AUTODESK **UNIVERSITY** 2014

One Model to Rule Them All: Using Revit to Produce Multiple Analytical Energy Models

Steve Gross, PE – Energy Analyst/Mechanical Project Engineer @ Interface Engineering

MP5879

Participants will learn how to use Revit software as a powerful and rapid geometry creation tool. As energy analysts and mechanical engineers, we often require multiple energy analysis programs to produce energy performance models, compliance models for Title 24 and Leadership in Energy and Environmental Design (LEED), load calculation models, and computational fluid dynamics (CFD) models. We can save significant time and money by using Revit software to create a single analytical model that we can export to various programs, including Simulation CFD software, eQUEST, energyPRO software, Trane's TRACE software, Integrated Environmental Solutions Ltd.'s Virtual Environment software, and California's new CBECC-Com software. This class will provide a detailed workflow for creating the geometry and exporting to the various programs, and you will discover tips and tricks to help troubleshoot and control the quality of your mechanical, electrical, and plumbing designs.

Learning Objectives

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

- Understand how to use the Revit software workflow for rapidly building an analytical model from an architectural model
- Learn how to use Revit software and the Green Building Studio service to export to multiple energy analysis programs
- Learn how to use Revit software to quickly import a full lighting design into an energy model
- Learn how to use Revit software to create CFD boundary condition geometry

About the Speaker

Steve Gross is an Energy Analyst and licensed Mechanical Engineer at Interface Engineering, Inc. His background is in Mechanical Engineering and Building Science, and his specialties include whole-building energy analysis, sustainable mechanical design, and mechanical control system optimization. Steve has substantial experience with successful projects, including those involving Leadership in Energy and Environmental Design (LEED), Net-Zero Energy and Water, energy audits/retrofits, and Passivhaus certification. He also has conducted research in the areas of model-based control, phase-change materials, and natural ventilation control strategies. He holds a B.S. in Mechanical Engineering from The University of Tulsa and a M.S. in Mechanical Engineering from Portland State University.

steveg@interfaceeng.com ; stevejgross@gmail.com

Acknowledgments

- Glen Walson, PE, LEED AP – BIM Manager at Interface Engineering
- David Horger – BIM/CAD Technologies Developer at Interface Engineering
- Stacey Lin, PE – Principal at Interface Engineering
- Aryn Bergman, PE – Mechanical Engineer at Tenderloin Consulting

Thank you for your pioneering efforts and contributions towards the development of this workflow and this AU class.

Introduction

Over the past several decades, detailed energy analysis has become an integral part of the design process for the built environment. Designers, Engineers, and Owners have come to rely on whole-building energy simulations and Computational Fluid Dynamics (CFD) to provide valuable feedback on the performance of their proposed designs to ensure the project meets its energy and occupant comfort goals.

As a part of this process, Energy Analysts and Mechanical Engineers often require multiple energy analysis programs to produce energy performance models, compliance models for Title 24 and Leadership in Energy and Environmental Design (LEED), load calculation models, and computational fluid dynamics (CFD) models (often requiring different software for each instance). The workflow presented here will give an in-depth demonstration of how a single analytical Revit model can be used to generate the input geometry for use in multiple analysis programs.

Rapid Analytical Model Geometry Creation

The power of Revit, in terms of energy analysis, lies in its ability to rapidly create an analytical representation of a building's various envelope components (i.e. walls, roofs, windows, and skylights). We use the term "analytical model" to differentiate between a detailed architectural model and a model that only contains the elements required to produce an accurate energy model.

Typically, the process begins with a detailed Revit model provided to the Analyst by an Architect. The Analyst then creates a simpler analytical model which can then be imported into various analysis programs via the Green Building XML (gbXML) schema. The following subsections provide a detailed workflow of this process.

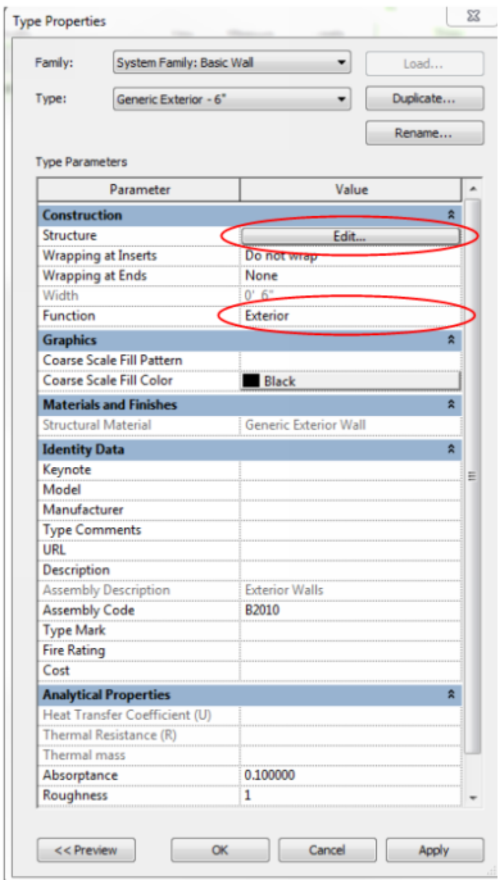
Project Template File

NOTE - Much of the information in this section has been previously presented at previous Autodesk University classes. Therefore, with permission from the Aryn Bergman, original author, I have reproduced it here with minor edits.

Creating a template file with the most commonly used families and views will help to expedite the creation of your analytical model. The idea behind using the template file is to preserve useful views, construction types, and schedules that will be used for every project. For example, it may be useful to have multiple 3D views in the template file; perhaps one showing the analytical model and architectural model together and another with only the analytical model. While each user will have a customized version of the template file, there are several items that should be included. The following subsections will provide suggestions on which items to include and how to set them up.

Envelope Families

Start with creating exterior, interior, and underground walls. Click on the structure "Edit" and use only one thickness as walls with varying thicknesses will introduce errors in your model. Create different materials for your interior and exterior walls so that you can assign different colors for identification purposes in your model. If an interior wall is mistakenly used as an exterior wall, then that surface will appear in the DOE2 or EnergyPlus model. Do the same for interior and exterior floors and roofs. Again make sure to use a standard thickness (i.e. 6") because in some models you will need to align roofs to floors.



For windows and skylights, create families that are only openings with an extrusion representing the glazing area. They should be modeled with a simple geometry; detailed geometry may cause errors in the energy model and will slow down CFD model run times.

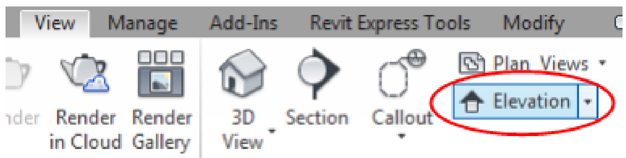
Doors are rarely used in energy models as ASHRAE 90.1 states that you do not need to model an exterior surface if it is less than 10% of the overall surface area of the building. If doors are needed, then follow the same guidelines for creating windows and skylights. Use simple geometry and a different material surface for identification purposes.

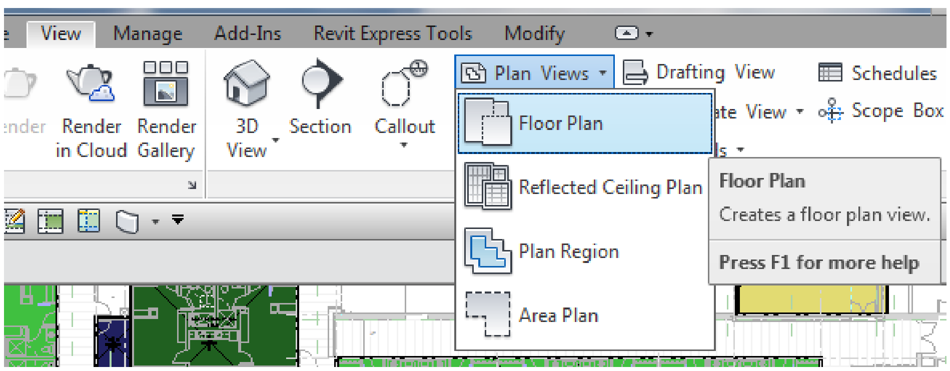
For building shading devices create a family type for the walls for vertical shading devices and a family type of roof for horizontal and angled shading devices. Thickness does not matter.

Views

Adding customized views to the project template will speed up the initial project setup as well as the creation of the analytical model. To begin, create an elevation view on the default floor plan that is present in the new project. Make sure the discipline is set to Architectural. Enter the elevation view and create a set of placeholder levels. Create a reasonable number of levels based on the type of projects you work on. My template file has 5 floor levels and a roof level. The floor-to-floor height doesn't matter at this point, as you will align the levels to the architectural model when you start the project.

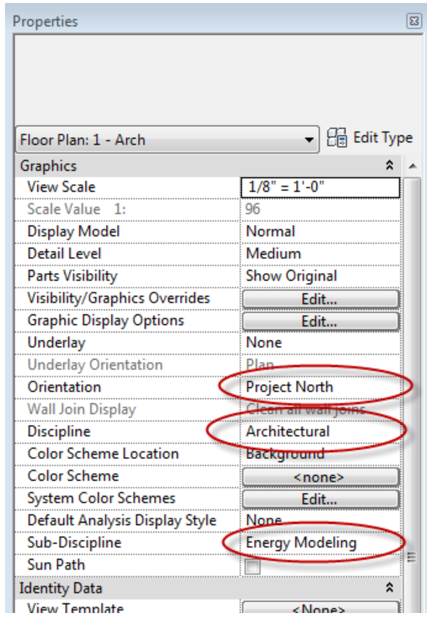
Then, create floor plans that correspond to the levels you've created to serve as place holders.





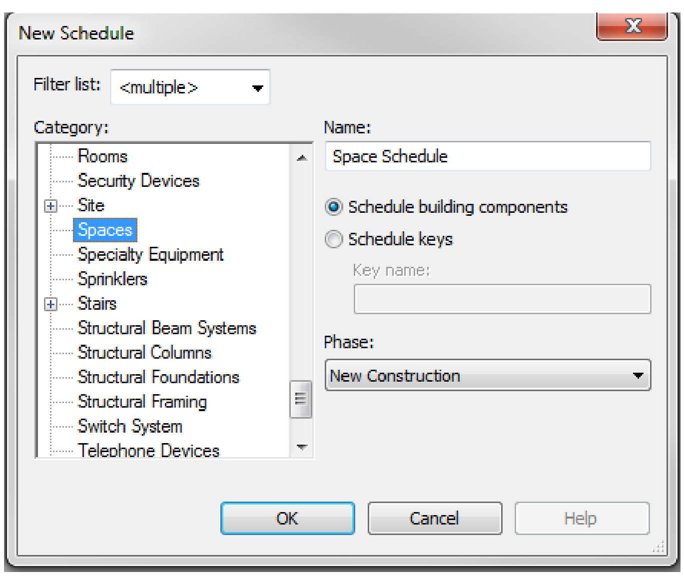
Next, create the remaining three elevations, one for each of the primary orientations. You can add additional elevations if required for specific projects.

It is useful to create floor plans and 3D views with the Discipline set to "Architectural" so that the architectural elements are more visible and easier to select. Create a sub-Discipline "Energy Modeling" for larger projects where you'd like to group views with the energy analysis geometry together in the project tree.

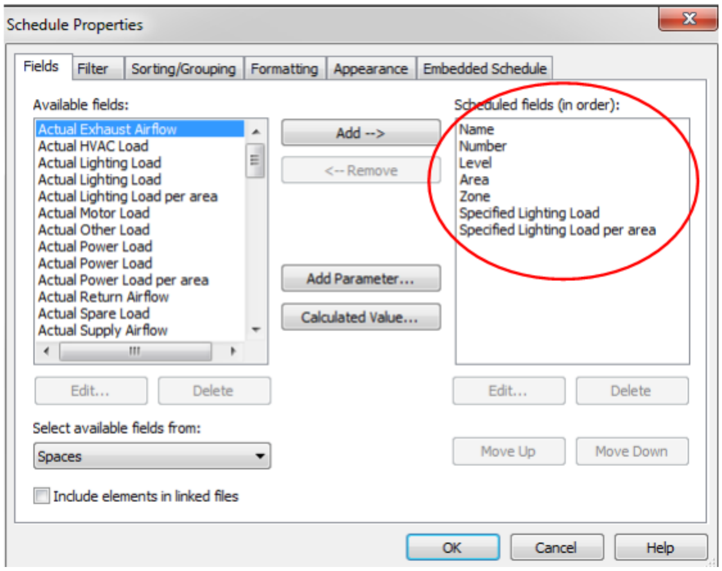


Create a view on the ground floor with the Orientation set to "True North." This will allow you to create the geometry in plan view, but be able to rotate the project True North anytime without affecting the orientation of other views. Keep the orientation set at "Project North" for all other views.

Space and HVAC Zones schedules are another feature to include in your template file that will save significant time. They can be used to display relevant information about the spaces you will be creating in spreadsheet form, such as Space Name, Zone Name, Level, Area, Space Height, and load information. In most cases, it is much easier to edit these quantities in spreadsheet form. On the Project Browser tree, right-click "Schedules /Quantities" and create a space schedule.



You can then add any of the information in the "Available Fields" section to be displayed in your schedule.



Creating the Analytical Model for Energy Modeling

In this section, creation of the analytical model will be discussed step by step. The general approach to this workflow is to "trace" over the architectural model, recreating a simplified geometry. Keep in mind that the workflow presented in this section is specifically for importing into energy modeling software like eQUEST and IES-VE. We will discuss modifications to this approach for CFD in another section.

Before jumping into the workflow, let's discuss some general rules to follow throughout the process.

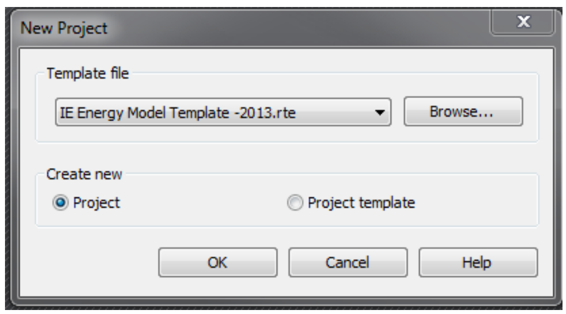
General Modeling Rules

1. Simplify geometry whenever possible – The analytical model doesn't need to reflect every detail of the building; it just needs to capture the features that significantly impact the energy consumption of the building.
    a. Exterior envelope components (windows, walls, roofs and floors) should be modeled as close as possible to the architectural
    b. Interior Zoning can be simplified.
        i. Group spaces with the same occupancy type and thermal exposure together into a single zone
        ii. Try to create rectangular shaped spaces whenever possible. There is no need to capture each nook and cranny of interior spaces
        iii. Don't include shaft or stairwell openings in floors and ceilings
        iv. Don't allow gaps between architectural elements
        v. Don't include columns, shafts and other building components that don't bound occupiable spaces.

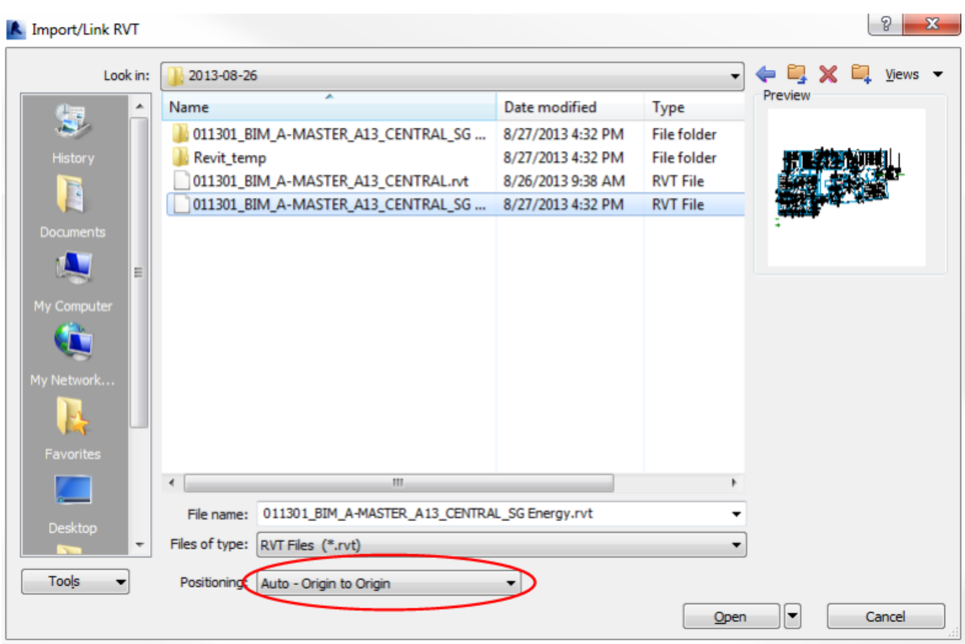
  c. Don't include doors unless they represent more than 10% of the exterior surface area of the building
2. Do not use design options. Save separate instances of the model to accomplish this
3. Do not use in-place families. Use the native tools for windows, walls, floors, roof, etc. In-place families do not translate to the energy analysis program
4. Under the Area and Volume computations (Analyze>>Space & Zones>>Area and Volume Computation) set the Volume computations to Areas and Volumes.

Step 1 –Link Architectural Model

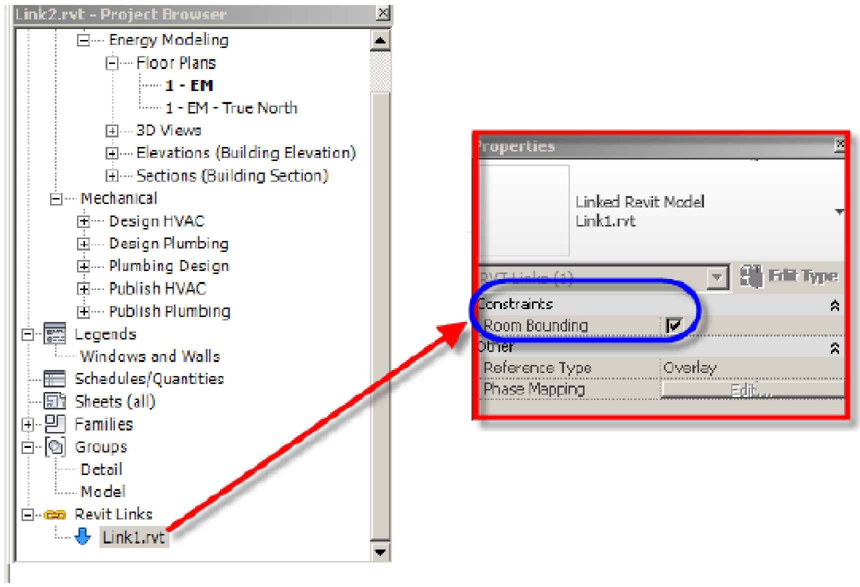
Begin by opening Revit and creating a new project. Start with the template file discussed in the previous section.



Navigate to the Level 1 floor plan, then navigate to the Insert tab on the ribbon and click "Link Revit". Navigate to a saved copy of the architectural Revit model you wish to use. Make sure to change the positioning to origin-to-origin

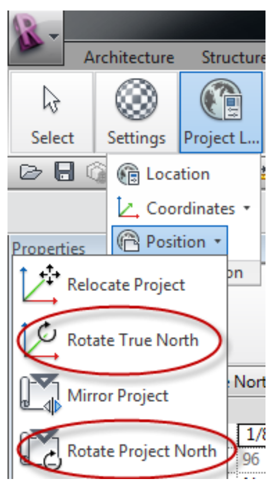


Make sure the "Room Bounding" building constraints for the linked models is NOT checked.



Step 2 –Set Orientation and

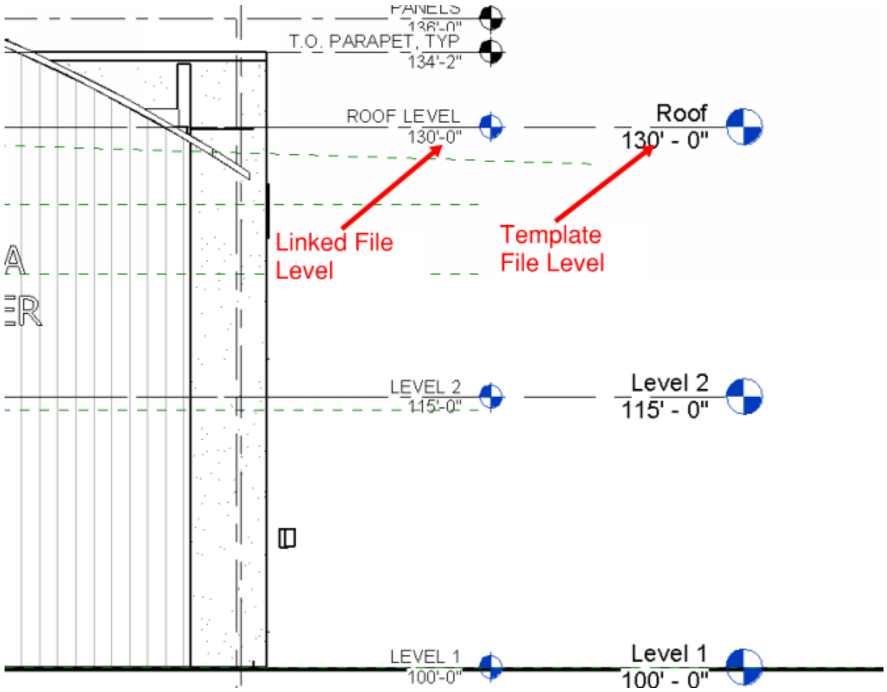
In the floor plan oriented to true North, click on Project Location under the Manage tab and either rotate True North or Project North to the correct orientation.



Step 3 –Align Levels

It is important to align the floor-to-floor heights of your template file with the architectural model so that you get an accurate accounting of exterior wall area.

Navigate to any of the elevation views. You will notice “Level” annotations belonging to both your template file and the architectural model. Use the “Align” tool to move the template file’s level to the same height as the linked file.



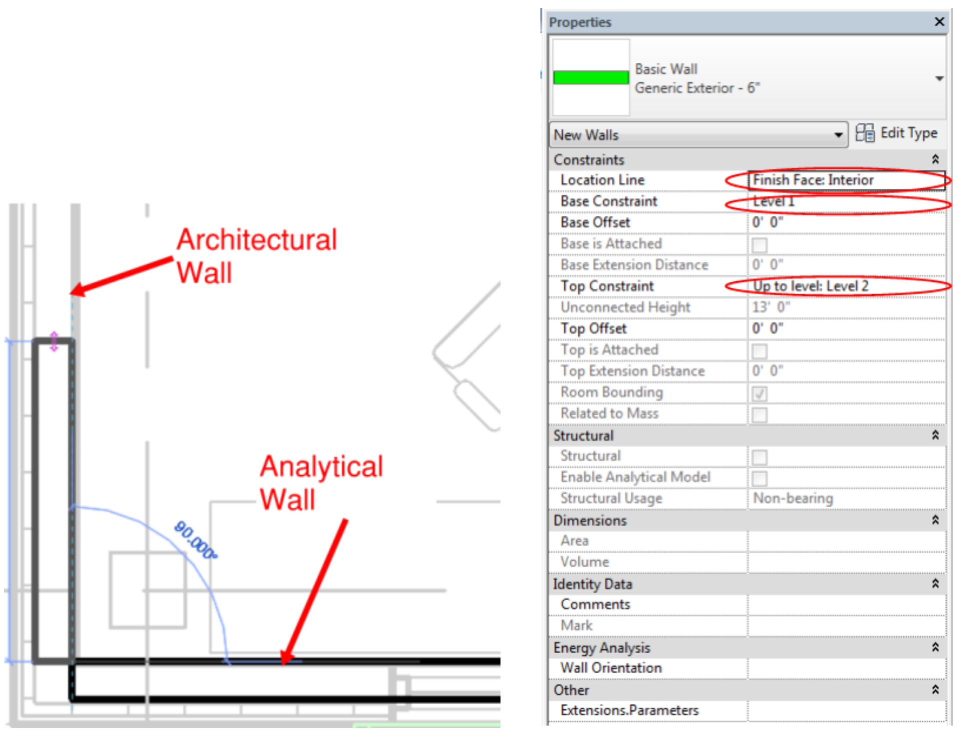
Now, when you open a floor plan in your model, you will see a floor plan from the linked model.

Step 4 – Draw Exterior Walls

Starting with the lowest level, begin by creating exterior walls by tracing over the linked model. Set the base constraint to the level you are working on and set the top constraint to the level above. It is often easiest and most accurate to set the “Location Line” to Finish Face: Interior. This makes the floor area bounded by the walls accurate.

Notice in the image below that the analytical wall is thinner than the architectural wall. This is irrelevant to the energy analysis. The only important aspect is an accurate floor and wall area.

Also note, by setting the linked model as halftone in visibility graphics makes it much easier to distinguish between analytical and architectural elements.



Continue tracing all the exterior walls to establish the shape of the building. Revit has several tools in the Modify tab of the ribbon that allow the geometry to be drawn much quicker than most energy modeling programs, such as trim/extend, align, array, copy, and paste.

One particularly useful item is the “Paste- Aligned to Selected Levels” tool. This comes in handy if you have a building with a uniform shape on multiple levels. You can simply draw the first floor and then copy up to the typical floors.

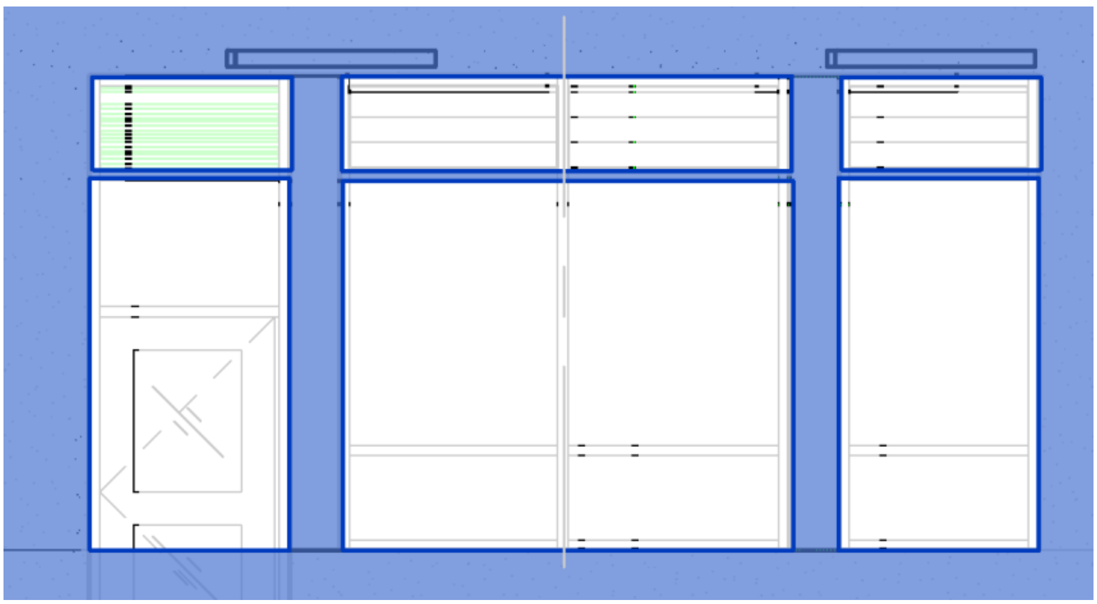
Items to Note:

1. You can use the same wall type for both above ground and below ground surfaces. The GBXML export process will differentiate between the two.

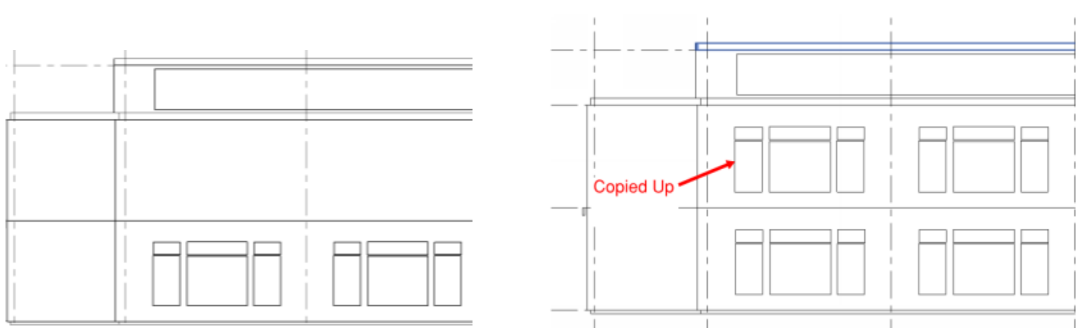
2. Don't make walls that traverse multiple levels. This will cause problems during export. Use the Copy and Paste tools to copy up walls that are identical on multiple floors.
3. Model spandrel glazing as an exterior wall
4. Only use one type of exterior wall. You must assign different wall types in your energy modeling program once it's been imported.
5. Model curved walls by breaking it up into several straight sections

Step 5 – Draw Windows

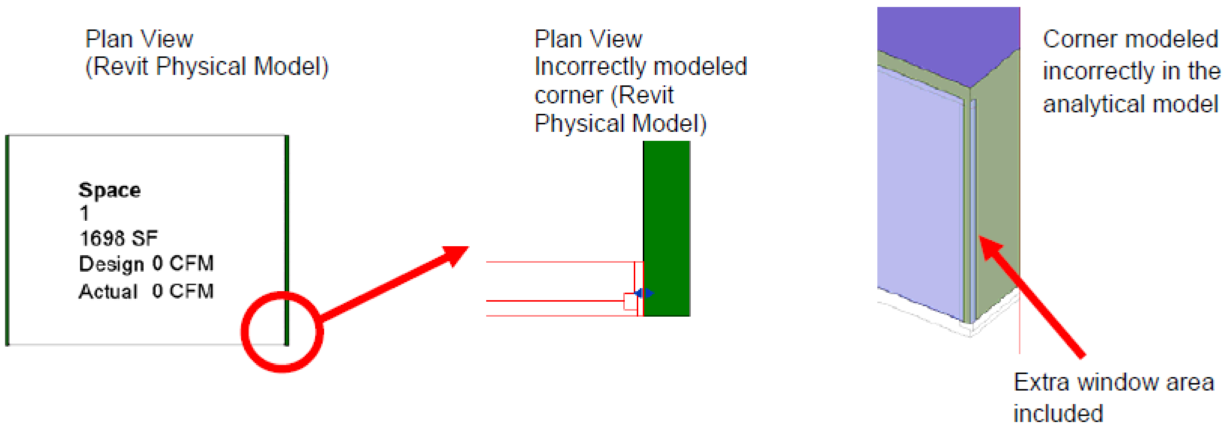
Draw windows in elevation view by placing a window at the same location shown in the linked model. Use a simple window geometry that doesn't have a complex frame. Simply place a default sized window over the window in the architectural model and use the align tool to match the dimensions.



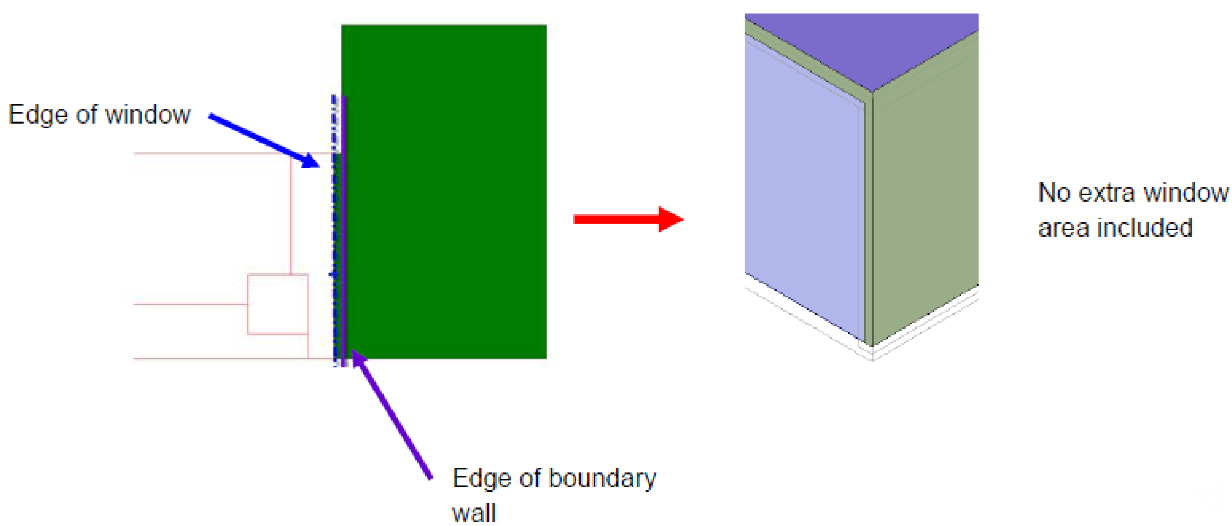
Windows can also be copied and pasted in floor plan view if the size and sill height are the same. If multiple floors have typical window arrangements, then you can use the copy and paste to selected levels tool in the same manner as with the walls. Simply lay out one of the floors and copy the typical windows up to other levels.



Offset the edge of the window from the edge of a bounding wall, floor, or roof by at least 0.5", even for curtain walls.



And here is an image of a window with the offset:

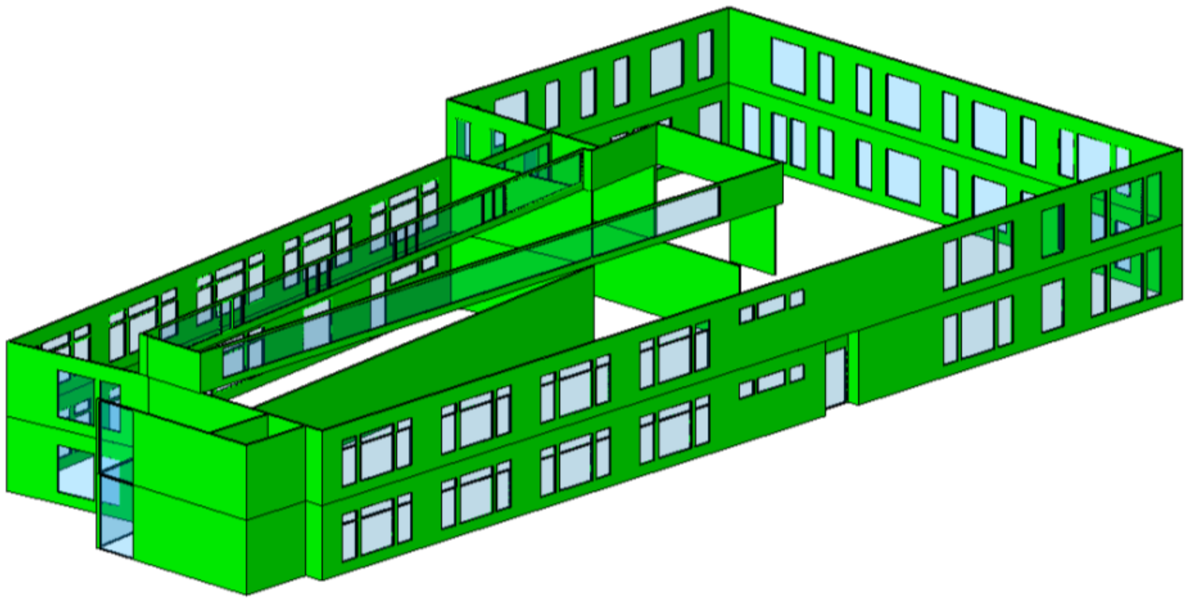


Items to Note:

1. If you have multiple windows at a fixed distance apart, you can you use the Array tool
2. Model curtain walls as a single pane of glazing (i.e. don't model the individual sections separated by mullions)

Progress so far…

At this point, you have the shell of the building and windows placed.



Step 6 – Draw Floors and Roofs

Draw floors using the “Pick Walls” tool: . Simply select all the exterior walls which define the boundary and a floor will automatically be drawn. As with other envelope elements, it can easily be copied up to other typical floors. Remember to use only one type of floor throughout the project. Model slab on grade as an exterior floor as well.

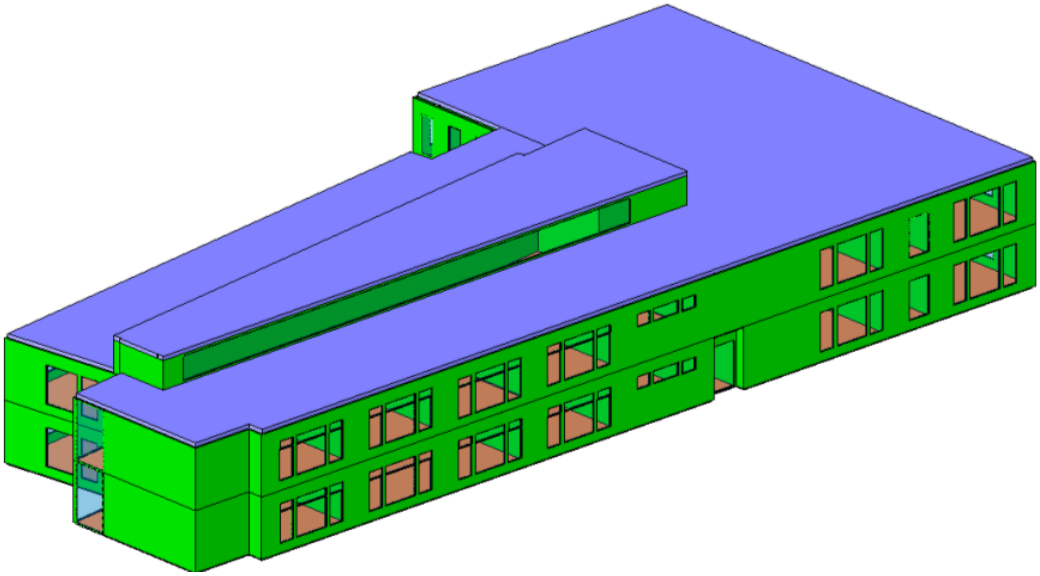
Roofs are drawn the same way. Use the same thickness for roofs as you used for floors.

Items to Note:

1. Use only one floor type throughout the project. It has been my experience that only exterior floor types (as opposed to interior) will export correctly through GbXML. Use exterior floors on all levels of the model.
2. After completing the floor, you will be prompted “Would you like walls that go up to this floor’s level to attach to its bottom?” Answer NO to this prompt. It can sometimes introduce errors otherwise.
3. Any floor that extends beyond an exterior wall should be modeled in two separate pieces; the section with an interior space above as a floor and the section with exterior above should be modeled as a roof; otherwise you will receive an error in your model.
4. Do not model stairwell or shaft openings in floors

Progress so far…

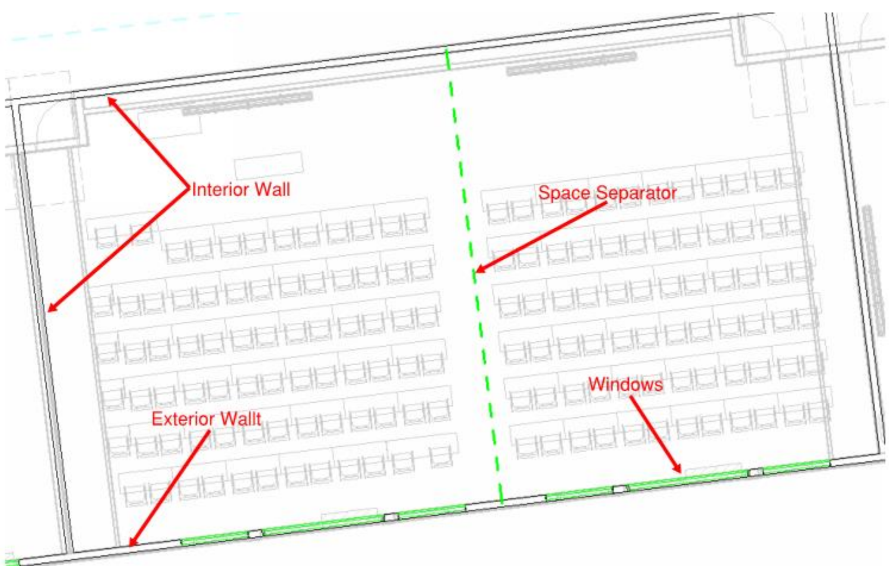
You now have a complete shell.



Step 7 – Interior Walls and Space Separators

Now that the exterior shell is created, the interior needs to be zoned according to thermal exposure and occupancy type. The same wall type used for the exterior can be used for the interior. The GbXML export process will differentiate between internal and external walls automatically.

Use space separators for zoning out spaces were walls don't exist (in open office spaces, for example). In eQUEST, this will result in walls composed of air. In IES-VE, this will result in walls with full-sized holes in them.



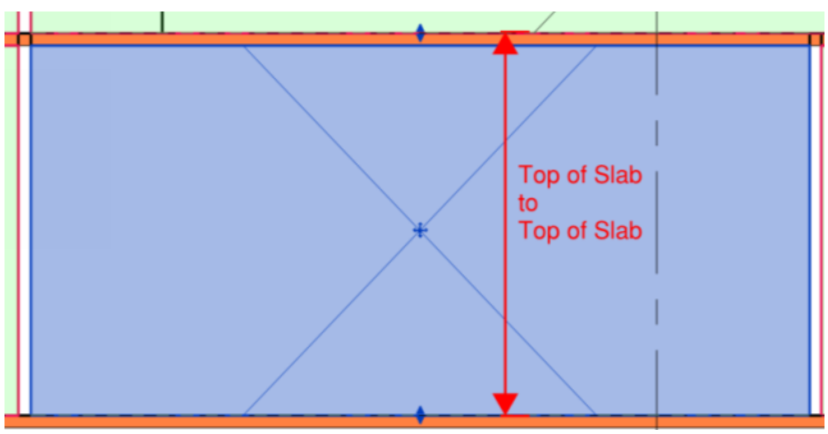
Items to Note:

1. Interior walls should always be full height (i.e. from level to level)
2. Space separators do not have a height associated with them, but will automatically go from level to level.

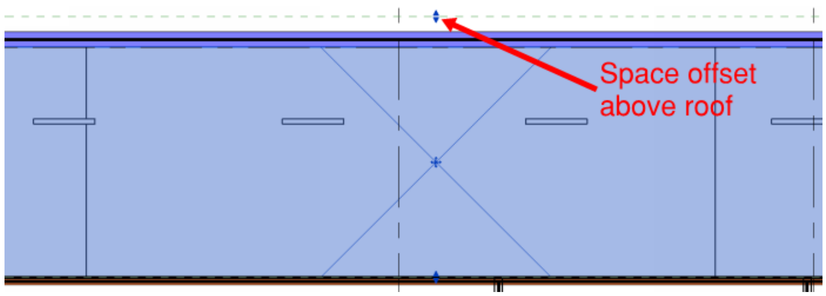
Step 8 – Create Spaces and Zones within Analytical Model

Now that you have all the necessary bounding elements in place, it's time to create spaces in the analytical model. Spaces are objects in Revit that hold all the relevant load data, such as exterior envelope components and lighting loads, that are exported to the energy modeling software. HVAC zones, or just simply zones, are intended to be a collection of spaces. For example, you might have 3 office spaces in a single HVAC zone. However, for the purposes of energy analysis, it is easiest and most straightforward to create one zone per space. Adding multiple spaces in a zone creates a level of complexity that is generally not necessary for energy analysis. Also, eQUEST does not allow you to have multiple spaces in a zone; the software is intended to work with one space per zone. Both elements are required for a successful GbXML export.

Click the "Space" button under the Analyze ribbon tab. Then, ensure that the space's height is floor-to-floor by setting the "Upper Limit" constraint to the level above. This is a critical step, because if the space does not encompass the bounding surface above (either the floor or roof), that surface will not export through GbXML. Use a section to inspect the vertical shape of the space to ensure it is properly aligned. Notice in the picture below that the space goes from top of slab to top of slab.



Another critical step is to use a limit offset for spaces which reside under a roof element. The offset should be at least the thickness of the roof construction element.



Next, use the “Place Spaces Automatically” tool that becomes available. This will create a space element in every fully bounded area. If you forget to set the space limit before placing the spaces, you can simply window select all and use the filter tool to select only the spaces. Once all spaces are selected, you can change the upper limit and offset globally for all spaces in the current view.

Next, create a HVAC Zone for every space by first selecting the space, then clicking the “Zone” button under the Analyze ribbon tab.

Note, in most instances, architects will use rooms in thier model. It is important to understand why we can’t use rooms for the purposes of energy analysis. The space element holds all the loads data associated with the space and rooms do not. So if you want to export LPDs and EPDs or any other of the load features, you must use space elements.

Items to Note:

1. Create one Space and HVAC Zone element for each zone desired in the energy model
2. Make sure space limits go from floor-to-floor.
3. For spaces on the top floor, use a limit offset of 1’ so that the space rises above the roof element. This will prevent false shading surfaces from showing up in the energy model

Step 9 – Space and Zone Naming

Naming spaces and zones can be the most tedious exercise involved with creating the analytical model. However, it is very important to have a name that will be allow you to identify the occupancy type once you have imported the GbXML file into the energy modeling software.

At Interface Engineering, we are employing two tools to help with this. First is a tool available on Autodesk’s website called the Space Naming Utility. This tool takes the Room name found in the architectural model and automatically applies it to the space. In most cases, you are able to identify the occupancy type based on this name and is thus an acceptable method. The second tool we use was developed in-house called the Create Zones tool. This actually creates 1 zone for each space automatically while also giving the zone the same name as the space. The combination of these tools has saved us countless hours of work.

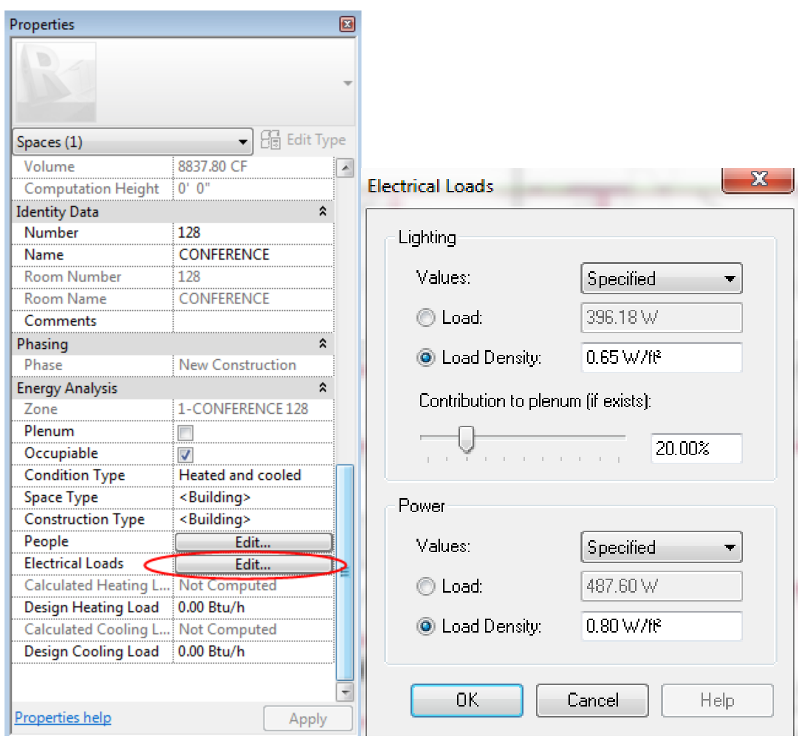
Items to Note:

1. Use a naming convention that will allow you to determine the zone's occupancy type once imported into the energy modeling software
2. Add-in tools can expedite the zone creation and naming process
3. eQUEST uses the Zone name from Revit
4. Trace uses the Space name from Revit
5. IES-VE uses the both the Zone name and a combination of the Space name and number

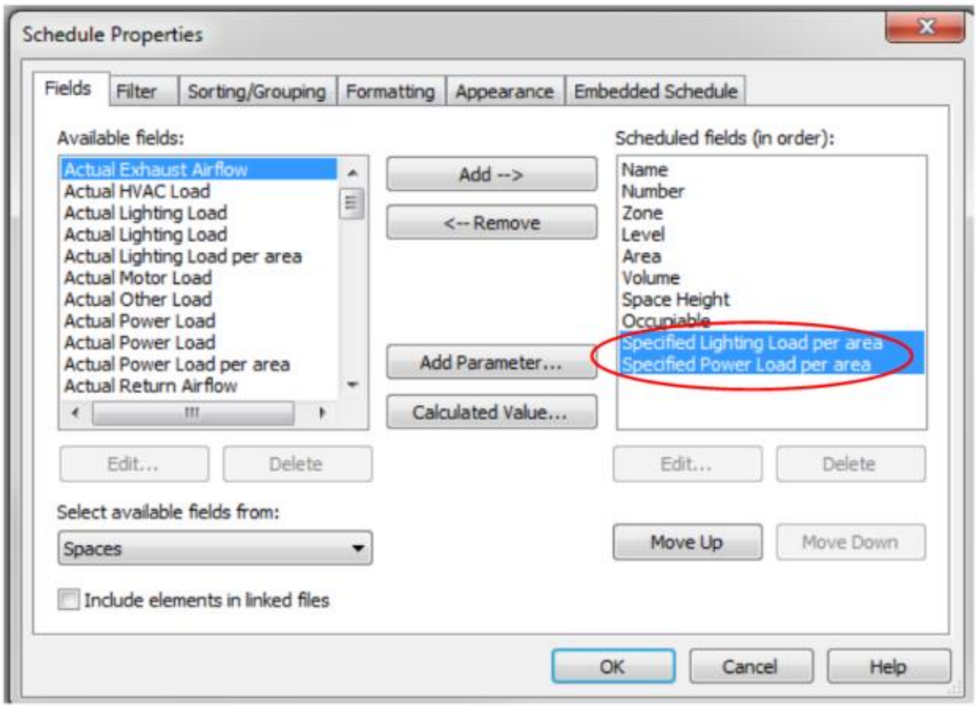
Step 10 – Setting Space Load Parameters

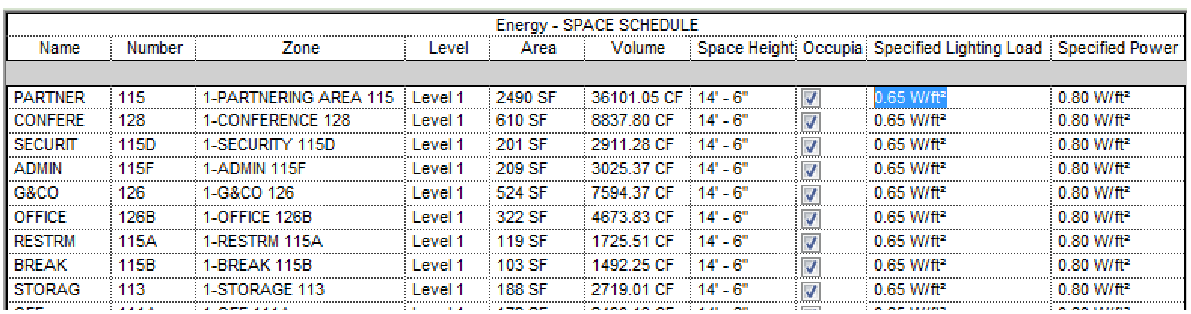
Revit allows you to specify load parameters in each space before it is exported to the energy modeling software. For example, lighting power density (LPD) and equipment load density (EPD) can be specified. While this information can be input in the specific energy modeling software you are using, it is often more convenient to do so in Revit. Its ability to schedule this information makes it very quick and easy to enter different LPDs and EPDs for different occupancy types.

Select the spaces you want to edit and click Edit button next to the Electrical Loads parameter in the Properties dialog. Then, you can specify the lighting and equipment loads in either Watts or Watts/ft^2. Unless you have actual lighting figures in the spaces (which will be discussed in a later section in detail), make sure to set the Values parameter to Specified.



Another approach is to select all the spaces and change the Values parameter to Specified. Then use a space schedule to enter the desired load values. Once you add the load parameters to your space schedule they are editable.

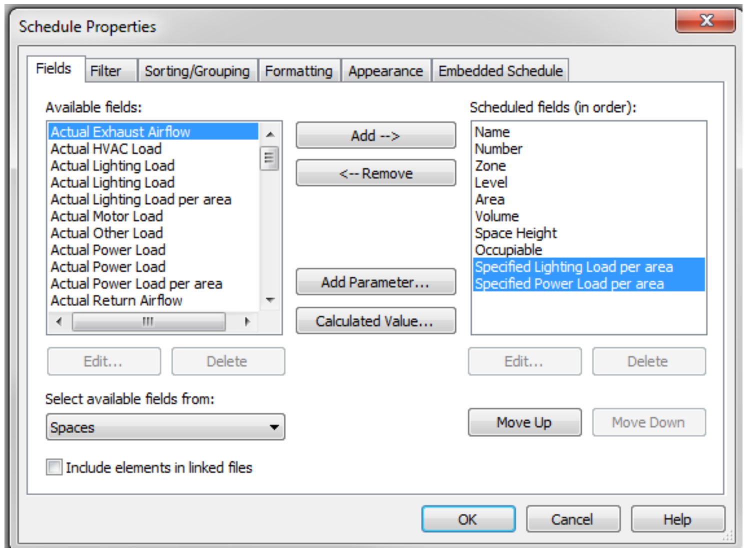




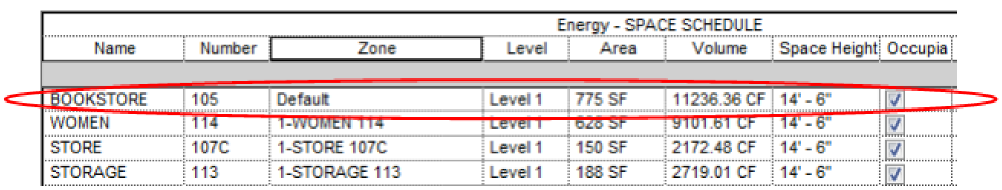
Energy - SPACE SCHEDULE									
Name	Number	Zone	Level	Area	Volume	Space Height	Occupia	Specified Lighting Load	Specified Power
PARTNER	115	1-PARTNERING AREA 115	Level 1	2490 SF	36101.05 CF	14' - 6"	☑	0.65 W/ft²	0.80 W/ft²
CONFERE	128	1-CONFERENCE 128	Level 1	610 SF	8837.80 CF	14' - 6"	☑	0.65 W/ft²	0.80 W/ft²
SECURIT	115D	1-SECURITY 115D	Level 1	201 SF	2911.28 CF	14' - 6"	☑	0.65 W/ft²	0.80 W/ft²
ADMIN	115F	1-ADMIN 115F	Level 1	209 SF	3025.37 CF	14' - 6"	☑	0.65 W/ft²	0.80 W/ft²
G&CO	126	1-G&CO 126	Level 1	524 SF	7594.37 CF	14' - 6"	☑	0.65 W/ft²	0.80 W/ft²
OFFICE	126B	1-OFFICE 126B	Level 1	322 SF	4673.83 CF	14' - 6"	☑	0.65 W/ft²	0.80 W/ft²
RESTRM	115A	1-RESTRM 115A	Level 1	119 SF	1725.51 CF	14' - 6"	☑	0.65 W/ft²	0.80 W/ft²
BREAK	115B	1-BREAK 115B	Level 1	103 SF	1492.25 CF	14' - 6"	☑	0.65 W/ft²	0.80 W/ft²
STORAG	113	1-STORAGE 113	Level 1	188 SF	2719.01 CF	14' - 6"	☑	0.65 W/ft²	0.80 W/ft²

Step 11 –QCing the model

There are several tools you can use to inspect the model to confirm it is set up properly. The space schedule is very useful for this purpose. Create a schedule with the fields shown in the picture below. The Space Height parameter is a calculated value which is simply the Volume divided by the Area.



First, confirm that each space has a corresponding HVAC zone by searching the Zone column for any instances of Default. If a space is in the Default zone, that means you haven't created an exportable HVAC zone for that space. Correct this by selecting the space and clicking the Zone button under the Analyze tab.

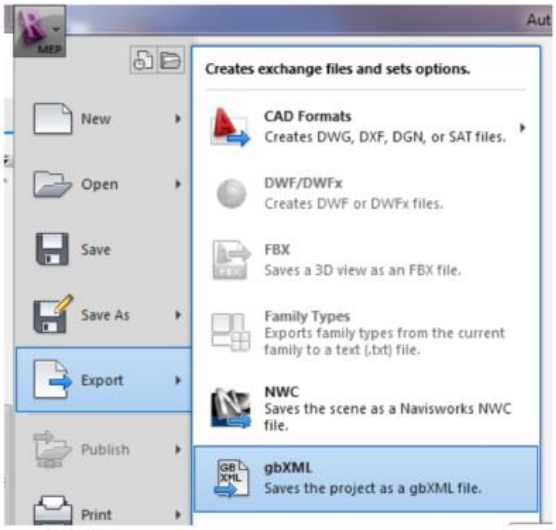


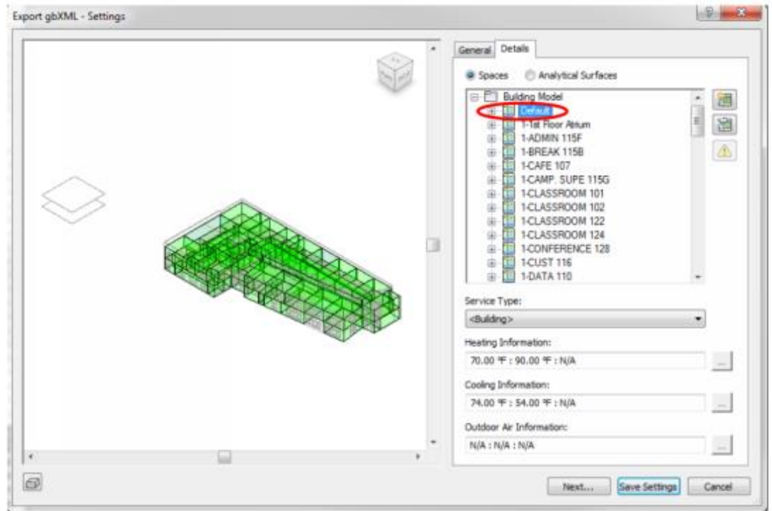
Energy - SPACE SCHEDULE

Name	Number	Zone	Level	Area	Volume	Space Height	Occupia
BOOKSTORE	105	Default	Level 1	775 SF	11236.36 CF	14' - 6"	☑
WOMEN	114	1-WOMEN 114	Level 1	628 SF	9101.61 CF	14' - 6"	☑
STORE	107C	1-STORE 107C	Level 1	150 SF	2172.48 CF	14' - 6"	☑
STORAGE	113	1-STORAGE 113	Level 1	188 SF	2719.01 CF	14' - 6"	☑

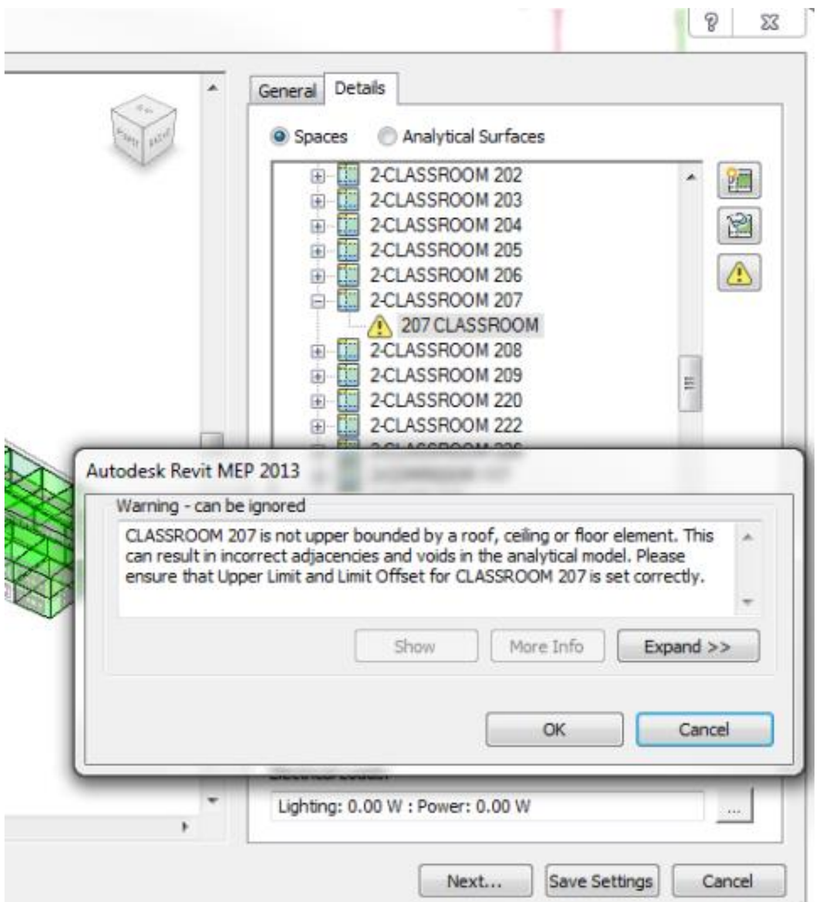
Then, inspect the schedule to ensure each space is assigned to a level and the space is marked as occupiable. Check the space height column to ensure the spaces span from floor to floor. If you see a space that is different than the rest or shorter than the floor-to-floor height, select the zone and correct the issue.

Another method of QCing the model is to look in the GbXML export dialog. Under the Details tab, you will notice un-zoned spaces will show up in the Default zone at the top of the list.





This dialog will also display various errors, such as have a space that is not fully bounded:

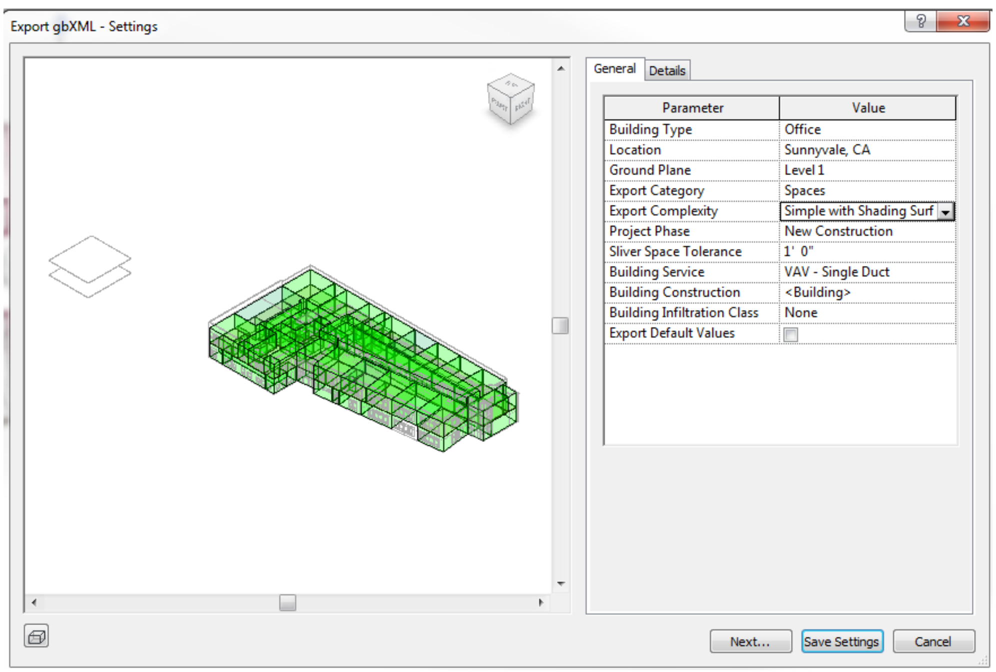


Step 12 –Exporting the Model via GbXML

Once you have inspected the model and corrected any errors, you are ready to export the GbXML file for use in the energy modeling software of your choice. Navigate to the Export to GbXML dialog

Edit the Energy settings (Analyze>>Energy Analysis>>Energy Settings)

- Set the Postal Code or city
- Set the Ground Plane – This dictates which surfaces will be underground vs above ground
- Set the Project Phase – Spaces must be placed in the same phase as the Project Information phase
- Energy Export Complexity set to Simple with Shading Surfaces – Keep in mind, the limit of shading surfaces in the model is 1024
- Set Sliver Space Tolerance. Leave the default value of 1' 0". Too much sliver space may allow light, solar radiation, and air flow thermal transfer between zones that in reality do not occur
- If you have specified LPD and/or EPDs, uncheck the export default values box



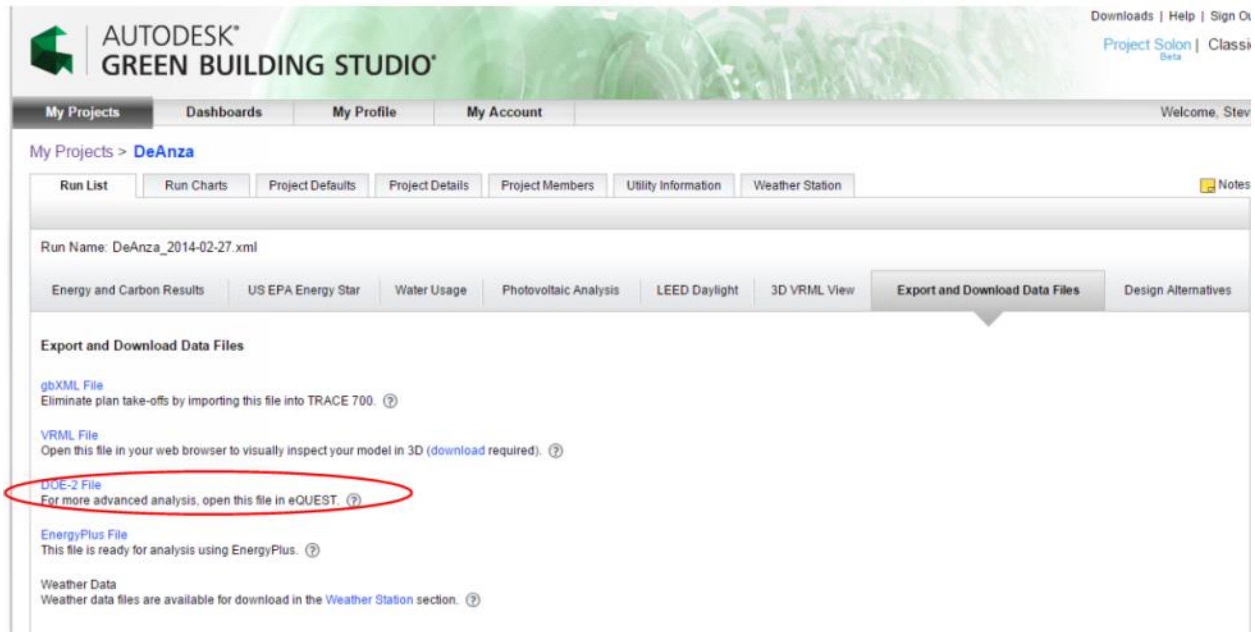
Then, click Next and save the file in a location of your choice.

That's it! You now have a complete analytical model with load parameters ready for import into the energy modeling software.

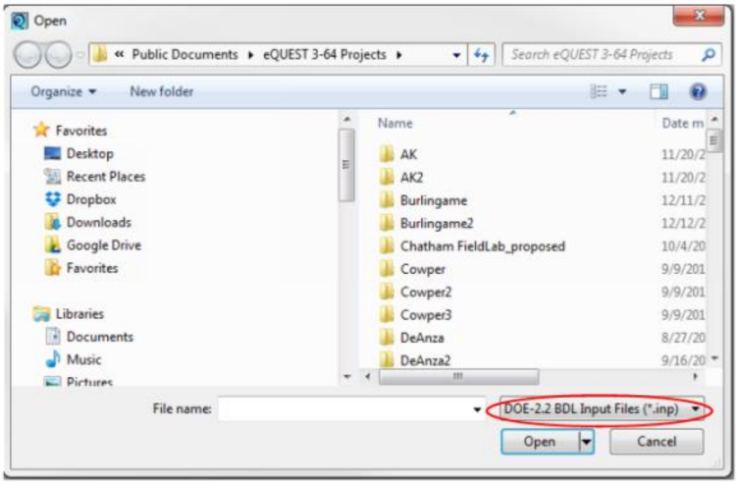
Importing the Analytical Geometry into Simulation Software

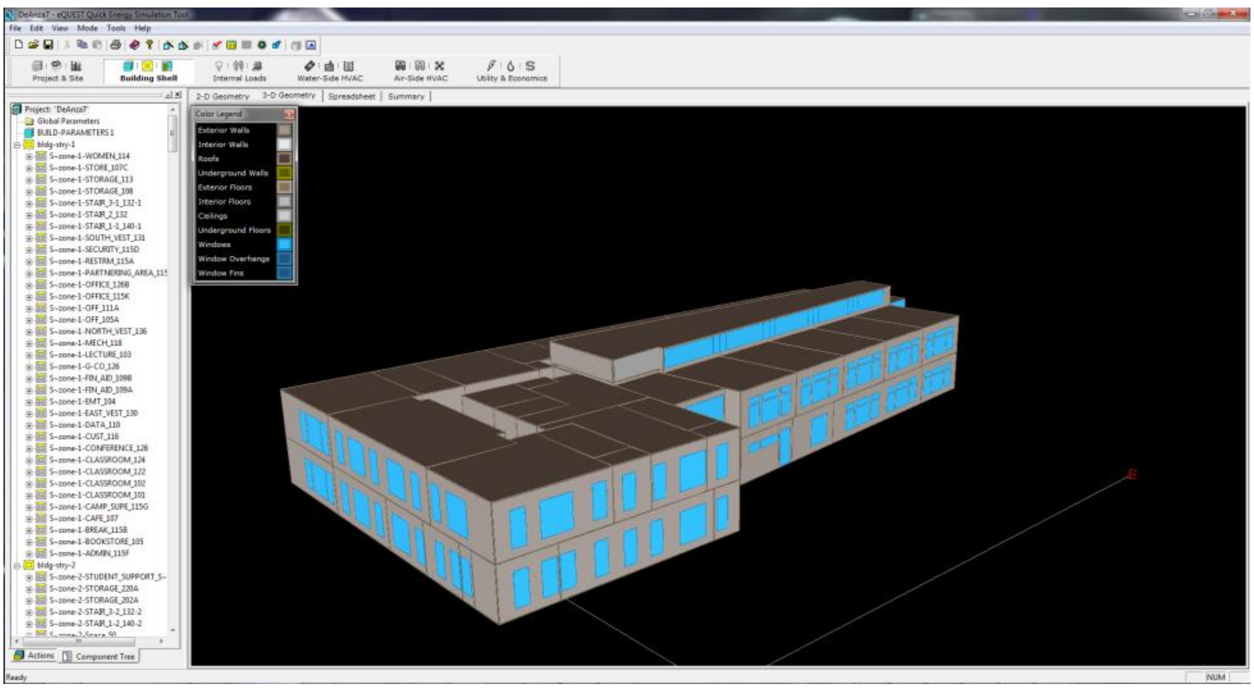
Importing to eQUEST via Green Building Studio

Because eQUEST doesn't have the ability to directly import a gbXML file, you must use Autodesk's Green Building Studio. Navigate to https://gbs.autodesk.com/ and create an account. Create a new project, fill in the relevant information and upload the GbXML file you've created. The website will then process the file in the cloud. Once processing is complete, you can download a DOE2 input file which can be found on the Export and Download Data Files page.



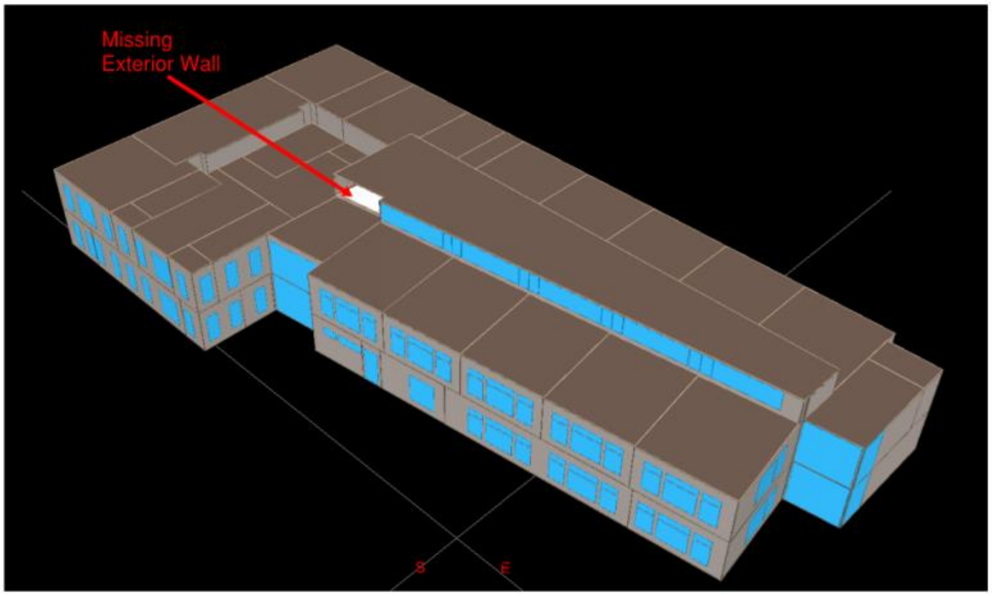
Next, open eQUEST and click "Select and Existing Project to Open", change the file type to *.inp and navigate to the file you've just downloaded from GBS. Now you are ready to start modeling in eQUEST!



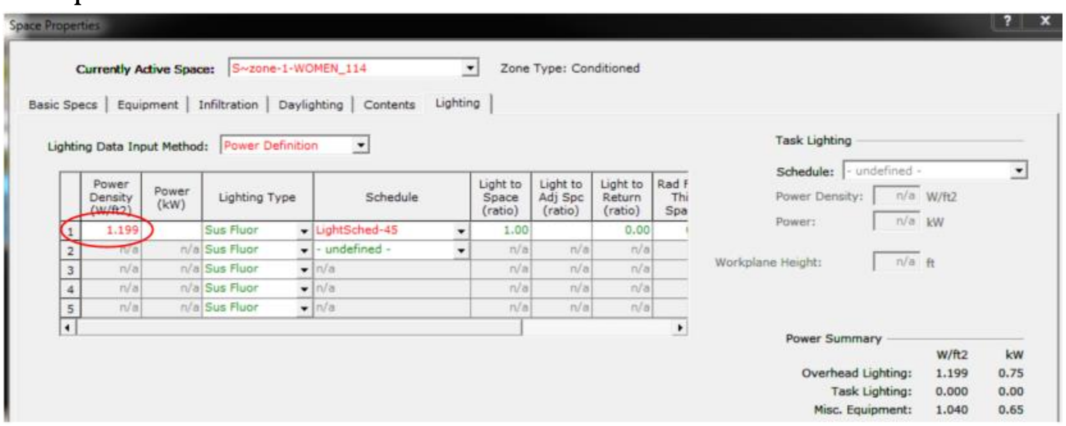


Items to Note:

- eQUEST uses the HVAC Zone name from Revit for both the space and zone name. The spaces are preceded by S~zone- and zones are preceded by zone-.
- Make sure to inspect the model in the 3-D Geometry tab for missing or incorrect (i.e. interior walls on an exterior surface or vice versa) surfaces. If you find errors, go back to Revit to troubleshoot the problem. It is most often caused by improperly aligned bounding surfaces.



- You'll notice that the load parameters you defined in Revit will be present in the space elements



Importing to EnergyPro

For projects in California, Title 24 energy compliance is required to acquire a building permit. EnergyPro is very useful for quickly producing the necessary Title 24 calculations and forms. It also has the ability to directly import the GbXML file without using Green Building Studio. That being said, it is a good idea to go through the eQUEST import process anyway because it provides a quick and easy way of inspecting the geometry with 3D representation of the model (which EnergyPro lacks).

Open EnergyPro and navigate to File -> Import -> Green Building XML. Then just navigate to the file you saved in step 12 in the previous section. That's it. Now you are ready to set up the remaining parts of your model in EnergyPro.

Items to Note:

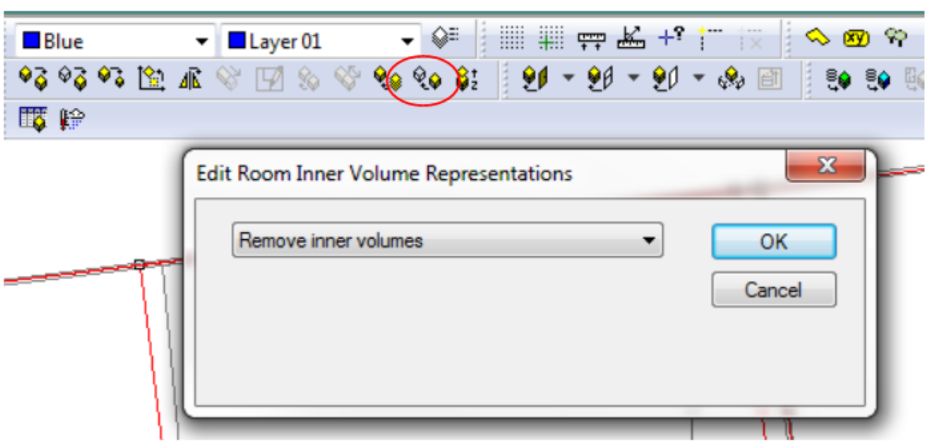
- EnergyPro references the Revit HVAC Zone name for its zone element, and the Revit space name for its space element. It will help to have a naming convention as described above, with both the space and the HVAC zone named very similarly in Revit.
- EnergyPro doesn't have a graphical representation of the model, so it is useful to use eQUEST to inspect the geometry before beginning modeling.
- Load parameters are imported into EnergyPro along with the geometry.

Importing to IES-VE

IES Virtual Environment is a very powerful analysis tool and many modelers are shifting towards using it for performance models. The import process is very similar to EnergyPro. You simply navigate to File -> Import -> gbXML file. In the next dialog, click Import and point to the file saved during Step 12 in the previous section.

During the import process, the program runs a special routine that "cleans up" any missing surfaces. This is a useful feature. Be aware that IES-VE does NOT import the load parameters from Revit.

It is important is to note that IES imports the thickness of interior walls used in Revit, while other programs like eQUEST and EnergyPro do not. You need to remove the inner volumes in the thickness of the wall. The ModelIT module has a button that does this for you automatically.



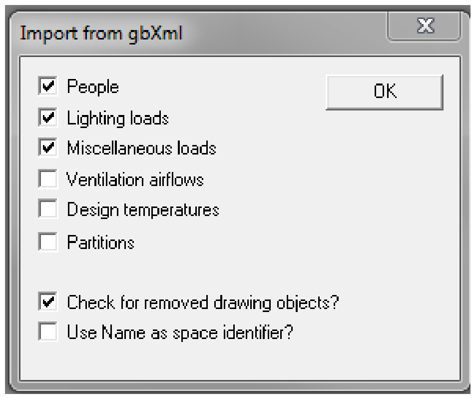
Items to Note:

- IES-VE references the Revit space name for its Room elements
- Load parameters do not import from Revit
- Remove inner volumes from interior walls

Import to Trane Trace

The same analytical geometry created for energy analysis purposes can be imported to Trane Trace for load calculation purposes. This can save considerable time, as the analyst and the designer can work from the same model. This also ensures consistent geometry and load assumptions. First, open Trace, create a new project then navigate to File-> Import gbXML and navigate to the file you saved in Step 12 of the previous section.

You will then be presented with a dialog asking which features from the gbXML file you would like to import. If you've entered load information in Revit, check the boxes for the items you want to import. If you are planning to enter load information in Trace, do not check any of the items.



Now you are ready to set up templates and proceed with your load calculation as you normally would.

Items to Note

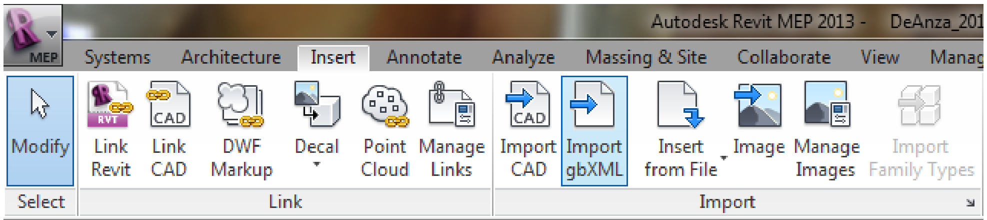
- Trace Rooms reference the Space name in Revit
- You can chose which load parameters you want to import from Revit.

Import Load Calculations from Trace Back into Revit

Once your load calculations are complete, you can import them back into Revit, which will then populate the calculated load parameters in the Space elements. This is useful for several reasons, including storing your load calculations, providing a visual quality control tool, and assisting with zonal system sizing.

In Trace, navigate to File->Export->gbXML and point to the same file you used to import into Trace. This will modify the file to include calculated airflow, heating load, and cooling load. Next, import the file into Revit. Navigate to the Insert tab, click import gbXML, and navigate to the gbXML file.

NOTE: This process relies on the naming scheme remaining unchanged. If you change the space or zone name in either Revit or Trace, the process will not work.

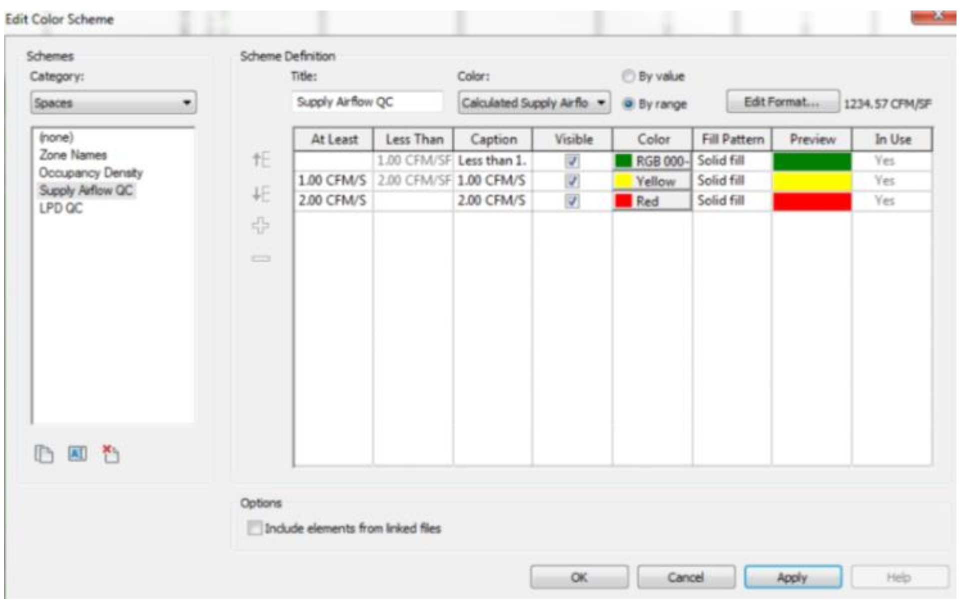


Now that you have the data in Revit, you can use it to assist with the design process.

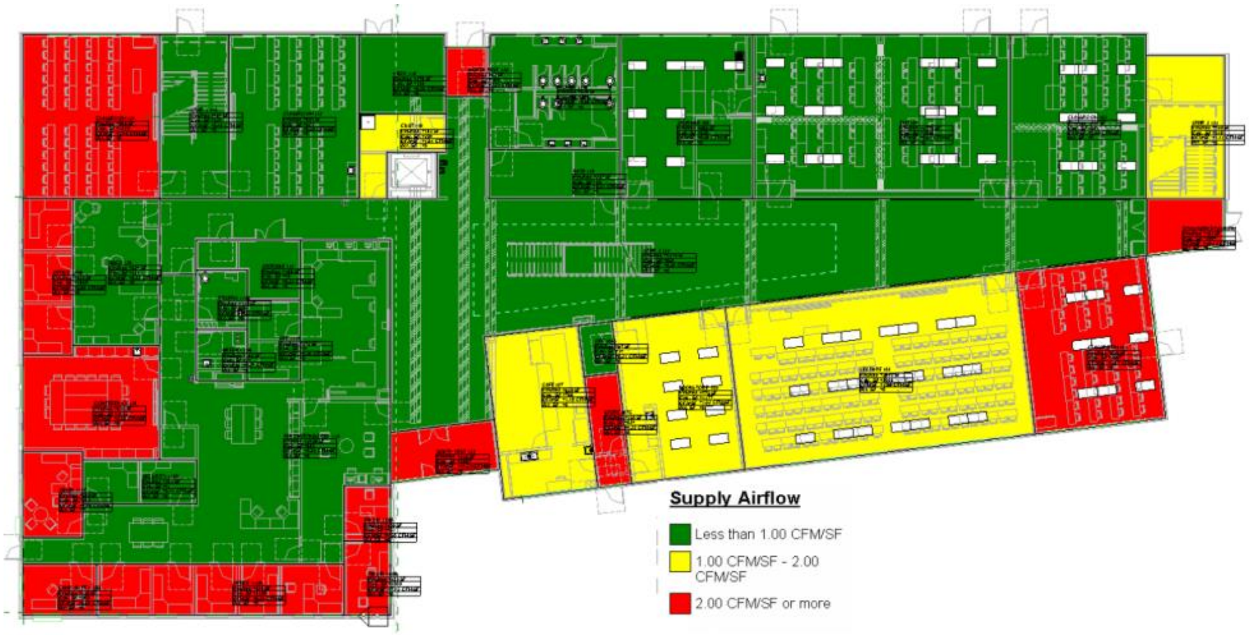
Using Loads Data from Trace in Revit

One useful way to use the loads data calculated in Trace is for a visual representation of the load. Create dedicated views under the Mechanical discipline, and then apply a Color Scheme Template which applies a color to the spaces based on CFM/ft^2. For example, for a typical office or school building, a common rule of thumb is a building average cooling airflow of 1 CFM/ft^2. Use the Color Scheme tool to give a visual indication of the distribution of airflow.

In the example shown below, I have set up three CFM/ft^2 ranges: less than 1, between 1 and 2, and greater than 2. Each has its own associated color.

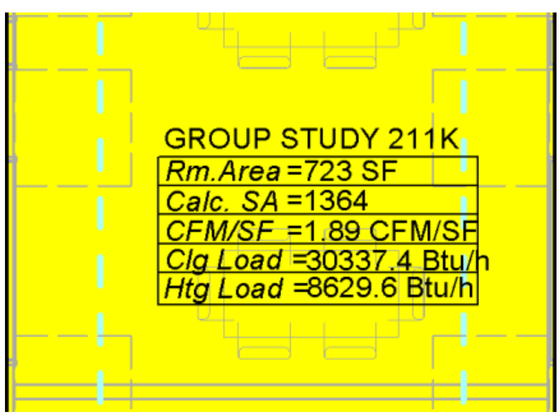


You would expect to see the highest airflows at perimeter zones (especially at the South and West) and low airflows at interior zones (especially corridors). The example below reveals this pattern.



Presenting the data in this way is very powerful for QCing the load calculation and understanding the distribution of air in the building. For example, if you noticed a red zone in the core of the building, it might indicate an error with the way the load calculation was set up. Perhaps an extra zero was accidentally added to the lighting power assumption.

You can also create custom space tags to display relevant information to assist with selection of diffusers and VAV boxes. In the example below, the space tag displays the room area, supply air volume, CFM/ft^2, and the peak heating and cooling loads.



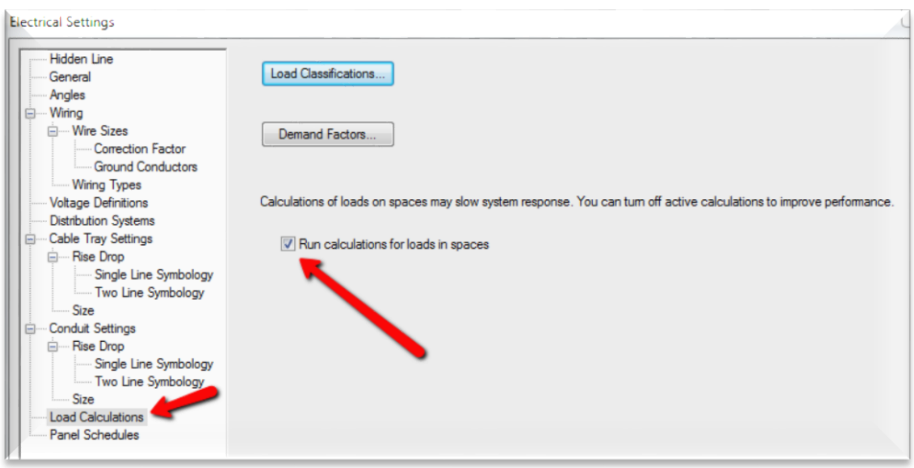
The approaches shown in this section are simply examples. There are numerous and creative ways a designer can use this data in Revit. Please be sure to share any novel ideas you have or are currently using in your design practice.

Importing a Revit Lighting Design into the Analytical Models

One key input when creating an analytical energy model is the lighting power density (LPD). The LPD (W/ft^2) is calculated by taking the total input power of the light fixtures in the space and dividing it by the floor area. A typical workflow would be to manually count the different lighting fixture types and quantities in each space in order to calculate the LPD. When a complete lighting design is available in Revit MEP (i.e. unique fixture families for each fixture type are placed in the MEP model), the LPD can be automatically calculated for each Revit space by copying and pasting the fixtures from the MEP model to the analytical model. A number of settings and elements need to be in place in order for this to work.

Revit's Electrical Settings

In order for Revit to calculate electrical loads in spaces, you must make some changes to the electrical settings. Open Revit's electrical settings dialog box and select "Load Calculations" from the selectable options. Place a check in the "Run calculations for loads in spaces" box. When making your energy model template discussed in the first section, be sure this box is checked.

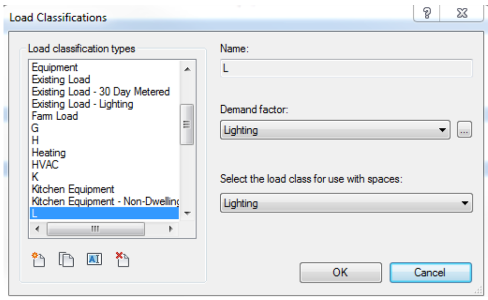


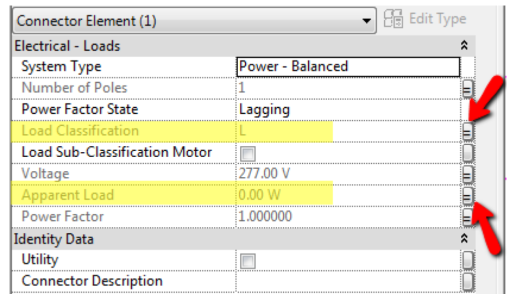
Revit Lighting fixture families

There are a few key components that need to be present on lighting fixture families in order to calculate LPD.

Unique fixture type: Each fixture type on the luminaire schedule must have a corresponding fixture type in Revit. Each fixture type must have its input watts parameter set to the corresponding scheduled value.

Electrical connectors: Every lighting fixture family must have an electrical connector. The load classification of the connector must be set to a lighting load classification. The apparent load parameter on the connector must be mapped to an apparent load parameter. The apparent load units should be set to Watts.





Lighting Fixtures must touch the Revit space: In order for the electrical load of the fixture to be read by the space element, the lighting fixture must be located inside or touching the space. This is typically not a problem when the space extents are from floor to floor, but can become an issue when room bounded ceilings are in the model.

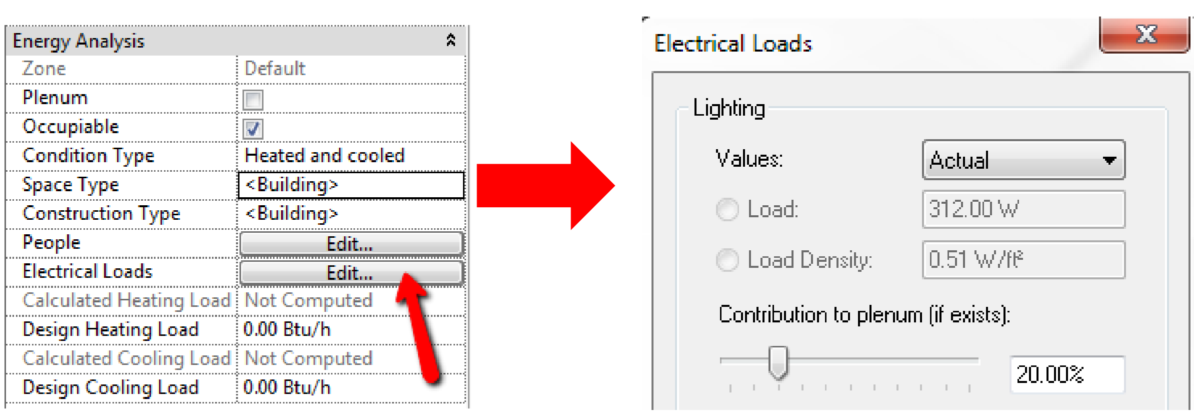
Remember to:

1. Verify the offset heights on the projects spaces.
2. Verify the heights of the lighting fixtures – If the fixture sits outside of the space boundary, move it to the space (actual location of the fixture inside the room is irrelevant for energy analysis purposes).
3. You can also use the Revit Room Calculation Point if a fixture doesn't touch the space.

Revit Space settings

Revit Spaces hold all of the useful load parameters which is exported through the gbXML file and imported into energy analysis programs. By default, Revit will calculate the LPD by the space type settings. If you want Revit to calculate the LPD by the quantity of lighting fixtures placed in the space then you need to override the default lighting load settings to Actual.

The electrical loads setting must be overridden manually for each space, but you can window select the entire building and filter out just the spaces to quickly make the needed adjustment.

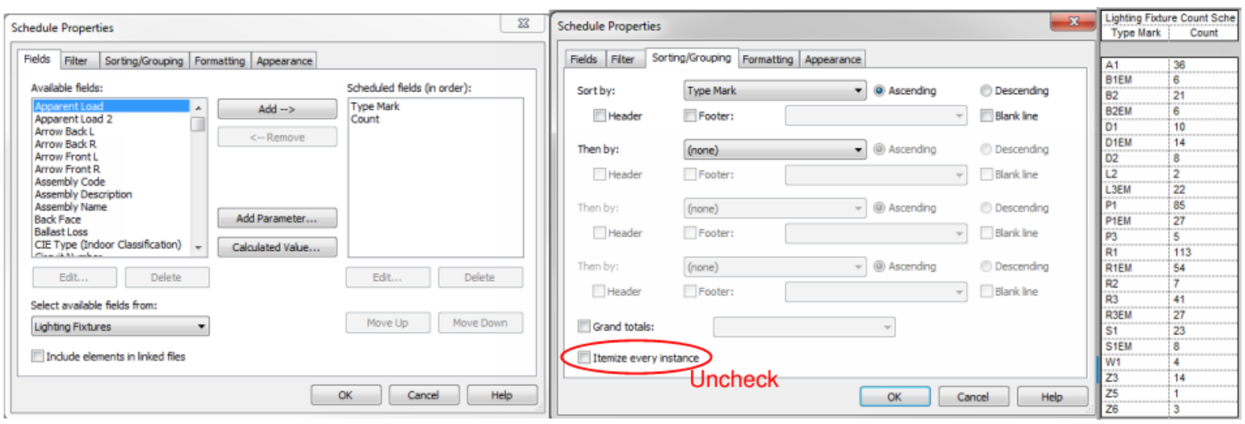


Copy and Paste Lighting Fixtures from MEP Model to Analytical Model

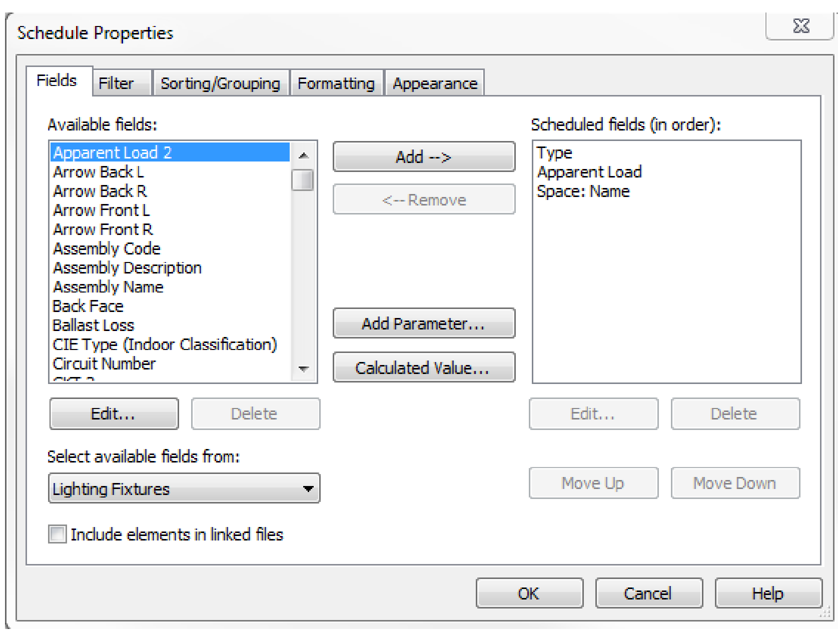
Now that we have the correct settings, copy and paste the lighting figures into the analytical model. In the MEP model, window select and copy all fixtures in each floor plan view using the filter tool. Then switch to the analytical model, navigate to the corresponding floor plan view, and use the Paste-Aligned to Current view to paste the fixtures. Note: This process needs to be done one floor at a time.

Use Revit Lighting Schedule to QC the Model

Once you have the fixtures imported, use schedules to verify that you have the same number of fixtures in both the MEP and analytical models. Create a Fixture Count Schedule in both models with the following fields and formatting features. This schedule can also be used to populate the Title 24 LTG forms



Next, make a Lighting Fixture Schedule with the schedule properties shown below.



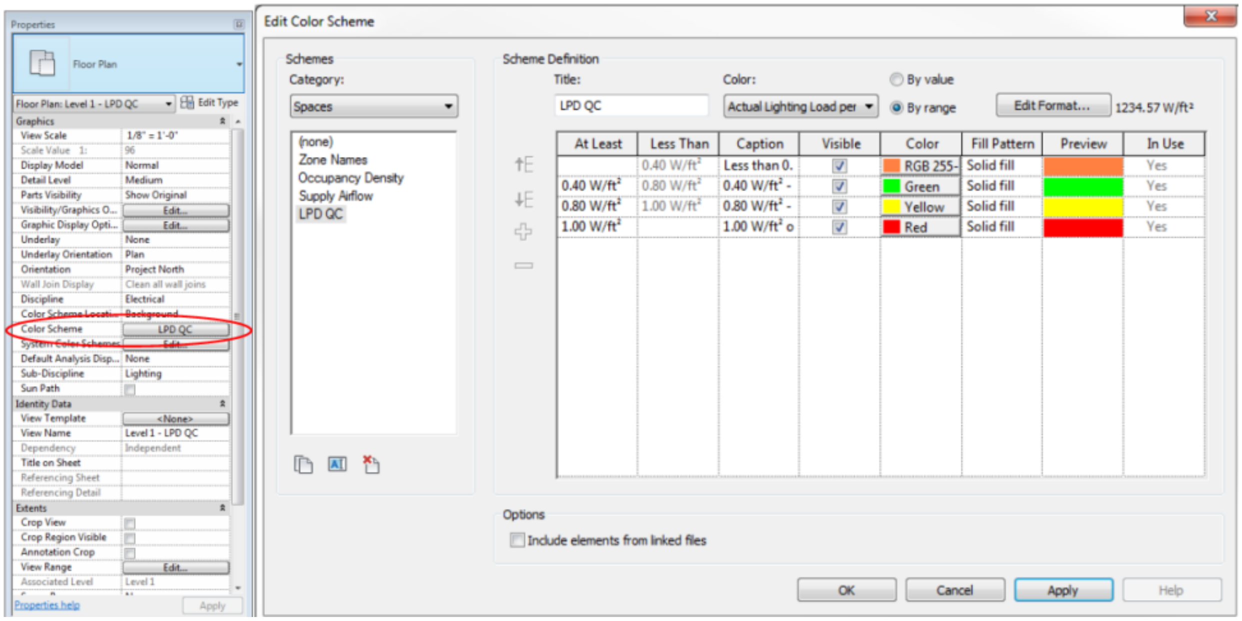
This will create a scheudle in the format shown below. This is useful for several reason. First, verify that the apparent load agrees with the luminaire schedule. Secondly, this allows you to verify that each fixture is placed in a space. If the Space:Name field is blank for any of the instances, this indicates a fixture is in the model, but not properly located in a space. Correct any issues before exporting the model.

Lighting Fixture Schedule		
Type	Apparent Load	Space: Name
1		
Z6 ext sconce	26.00 W	
Z6 ext sconce	26.00 W	
Z6 ext sconce	26.00 W	
3		
22		
B1EM	37.00 W	1st Floor Atrium
B1EM	37.00 W	1st Floor Atrium
2		
R3	18.00 W	1st Floor Atrium
R3	18.00 W	1st Floor Atrium
R3	18.00 W	1st Floor Atrium
R3	18.00 W	1st Floor Atrium
R3	18.00 W	1st Floor Atrium
R3	18.00 W	1st Floor Atrium
R3	18.00 W	1st Floor Atrium
R3	18.00 W	1st Floor Atrium
R3	18.00 W	1st Floor Atrium
R3	18.00 W	1st Floor Atrium
R3	18.00 W	1st Floor Atrium

Use Color Schemes and Custom Space Tags to QC the Lighting Design

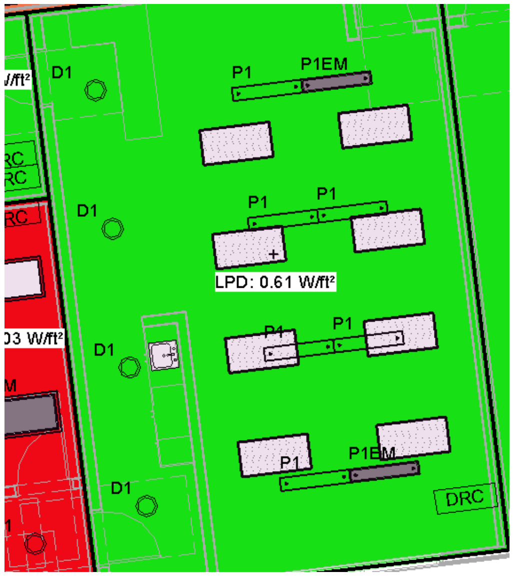
Another useful way to QC both the model and the lighting design is to apply a color scheme with LPD ranges to a set of floor plan views. Similar to the way the load calculation data is QCed from Trace, the color scheme tool will provide a visual cue that there may be an issue with the lighting design or placement of fixtures.

Create a color scheme similar to the one shown below. It uses four colors to indicate the performance of each space. Orange is displayed if the LPD is very low, green if it is in an optimum range, yellow if it is high but not unreasonable, and red if it is very high. The LPD ranges you chose will be different for each project.

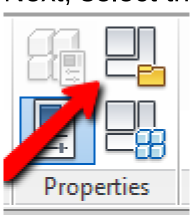


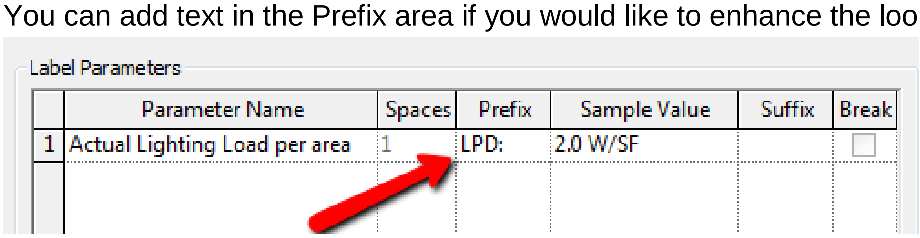
This will create a floor plan view similar to this:

Another interesting tool to use is a custom space tag which tags each space with the LPD.



Creating this space tag is very simple.

1. Start of by creating a new family and using the Generic Tag.rfa family template.
2. Next, select the Family Category and parameters button.
3. 

4. Select **Space Tags** from the list of categories. Then close dialog box.
5. From the Create Tab, select the Label button. Click into the working view to place the location of the label. The Edit label dialog box will open.
6. The units of Lighting power density is W/sf. Select the parameter "Actual Lighting Load per area" from the available fields list. Double clicking or selecting the Green arrow will move the parameter over to the label parameters list.
7. You can add text in the Prefix area if you would like to enhance the look of the tag.
8. 

9. Locate the label at the intersection of the reference planes. The intersection of the reference planes will be the insertion point of the tag.
10. Last, save the space tag family and load it into your project.

Using Revit to Create CFD Model Geometry and Boundary Conditions for Autodesk Simulation CFD

Autodesk Simulation CFD is a powerful computational fluid dynamics simulation software that is capable of using Revit as its geometry creation user interface.

General CFD Modeling Rules

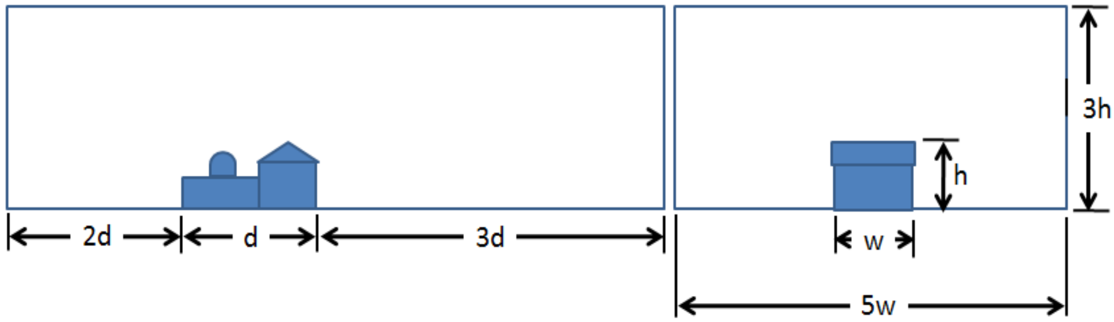
The following general rules apply when using the Revit plugin to export geometry to Simulation CFD using the:

- Keep geometry as simple as possible.
- Be careful to avoid offsets in geometry. Be sure to use the align tool for stacked and adjacent walls.
- Always export the model from a 3D view. If you export from a floor plan or elevation, only the elements in the active view will export.
- Only objects and features that are visible in the active 3D view will be exported.
- Use visibility graphics and the hide elements commands to hide unnecessary objects to reduce geometric complexity and computational time. You can also use section boxes to exclude unnecessary elements.
- Create independent boundary surfaces using independent building components. For example, if you want to analyze how a sun spot on a floor affects interior conditions, you need to create two independent floors in Revit (one that will be the warm boundary condition in the simulation and one that represents the remaining floor that is not in the sun spot).
- A space fully enclosed by at least 6 surfaces will result in an interior volume element in the CFD program. If the space is not fully bounded, you will not get an interior volume.
- To create an exterior volume around a building, create walls, roofs, and floors that surround the building with appropriate dimensions.

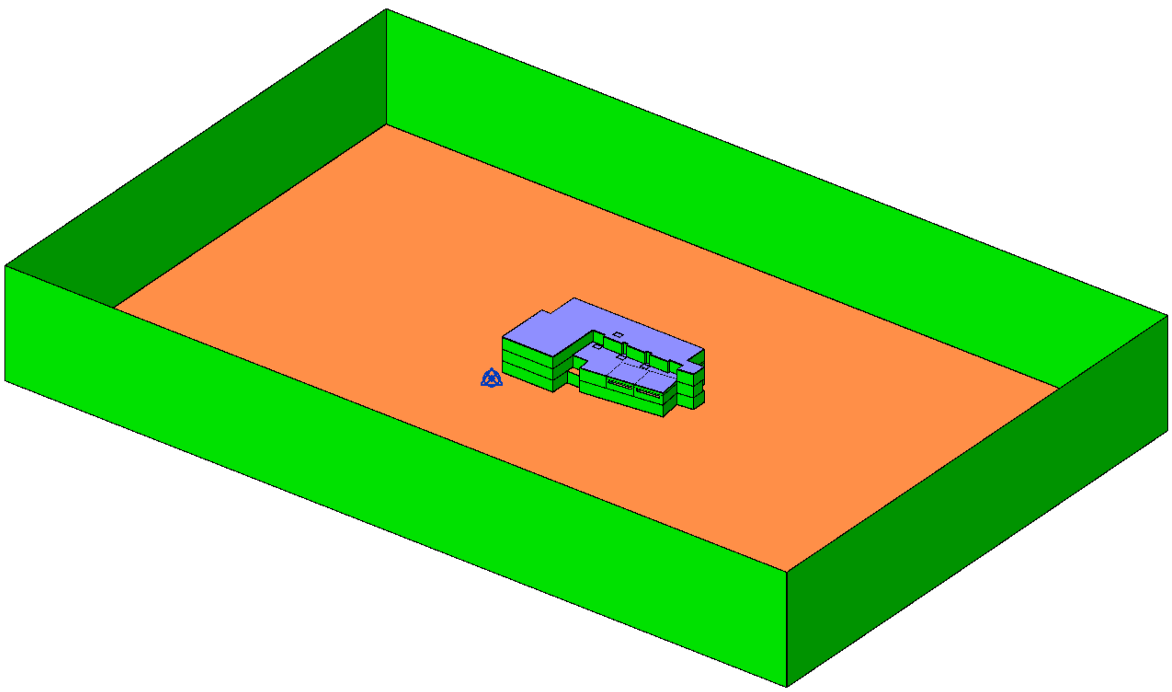
External Flow Models

External flow models are used to investigate how a given wind pattern interacts with a building. For example, you may wish to determine the wind pressure at a window for the purpose of investigating natural ventilation. In this case, we can use the same analytical model geometry we created for energy analysis.

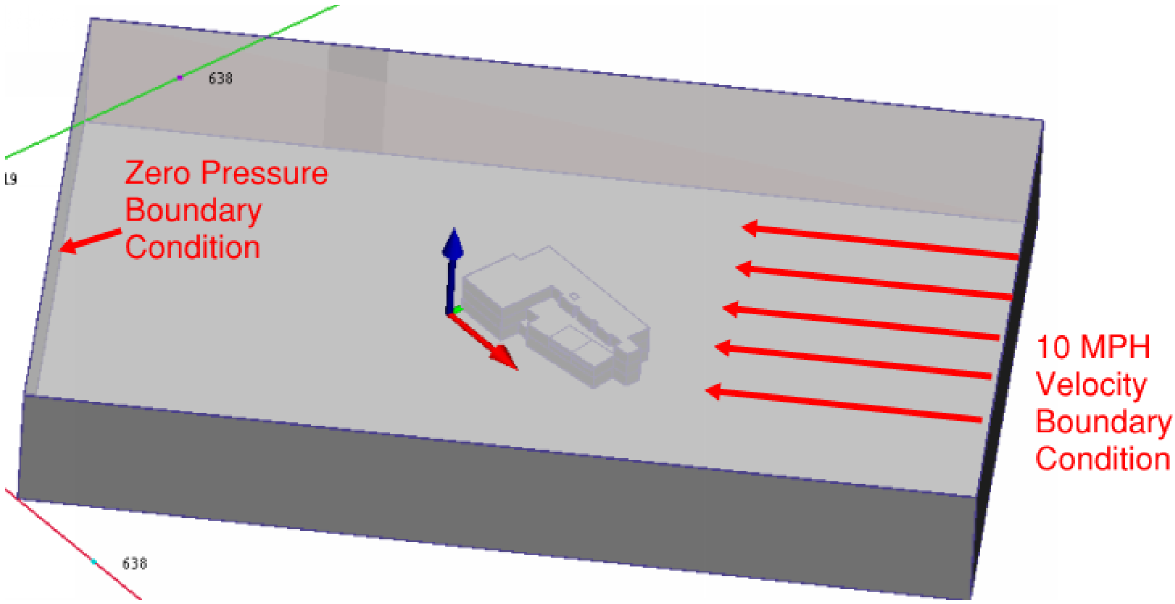
For an external flow simulation, you need to create an external volume around the building using walls, floors, and roofs according to the dimensions shown in the image below:



Here is an example of the analytical energy model geometry modified to include the walls, floor and roof that establish the extents of the external volume (the roof is hidden for demonstration purposes):



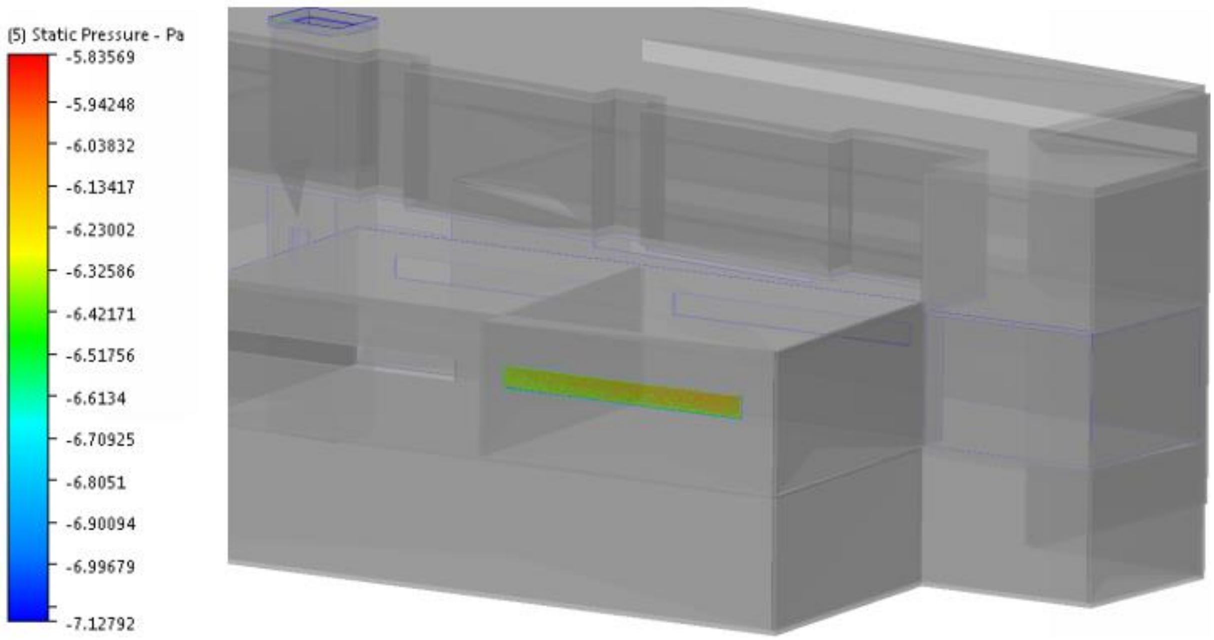
And here is an image of the geometry once it's been imported into Simulation CFD:



In order to simulate the average wind conditions, a velocity boundary condition is applied to the left-most surface of the external volume (the inlet). A zero pressure boundary condition is applied to the downwind surface to allow the "wind" to exit the solution field (the outlet). Notice that the building is situated off-axis from the external volume. This represents the average direction the wind blows throughout the year on this building's site.

Note, an air velocity boundary condition can only be simulated (accurately) as normal (perpendicular) to the one of the external volume surfaces. Slip/Symmetry boundary conditions are applied to all other surfaces of the external volume so that flow is not impeded by these surfaces. There is no need to delete any of the interior features that were created for the energy model geometry. Simply suppress those elements from the simulation once imported into Simulation CFD so they are not meshed into the flow field.

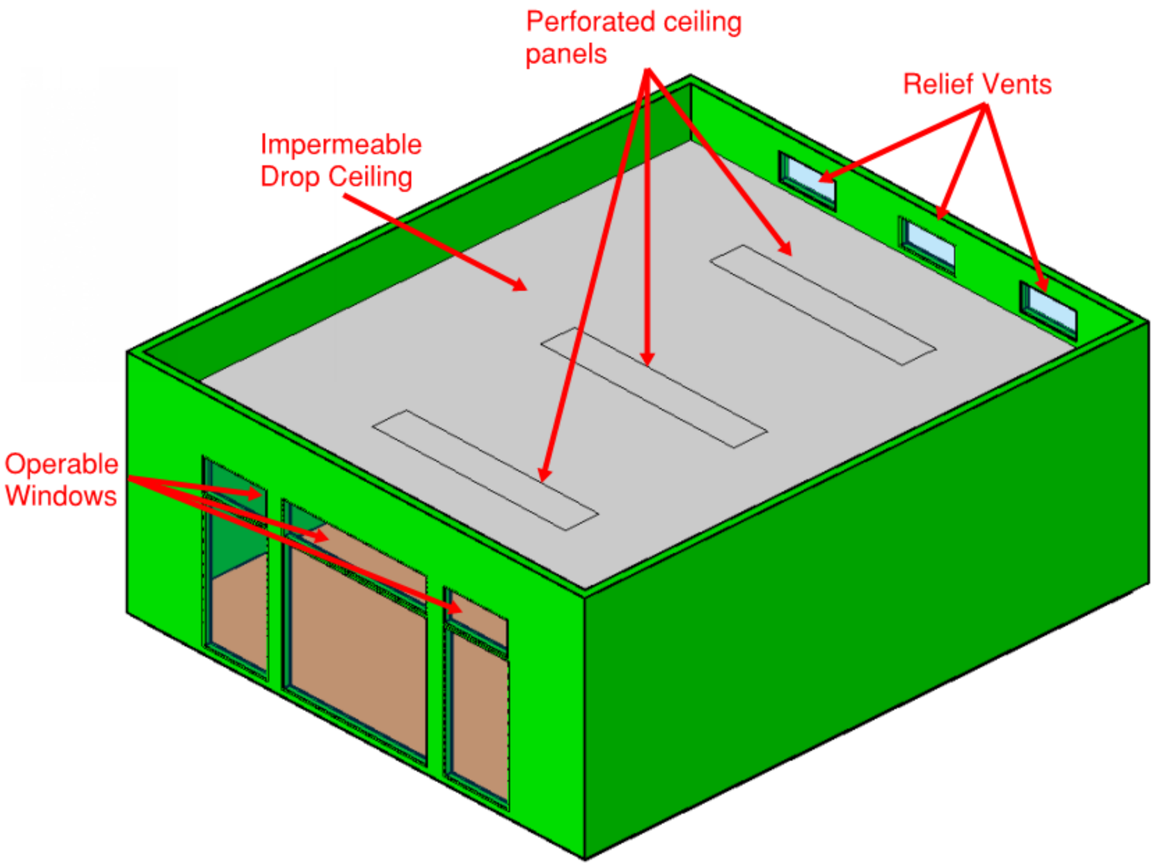
Here is an image showing the resulting static pressure on a window on the leeward side of the building:



Internal Flow Models

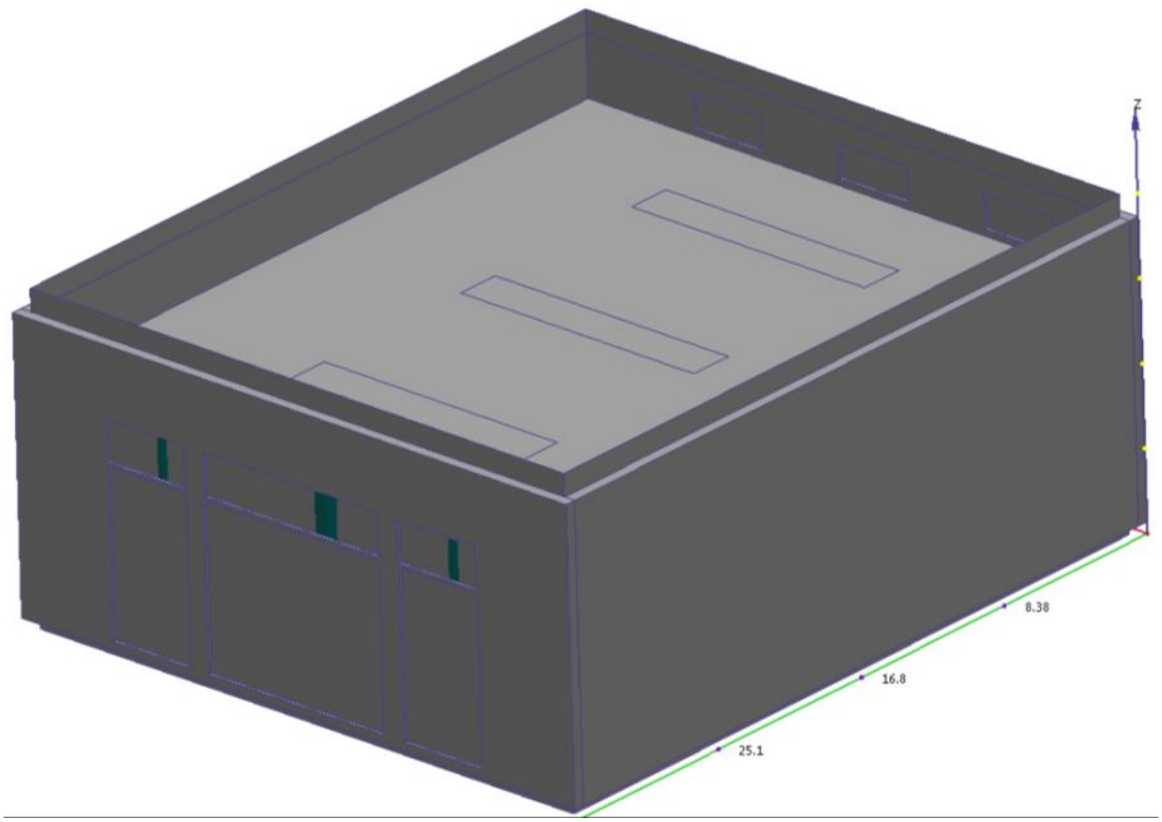
Internal flow models typically involve investigating air temperature and velocity distributions inside a specific space of the building. It is usually impractical to simulate the all the pressure relationships inside entire building so it's best practice to investigate one room at a time. In this case, we can hide all the irrelevant portions of the energy model geometry in Revit using the Hide Elements command.

Below is an example of an investigation of the performance of a naturally ventilated classroom. The natural ventilation system includes a set of three operable windows to allow outside air to enter the classroom, three perforated ceiling panels to allow air to transfer from the occupied space to the plenum, and three relief vents the relive air from the plenum into the adjacent corridor. The roof is hidden in the image below for demonstration.



The operable windows are modeled in Revit as simple window instances. It's important to have independent windows for the non-operable windows below, so that boundary conditions can be applied only to the portion that is ventable. Similarly, four separate ceiling elements have been modeled in Revit in order to differentiate between the perforated portions and the impermeable portion. The relief vents are also modeled as simple window instances.

Here is an image of the model once it has been imported into Simulation CFD, again with the roof hidden for demonstration purposes. Volumetric flow rate and temperature boundary conditions have been imposed on the operable windows to simulate outside air entering the space. The three perforated ceiling panels have been assigned as resistance material with 18% free area to simulate the perforated panels. The relief vents have a zero pressure boundary condition to serve as the outlet.



Here are some snap shots of the results of the simulation showing the air velocity and temperature distributions.