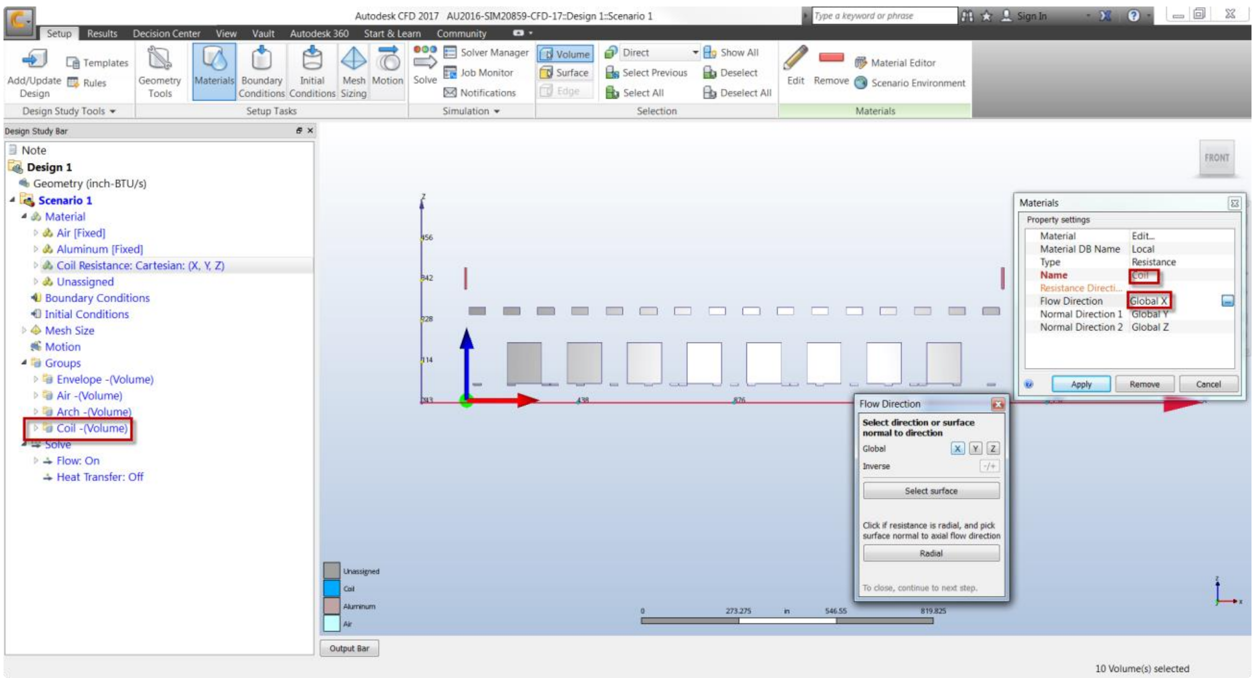


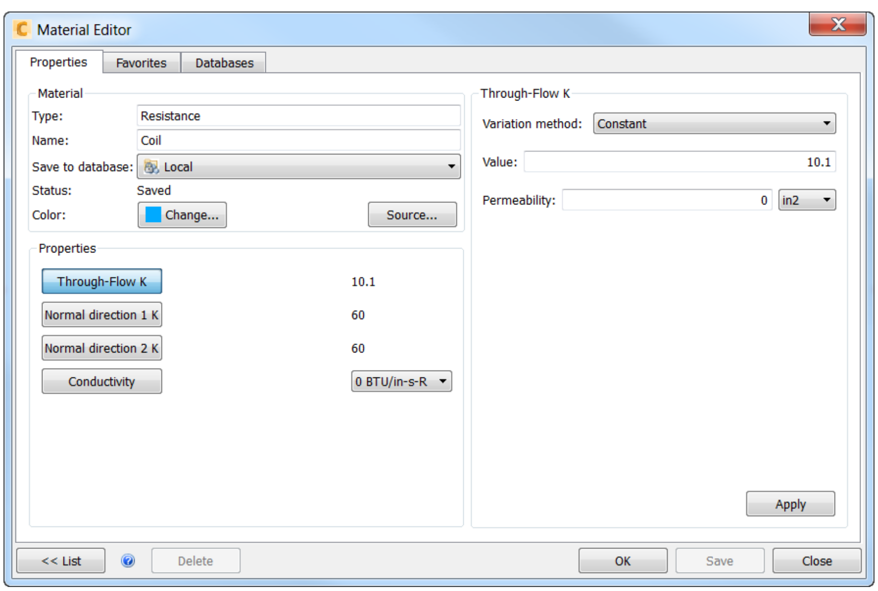
- Change the view cube to **Front**, select the cooling coils, create group **Coil**, and assign them to **Local>Resistance>Edit…**

- Create new resistance ***Coil*** with **Through-Flow K Constant value = 10.1** (calculated in CFD input worksheet)

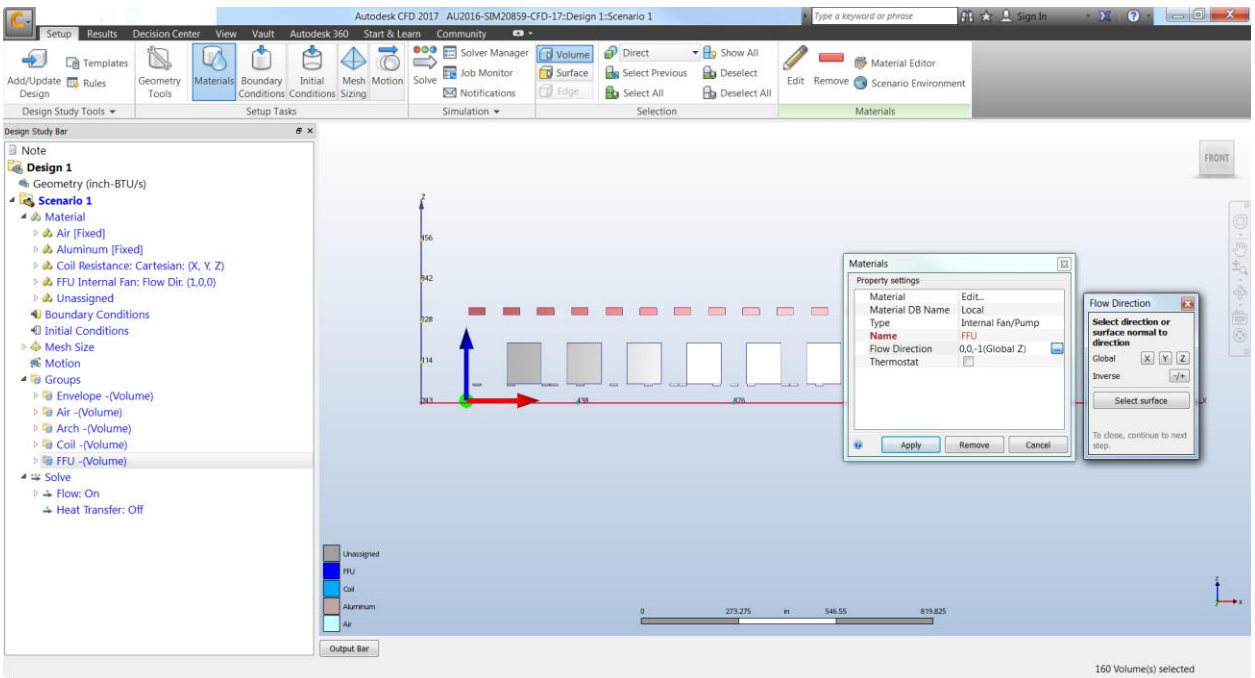
- Set the remaining **Normal direction 1 & 2 K** set to at least 5x the **Through-Flow K** value to force air in the through direction.

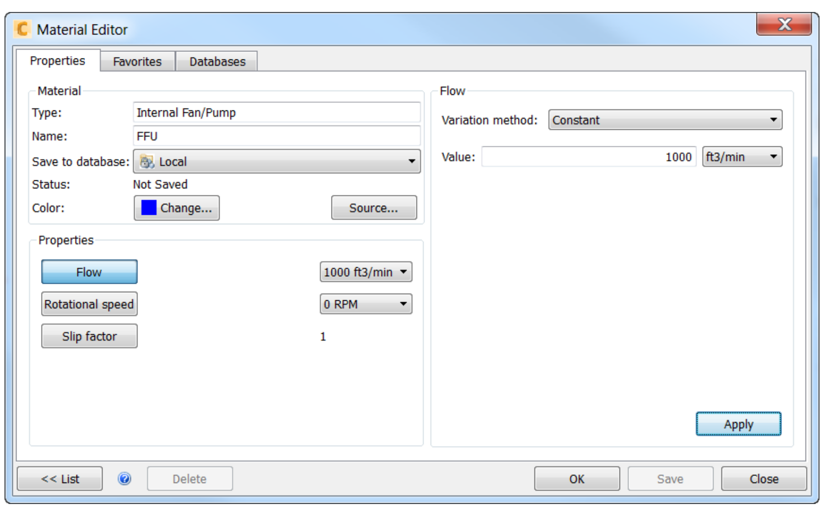
- Ensure the Flow Direction is in the X direction. Select the **Coil** group and hide it.





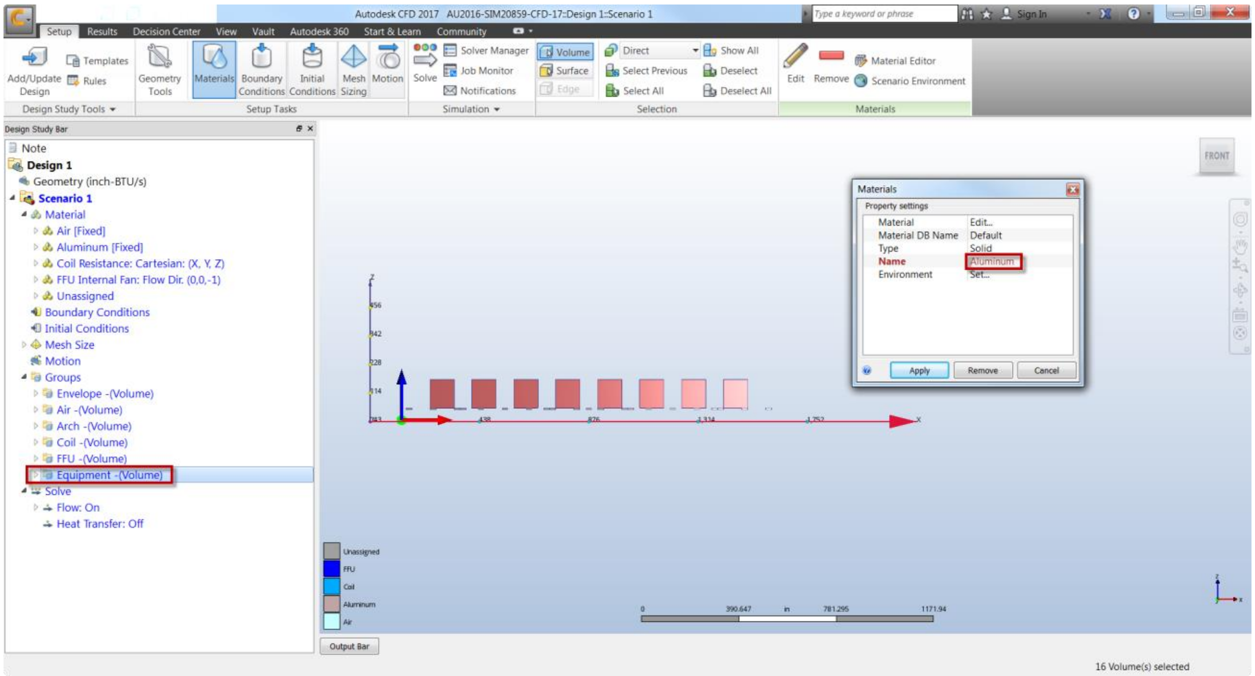
- Select the FFU's, create group **FFU** and assign them to **Local>Internal Fan/Pump>Edit…**
- Create new Internal Fan/Pump ***FFU*** with **Flow Constant value = 1000 ft^3/min** (calculated in CFD input worksheet).
- Ensure the Flow Direction is in the **-Z** direction.
- Select the **FFU** group and hide it.





- Select the Equipment, create group **Equipment** and assign them to **Default>Solid>Aluminum** (they will be suppressed for this exercise).

- Select the **Equipment** group and hide it.

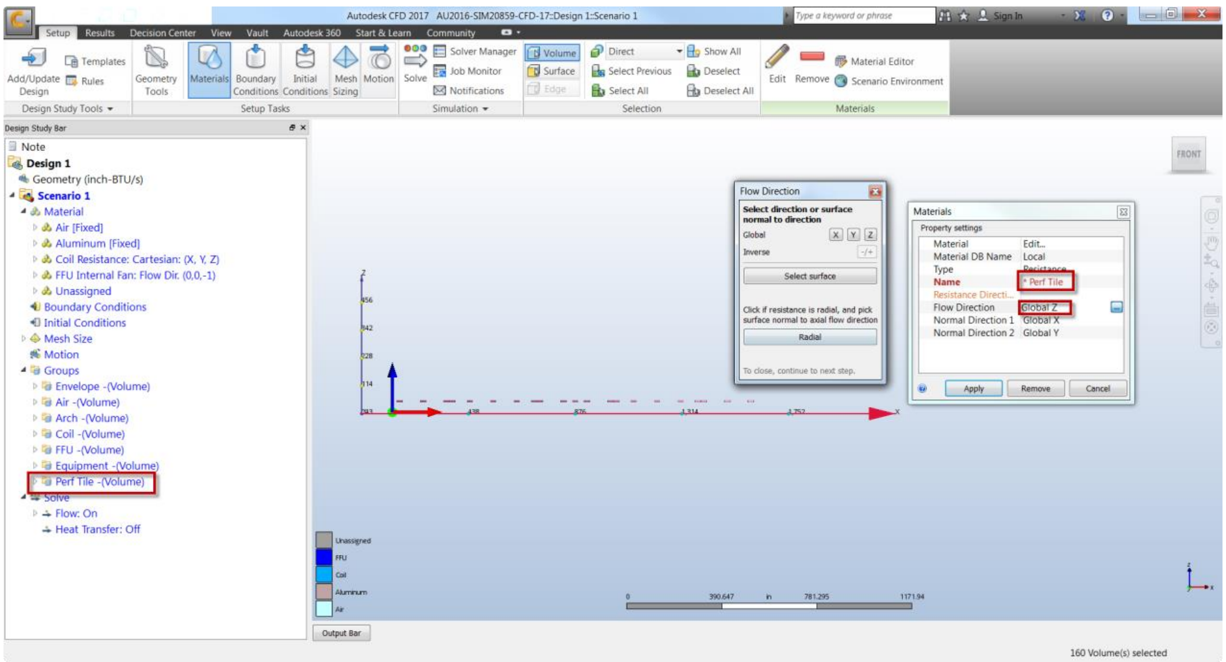


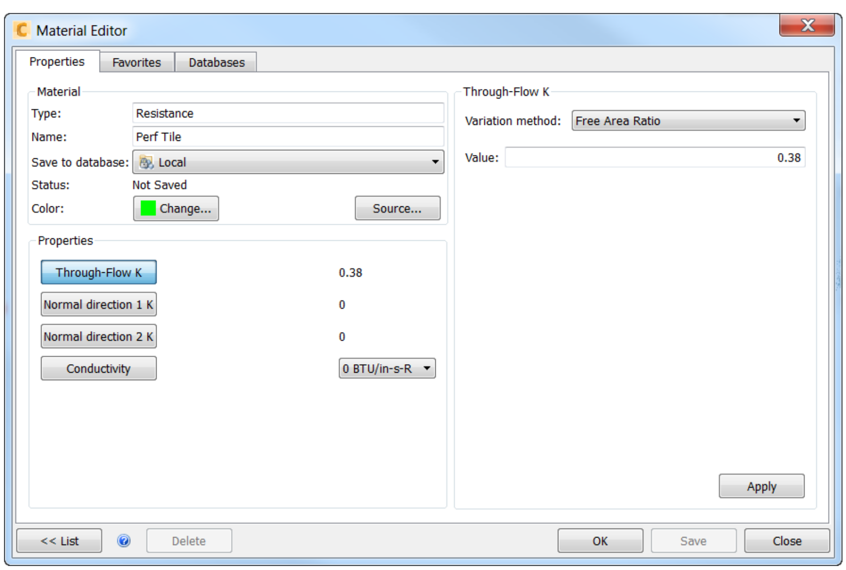
- Select the perforated tile, create group **Perf Tile** and assign them to **Local>Resistance>Edit…**

- Create new Resistance ***Perf Tile*** with **Through-Flow K Free Area Ratio = 0.38** (from M+W Products data in CFD input worksheet).

- Set the **Normal direction 1 & 2 K** Free Area Ratios to **0** to prevent non-through-flow direction. Ensure the Flow Direction is in the **Z** direction.

- Select the **Perf Tile** group and hide it.

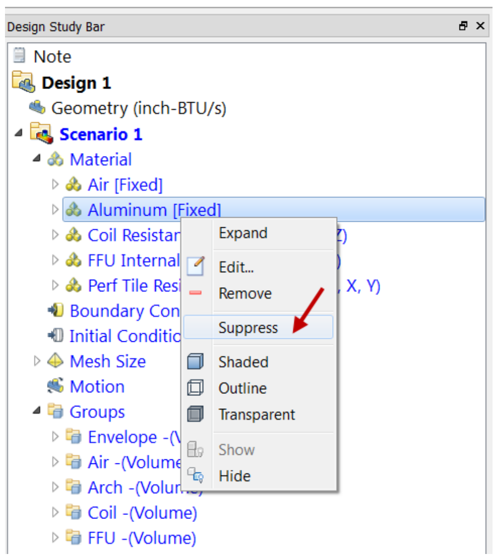


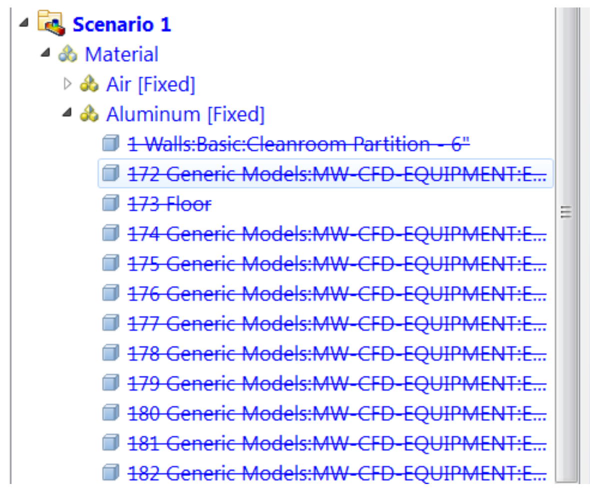


Suppress The Obstruction-Only Materials

Recall that we set many materials that would be merely obstructions in the model to **Aluminum** for suppression later.

- In the Design Study Bar>Design 1>Scenario 1>Material>Aluminum>Right-Click>Suppress.
- Expand *Aluminum* and notice that all objects under it are now struck through.



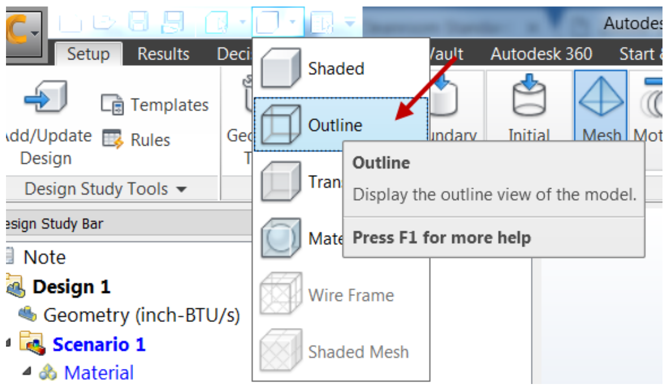


Mesh The CFD Flow Model

With all of the object Materials assigned and Scenario Environment set, it is time to Mesh the model for Flow analysis.

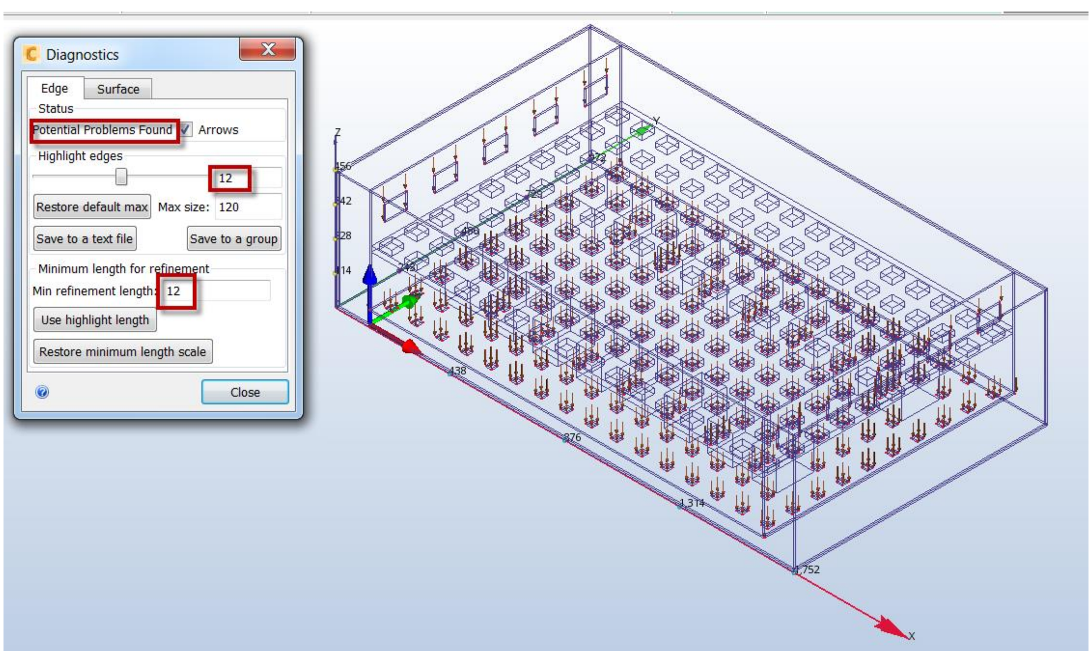
While more advance meshing techniques exist, in this exercise, we will use Automatic Meshing with specific object refinement to ensure the generally recommended 3-4 elements through resistance and internal fan objects.

First we will review the settings and any potential problems in Mesh Diagnostics. Keep in mind again that we specifically limited the minimum object dimension in our 3D model to 6". It is beneficial to set the visibility to **Outline** mode during this task.

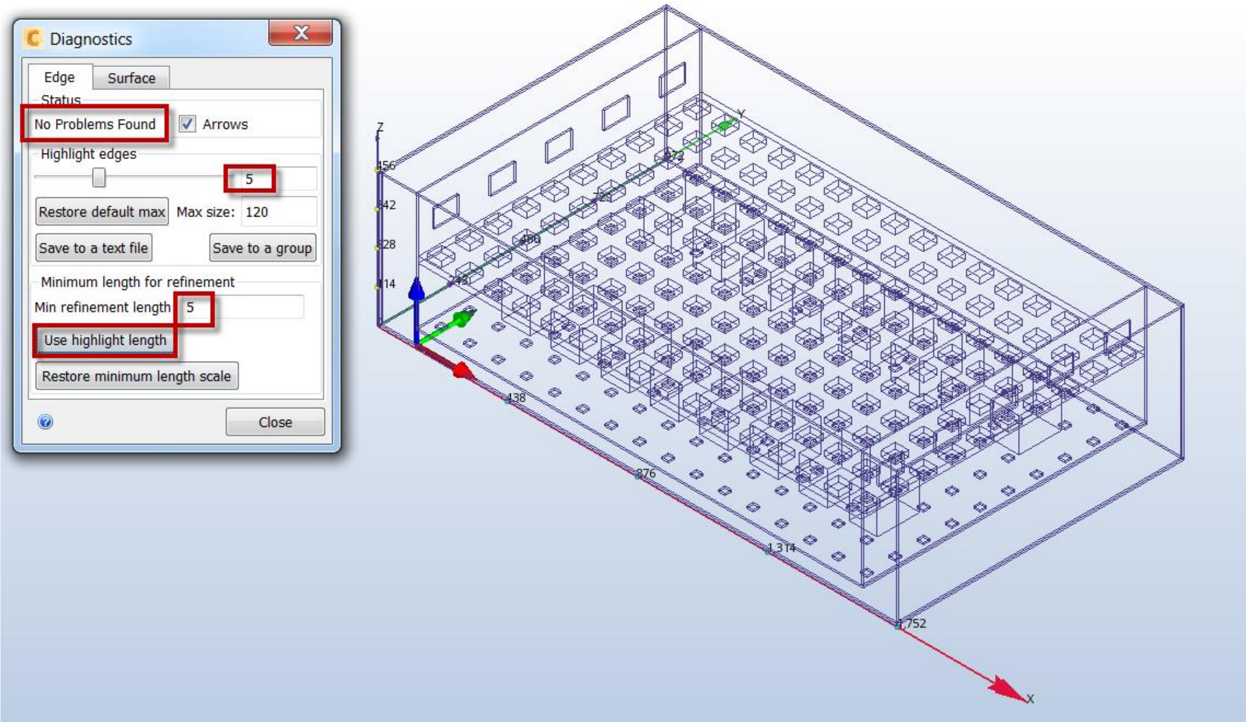


In the **Setup** ribbon, with **Setup Tasks>Mesh Sizing** selected, go to **Automatic Sizing>Diagnostics**.

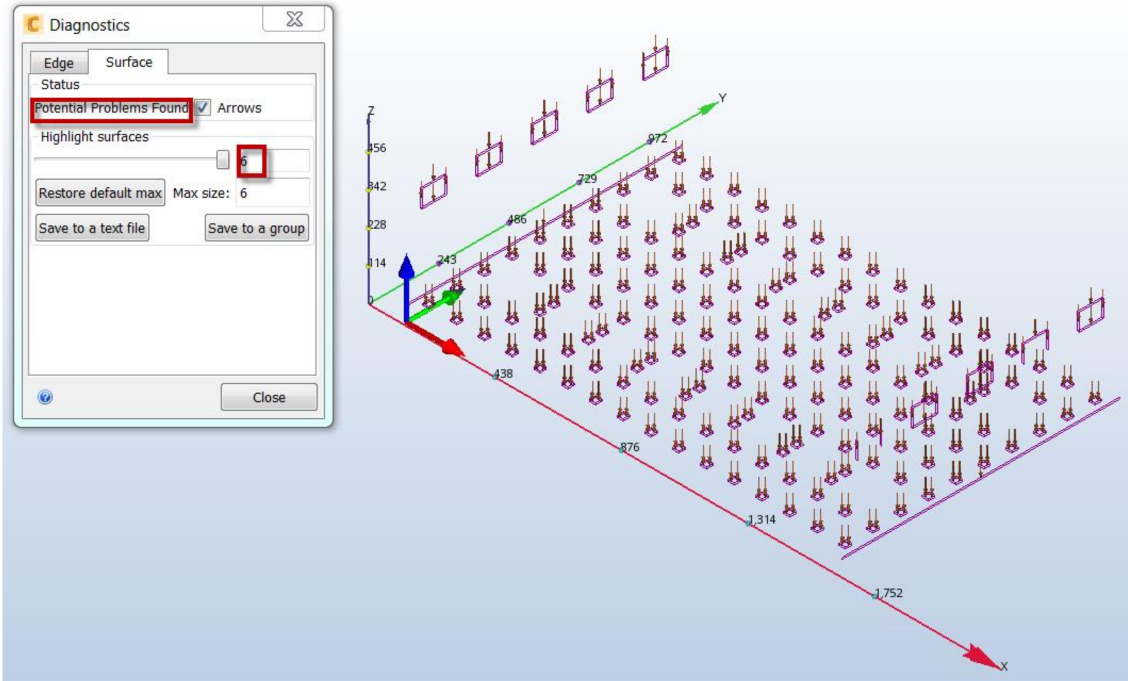
Notice that *Potential Problems Found* is displayed in the *Status* area and that there are arrows pointing to the corresponding problem areas in the model. Notice also that the *Highlight* slider is set to 12 (inches), which will highlight any edges at or less than 12".



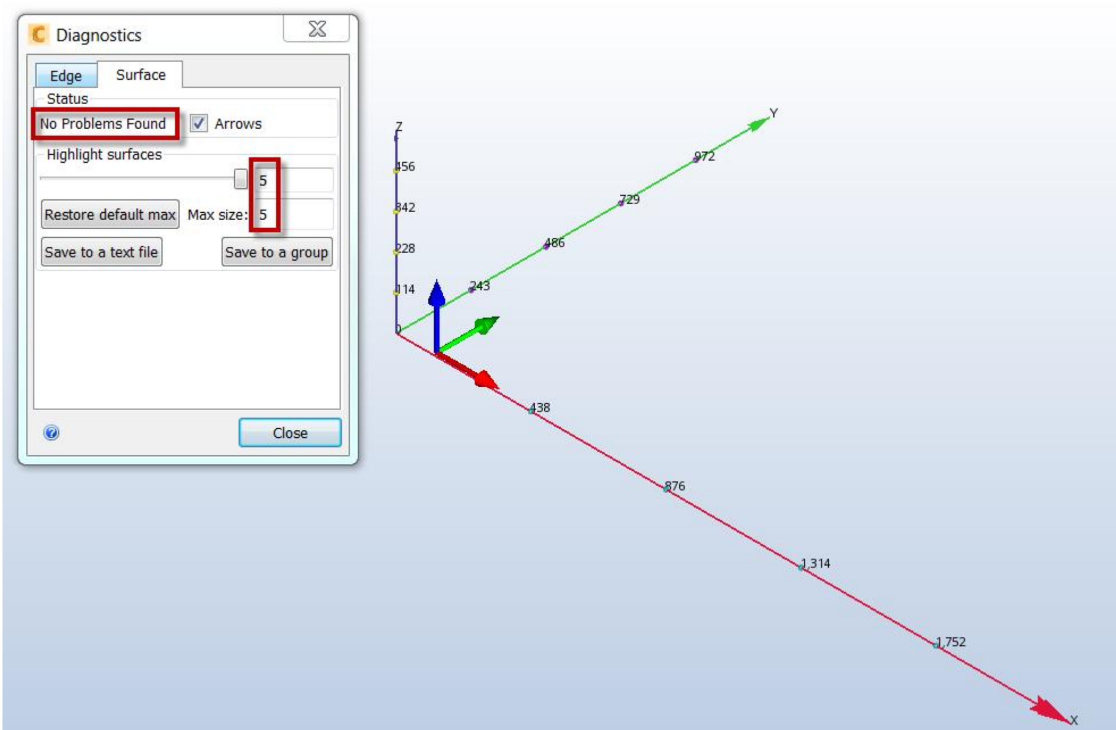
Now enter 5 in the *Highlight edges* field and notice the arrows disappear and the *Status* reports *No Problems Found.* Click **Use Highlight Length** to set *Minimum length for refinement* slightly below our 6" minimum 3D model dimension.



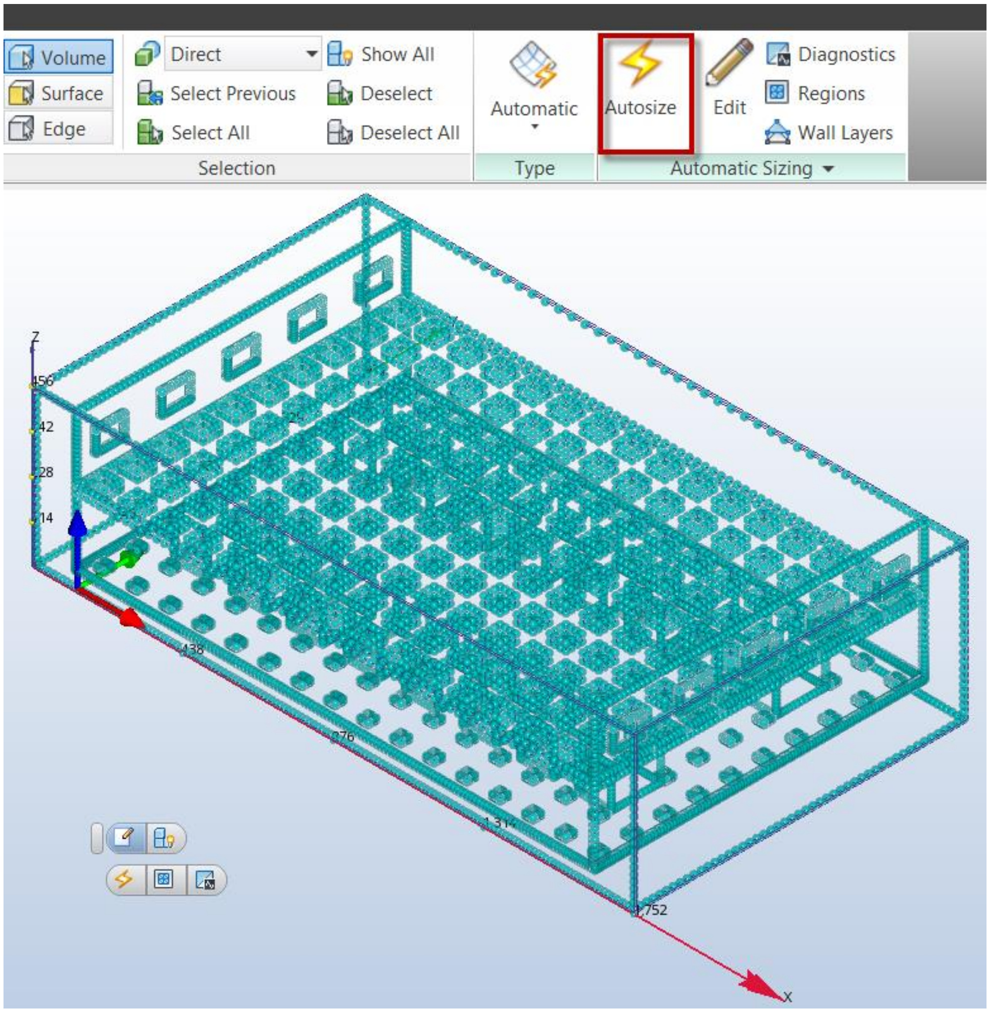
Now click on the **Surface** tab and notice again that with Highlight set to 6 that we have potential problems indicated.



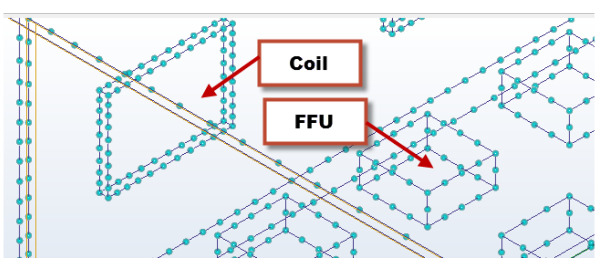
Set *Highlighted surfaces* to 5 and notice the problems clear up. Enter 5 in *Max size* and click **Close.**

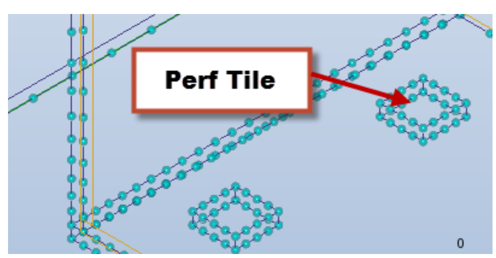


Now we are ready to Mesh the model. Click **Autosize**. The mesh seeds should appear fairly quickly.



Zoom into an FFU and notice that there are only 3 seeds and 2 elements through the volume. There are only 2 seeds and 1 element through the Perf Tiles and Coils. This will not be adequate to get a good velocity profile out of the FFU's or accurate pressure loss through the coils and perf tiles.

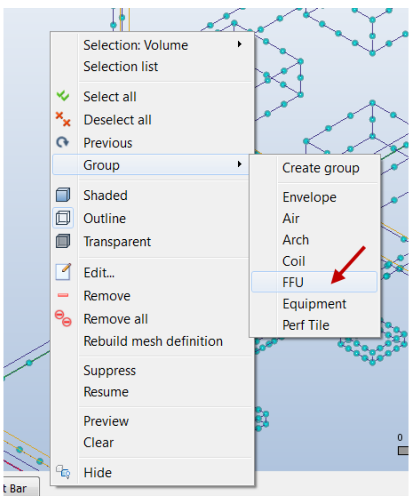


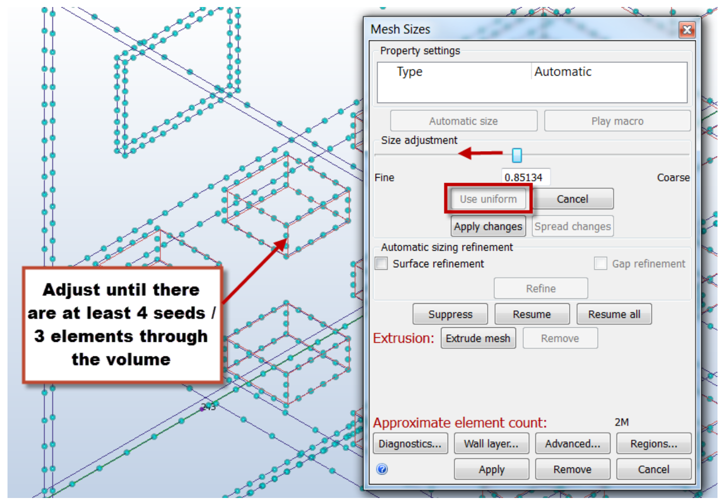


We need to refine the mesh on these objects to get at least 3 mesh elements minimum through the resistance and internal fan, as prescribed by this CFD Help article.

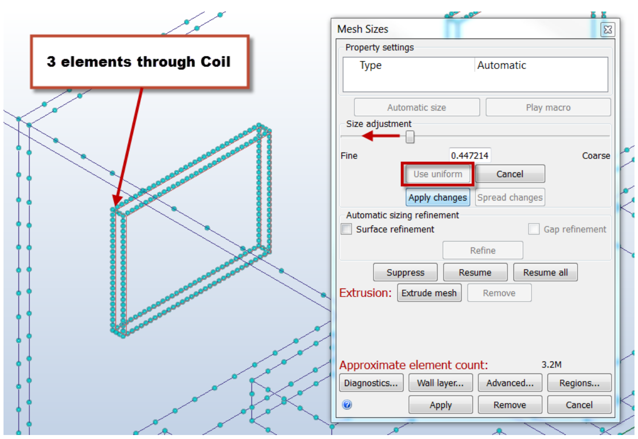
http://help.autodesk.com/view/SCDSE/2017/ENU/?guid=GUID-C87366A9-0FBC-4940-B234-327588E85E0D

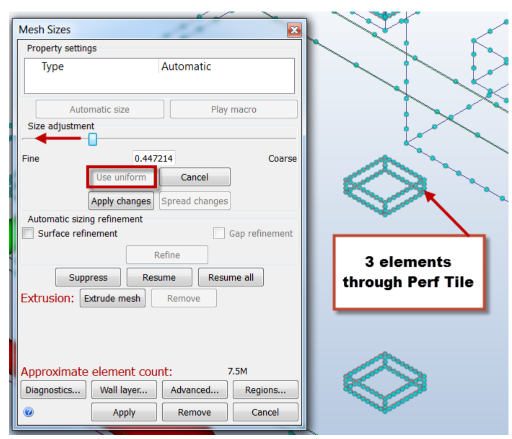
We will now leverage the groups we created earlier to select and edit them. Right-click anywhere in the domain space (not on an object) and select **Group>FFU.**





Repeat for the **Coil** and **Perf Tile** groups.





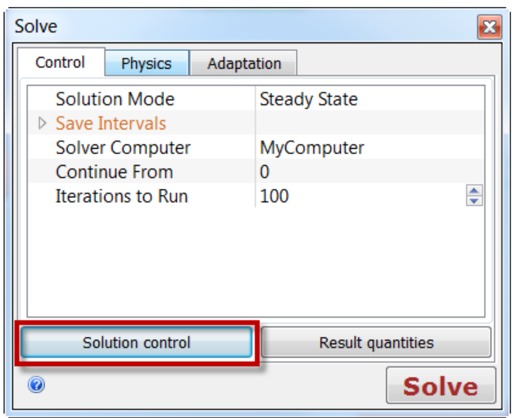
Tip: Note that it is good practice to do small individual component analyses to establish the number of elements required to give accurate results. It is highly recommended that you do a simple, scaled-down wind tunnel analysis on each type of internal fan and resistance to validate your meshing element quantity decision for those objects. It is often much more difficult and time consuming to determine results accuracy on individual objects in a model with hundreds of them.

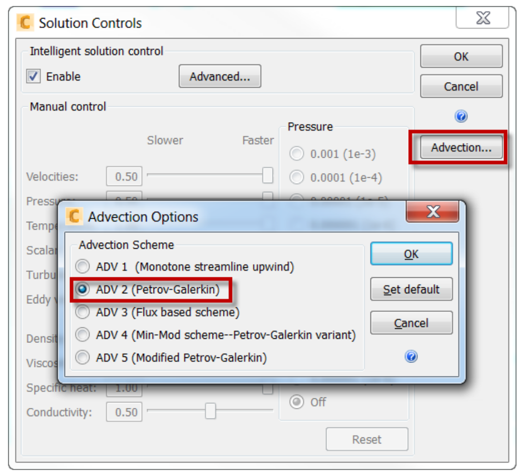
Solve The CFD Analysis Flow Model

With the Scenario Environment set, all of the object materials assigned and meshing complete, it is finally time to solve the flow analysis model.

Click **Solve** to open the settings. Most defaults are acceptable for this type of CFD analysis, but we do need to set **Solution Control>Advection>to ADV 2** for this type of flow analysis that may later have convective heat transfer applied, as described in the CFD Advection Schemes Help article.

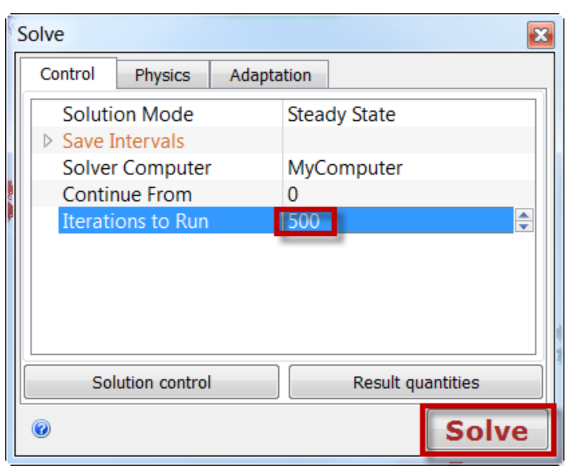
http://help.autodesk.com/view/SCDSE/2017/ENU/?guid=GUID-F691B334-CCE2-47E9-B6C4-21666712C163





Tip: Any model should be preliminarily run for only a couple of iterations to check for errors and flow directions. This can actually be done after clicking **Autosize** and before further refinement as the number of fluid elements is MUCH less at this point and the model with mesh and run more quickly.

This particular model likely wont converge within 100 iterations. 200-300 iterations are expected. We can set it higher as CFD will stop at convergence.



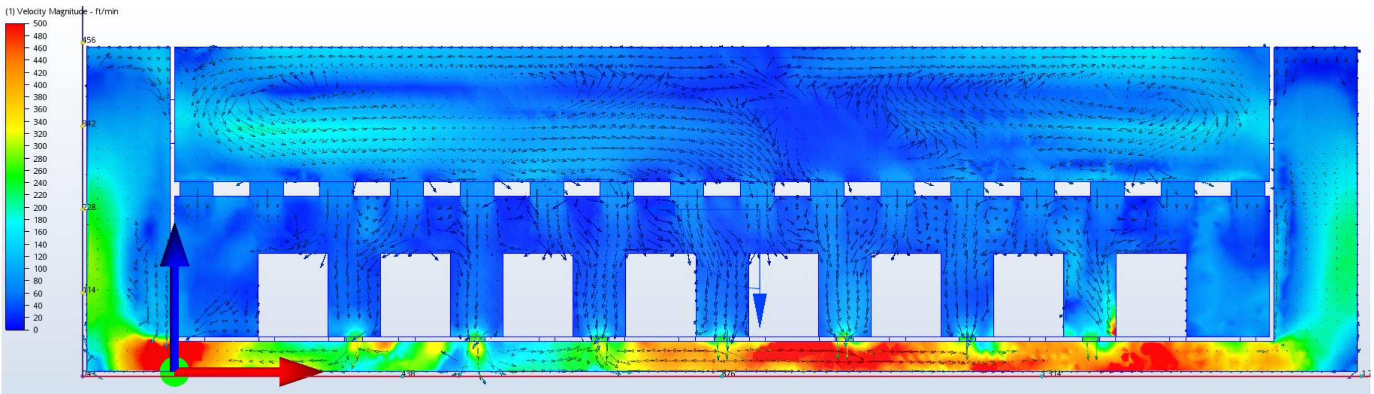
View the CFD Analysis Results

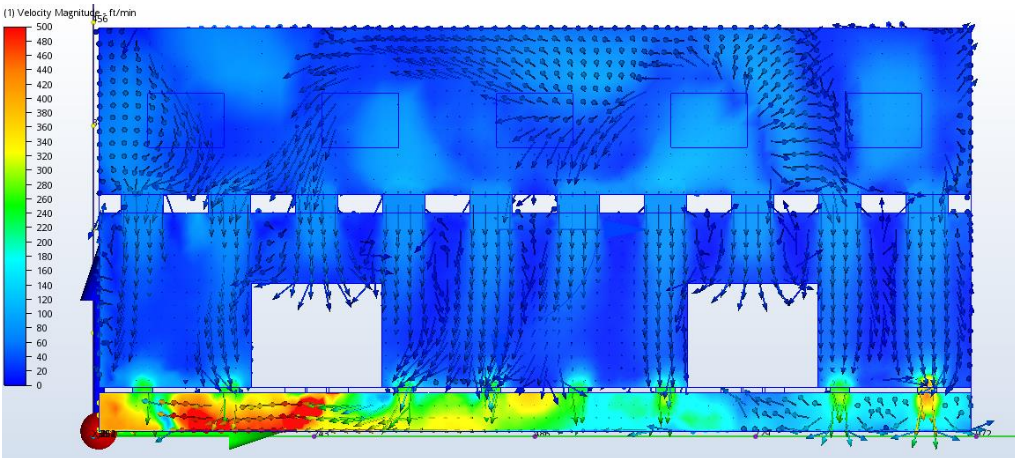
After several hundred iterations to reach convergence, the CFD flow analysis results are ready to view. There are many different ways to view and compare results in CFD, including Planes, Particle Traces, Iso Surfaces and Iso Volumes. There are also several methods to extract bulk data out of CFD for further external analysis and validation.

Detailed procedures for each method can be found in this CFD Help article and subsections.

http://help.autodesk.com/view/SCDSE/2017/ENU/?guid=GUID-BDC6758D-BB45-43E9-B4E7-233904B36232

For this analysis, we will look at a couple of representative 2D Planar sections.





From the results above, we observe that…

- Flow is dominantly vertical and laminar through the cleanroom at the at a 30-60 FPM velocity around the equipment
- Velocities reach 500 FPM in isolated areas under the RMF
- Chase velocities are mostly below the max design of 300 FPM except at the interface of the under-RMF and chases.

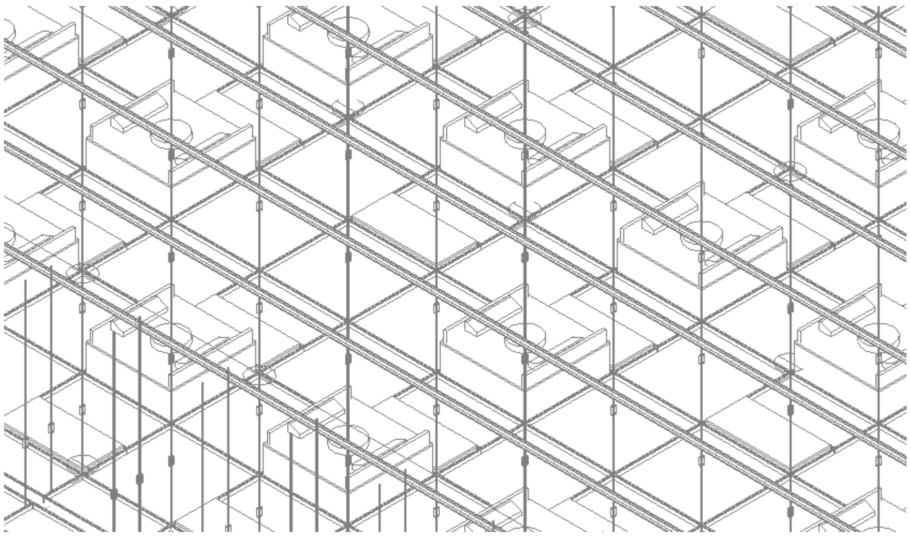
It can be concluded that this cleanroom is operating within the design velocity parameters.

Examples of Revit Design Geometry That Can Cause Issues With CFD

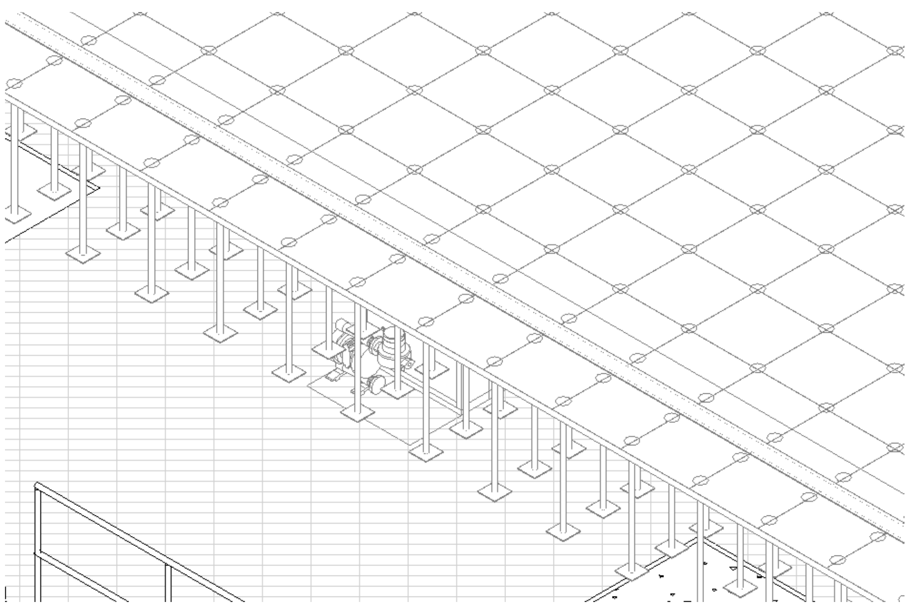
Overview

In this document, we went through a very specific process to ensure a simple, clean, clash free 3D model suitable for CFD analysis.

Here are some examples of the same corresponding Revit geometry for Architectural Cleanroom ceiling systems, FFU's, and raised metal floor systems that is far to detailed and complex for CFD analysis.



ARCHITECTURAL DESIGN MODEL: M+W CLEANROOM CEILING SYSTEM WITH FFU'S



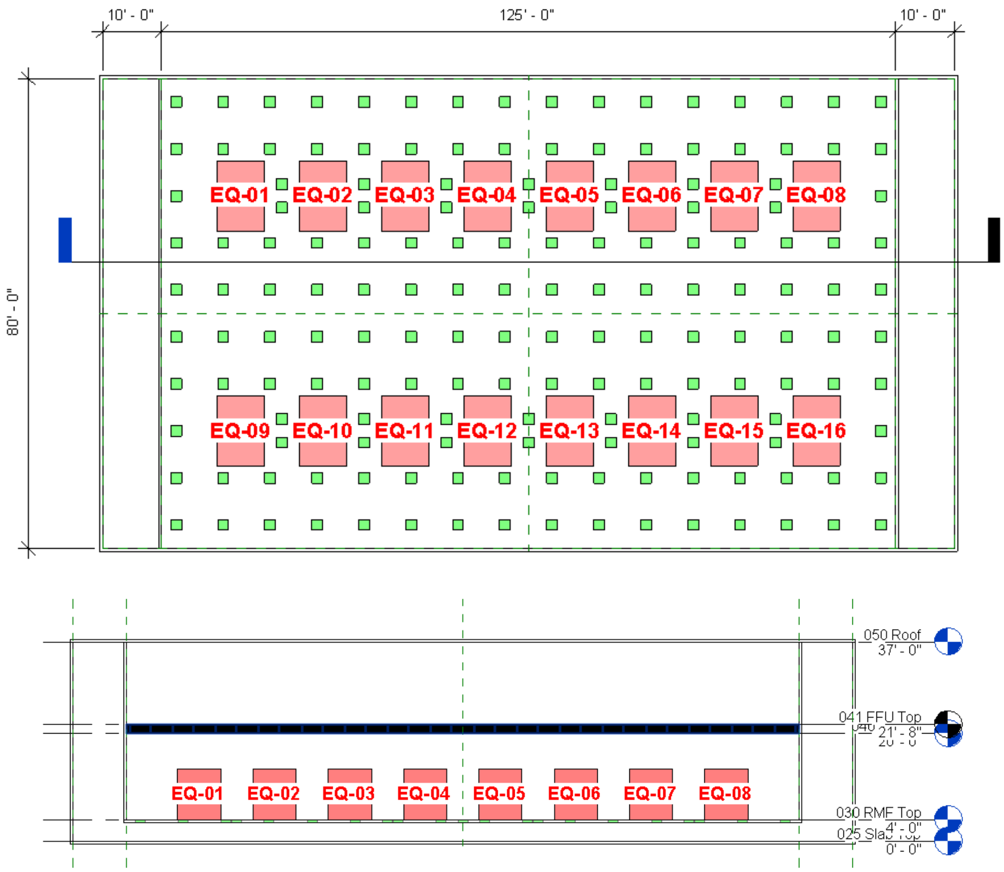
ARCHITECTURAL DESIGN MODEL M+W RAISED METAL FLOOR WITH SUPPORTS

Leverage Revit Data and Parameters To Manage The CFD Project

Overview

Revit offers powerful data and visualization tools for documentation and object management. As already demonstrated in this document, worksets and color filters greatly improve visibility and object management.

The most fundamental parameter in Revit is **Mark**. A unique tag can be placed on any instance or type of an object in Revit, which can be correlated back to equipment schedules, specifications, calculations, etc. With a unique Mark in place across multiple data sources, data can be manually or automatically looked up and sync'd. This can be done with tools as simple as Excel or complex as a Sharepoint report page with an SQL database backend.



SAME TAG INFORMATION CAN APPEAR IN ANY VIEW

Revit places some general default parameters on all objects in the model. Custom parameters can also be created and assigned to the different object categories if desired. Revit schedules and tagging can be used to report information in any view, whether plan view, section or 3D. This can facilitate mass editing of the same parameter on multiple elements, a dimensional change on a volume type, issue comments at the object level, object revision history, etc.

<CFD Schedule>

A	B	C	D
Family and Type	Comments	Issue	Count
MW-CFD-EQUIPMENT: EQUIPMENT	Owner process equipment	Layout out of date as of 10/27/2016	16
MW-CFD-FFU: 48" x 48"	M+W Silent FFU 1000 CFM	Need to change airflow to 1200 CFM	160
MW-CFD-TILE PERF: MW-CFD-TILE PERF 6"	M+W Perforated Tile 38% Free area	Architects want to change layout yet again	160

REVIT DEFAULT PARAMETERS CAN BE LEVERAGED IN DIFFERENT WAYS