BES323521

Revit modeling strategy for successful CFD Analysis: an iterative approach

Leonardo Chonkan
Circuito-JCI

Learning Objectives

- Use Autodesk Revit to create CFD-Ready geometry
- Export building geometry from Autodesk Revit directly into Autodesk CFD
- Use CFD's geometry/meshing tools to simplify the imported geometry
- Create rules to assign materials and run a simple analysis to validate meshing

Description

An efficient CFD Analysis process should take into account the quality of the information being fed to the software. This is of particular importance when it comes to geometry. Adequate meshing is crucial not only for the convergence of a solution but also for the quality an ease of interpretation of results. This instructional demo will demonstrate the practical implications of managing large Revit models from multiple sources and having to integrate them a single mesh for analysis. We will start by creating some geometry in Autodesk Revit and walk through the steps of simplifying and refining that geometry for successful meshing in CFD. We will be using the Import/Export capabilities of Autodesk Revit and CFD before finally setting up a simple design study to validate the imported geometry. This class will feature an iterative approach used for modeling a project in Costa Rica as a case study. Going from basic geometry and then expanding into a more realistic scenario.

Speaker

Leonardo Chonkan is a Civil Engineer with a passion for technology who works as a BIM Coordinator in Costa Rica's largest MEP Design firm. He began teaching people how to use Autodesk Revit for Construction Modelling while completing his graduation project on Building Information Modeling in 2015. Since then he worked for a Land Developer, a General Contractor and a BIM Consultant before finally arriving at his current position with Circuito-JCI, where he is directly involved in the planning and execution of BIM Projects being delivered in Costa Rica, Mexico and Panama. Leonardo is an Autodesk Certified Professional and a member of Autodesk Developer's Network. He regularly teaches BIM related subjects at Universidad de Costa Rica, Universidad Creativa and Cámara de Construcción Costarricense.

About Johnson Controls Inc.

Johnson Controls' Center of Excellence Network enables global customers to streamline and standardize their security, HVAC, mechanics and fire systems on a global basis. It is Johnson Control's goal to serve as a key player in driving innovation and delivering unique value to our strategic customers. Customers who leverage the CoE Network benefit from local talent which is backed by Johnson Controls' 140+ years of experience and breadth of technology expertise, freeing them to focus on moving forward with new, differentiated solutions.

About Circuito

Our Circuito CoE provides services and integrated engineering solutions with a large focus on MEP design. Since 1975, Circuito has helped architects, corporations and business owners to obtain successful, sustainable solutions that are innovative and quality-driven.

- 140+ Circuito employees
- Three (3) Circuito offices throughout Costa Rica, Mexico
- and Panama
- 1,500+ customers served
- 3,500+ projects delivered
- 20+ high-level certifications held

About Zavia Capital

ZAVIA CAPITAL is a real estate development company based in Costa Rica focused exclusively on identifying, acquiring, and developing successful and innovative projects in Costa Rica and the United States. ZAVIA Capital is focused on creating value by developing projects with a high added value within growing industries.

About Marlo Trejos Arquitectos

Studio Quinientos Seis, also known as Marlo Trejos Arquitectos, is a design practice based in Costa Rica. They focus on projects that range from Interior Design to large scale urbanistic projects. Their creative, innovative and sustainable thinking has helped establish their reputation in Architecture and Urban Design. For the past two decades they have developed an integral and environmentally friendly approach to architecture and interior design. They see sustainability as the only way to produce long lasting buildings that produce positive outcomes in their environment.

Table of Contents

Introduction

What is CFD?

Computational Fluid Dynamics is a branch of fluid mechanics that uses numerical analysis to solve problems that involved the movement (flow) of fluids. The equations characteristic of this type of analysis are impossible to solve by hand. CFD enables engineers across the world to get a better understanding of the behavior of fluids in buildings, machines, products and a wide variety of other applications by leveraging the capabilities of today's computers. Autodesk CFD is a Software solution that brings Computational Fluid Dynamics to an integrated environment where data is coming in and out of the software as part of a larger, integrated process.

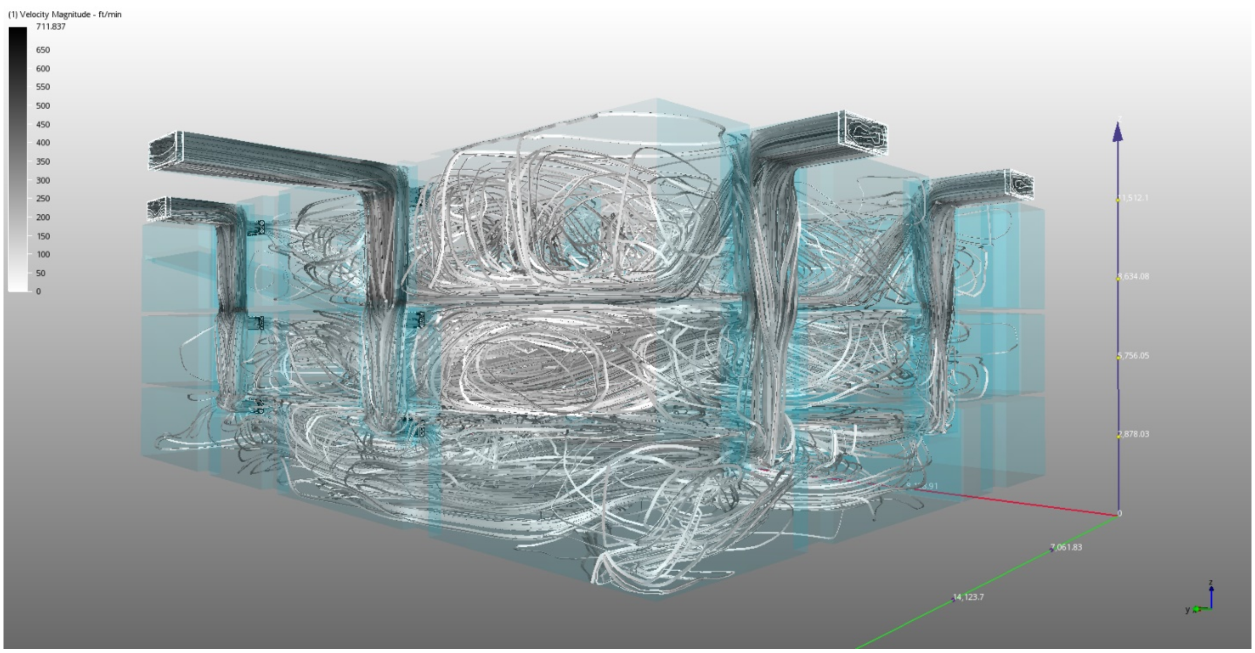


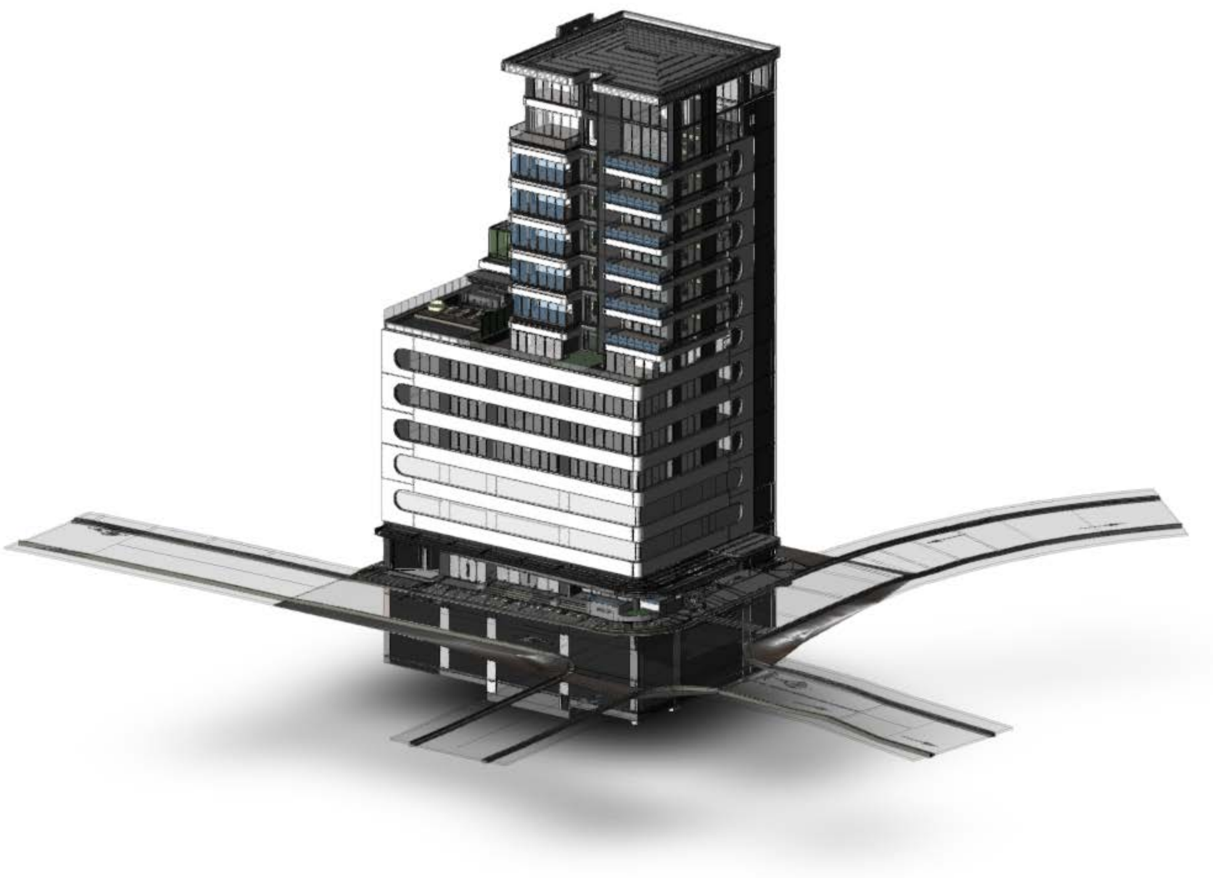
Figure 1. Sample Simulation of Basement HVAC Application

Why use CFD?

By using CFD, designers can take advantage of today's computational advancements to gain insight on their designs and make better decisions in less time. In AEC applications, for example, engineers are using CFD to solve increasingly complex scenarios dealing with internal ventilation, comfort, contaminant extraction, clean rooms, HVAC applications and more.

About the Selection of the Sample Project

In this example, we look at how Autodesk CFD was used to make decisions on the number of inlets/outlets required for extracting Carbon Monoxide (CO) out of a three story basement. The criteria for this analysis was the minimum volume of air to be extracted and the maximum permissible speeds for air coming in and out of the basement. The example provides a level of geometric complexity characteristic of a real project and poses an interesting challenged that can be solved with the aid of CFD.



Overview of the Modeling Process

The process demonstrated in this class can be described in terms of the diagram shown in **Figure 1**, which is a typical cross-functional flow chart where tasks have been divided according to the software in which they take place. The process starts when

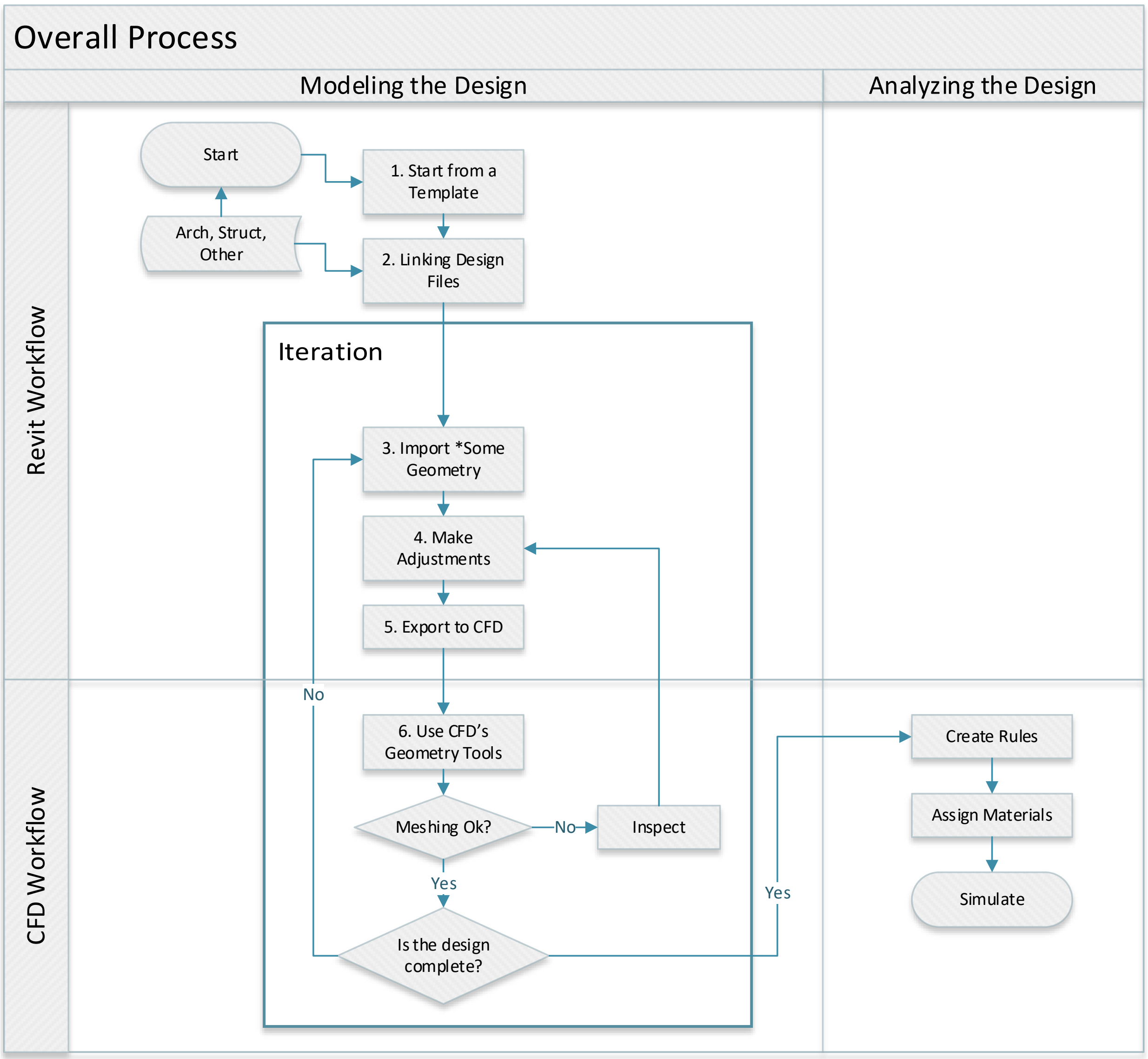


FIGURE 2. FUNCTIONAL DESCRIPTION OF THE ITERATIVE PROCESS

Use Autodesk Revit to create CFD-Ready geometry

Once you have got a problem to solve, it is time to start your CFD Model in Autodesk Revit. This section covers the use of Autodesk Revit 2018 to generate simplify building geometry ready to be exported to Autodesk CFD. The export process is carried out through the CFD Export Add-In that ships with Autodesk CFD.

Start with a template (or not)

If you are in this class and you are not a friend of mine, chances are you don't have a Revit CFD Project Template. Don't stress. Templates are a natural result of continuous quality work. You will have one by the time you have done this a couple of times.

When creating a Project Template for your CFD Models in Autodesk Revit, you should consider having the following items:

- Wall Types
- Floor Types
- Ceiling Types
- View Templates for Modeling / Exporting
- Materials
- Naming Conventions
- Place Holders for Architectural/Structural Models

This class will use a Flow Only simulation to validate our imported geometry. The use of thermal properties and thermal analysis are not covered in this class.

Link-in Design Files

In a typical Revit project there will be at least two companies sharing Revit models in the early phases of a project. When the business case for a CFD analysis has been established, a next step is to create the CFD analysis model. There are many ways to accomplish this task, but this class focuses on re-using other consultants data as a starting point for the modelling process. This approach speeds reduces the time required to model the geometry and allows the users to concentrate on other aspects of the modelling process.

The process for liking the files and setting up the file is as follows:

1. Start by using a Project Template and linking the Architectural and Structural models:

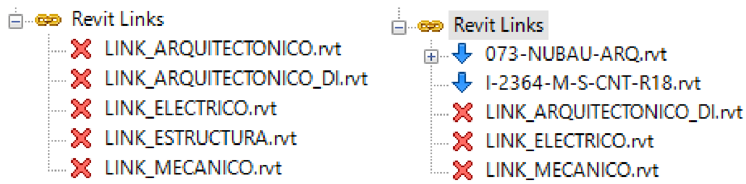


FIGURE 3. LINKING DESIGN FILES USING A TEMPLATE

2. Save the Project and make it into a Central Model so that you can use Worksets for separating the different sources of information from your actual CFD Model. At this point, you can also set up Worksharing Display Settings into something meaningful:

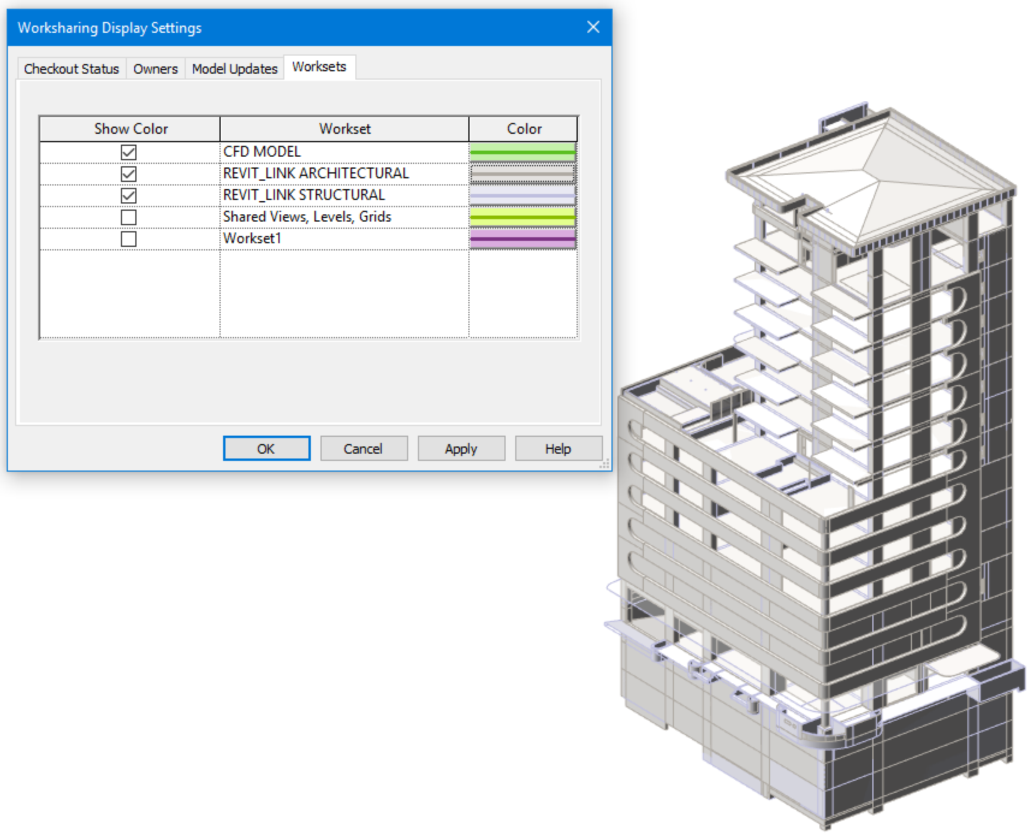


FIGURE 4. SPLITTING OF THE PROJECT INTO WORKSETS TO CONTROL VISIBILITY OF DATA

We use Worksets in this example as means of dividing the Project. You can use as many Worksets as you want given that they make sense to you. **The important concept is separating imported data from the data to be exported**. In this sense, your Revit Project is acting as a bridge for getting design information from other consultants into CFD.

If you want to avoid the use of Worksharing, a simple Text parameter would be sufficient to achieve the classification of elements. By using Worksets and Worksharing features, however, we can take advantage of the built in visualization and model handling features of Revit that are exclusive to Worksets.

3. Once the models are in place and you have performed standard linking duties such as **copying/monitoring levels**, you can **now create as many working views as you like** in order to understand the design and all of its portions.

Importing Geometry

Even in early phases, most AEC Models have a level detail greater than that required for a successful and efficient CFD Analysis. That is, the BIM Uses as defined in the BIM Project Execution Plan often require a Level of Detail (Geometric) often unnecessary in a CFD Analysis.

Having geometry with excessive detail will affect the software's (CFD) ability to create a successful mesh. Careful consideration needs to be given as to which elements affect the behavior of fluids in the simulation and to the amount in which they could an impact on the results. The Geometry Modeling Techniques for AEC Applications section of the Official Product Documentation provides guidance about Model Complexity, Level of Detail, Model Integrity and Associativity best practices.

Copy/Monitor

To cope with the inevitable changes in Design, Copy/Monitor should be used whenever possible. The following categories allow for standard Copy/Monitor procedures:

- Walls (include openings)
- Floors (include openings)
- Columns
- Levels, Grids, Coordinates.

Paste In Place

There are model categories that don't allow the user to Monitor them. We can, however, copy those elements into our project and reference them to elements that are being monitored in order to be able to track changes. Special care needs to be given to non-monitored objects to ensure coordination between disciplines. The following items should be copied from linked models when it has been determined that they could have an impact on the results:

- Ramps
- Structural Framing
- Openings
- Ceilings
- Casework
- Furniture

What To Import

The easiest way to start filtering objects that don't need to be exported is to create a 3D Working View, Open the Visibility/Graphics Dialog and Turn Off all categories. You then start adding categories to your view as necessary. This exercise will make you think of each possible component and whether or not it needs to be exported into Autodesk CFD. After doing this, you end up with a clear view of what needs to be imported.

Figure 5 and **Figure 6** provide a good example of the simplification made possibly simply by filtering out unwanted categories in the Visibility/Graphics Dialog.

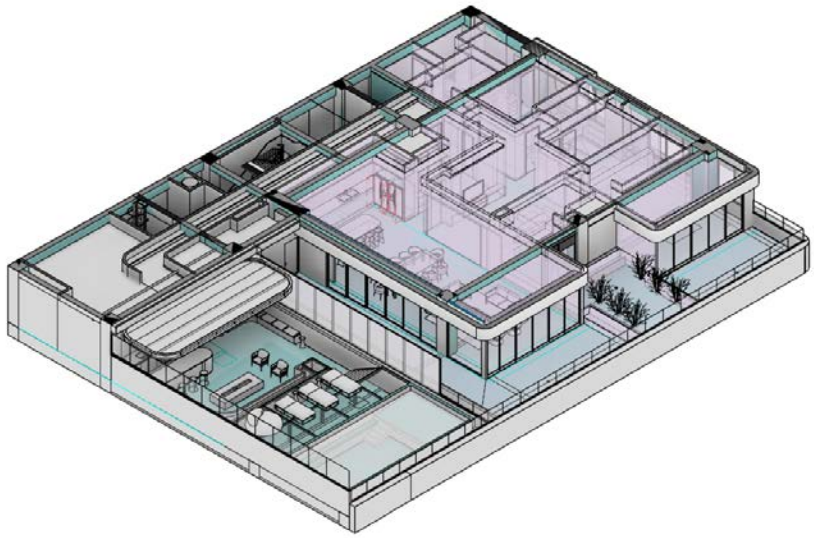


FIGURE 5. 3D WORKING VIEW BEFORE APPLYING FILTERS

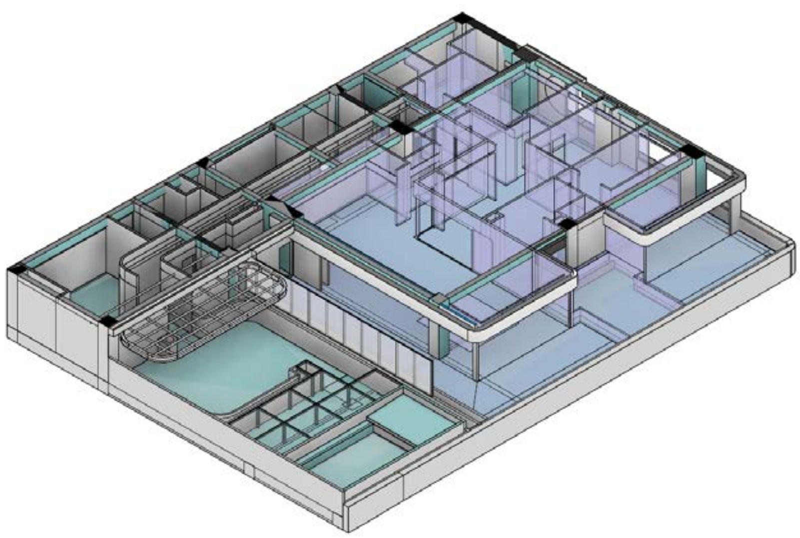


FIGURE 6. 3D WORKING VIEW AFTER FILTERS HAVE BEEN APPLIED

Copying Floors

In this example, we start from the bottom up by creating the Slab-On-Grade on S300 (the lower basement level). This is where our first simplification occurs. The architectural design team has created individual floor slabs so that they don't overlap with structural beam foundations. The structural team has created a combination of Slab Foundations and Structural Framing Foundations to represent what's happening in S300.

Our CFD model will include neither one, but a single slab representation of the concrete elements at that level. In most cases like this one, it is easier and more convenient to simply draw the edges of floor slabs aligned to the exterior wall faces. **Figure 7** shows the application of this principle to the floor slabs of **Level S300** and **S200**.

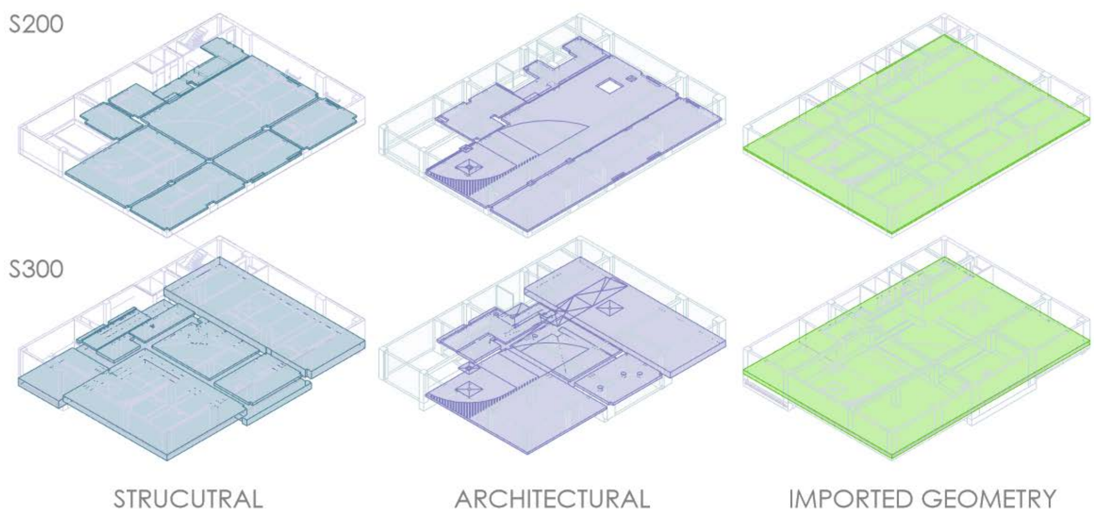


FIGURE 7. FLOOR SLABS FOR S300 AND S200.

Copying Walls and Columns

So far we have imported only two floors slabs into our model. It is time to make our first enclosed volume by importing enclosing walls and also internal walls that surround our Design Study. Not all walls and columns need be copied into our CFD Model. **Only the ones that enclose our Design Study or pose a significant restriction to internal air flow should be copied.** To copy Walls into our model, we can use the Copy/Monitor Tools provided in Autodesk Revit.

A couple of best practices:

- All structural walls are to be copied from the structural model.
- Only non-structural walls are to be copied from the architectural model.

When copying columns, consider if they have an effect on the Design Study or not. In this example, we are only copying internal structural columns that contribute to enclosing the internal volume to be studied. **Figure 8** shows the all walls and columns that were copied into S300. Our first iteration consists of the lowest basement level, so we can go ahead and copy the S200 floor slabs to enclose the S300 volume.

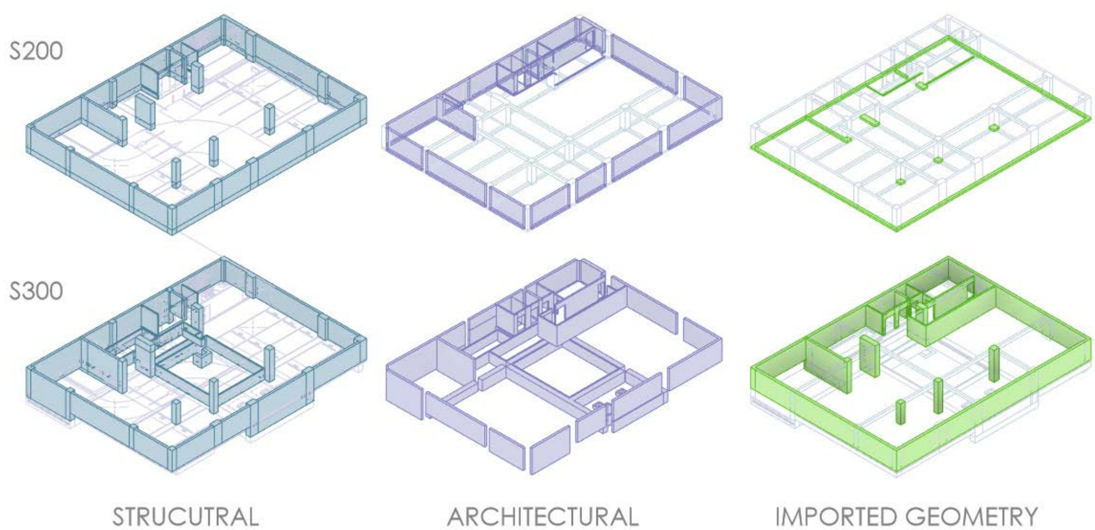


FIGURE 8. COPYING WALLS AND COLUMNS IN S300.

Adjusting the Model

Before we attempt to simplify the geometry of the newly imported elements, we must choose an appropiate type mapping for all walls, columns and floors imported so far. The desired dimensiones of elements is to be set before we can even begin to align them.

The following simplifications are made in order to facilitate our task:

- The thickness for all walls and floors is to be matched with the orignal type in the structural model.
- Dimensions for columns were obtanied using the Copy Original Type option for the Copy/Monitor Command.

<WALL SCHEDULE BY LEVEL>			
A	B	C	D
Type	Length	Base Constraint	Width
CFD CONCRETE 15 CM	2.82 m	S300	0.15 m
CFD CONCRETE 15 CM	2.56 m	S300	0.15 m
CFD GYPSUM 20 CM	11.10 m	S300	0.20 m
CFD GYPSUM 20 CM	2.84 m	S300	0.20 m
CFD GYPSUM 20 CM	7.85 m	S300	0.20 m
CFD GYPSUM 20 CM	4.61 m	S300	0.20 m
CFD GYPSUM 20 CM	4.23 m	S300	0.20 m
CFD GYPSUM 20 CM	2.03 m	S300	0.20 m
CFD CONCRETE 30 CM	4.79 m	S300	0.30 m
CFD CONCRETE 30 CM	1.82 m	S300	0.30 m
CFD CONCRETE 35 CM	22.56 m	S300	0.35 m
CFD CONCRETE 35 CM	30.31 m	S300	0.35 m
CFD CONCRETE 35 CM	30.40 m	S300	0.35 m
CFD CONCRETE 35 CM	22.13 m	S300	0.35 m

Visual Inspection of Imported Objects

There are a number of quality checks that we can perform on the Revit side to improve interoperability with Autodesk CFD. From the CFD's Documentation:

- Ensure walls, floors, and ceilings meet cleanly. Avoid small overlaps or gaps. This prevents a high concentration of very small elements where these features meet.
- To review details of the construction, use the Thin Lines view option.
- To ensure walls are cleanly lined up, use the Join and Align commands.
- Avoid small interferences and gaps between structural elements (such columns and beams) and neighboring geometry.

Please refer to Geometry Modeling Techniques for AEC Applications guide for more information on Model Complexity, Level of Detail and Model Integrity.

Check for Interferences

It is recommended that you run an interference check on the geometry to be exported before launching into CFD. You should adjust overlapping geometry before creating your Design Study. Figure 9 illustrates the simple but powerful process of selecting the elements to be exported and running an Interference Check on those elements.

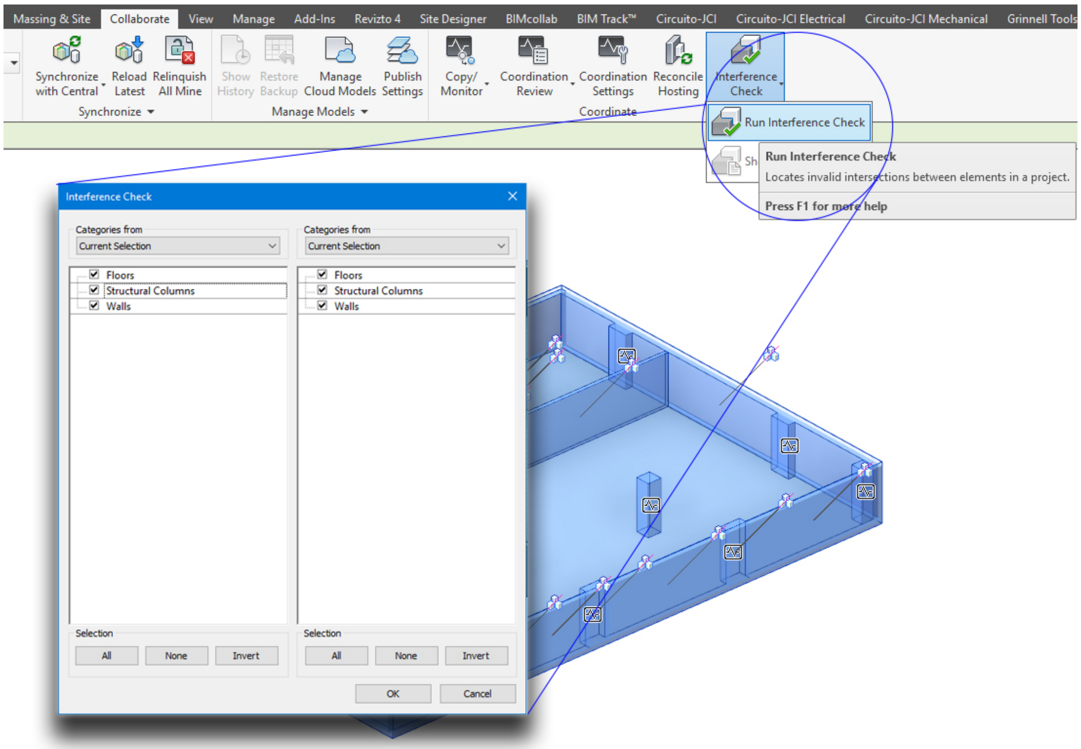


Figure 9. Use Revit's Interference Check before launching into CFD

Materials and Naming Conventions

The choice of which Family Type and Material to use ultimantely depends on the the analysis to be performed. Thermal applications require thermal properties, and so on. In this example we are using a Flow Only simulation to valide appropiate meshing, so properties other than dimensions are irrelevant to the results.

Export building geometry from Autodesk Revit directly into Autodesk CFD

So far we have created our first level and simplified its geometry using our best judgement. We checked for interferences and fixed all of them. **Figure 10** shows the original geometry coming in from both the architectural and structural models and the simplified version after applying several adjustments to the imported geometry. All elements have been aligned and joined to avoid interreferences and small gaps.

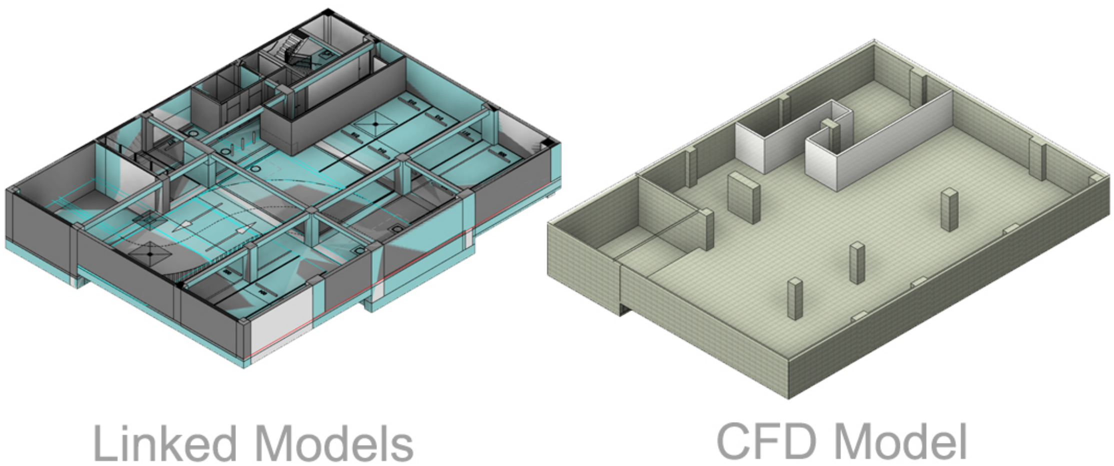


Figure 10. Imported Geometry vs Exported Geometry

Understanding the Flow Volume

For internal ventilation or forced ventilation AEC applications like the one featured in this class, **you generally want to make sure you have an air-tight volume** or something that is very close to one. This is also known as the *flow volume*. For the basement shown in Figure 10, the flow volume is not really enclosed because of the vertical shaft shown on the left side. You can either close the gap in the upper level floors slab directly in Revit or use CFD's geometry tools to fill the gap and enclose the internal volume. The later option provides useful when there aren't many void to fill in the model. For one vertical shaft, it is relatively easy to fill the gaps in CFD. For a Building with many openings it is often convenient to fully enclose the flow volume directly in Revit.

The Model Assessment Tool (MAT)

There are times when even though you managed to import your model, CFD won't be able to turn it into a mesh or the automatic meshing tools produce a mesh that doesn't fit analysis requirements**. Figure 11** shows the error message displayed when you try to launch *from a model* that has too many inconsistencies. Similarly, **Figure 12** shows an error message associated with silver surfaces or tiny edges in the geometry. The model assessment toolkit is an Add-In for

Autodesk CFD that helps the user identify potential meshing issues before actually creating the Design Study.

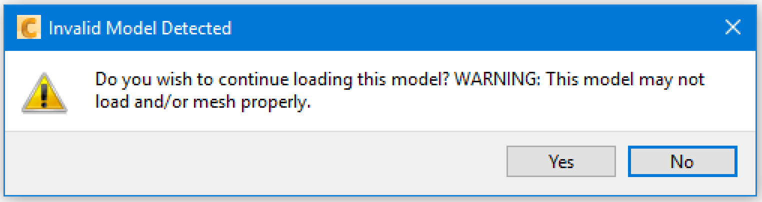


Figure 11. Invalid Model Detected

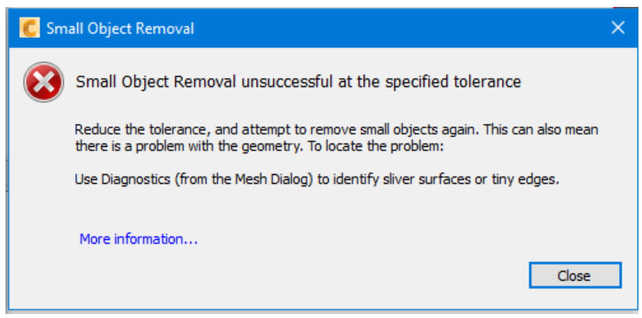


Figure 12. Small Object Removal Failure

By using the Model Assessment Toolkit you can inspect the model for the following known erroros:

Short Edges

Small edges can introduced errors when meshing by requiring an excessively fine mesh in those areas. Fine meshing requires more computing, so it needs to be avoided when the elements are so small that it is not expected they impact the final solution. Small edges can also lead to “discontinuous behavior”. The latest behavior should be avoided at all costs as it can cause inaccuracies in the solution. **Figure 13** shows our 1st Iteration on the MAT using the Edge Lengths Tools to identify potential issues. We see that the MAT uses the shortest edge as the filter for edge length by default. Thus we get 50 potential issues with a length of 200 mm. This happens to be intentional and it corresponds to the thinnest elements in the model (walls, slabs). If we set the filter to look for edges shorter than 199 mm, for example, we get ZERO potential issues in the model due to small edges. In general, **we should try to avoid generating edges that are shorter than our thinnest element**. This also means that we should not look for edges larger than our model’s shortest parts, as this will result in very large number of results that are not relevant.

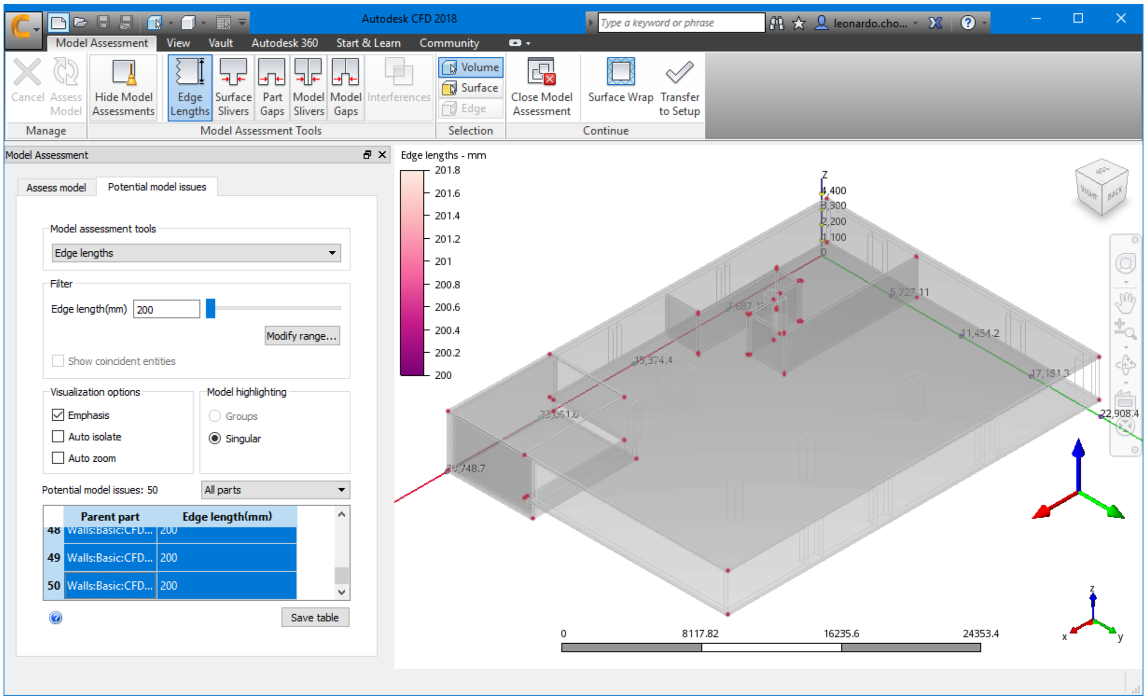


Figure 13. Identify Short Edges using the MAT (200 mm length)

Part Gaps

When you join two elements that intersect, Revit identifies the overlapping solids and generates a cut (gap) in the one of the elements (depending on the join order). Even if the elements look continuous in Revit there might be potential issues associated with the cuts on those elements. Part gaps can cause discontinuity issues and mesh scaling issues and they generally need to be fixed for adequate meshing. **Figure 14** shows the results found when looking for gaps smaller than 470 mm in the model. We notice that the walls surrounding the elevator shaft have been cut in order to be join with the S300 floor slab.

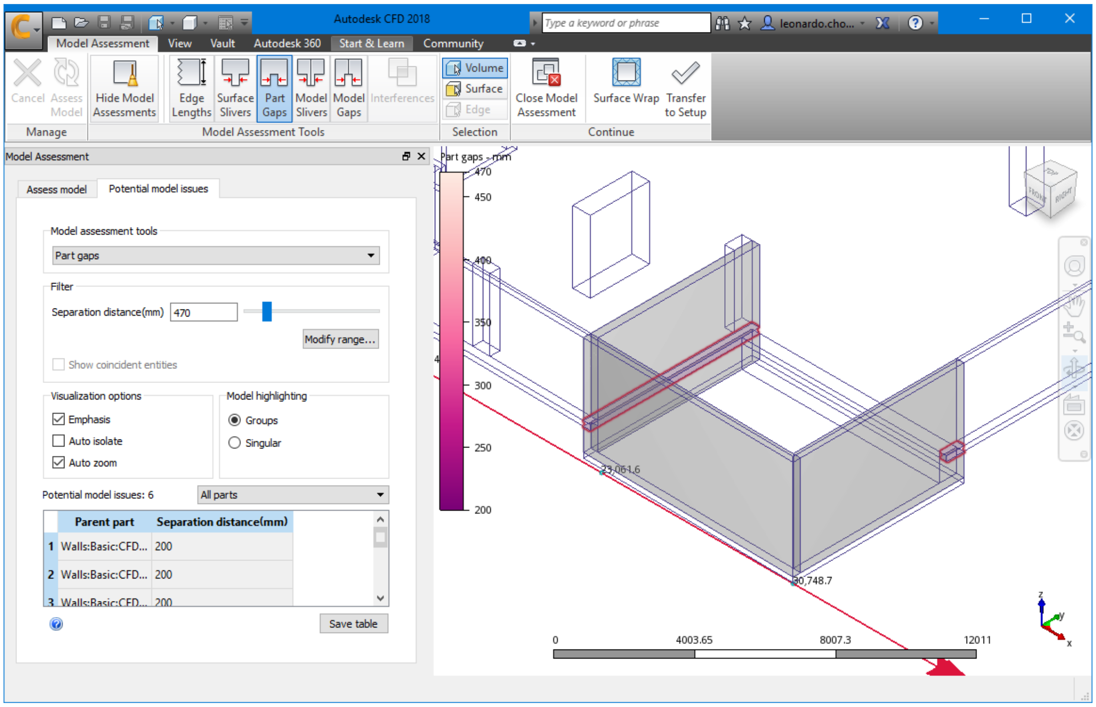


Figure 14. Identify Part Gaps using the MAT (1st Iteration).

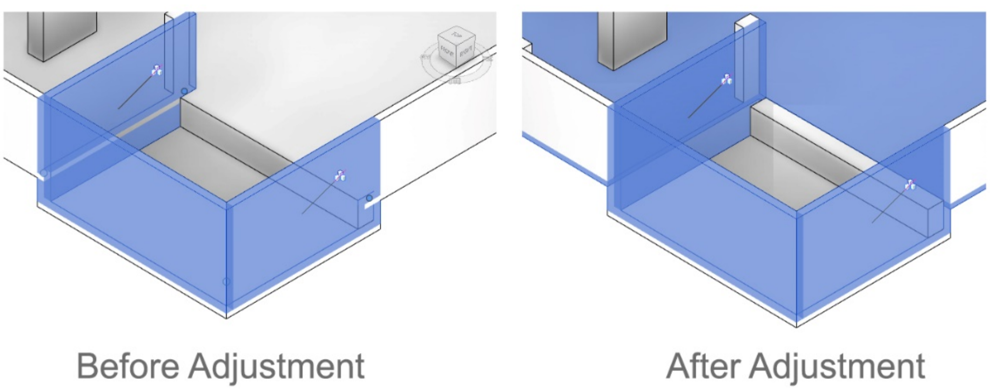


Figure 15. Adjusting the Floor Slab to Avoid cutting walls

The problem can be fixed by either splitting the walls in Revit or adjusting the slab edge. In this case it makes sense that we adjust the slab edge as these are continuous retaining walls and are not actually attached to the floor slab. **Figure 15** shows the adjustments made to the floor slab in level S300 so that it doesn’t cut the walls of the elevator shaft**. Launching the MAT after this modification resulted in ZERO part gaps.**

Model Gaps

A model gap is a small separation between two adjacent model parts. It is the opposite of an interference, and it has a similar effect on meshing and performance: it causes discontinuity. Model gaps are also a major enemy of having properly enclosed volumes. This is a serious issue as it will keep you from being able to run internal ventilation simulations and will introduce inaccuracies near the building when using external volumes as the flow volume. Model gaps are really hard to locate by simple visual inspection. **Figure 16** shows the results of launching the MAT's model gap assessment on the S300 level in our example. There are 36 potential model issues indicating a minimum separation of 0.00094944 between elements that are touching in Revit. Even though these elements are referenced to the same plane, CFD ***will** have* trouble creating an internal volume off this geometry because these small inaccuracies.

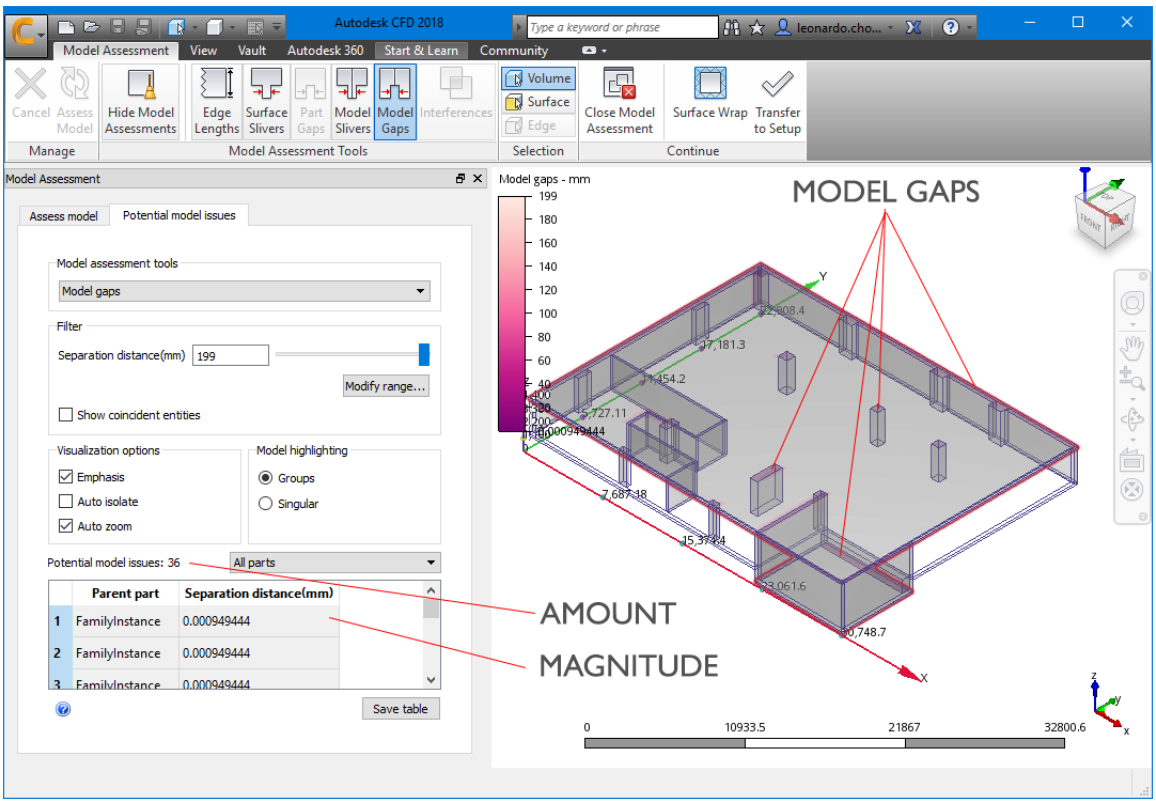
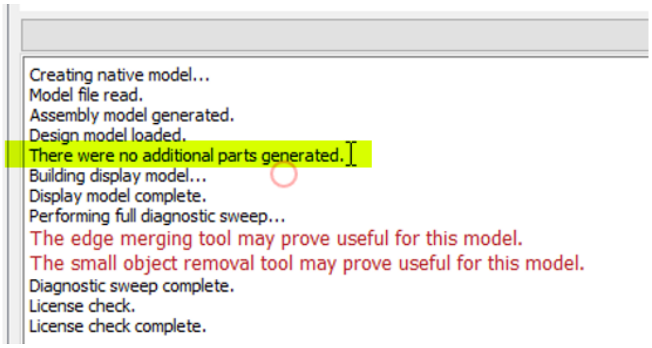


Figure 16. Detecting Model Gaps in the MAT (1st Iteration)

If we launched CFD at this point, we would get a message that reads "*There were no additional parts generated*".



This means that even though we enclosed the volume in Revit, CFD couldn't create an internal volume from the imported geometry. Internal volumes need be properly enclosed for us to be able to assign boundary conditions on internal ventilations examples.

To fix the model gaps, we can create an intentional overlap between the columns, walls and floors highlighted in **Figure 16.** We then use Revit's join command to cut the overlapping geometry and create a monolithic volume. **Figure 17** shows the S300 model after all basement walls and columns have been assigned a Top Level Reference equal to that of the S200 floor slab. We use the Join Command on the external wall. The internal walls are overlapping but they wont be joined in Revit because they have different materials assigned. This is OK and it wont be a problem.

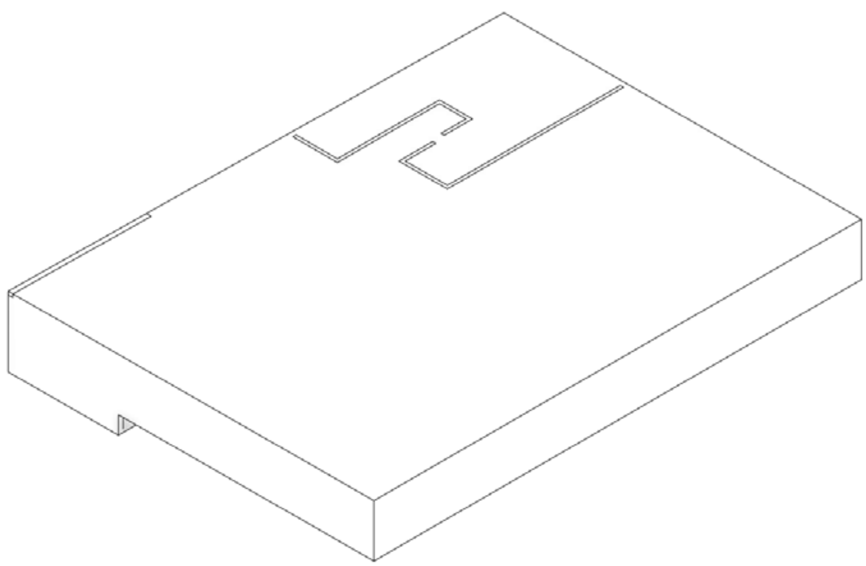


Figure 17. Intentionally overlapped geometry before joining

Figure 18 shows the 2nd iteration on removing model gaps. In this case we extend the wall in the negative Z direction to fully overlap with the elevator's floor slab 1100 mm below S300. **Re-launching the MAT after applying this fix and joining the two**

elements in Revit results in no gaps between them. We finish up by performing a a final check with a big tolerance (in order to detect bigger gaps) and realize that some columns that are next to walls also have gaps. We apply similar overlapping, aligning and joining techniques to finally get no model gaps in our 3rd iteration using the MAT (not shown).

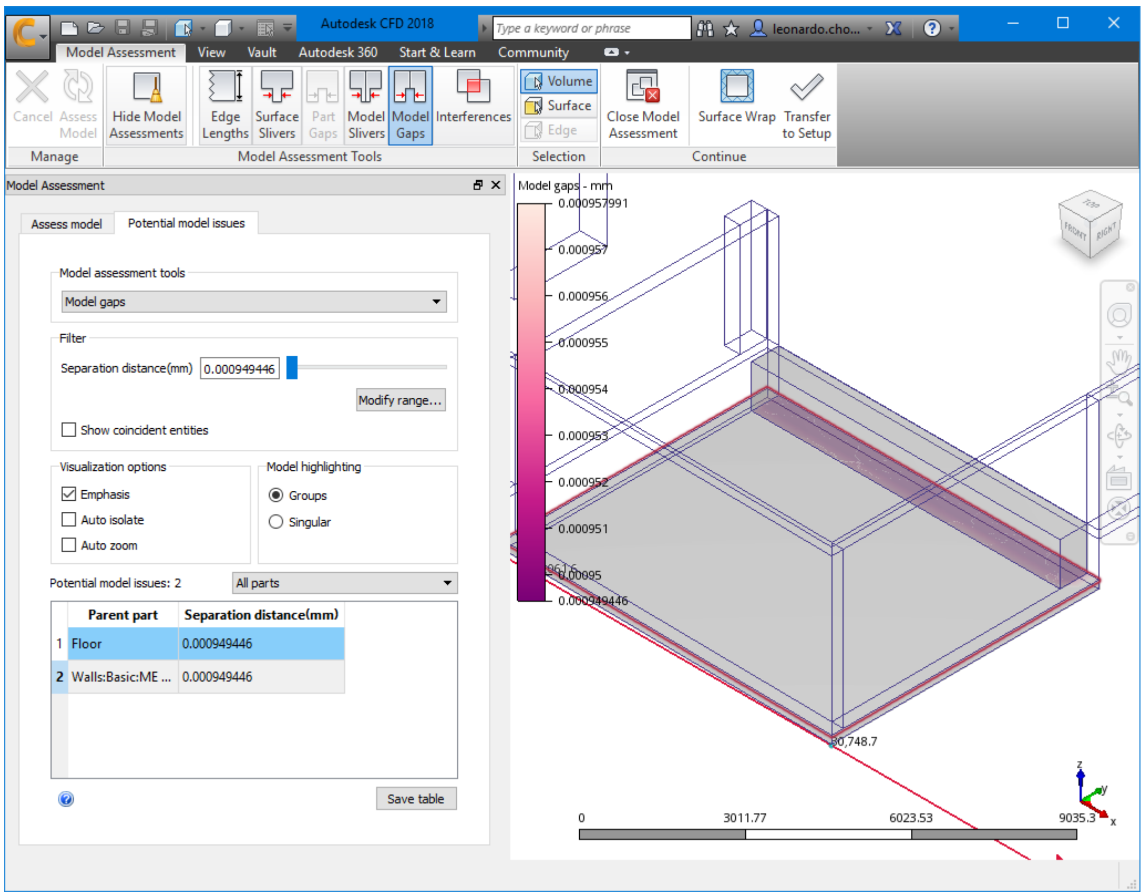


Figure 18. Detecting Model Gaps in the MAT (2nd Iteration)

If you enclosed your volume in Revit and checked for model gaps, it is most likely that CFD will automatically create an internal volume for you when bring the geometry in. This is a big time saver as you don't want to have to create it every time you update your design geometry to evaluate different options.

Interferences

Interferences between model parts can cause unexpected behavior when transferring geometry into CFD. Despite running Revit's built-in interference check, we find that the MAT detected 23 interferences between model parts. The highlighted parts in **Figure 19** are overlapping by a less than 0.001 mm. By inspecting the results, we see that most columns are overlapping with their base level's floor slab. However, they don't overlap with the top level's floor slab. We can go back to Revit and check on those columns to further investigate the problem.

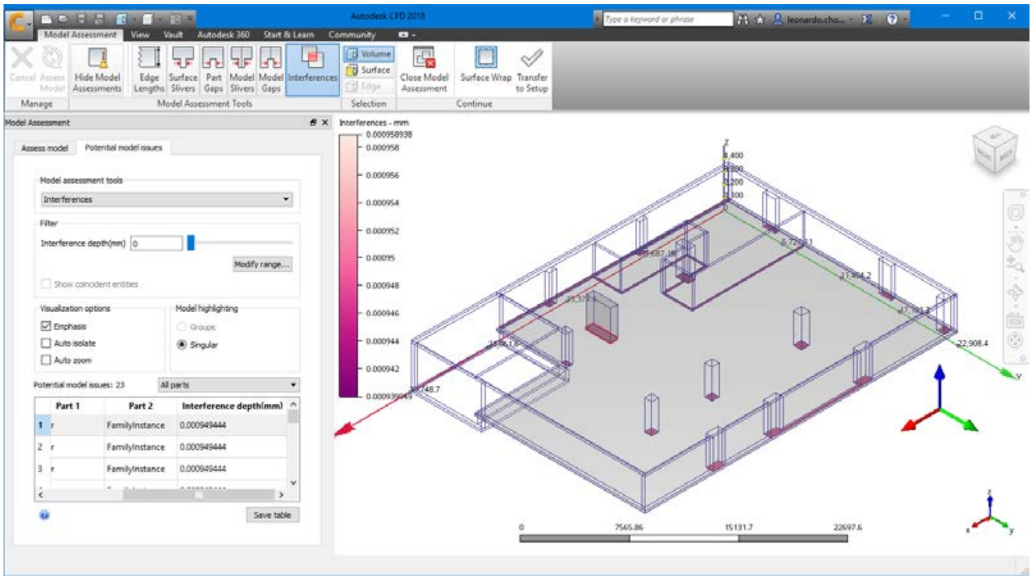


Figure 19. Check for Interferences with the MAT (First Iteration)

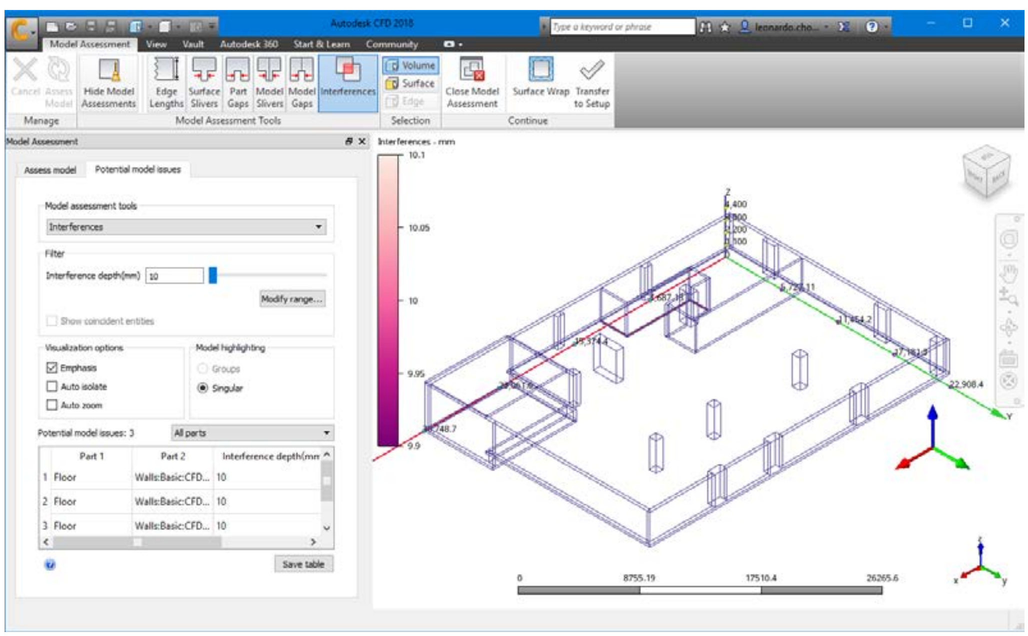


Figure 20. Check for Interferences with the MAT (First Iteration)

We also notice that those 3 interferences correspond to 3 walls that overlap by 1 cm with their base floor slab. We actually caused this in our last iteration and we can easily fix it by attaching those walls to their base level or by manually joining them. The final iteration resulted in zero interferences detected by the MAT (not shown).

Direct Launch vs MAT Transfer

After using the MAT to assess the model against known geometry issues, we are finally ready to launch our Revit model into CFD. We can do this by using the “Transfer to Set Up” button shown on the MAT’s interface (Figure 20) or by using the Launch Active Model Tool directly from Revit. Figure 21 illustrate the two different ways of Launching CFD from Revit. So far we have used the MAT repeatedly on our S300 basement so can use the second method now to Create a new Design Study. The steps are described in the following section.

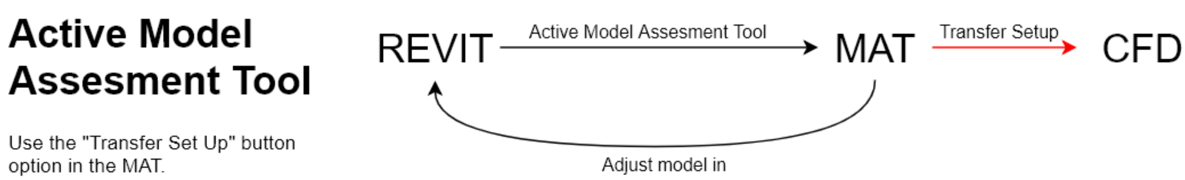


Figure 21a. Active Model Assessment Tool

Figure 22b. Launch Active Model Tool

Creating a new Design Study

The process of creating a new Design Study is as follows:

1. In Revit, make sure you are located inside a 3D View that shows only the elements you want to export to CFD.

2. Verify that you have Autodesk CFD installed and that you installed the CFD Exporters for Autodesk Revit.

3. In the Add-Ins Ribbon Tab, locate the Autodesk CFD 2018 group and click on the Launch Active Model.

4. When you first Launch CFD from either Revit or the MAT, you will be asked if you creating a **new Design Study** or update an existing one. In this case, we want to create a new one and we will update it in the future. Please refer to CFD's documentation for more information on Design Studies and Design Scenarios.

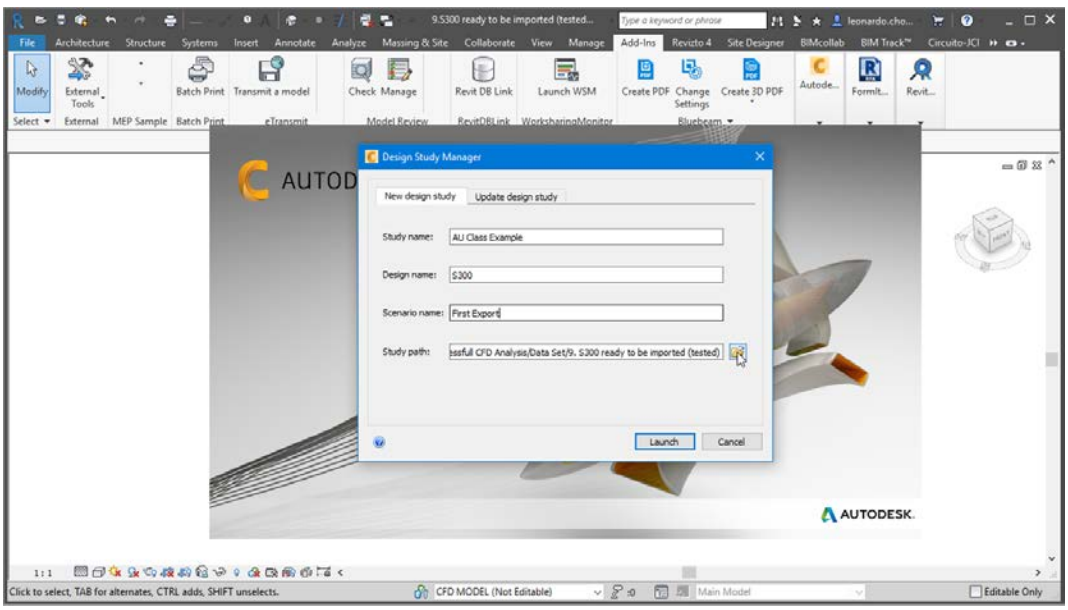


Figure 23. Design Study Manager - New Study

5. Click **Launch** to open an instance of Autodesk CFD with your Revit model in it.

6. Unless you get any fatal errors, CFD will open and the Geometry Tools dialog will automatically open. In most cases your model will benefit from using geometry tools, but we will close it for now and come back it later.

7. Take a loot the imported geometry and check the Message Window to see the results of your import. The messages in red are important for obvious reasons,

but we are also looking to see if any additional parts were generated to represent internal flow volumes. Figure 24 shows the CFD interface right after a new Design Study has been created. We confirmed that "***There were two additional parts generated***" and that the model is complete. The message window also advises to use the edge merging tool on this model. We will do so in the following sections.

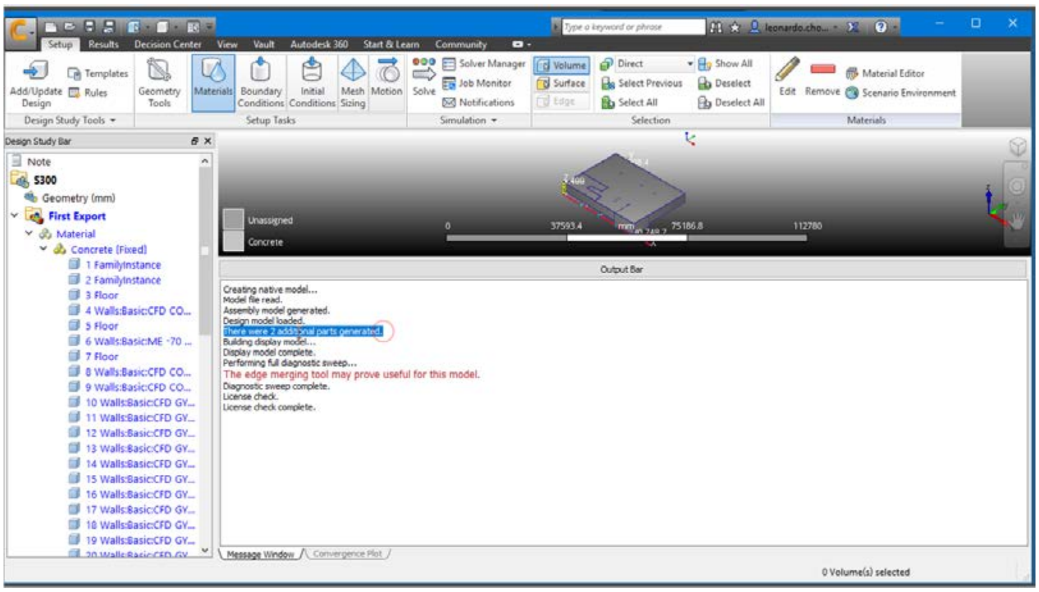


Figure 24. New Design Study After Creation

That's it! We have successfully imported geometry from Revit into CFD. We can now use CFD's tools to simplify the imported geometry to make sure it meshes properly.

Updating an Existing Study (iteration)

Before simplifying the geometry using the Geometry Tools, let us add some more levels to our design study using the insight we gained by iterating so many times on the S300 level. At the end of each level (or every other level) it is highly recommended that you try to mesh the geometry in CFD by assigning materials and running the solver with zero iterations. This is a health check that you want to perform regularly to avoid modeling problematic elements and having trouble identifying them in the future. Of course, you can always use the MAT to check on the newly added geometry and avoid having to launch CFD several times. In this sense, there are two levels of iteration: one is the described in Figure 21a. Active Model Assessment Tool where we go back and forth between Revit and the MAT to reduce the chance of getting errors in CFD. The other one consists of Launching CFD and building a test mesh to validate chunks of newly added geometry until our Design Study is complete. The procedure for creating a test mesh is outline in a different section of this document. **Please perform that procedure before using this section of the document.**

Second Iteration

For our second iteration we added the walls, columns and floors slabs in S200 and S100. We also added the gate for the car elevator shaft at each level. We applied the same modeling techniques we discovered in the first iteration and probably did it in one sixteenth of the time. a technical stand point, this is no longer your first CFD Project, right?

Figure 25 shows the geometry exported on the 2nd iteration before fixing interferences in Revit. There are missing elements on Figure 25 and that is OK. Those elements will be added once we have a working mesh for the main components of the building envelope. That being said, we will call this Design Iteration 2 and it will be an update to the Design Study we created in the previous section.

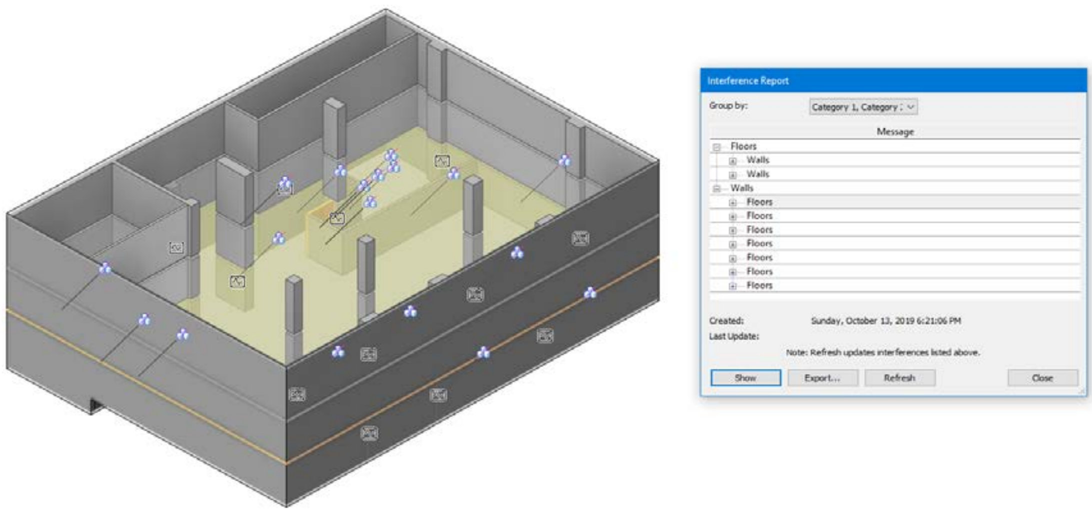


Figure 25. 2nd Iteration Revit Export

The process for updating an existing Design Study is a follows:

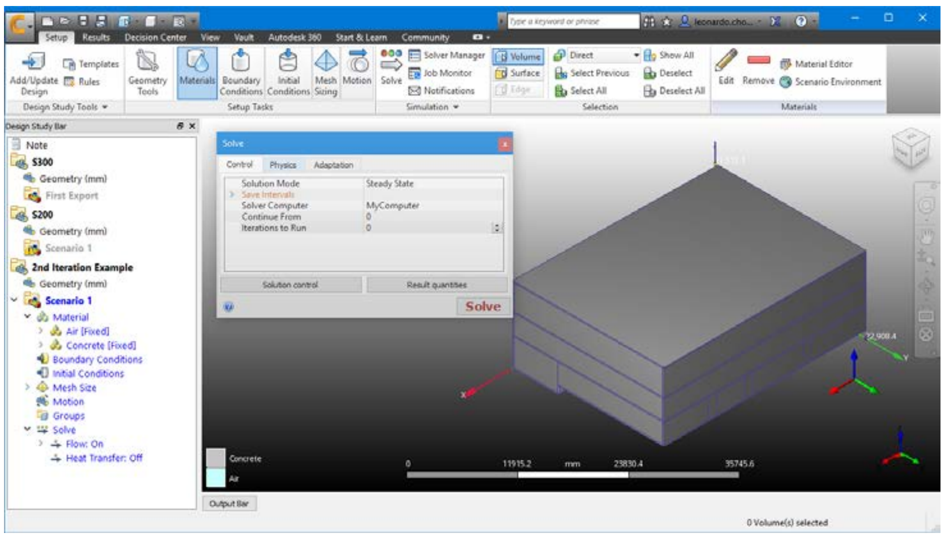
1. Transfer your set up or launch directly in the same way you did when you first created the Design Study.

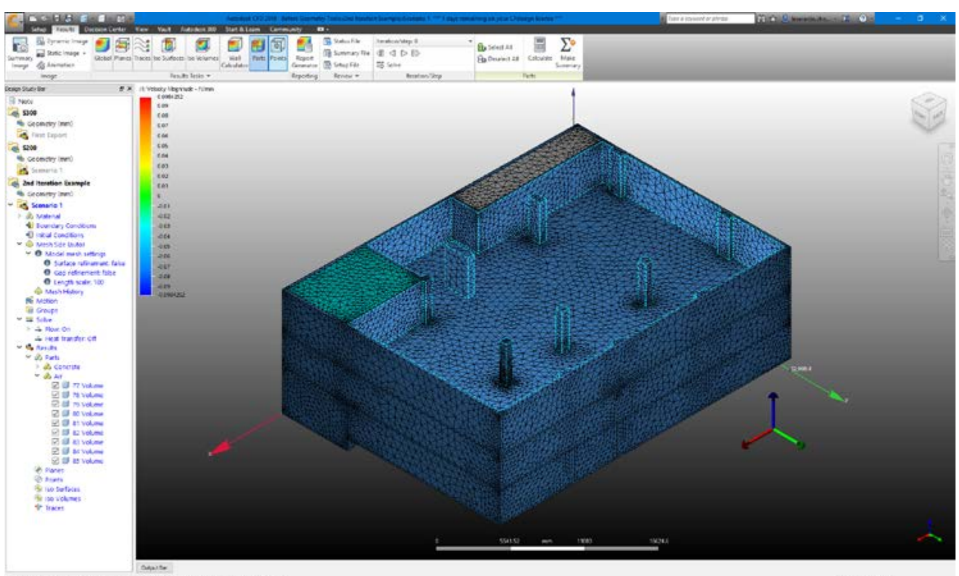
2. The Design Study Manager dialog shows up. Choose an existing Design Study and one of the following options:

    a. If you click on an Existing Design Study, it will only let **you ADD a Design to the Study**.
    b. If you click on an Existing Study, it will give you the option to **Update the Design.**

3. Click on Add Design to Study.

4. Once in CFD, you will see 2nd Iteration Example in the Design Study Bar. This allows us to keep a record of all our models iterations so that we can create test meshes for each one of them and go back to the last working one in case that newly added geometry isn't properly modeled in Autodesk Revit.





Use CFD's geometry/meshing tools to simplify the imported geometry

Geometry Tools

Before creating a mesh in CFD is often convenient (sometime necessary) to simplify the geometry even further using CFD's built-in Geometry Tools. You can get rid of small edges, small objects and create additional volumes in order to preprare your model for meshing. When you first launch a model from Revit into CFD you will be prompted to use Geometry Tools to simplify it's geometry. For learning purpose we have been omitting when creating and/or updating the Design Study. If you click on Setup > Setup Taks > Geometry Tools you can access the same Dialog at any time after the creating of your Design Study. Please note that using the geometry tools after meshing will result in the deletion of current mesh and required the user to create a new one. This is OK. However, it is more convenient to use Geometry Tools right after the Design Study has been created to avoid having to perform mesh operations repeatedly.

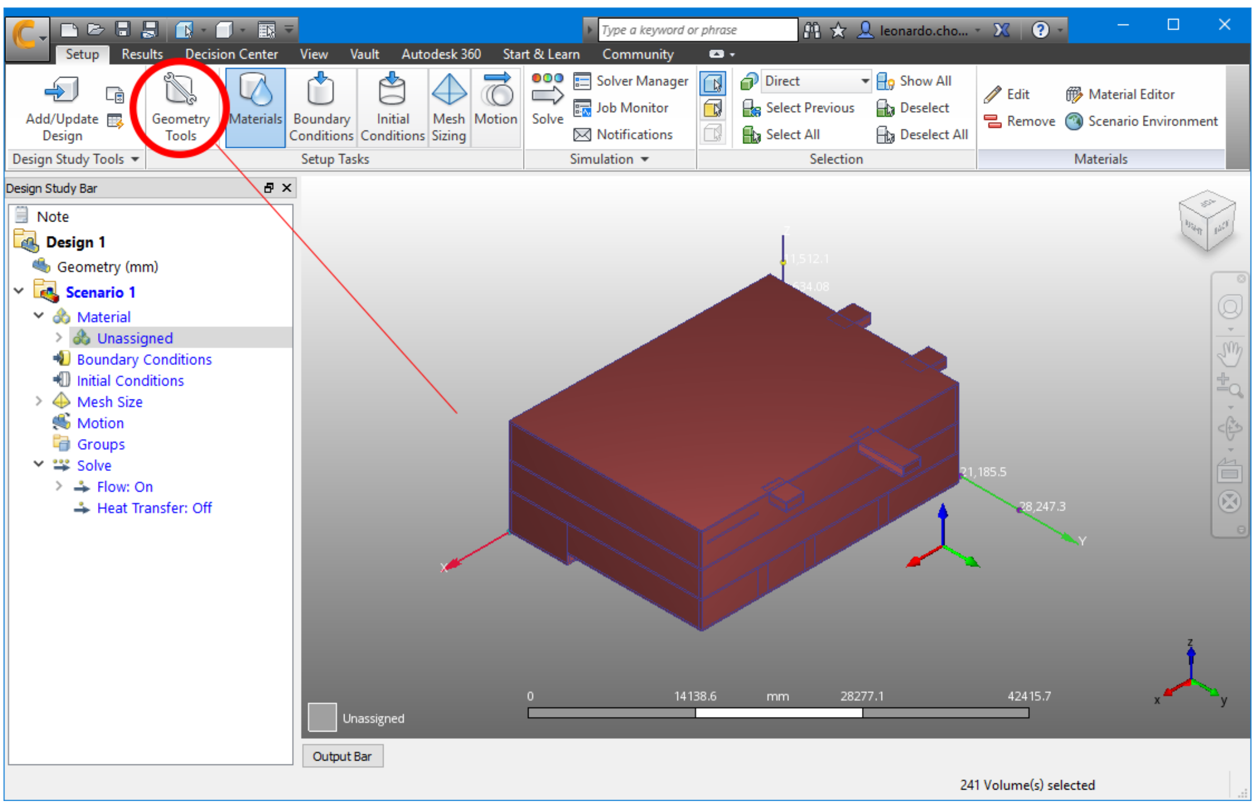


Figure 26. Accessing the Geometry Tools Dialog

Merging edges

Use the Edge Merging tab to get rid of small edges that meet an angle less than a specified tolerance. This helps to reduce the overall mesh density and results in faster analysis times. **Figure 27** shows the Edge Merging Tool in use. The user can adjust the slider to select a threshold angle. CFD calculates the number of edges and optionally highlights the edges to be removed in the model.

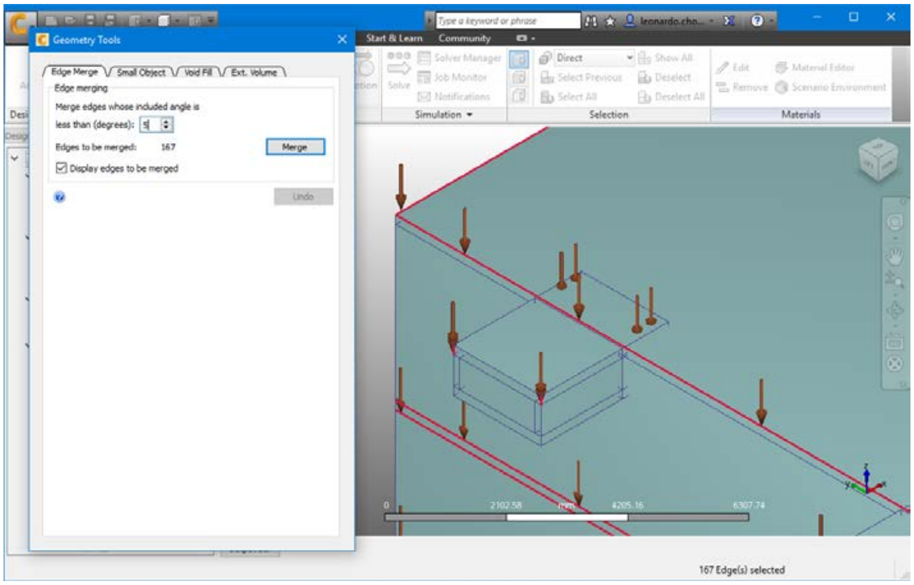


Figure 27. Edge Merging Tool

Removing small objects

A very similar concept is applied when using the Small Object Removal tool. Indeed, the small edge removal tool is to be used **prior** to removing small objects.

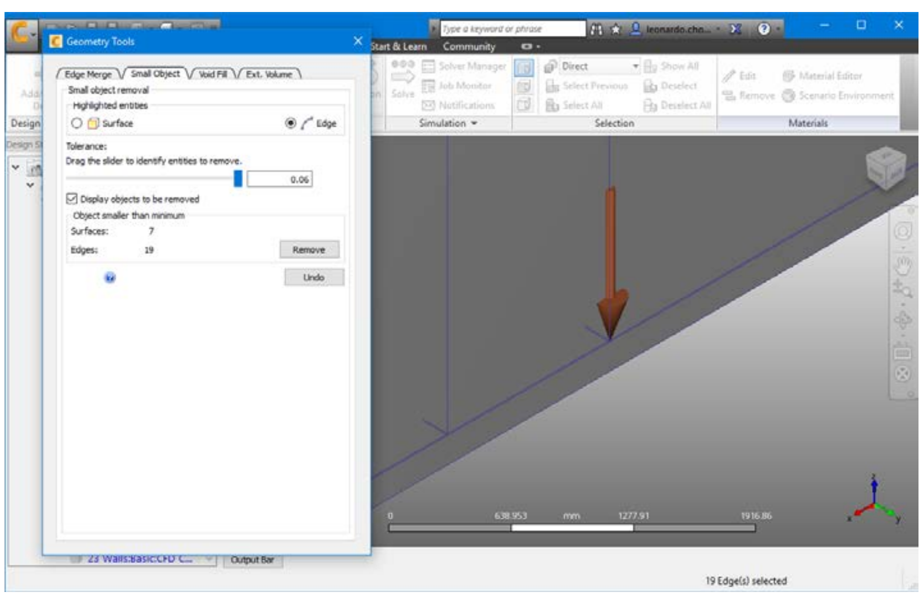


Figure 28. Small Object Removal Tool

Filling Voids

Following our previous discussion on internal volumes, the Void Fill tool provides useful when your design's geometry doesn't actually contain an enclosed flow volume. The fill void tool allows the user to select co-planar edges that might define the interfacing surface of your internal volume with the exterior. Figure XX shows a selection of four edges to be used for building a planar surface, which in time will generate a new internal volume that exists only in your design study. You can confirm these actions in the Message Window after you use the tool. There should be a message that reads "There were 1 additional parts generated". This is the same message you would get if you enclosed the volume in Autodesk Revit prior to importing the geometry.

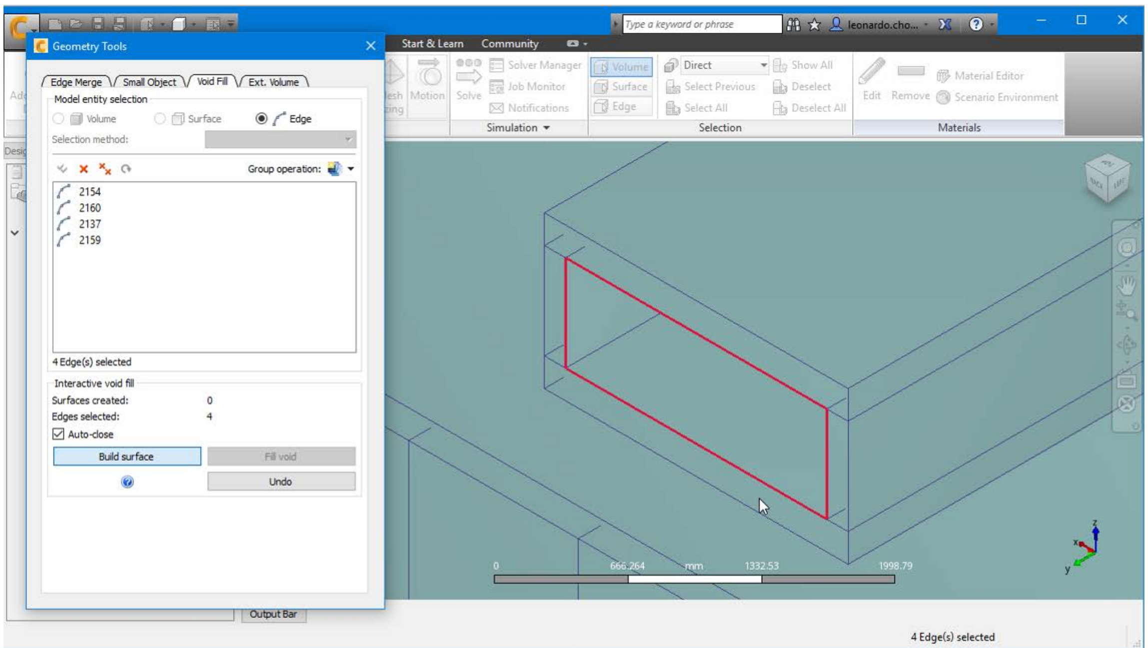


Figure 29. Void Fill Tool

Creating External Volumes

The external volume tool is used when you need to define an external domain for your simulation. This is normally the case when you use CFD to analyze natural ventilation scenarios such as the flow of air coming from exterior of the building. It is also necessary in wind loading analysis. **Figure 30** shows an example an external volume being created for our basement. In our example, we are only interested in the internal air movement as the basement is fully enclosed by gates. The external volume created here is for illustration purposes and it is not required for our analysis.

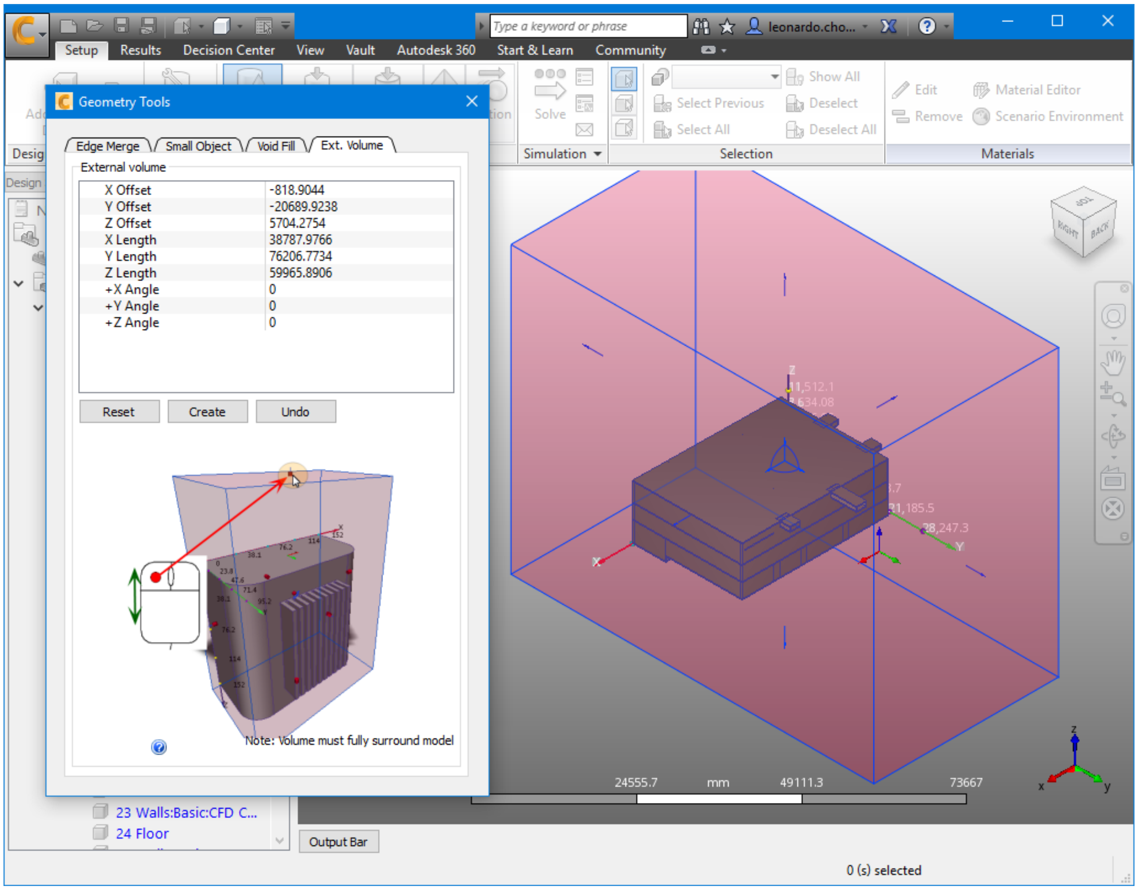


Figure 30. External Volume Tool

Pro tip: Use geometry tools only once you have completed your design and before you go into analysis.

Create rules to assign materials and run a simple analysis to validate meshing.

Assigning materials

To assign materials to the imported geometry, Click on Setup > Setup Tasks > Materials. Then select one or many parts in the model and Click on the Edit button. You can also select elements directly from the Design Study Bar shown on the left of the user interface in **Figure 31**.

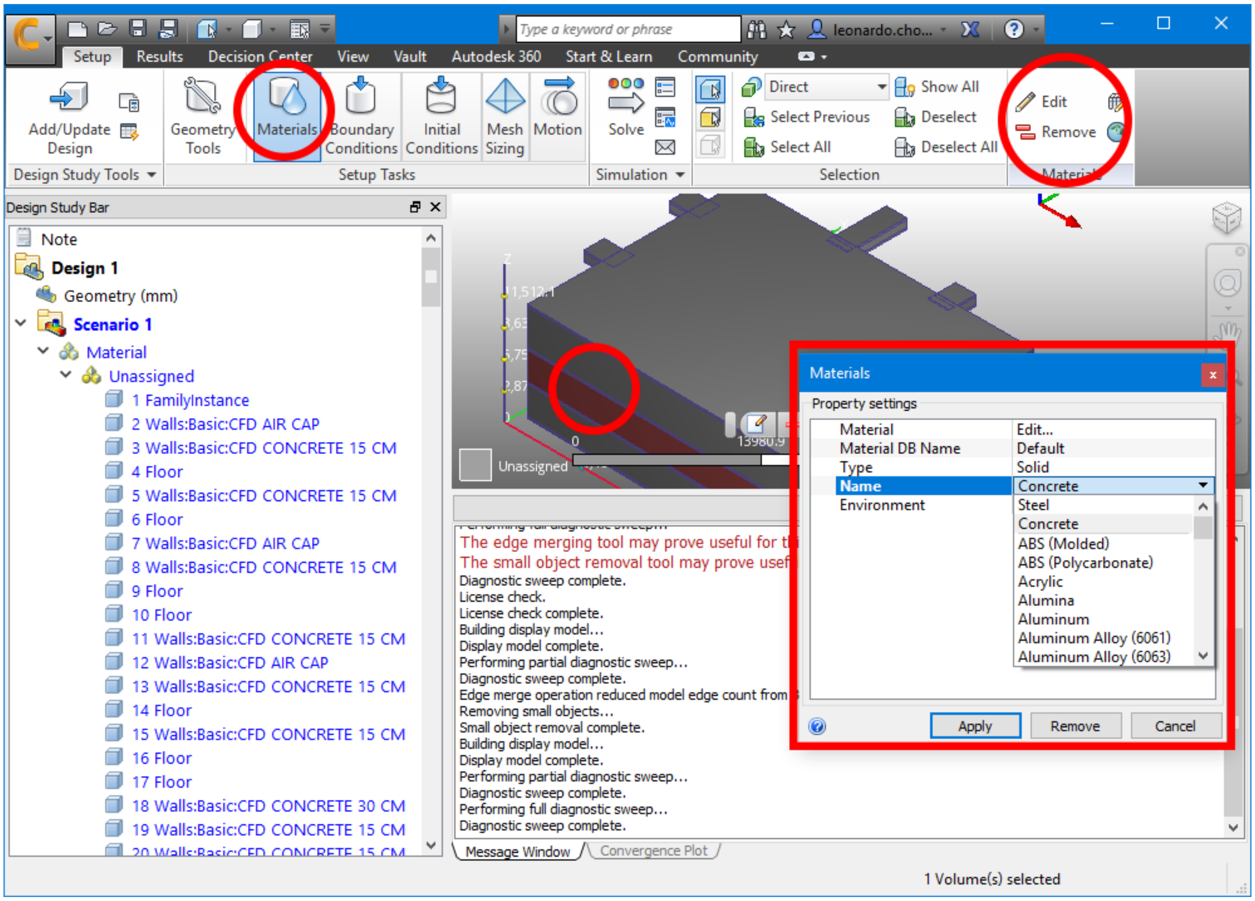


Figure 31. Assigning Materials

Rule Creation

Revit Categories, Family Names and Type Names are transferred into CFD as part names to improve the associativity between analysis models and BIM models. In Figure 31 we notice that CFD detected the Floors category and the Walls Category. It didn't detect the structural Columns (see line 1, 1 Family Instance) and it didn't detect the Floor's family or type name. You can take advantage of associativity by giving appropriate names to family types so that you can later create rules that automatically assign materials base on the contents of the part name. You can create rules either manually or by using an existing setting that needs to be replicated. In our example we will use an existing material assigned to a part to create a rule that assigns the same material to parts with similar names. You can refer to the Rules section of the documentation for detailed instructions on how to manually create rules. The later is useful when creating templates and setting standards for your company or practice.

To create a new Rule from our previously assigned material, we can select the part from the Design Study Bar, right click it and associate this part name to a material as shown in **Figure 32.**

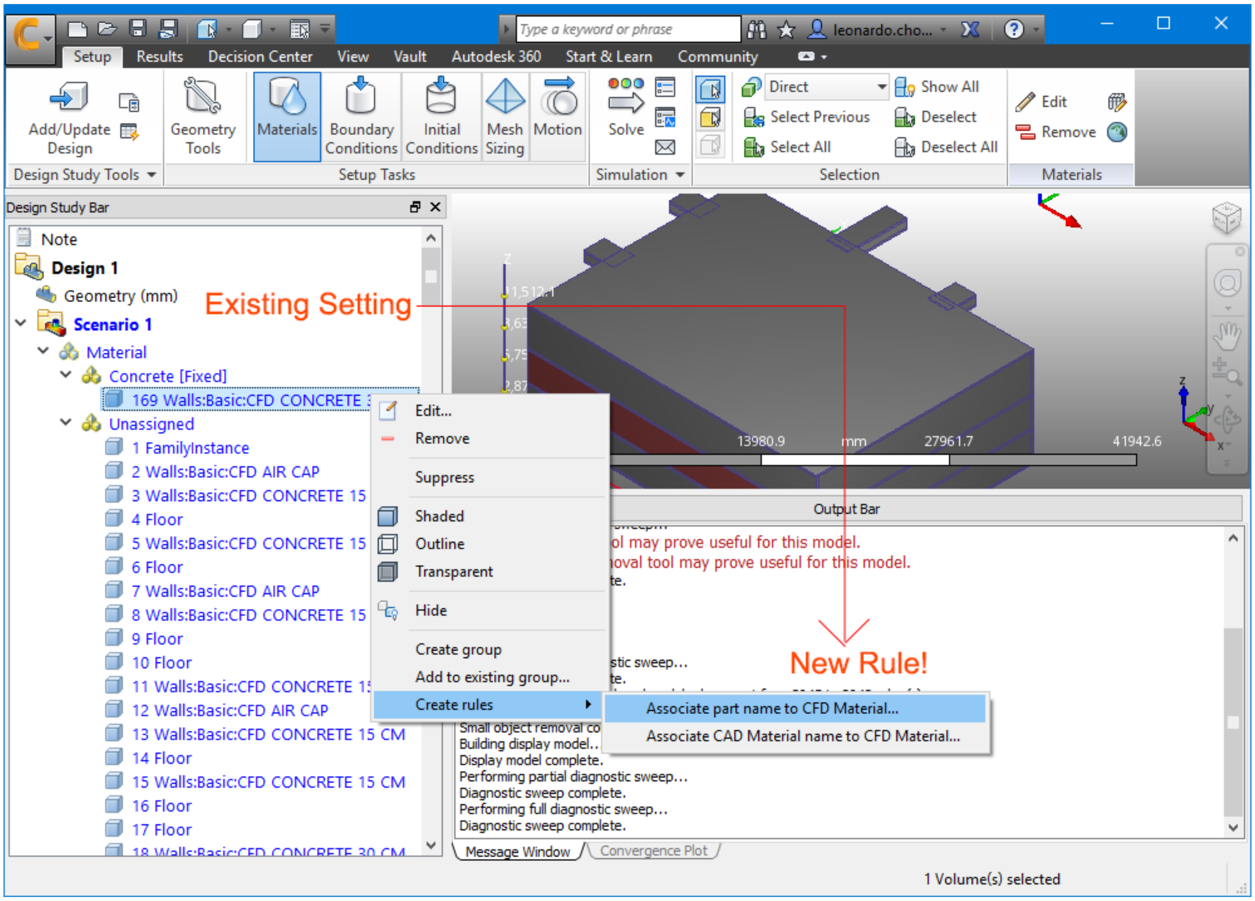


Figure 32. Create Rules from Existing Settings

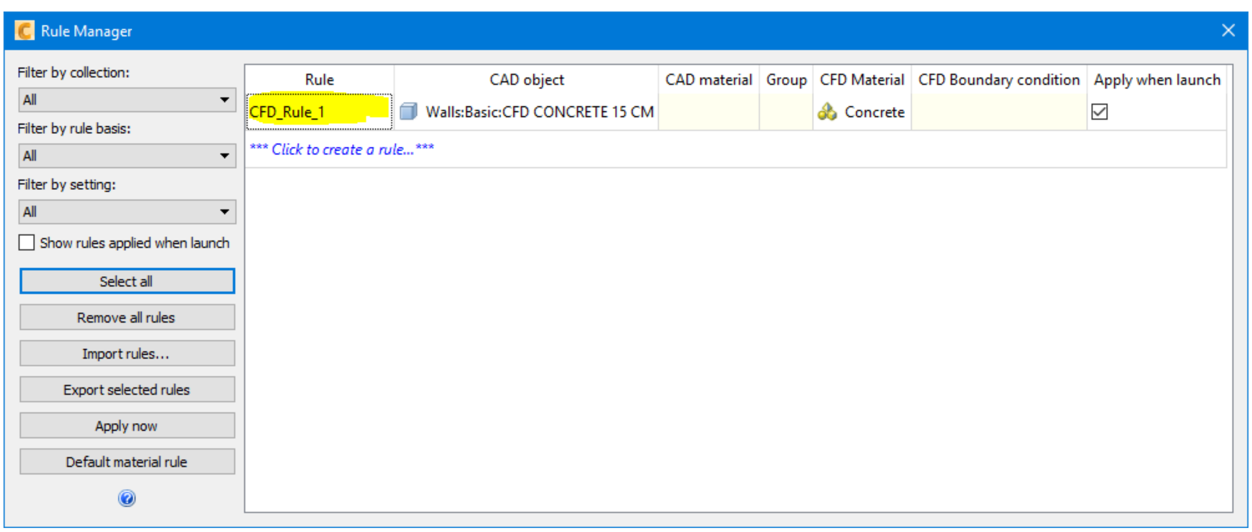


Figure 33. Rule Manager Dialog

Once in the Rule Manager dialog, click on the newly created rule (highlighted in yellow) to change in its name and verify its criteria. By clicking on the Apply Now button we can automatically assign the Concrete Material to All Elements with the same part name. Rules don't necessarily have to include the full part name. This allows the user to create a more inclusive rule from a previous one. In **Figure 34** we have modified the name and criteria for the concrete walls rules to allow for all part names with the word "CONCRETE" in them to be included by the rule. We also clicked on the ***Create a rule*** option to bring out the Rule Creation dialog. In here, we are created a rule name Floor that assigns a concrete material to all parts with that same name.

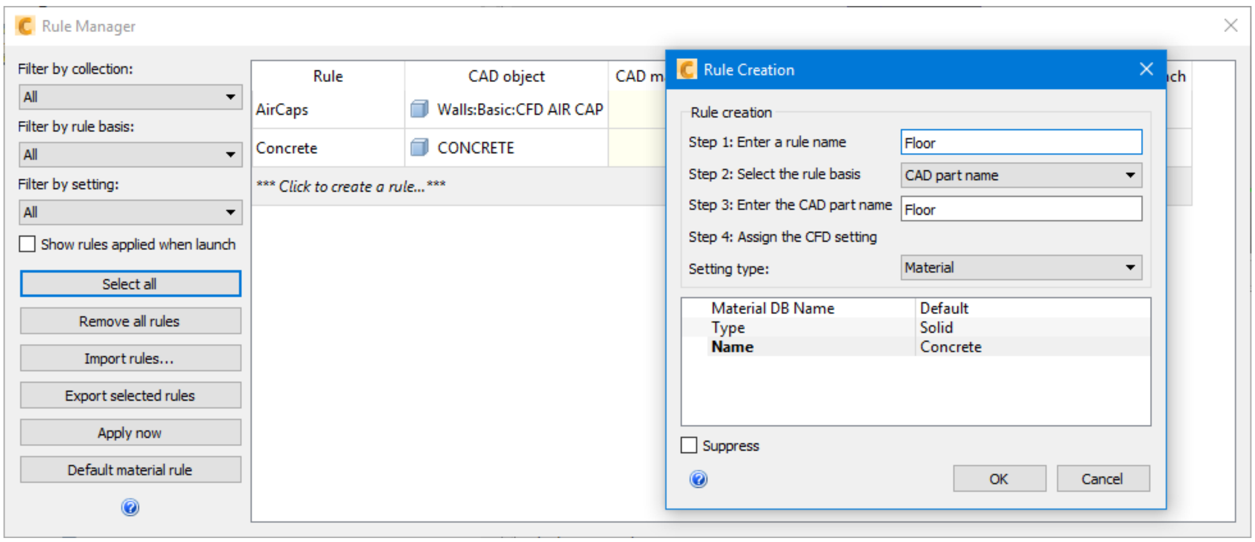


Figure 34. Creating New Rules

Carrying out this procedure in the initial iterations of the Design Study can save you a lot of time assigning both materials and boundary conditions during design iterations. Of course, a good naming convention and good modeling practices will be of great help in this effort. Also, if you create rules for one Design Study you have the option of running them automatically in all Design Studies upon Launching CFD. For more detailed discussion on Design Study Automation and creating other types of rules please refer to the Official Product Documentation.

Run a simple analysis to validate meshing

So far we have a model with parts and materials assigned to them. At this point we can create a test mesh to check if the imported geometry is definitely ready for analysis. For testing purposes we can use the autosize meshing tools and refine the mesh later. You can navigate to the Mesh Sizing Set Up Task and Click on the Autosize icon. This is the easiest way to get you started and the process is depicted in **Figure 35.** By clicking on the Autosize button the user can see preview of the nodes to be located on the edges of the geometry. By exploring this view the user can get a good sense of the size of the mesh and how the size of the parts affects the mesh. In this step, the user can also make adjustments before generating the mesh.

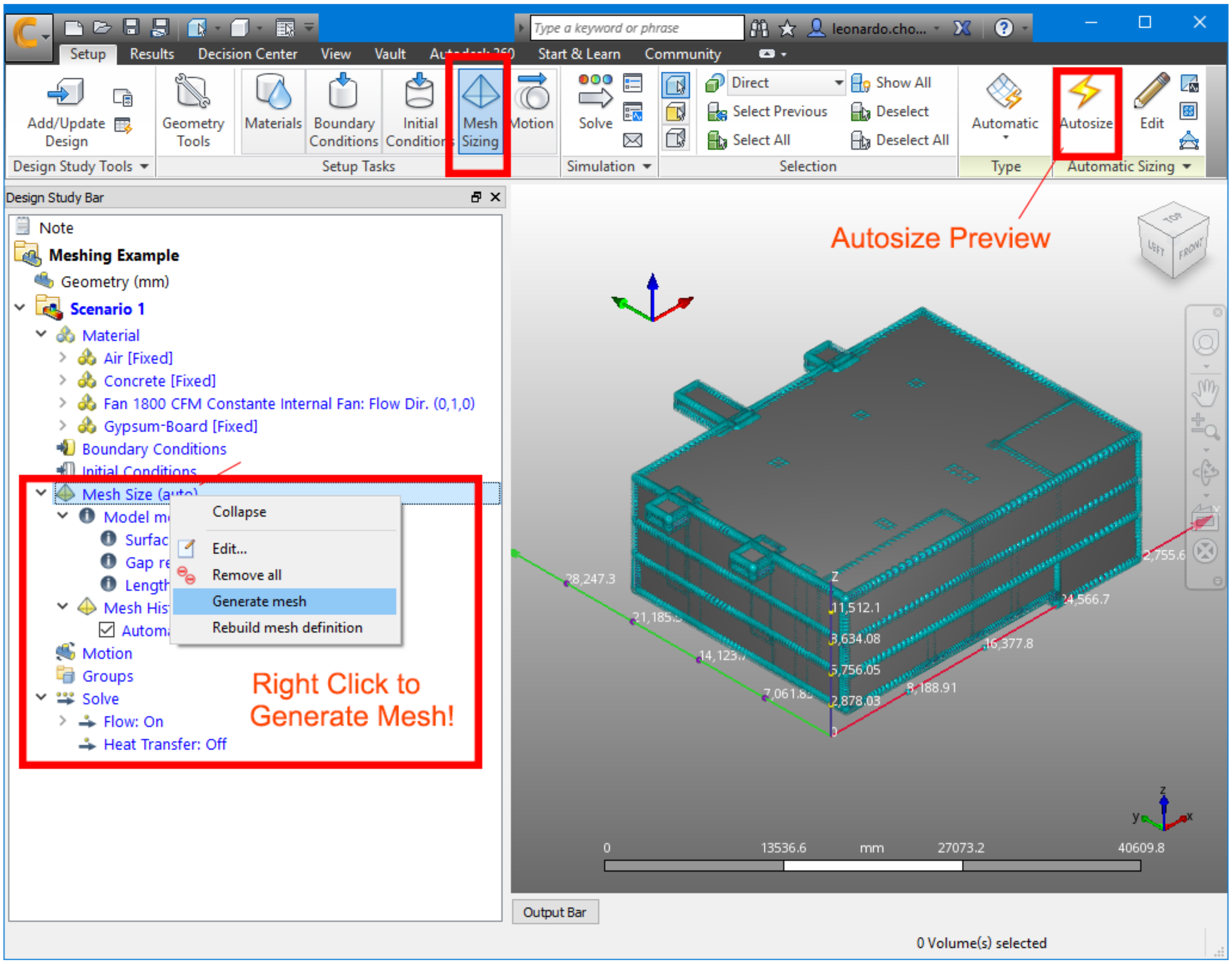


Figure 35. Mesh Auto Sizing

To actually build the mesh, the user has the choice of either running a simple analysis with 0 iterations or generating the mesh before actually using the solver. The later is useful for testing purposes because you don't have to worry about boundary conditions or other aspects of the solution. To create a mesh, Right Click on the Mesh Size icon located in the Design Study Bar and select Generate Mesh. This process is also shown in **Figure 35**.

If there are any geometry errors that prevent the model from meshing, the user would get an error message at this point. Otherwise, the Message Window will display "Meshing completed". The user can now inspect the create mesh and any results generated (if an analysis was run). **Figure 36** shows the mesh created for our Design Study example.

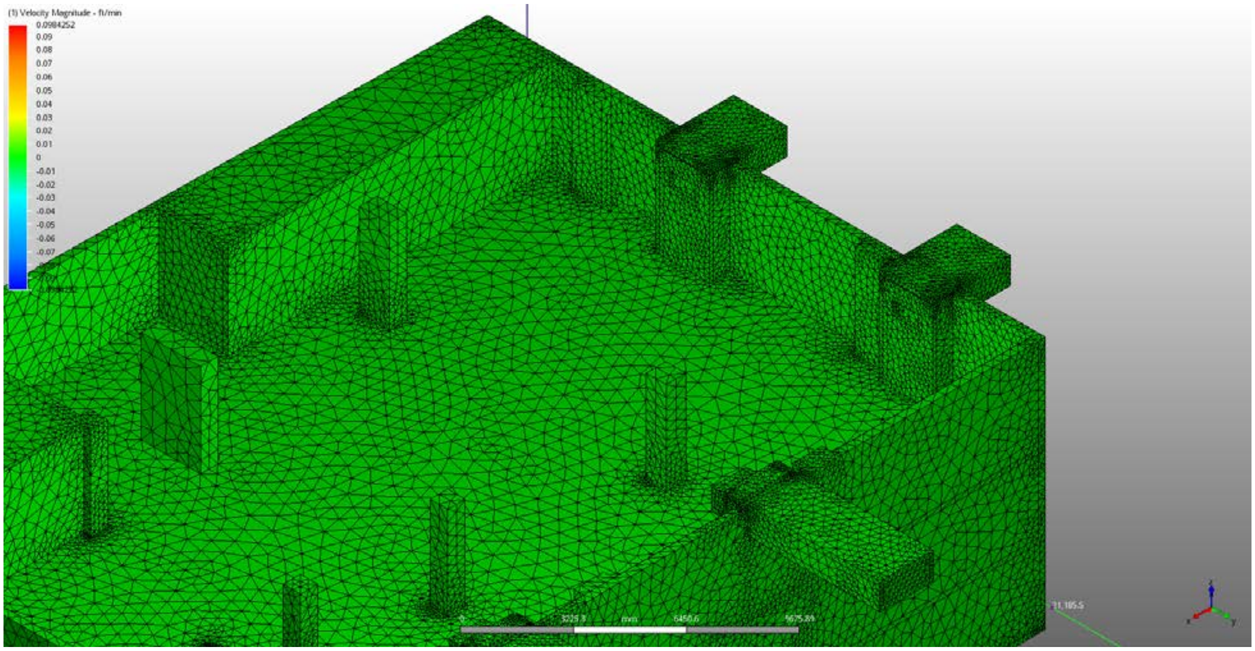


Figure 36. Meshing completed successfully

Final Thoughts

Hopefully, by the end of this demo you will have the confidence to start using CFD right away. It has been demonstrated that following a systematic approach you can create complex design studies by making additions to smaller ones.

Getting an initial mesh is an important part of the process and the mesh quality is a crucial factor in speed and convergence. The aim of this class was to help you overcome the initial obstacles that I encountered when running my first simulations. I encourage you to further investigate on meshing and solving. You now have the tools to run many simulations on your first mesh and evaluate different scenarios and boundary conditions.