THE COOPER UNION

ALBERT NERKEN SCHOOL OF ENGINEERING

A STRUCTURAL ENGINEER'S GUIDE TO USING REVIT STRUCTURE

by

Calogero Castania

A thesis submitted in partial fulfillment

of the requirements for the degree of

Master of Engineering

May 4, 2010

Professor Cosmas Tzavelis
Advisor

The Cooper Union for the Advancement of Science and Art

THE COOPER UNION FOR THE ADVANCEMENT OF SCIENCE AND ART

ALBERT NERKEN SCHOOL OF ENGINEERING

This thesis was prepared under the direction of the Candidate's Thesis Advisor and has received approval. It was submitted to the Dean of the School of Engineering and the full Faculty, and was approved as partial fulfillment of the requirements for the degree of Master of Engineering.

____

Dean, School of Engineering – May 4, 2010

____

Professor Cosmas Tzavelis – May 4, 2010
Candidate's Thesis Advisor

ACKNOWLEDGMENT

First, I would like to acknowledge my thesis advisor, Professor Cosmas Tzavelis, for his guidance in the completion of this thesis project. It was by his suggestion that I chose this topic, and he has continued to provide me with advice and support along the way. I am grateful to him for all he has done. I would also like to thank the rest of the faculty at The Cooper Union, for the influence that they have had on me as a student throughout my many years of study at the school.

Another acknowledgment must go to my fellow structural engineers at DeSimone Consulting Engineers, including both my peers and supervisors. Aside from the input they have had on this thesis project in both direct and indirect ways, they have more importantly provided me with the atmosphere that has helped me to develop as a structural engineer. Their eagerness in imparting their vast amounts of experience and knowledge has influenced me greatly in my profession.

Finally I would like to acknowledge my family and friends, who always support me in all that I do. Thank you.

ABSTRACT

The present Master's thesis seeks to develop a better understanding of how to model a project within Revit Structure by applying structural engineering industry experience. The goal is to provide a guide to students for the process of producing a structural model with the intent to form construction documents for a project. An assessment of the software's user interface, including a comparison to the current industry and education standard software was performed. A suggested procedure for modeling a project was developed. This procedure involves performing the required modeling tasks in an order that allows the user to best understand the modeling process of a project. Suggestions were given for the procedural steps of producing a model based on structural engineering industry experience with the intention to maximize the efficiency and usefulness of the structural model in a project setting. Comments regarding the use of Revit Structure on real New York City construction projects were collected from experienced structural engineers at an established structural engineering firm. Advantages and disadvantages were considered from the perspective of the structural engineer, including the concerns and input of other clients and consultants that the structural engineer works with throughout the development of a project. Ultimately, conclusions were formed on the usefulness of the program and its place within the industry.

TABLE OF CONTENTS

LIST OF FIGURES

1. INTRODUCTION

The engineering industry has evolved over the years in the way that it creates drawings for the purpose of constructing projects. Before the dawn of the computer, drawings were drafted entirely by hand. After computers started becoming a part of every industry, programs were developed to draft architectural and engineering drawings on the computer. Computer programs such as AutoCAD by Autodesk are used to create and modify construction documents in electronic versions that can be plotted to produce physical hard copies. AutoCAD now dominates the industry for this purpose, and is a staple in every office related to the design and production of construction projects. Drafting programs, however, are limited in their ability to aid in the design and construction process.

Building information modeling (BIM) is most commonly defined as a process for creating a digital representation of the physical and functional characteristics of a building. It usually consists of a three-dimensional model that aids in the efficiency of the design and construction process. BIM is intended to incorporate the information that is necessary to see a project through the different phases of it's anticipated life. Beginning with the development of the building project, BIM can assist designers in the process of creating the architectural, structural, and mechanical, electrical, and plumbing (MEP) components of the building. These components involve a broad list of design consultants that will provide information to be incorporated into the building design. BIM can also include abilities that assist in the construction of a project such as material takeoffs and scheduling. Additionally BIM can also integrate information for some of the building's systems' processes during maintenance and service. The organization and compilation of all of this information into a single model can help streamline the process of handing over a project from the design phase to the construction phase and ultimately through the building maintenance and service phase.

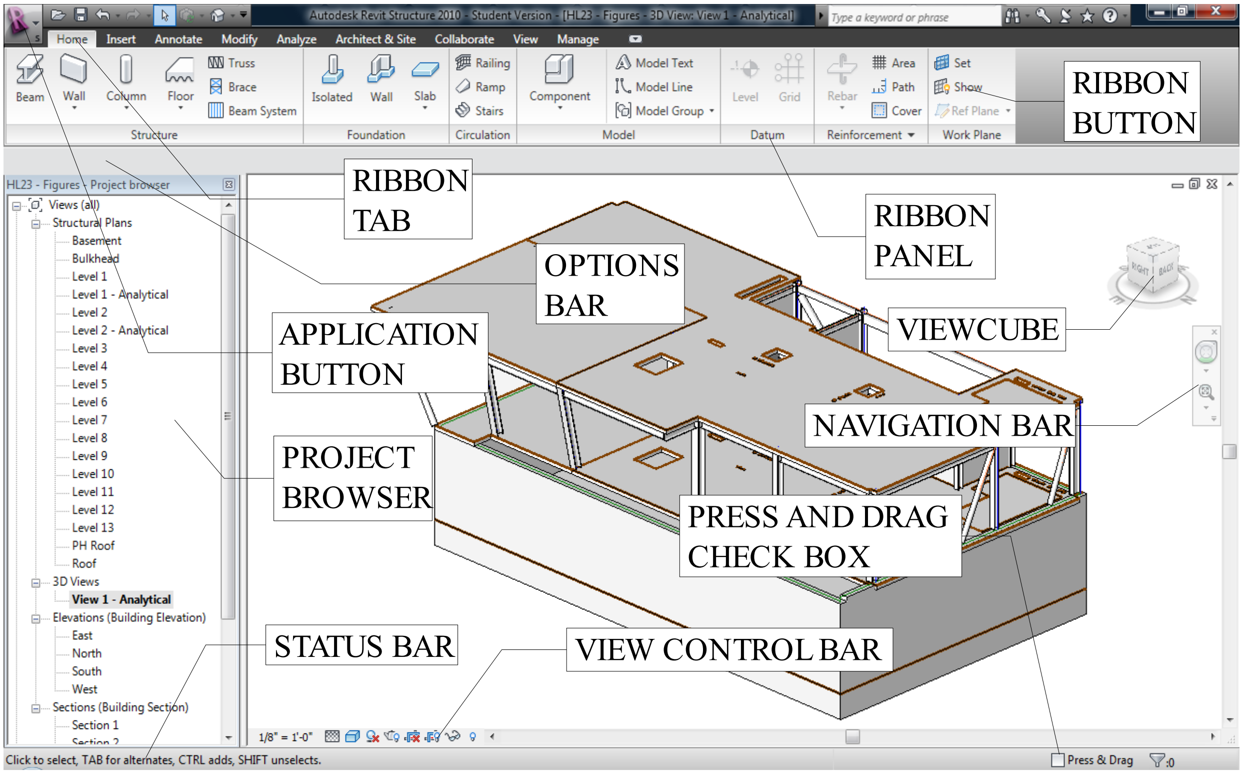
Aware of AutoCAD's limitations, Autodesk began making efforts to provide a product beyond drafting software that would aid in multiple facets of the industry. In keeping with such efforts, it purchased software called Revit from the company Revit Technology Corporation. Revit was designed for the purpose of architecture, and was developed by a team that consisted of members with a design and construction background.

Revit Structure is a BIM program from Autodesk that allows structural engineers to develop a single three-dimensional structural model to aid in the design of and produce the required documents for a building construction project. The edition of the program used for the purposes of this thesis was Revit Structure 2010.

2. THE REVIT USER INTERFACE

The Revit Structure user interface is different from previous versions of AutoCAD. There is a learning curve when trying to adjust from one's experience with AutoCAD to being able to use Revit productively. This does not mean that Revit cannot be learned within a reasonable amount of time, it just merely suggests that one cannot expect to immediately jump into using Revit comfortably because it is an Autodesk product like AutoCAD.

One major difference between the two products is that Revit is element based, whereas AutoCAD is geometry based. The significance of this in Revit is that each element that is created is more than just a drawing or graphical representation of the structure. These elements contain much sophisticated information regarding the parameters and uses of the different pieces of the structure. Another significant difference between the two programs is that with Revit all of the information for a structure is meant to be contained within one three-dimensional model. This is vastly different from AutoCAD, in which the industry standard is to have a different drawing file for each sheet of a drawing set. Therefore each floor plan is located in a different drawing file from the others, providing no connectivity amongst them. Creating a model has both distinct advantages and disadvantages that will be discussed. Other differences between the programs are seen in the way users access the tools of the program to perform different functions.



<u>Figure 1:</u> The Revit User Interface and Toolbars

Ribbon

Revit has moved away from the multiple toolbar appearance that was found in previous versions of AutoCAD to a more distinct and organized form of accessing the different tools, commands, and views. One example of this is the **Ribbon** along the top of the screen that contains the available tools broken down into a button, panel, and tab system. The tabs are general categories that the user will need to become familiar with in order to know where the different tools are located. The tabs are grouped in such a way that most of the tools that will need to be used around the same time are in the same general vicinity. The tabs are then further broken down into panels, which group individual buttons of a similar type together. There are also action tabs, which only appear when performing certain tasks. For instance, when clicking on an element button to create a particular object such as a beam, a **Place Beam** tab will appear that contains all of the modeling tools that will need to be used to properly create a beam.

Project Browser

A new feature altogether is the **Project Browser** window located on the left side of the screen. This window contains all of the views with which the user can access the model. This is where the user will switch between structural floor plans, building elevations, section cuts, framing elevations, and the 3D view, among others. There is also a list of other items that can be viewed from the **Project Browser**, including schedules, sheets, and families.

Schedules are a necessary part of practically all drawing sets, and contain specific detailed information on certain elements such columns, beams, footings, or pile caps, among others. The information is tabulated in these schedules to provide easy access to this information in one place without the need to constantly flip back and forth between numerous drawings. Sheets will become the finished product for each drawing, and will show the titleblock and other relevant project information regarding the drawing that is not displayed by the model. Sheets will be created to display the different views of the project, on which the information needed to construct the project will be provided. There are also families on the **Project Browser**, where the different element types that are loaded for each category can be displayed, and double-clicked to edit their **Type**

Properties. Not all of the available families are loaded at the start of a project because memory is consumed by loading families. To help the program run efficiently, it is best to only load the families that are required. The **Project Browser** is a very helpful feature to navigate the immense amount of views and information that will be contained within the single three-dimensional model.

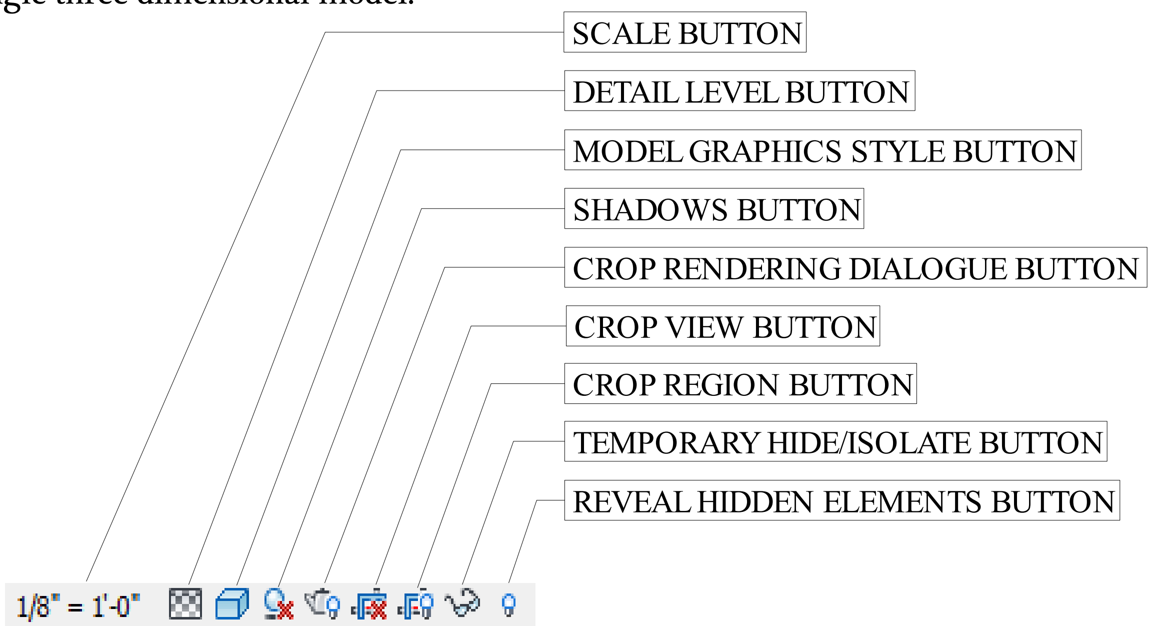


***Figure 2:* The View Control Bar Buttons**

View Control Bar

At the bottom of the viewing screen, there is a **View Control Bar**, which allows the user to adjust different options to optimize the current view. The **Scale** is also shown and can be edited on this bar. The **Detail Level** button gives the ability to adjust the amount of detail from the options of **Coarse, Medium,** or **Fine**. While in the 3D view, it is recommended to keep the **Detail Level** on **Fine** if the computing power is available to use the model and program efficiently. Although it should not have a dramatic effect, if this greatly slows down the performance of the program, then putting the **Detail Level** on **Medium** should suffice for most three-dimensional viewing needs. Having the **Detail Level** on **Coarse** is not nearly helpful as the other levels because it removes a great amount of detail from the model. On the **Coarse** setting, the user is no longer able to see structural elements such as beams, columns, or braces, as they are replaced by lines for graphical simplicity. In the plan view, however, the **Coarse** setting is recommended in order to graphically show the elements as simple lines, as this is industry standard for framing plans. The higher level detail settings in plan view can be used to display elements in different ways depending on the needed use at the time. The **Detail Level** button on the **View Control Bar** allows access for switching between these settings, and with a very fast refresh time after switching, it makes trying the different levels of detail a quick and easy process.

The **Model Graphics Style** button allows the user to change the appearance of the model graphics in the viewing screen. The different graphic options include **Wireframe**, **Hidden Line**, **Shading**, and **Shading with Edges**. The **Wireframe** setting shows the outlines of every element, and displays the remaining portion as transparent so that members behind other members are visible. **Wireframe** is suggested for use in plans views, or any view where it is necessary to see a background drawing. For instance, the **Wireframe** setting needs to be turned on when viewing a slab edge background drawing in order to facilitate tracing the slab edges for new floor elements or opening elements. It should be noted that when viewing a particular level plan view with **Wireframe**, the levels below will not be able to be seen, which offers clarity when working on a single level. The **Wireframe** setting may also prove to be useful at times in the 3D view when trying to see elements that are behind others, but generally it is more recommended to hide particular elements for this purpose. The ability to hid elements is explained further below in the portion describing the **Temporary Hide/Isolate** button.

The **Hidden Line** setting shows the same outline of elements as the **Wireframe** setting, but with **Hidden Line** the remaining portion is white as opposed to transparent. This is a useful setting when viewing an elevation or section view, since it shows only the foremost elements and not the ones behind. The **Shading** setting shows different levels of shading on the elements depending on the angle with which they are being viewed. No outlines are shown on the elements, making it more difficult to discern where one element ends and another begins. While it gives the opportunity to produce some interesting views of the model, it is most likely that the **Shading** setting will be the least used of all the model graphics styles available to the structural engineer.

The last option on the **Model Graphics Style** button is the **Shading with Lines** setting. **Shading with Lines** shows the the same shading effects as the **Shading** setting, except now with an outline given to all elements, similar to the **Hidden Lines** setting. This provides a very clear view of the structure, and is strongly recommended as the most useful setting for viewing the model in 3D. It is also recommended as an additional option for elevation views, providing a similar view to the **Hidden Line** setting, with slightly more detail by shading members shown at different angles or of different materials.

The **Shadows** button allows the users to turn shadows on or off for the model, which display how elements would block light from getting to other parts of the structure. This is not a particularly important tool for structural engineers, but it is available. It is also possible to change the location of the light source.

The **Show Rendering Dialog** button opens the **Rendering** window, where the user can find the **Render** button and specify the settings for rendering the model. Rendering the model will provide the model with a very realistic look for all of the different structural elements based on their materials. This ability requires an immense amount of computing power, however, and should only be used at times when the user does not need to perform any other actions and can leave the computer processing for an extended period of time to produce the rendering.

The **Crop View** and **Crop Region** buttons allow the user to manipulate a viewing area to display only elements that are within the viewing area boundaries. This is similar to a viewport in AutoCAD. By putting the **Crop Region** button to the **Show Crop Region** setting, the boundary of the viewing area will be displayed as an outline around all or a portion of the model. The **Show Crop Region** setting is on when the light bulb adjacent to the crop symbol on the button icon is lit, and the setting is off if the light bulb is off. Clicking the button once will switch between the on or off setting. Note that it may be necessary to zoom out in order to see the crop region viewing area outline.

Toggling the **Crop View** button on or off will remove the portion of the model outside of the crop region from the view. When the **Crop View** is off, this is graphically represented by a red X adjacent to the crop symbol on the **Crop View** button. When on, the red X is removed, showing just the crop symbol. Clicking the button once will toggle between having the setting on or off. Turning the **Crop View** on effectively creates a condition similar to a viewport in AutoCAD, by only showing the model within the editable boundary. This tool plays an important role when creating sheets that will require having different portions or views of the model on a single sheet, with each view bounded by a set amount of viewing space.

The **Temporary Hide/Isolate** button allow the user to hide or isolate a particular element or category. This ability is useful when wishing to view only one or a few elements or category types at time. For example, all of the grid lines can be viewed by themselves by clicking on one of them, and then selecting **Isolate Category** from the **Temporary Hide/Isolate** menu. This will leave only the grid lines in view, hiding all of the other categories. The same could be done for all of the columns or other structural elements. A combination of multiple categories could also be isolated. The user can also view a single element such as a grid line by itself, by selecting the **Isolate Element** option. Multiple elements or categories can be selected at once, and choosing to isolate them will affect all of the elements or all of the categories selected.

Selecting **Hide Category** does the reverse of **Isolate Category**, by removing the selected element category from the view, and leaving everything else revealed. In this same way, **Hide Element** removes a single or multiple selected elements from the view, leaving all other elements in that category, as well as all other categories. Multiple elements or categories can also be hidden at one time if they are all part of the same selection set.

Hiding an element or category is useful to view the framing plan below a floor when the slab is blocking the view. For example, the floor element can be selected, and by choosing the **Isolate Element** option, that particular floor will be removed from the view. The same could be done to view all of the framing for an entire structure, by choosing **Isolate Category** to remove all of the floors, as opposed to just the one particular floor selected. The **Temporary Hide/Isolate** button allows much more specific control over what elements are viewed than the **Wireframe** setting of the **Model Graphics Style** button.

The elements that are hidden after using the **Temporary Hide/Isolate** button can be returned to the view by selecting the **Reset Temporary Hide/Isolate** option from the **Temporary Hide/Isolate** button menu. The settings can also be permanently applied to the view by selecting **Apply Hide/Isolate to View**.   To temporarily toggle the elements or categories back into the view, the user can use the **Reveal Hidden Elements** button. This displays a cyan colored outline of all of the elements that were previously hidden. The isolated element will remain in the same graphics style that it was in when it was isolated. When the **Reveal Hidden Elements** button is on, it is represented by the light bulb in the button icon being lit, and when the button is off the light bulb is not lit.

Multiple buttons from the **View Control Bar** can be used in combination to provide different results and views. For instance, the **Detail Level** and **Model Graphics**

Style buttons can change settings for elements that were isolated or hidden using the **Temporary Hide/Isolate** button. The user should become familiar with the different viewing options to create ideal views for the different tasks that will need to be performed when creating and using a model.

Zooming and Orbiting

There are dedicated toolbars for zooming and viewing the model in different ways in both 2D and 3D views. A **Navigation Bar** on the right side of the screen displays two major buttons. While in a two-dimensional view, the top button opens the **2D Wheel** and the bottom button is a customizable zooming button. While in a three-dimensional view, the top button opens the **Full Navigation Wheel** and the bottom button remains the same customizable zooming button. In addition to the **Navigation Bar**, there is also a **ViewCube** with which to manipulate the views of the three-dimensional model.

The **2D Wheel** is available while in a plan, elevation, section or other two-dimensional view of the drawing. It contains the buttons **Zoom**, **Rewind**, **Pan**, and a drop-down menu. Clicking and holding down the **Zoom** button allows the user to zoom in or out from the model by dragging the mouse arrow. Dragging the arrow up or right will zoom in, while dragging it down or left will zoom out. The **Rewind** button allows the user to go to previous views. Clicking on the button once will return to the most recent view, while holding the button down allows the user to see multiple previous views that can be revisited by dragging the mouse arrow to the thumbnail pictures of the views. The **Pan** button allows the user to move around the view in the current zoom. This is done by clicking and holding the button down, and moving the mouse around as desired. The drop-down menu on the **2D Wheel** gives the user access to a few additional buttons, including a **Fit To Window** option, which zooms to the largest view that will fit the full model.

The **Full Navigation Wheel** for use in the three-dimensional views offers the user different options for exploring the model. It has **Zoom**, **Pan**, and **Rewind** buttons similar to the **2D Wheel**. In addition to these, however, there is also an **Orbit** button that allows the user to orbit the model around a particular pivot point. The user can place the pivot point at a location within the model by clicking and dragging the **Center** button, which is in the middle portion of the wheel. The **Walk** button allows the user to experience the model by moving along a path within it. It is, however, a relatively complicated viewing tool that is not particularly useful for structural engineers. The **Look** button allows the user to keep their current viewing location, while looking in different directions to see multiple angles of the model from the same perspective. The **Up/Down** button performs a similar ability to panning, except it is restricted to moving only up or down along a straight line.

The customizable zooming button on the bottom of the **Navigation Bar** can be changed by clicking on the drop-down menu below the button. The list of available buttons is then displayed and includes **Zoom In Region, Zoom Out (2x), Zoom To Fit, Zoom All to Fit, Zoom Sheet Size,** and **Previous Pan/Zoom**. The default is the **Zoom In**

Region button, which allows the user to create a zoom box around a particular area to achieve a close-up view of that area. The **Zoom Out (2x)** button zooms out in large bursts. The **Zoom To Fit** button zooms to fit all visible objects on the screen, and the **Zoom All To Fit** button zooms to fit all visible objects and visual aids on the screen. The **Zoom Sheet Size** button zooms to display the model within the sheet size. The **Previous Pan/Zoom** button returns to the last view that was edited by zooming or panning.

It is recommended to use the **ViewCube** for most of the three-dimensional viewing needs. The user can simulate the views is would like to achieve on the model by performing them on the cube. For instance. It provides the user with ability to orbit the model by clicking and dragging to orbit the cube. Particular points on the cube, such as one of the sides, edges, or corners can be clicked on to provide a full model view of that particular point. The **Home** button near the upper left hand corner of the cube allows the user to return to a specific default viewing angle. The user can specify this viewing angle by clicking on the arrow near the bottom right hand corner of the cube, and selecting **Set Current View As Home**. Other viewing and direction settings are available through the same arrow. The **ViewCube** provides the user with the ability to quickly and easily manipulate the views of the three-dimensional model.

Options Bar

The **Options Bar** is a toolbar that appears directly under the **Ribbon** after or during certain actions are performed in Revit. Some of these actions include selecting an element or creating a new element. The **Options Bar** provides faster ways to enter information about how to model an element, and helps define the element without the immediate need to access the properties windows of the element. This ability allows the user to create elements more quickly. The **Options Bar** also helps draw or modify certain elements in ways that streamline the modeling process. Examples of this include options such as rotating elements immediately upon placement, or defining vertical elements' level extents. There are a great multitude of options on the **Options Bar** for the various functions that need to be performed throughout the modeling process. Refer to the individual sections that follow for specific information regarding the **Options Bar** while performing particular modeling tasks.

Status Bar

A **Status Bar** that provides information to aid in the modeling process exists on the bottom of the screen. While in the middle of performing a particular command, it could give an instruction in the bottom left hand corner on the next action that needs to take place. For instance, after clicking on the **Beam** button on the **Structure** panel of the **Home** tab, the **Status Bar** suggests "click to enter beam start point". After this action has been completed, it will read "enter beam end point". Brief and direct instructions such as these are helpful to the user in understanding how certain actions are performed while modeling. Note that the **Status Bar** will only make suggestions for the next step of the current command, disregarding the need for tweaking options and settings to perform the command as desired. It is only intended as a guide through the actions, and not a thorough explanation of every step required to accurately model an element.

The **Status Bar** can also display information for a particular element by placing the mouse arrow over that element. For instance, if the user places the mouse arrow over a column element, which then outlines that element in purple, the **Status Bar** will display the column's family and type. For other elements it will display other information, such as the tag that was given to a grid line. There is also a counter on the far right side of the **Status Bar** that shows the number of elements selected at one time. Clicking on this counter opens up the **Filter** window, allowing the user to remove some of these selected elements from the selection set by category. See the portion regarding the **Filter** ability in the *Selecting Elements* subsection. The **Status Bar** should be referenced often to verify that the correct element is going to be selected, how many elements are selected, or for hints regarding the next step to complete for a command.

Press and Drag Check Box

The **Press and Drag** check box in the bottom right hand corner determines whether or not items can be clicked on and dragged by holding down the mouse on the selection. If the check box is off, then clicking and holding the mouse button on the item will only create a selection box from the point initially clicked. See the subsection below on *Selecting Elements* for more information regarding selection boxes. If the **Press and Drag** check box is on, then clicking and holding the mouse button on the item will allow the user to drag the item to a new location that is determined by when the mouse button is released.

Selecting Elements

After elements are clicked on in Revit Structure, they are highlighted in blue, signifying that they have been selected. Upon selecting an element, a modification action tab is initiated, allowing the user to modify the current selection by accessing the list of editing buttons on the ribbon.

An ability that has remained the same as it was with AutoCAD is the use of selection boxes. Selection boxes are made by clicking on a point in blank space and holding down the mouse button while moving across the screen. Note that if the **Press and Drag** box is turned off, the user can click and hold down the mouse button on blank space or an element to begin a a selection box, as mentioned above. Starting a selection box and moving to the left will select any element that is touched by the selection box. Starting a selection box and moving to the right will only select elements that are completely encompassed by the selection box. These directions are true regardless of whether the selection box is made moving up or down. Therefore, moving either to the right and up or to the right and down will perform the same function, just as moving either to the left and up or to the left and down will also perform the same function.

If the user has selected multiple items by using a selection box, there may be some elements within that selection set that the user does not want to be included. One way to remove elements from a selection set is to use the **Filter** ability. After multiple items have been selected, the **Multi-Select** action tab automatically opens up, displaying the **Filter** button on the **Filter** panel. Clicking on the **Filter** button opens up the **Filter** window. It can also be accessed by clicking on the selection counter on the far right of the **Status Bar**, as mentioned in the *Status Bar* subsection. The **Filter** window displays the different categories of items selected, with a check box adjacent to the name of each category. A count is also displayed showing the quantity of items within each category that is currently selected. The user can filter out any undesired categories by unchecking the boxes adjacent to those element categories, and only keeping the ones to remain selected. Clicking **OK** will keep only the categories with the check box turned on selected, and will deselect any categories that had the check box turned off.

While the **Filter** button allows the user to remove categories of elements from the selection set, there is also a way to deselect individual elements from a selection set. This is an ability that has not changed from AutoCAD and involves either adding items to or removing items from a set of selected items. The user can hold down the Ctrl key to click on different elements to add them to the current selection set. While the Ctrl key is being held down, the mouse arrow will have a small plus sign next to it to indicate that elements will be added to the selection set. To deselect elements from the current selection set, the user can hold down the Shift key while clicking on selected elements. While the Shift key is being held down, the mouse arrow will have a small minus sign next to it to indicate that elements will be removed from the current selection set.

A different option for selecting multiple elements at once is the **Select All Instances** ability. This is available by right-clicking on a particular type of element, and then clicking on **Select All Instances**, which automatically selects all members that are the same **Type** as the element that was clicked on. The element **Type** is one of an element's defining properties, such as a particular wide-flange beam shape or a particular concrete section. The ability to **Select All Instances** is useful to determine the quantity of elements of a particular type in the current view. It also allows the user to edit a parameter of the elements' properties all at once while the elements are selected.

Since the model in Revit is three-dimensional, there are often times when it is difficult to select elements because multiple elements are overlapping each other. It is possible, however, to rotate through alternates for overlapping elements until the desired element is displayed, and can then be clicked on to be selected. It is generally recommended to place the mouse arrow over the outline of the desired element, as in many instances this is the only place an element can be clicked. To rotate through alternate elements, press the Tab key when the mouse arrow is over the outline of the desired element. This will switch the purple outline from one element to the next, and can be repeated until the desired element is outlined in purple, and can be clicked on to be selected.

One way to verify that the correct element was selected is to view the model in 3D by accessing the 3D views on the **Project Browser**. By manipulating the view until the desired area is displayed, the user can be certain that the desired element was selected. This is generally a recommended habit to have after not only selecting elements, but also after modeling elements in a project. After creating an element or group of elements, it is strongly suggested that these elements are examined in the 3D view to be sure that the structure was created as desired. This same procedure should be followed when copying or moving elements, or just modifying elements in general. Note that datum elements such as levels and grid lines, however, are not viewable in 3D. The 3D view is an extremely useful tool to verify that a structural element snapped to the correct location and at the correct elevation.

While it is often difficult to determine if an element was modeled properly in the two-dimensional views, there are a few ways to more clearly understand how the element was modeled from within these views. One way to do this is by simply selecting the element, which displays some of the elements characteristics. An example of this is when viewing framing elements in a structural floor plan. Lines that do not extend to the full length of beams are used to represent the actual shapes for drawing clarity, making it difficult to determine if beams have snapped to the correct locations. Aside from viewing the model in 3D to verify the placement of a beam, the user can click on the beam to select it for more information. This will display two solid blue dots at either end of the beam, noting the actual location of the start and end point of the selected beam. The same is true for columns, walls, braces and other elements, depending on the current view.

3. THE LIFE CYCLE AND DESIGN PROCESS OF A BUILDING

Before attempting the modeling procedure for a project, a user should understand a building's life cycle, and particularly the general design process. The life cycle of a building is typically considered to include its design, construction, use, maintenance, and demolition. While this thesis is geared towards the modeling of a project for the design and construction portions of the building life cycle, BIM programs can be useful throughout the entire building life cycle, as previously mentioned. The structural design engineer is generally only involved in the design and construction of the project, and rarely has a role beyond these phases. The use and maintenance portions of a building begin after it has been constructed, and is occupied for the purpose that it was designed. Structural engineers may only play a role during these phases if additions to the structure or an evaluation of the structure is requested. The demolition stage includes the decommissioning of a building that will no longer be used. The building and its systems may be dismantled or destroyed as required. A demolition specialized structural engineer may be involved in the ultimate demolition of a project at the end of its life, but it is unlikely that the design engineer will be a part of the building's demolition unless it is only a renovation and not truly a demolition.

The design process includes the conception of the ideas that will shape the project through to the final construction documents that will be used to build the project. The first stage of the design process is the Schematic Design phase, in which information about the project is discussed with the client and design team in order to meet the client's desired result. The structural engineer will begin to address the needs of the structure by providing multiple options for the building structural system. This often includes considering the different material options available for a project, such as reinforced concrete or structural steel. Different design concepts are explored in an attempt to find the best direction for the project. The structural engineer may also try to identify the project's specific design challenges in this phase. Raising the particular challenges of a project early in the design process will force the entire design team and client to acknowledge the existence of the challenges and the resulting effects on the complexity of the project. Typically basic floor plans will be produced during this phase for initial pricing comparisons. Ultimately the goal of the Schematic Design phase is to figure out the path that the design will take, and which of the studied schemes will be the direction to follow for the project.

The next stage of the design process is the Design Development phase. During this phase, the structural engineer will begin to refine the design of the project more thoroughly as more information becomes known. As the architectural and MEP design progresses, the structural engineer can begin to assess and accommodate the structural requirements of the project as well. The structural engineer will devise a working system for the different aspects of the structure, including how the structure will resist the gravity and lateral loads it will experience to both its superstructure and foundation. It is suggested that as many design decisions as possible should be made during this phase in order to prepare for the next design phase to produce construction documents.

The final stage of the design process is the Construction Documents phase. By the end of this phase, the completed design of the structure needs to be finalized and displayed on the documents that will be used to construct the structure. By the end of this phase, all of the details of the project for every member need to be designed and identified. It is common for these documents to be produced and issued in portions, such as fifty percent construction documents or one-hundred percent construction documents. Releasing the documents in portions helps meet specific goals for the design team, and also allows accurate pricing of different components of the project. The construction documents will be used to acquire bids from contractors, and allows them to obtain cost estimates from the subcontractors that will provide the required trades necessary to construct the project.

After the design of project is completed, the structural engineer will usually take part in the construction administration portion of a project. During this process, the structural engineer will review construction submittals and be available to answer any questions that are raised during the construction of the project. The construction submittals include shop drawings that are produced to detail the exact types, quantities, and sizes of the structural material to be used during construction. Questions that are raised by the construction managers will include requests for information (RFIs) and other formal methods by which clarification can be requested for the issues that arise during construction. Field visits and inspections may also be included as part of the structural engineer's scope of services during the construction administration process.

The structural engineer's role in this phase is to oversee the implementation of their design by providing approvals and clarification for the structural aspects of a project during construction. The structural engineer must be able to resolve any problems relating to the structure that develop while the project is being constructed. A BIM program can assist in the design and construction administration process by providing a detailed model that will aid in understanding and resolving the intricacies of a project.

While the above summary is a vague generalization of the typical design and construction phases often encountered by a structural engineer on a project, most projects will present different challenges and situations that will inevitably affect this process. Different schedule constraints or involved parties may have small or dramatic effects on how the project will progress and will ultimately be designed.

4. THE RECOMMENDED MODELING PROCEDURE

The procedure for developing a model begins with suggestions for how to start the project within the program. Linking background drawings or a reference model for use as a guide when modeling will allow the structural engineer to determine the geometry of the structure and the structural requirements of the project. Levels form the vertical datum to which the structural elements will be referenced. Grid lines will be referenced as the horizontal datum for the structural elements.

With background drawings or a reference model, levels, and a grid system, the user can begin modeling the structural elements. The vertical column and wall elements should be modeled first as these will support the remaining structure for the floors. Next, the beams and braces should be modeled to provide support for the structural floor. With the framing members in place, the structural floor can be modeled. Openings in the structural floor, such as penetrations for architectural, mechanical, electrical, plumbing or logistical construction requirements can be modeled after the structural floor. Finally, the foundation of the structure is modeled last. The reason for modeling the foundation last is that the superstructure above it needs to have been designed to a working level in order to develop the necessary design requirements for the foundation.

5. STARTING A NEW PROJECT

The first step of creating a model is starting a new project, and the user begins by clicking on the **Application** button, placing the mouse arrow over the expansion arrow adjacent to **New**, and selecting **Project**. This opens up the **New Project** window, which allows the user to browse for the location of the default structural template file. The default structural template file has the **Project Browser** already set up with recommended views such as **Structural Plans**, **3D Views**, and **Elevations**. It is also possible to start a blank project without the use of any template files by selecting **None** under the **Template File** section of the window. It is recommended at first, however, to choose the default structural template file to begin a project. The user can also customize and save a template file that contains the desired views and elements for use on future projects.

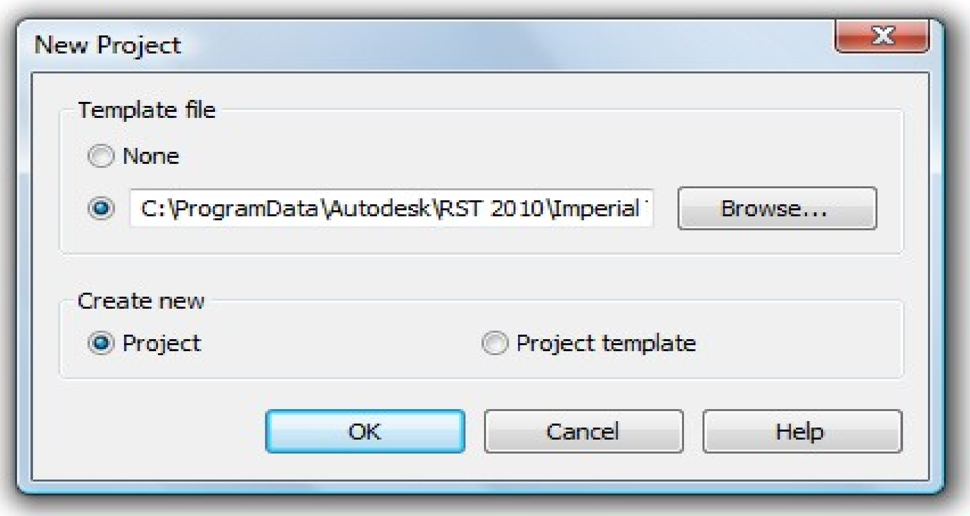


***Figure 3:** The New Project Window for Starting a New Project*

After making the desired **Template File** selection, the user can choose to create a new **Project** or **Project Template**, by clicking on the appropriate selection. Selecting **Project** allows the user to create a new project file in which to begin a model. Selecting **Project Template** instead gives only the ability to create a template file. It is suggested to select **Project** as this gives the flexibility to save the file as either a project or as a template. If **Project Template** is selected, the user will only be able to save the file as a template file, and not as a project file. Therefore to begin, select new **Project** and then click **OK**.

6. LEVELS

Adding the required levels of structure to a model is an extremely important step in development, as these levels will be used and referenced in many ways during the modeling process. The levels are reference planes to which the structure will be bound. Therefore, if floor to floor heights or floor elevations change, the structure will automatically accommodate these changes.

Like many different aspects of a project, this is an area that may see changes throughout the life of a project. An architect may go through multiple iterations of a building layout, at times requiring the addition or deletion of levels. As the architect determines the different uses of the spaces, the sizes and layouts of the spaces, and how the different spaces will stack from floor to floor, the structural engineer must accommodate the design requirements.

This may mean that some structure will be at an intermediate height between two main floor levels. This is not uncommon if there is a mezzanine level, a loft level, or an area with a particularly large slab drop. A new level in Revit can be created to aid in the modeling of this intermediate level. The levels of a structure will often need to be modified in particular ways. Examples of these modifications include the possibility that floors will be renamed and floor elevations will be changed. Levels should be created or modified in the elevation view.

Adding A New Level

From the elevation view, the user can create a new level by clicking the **Level** button on the **Datum** panel of the **Home** tab. The **Element Types** for a level provide a few different options for how the level lines will be displayed. The **1/4" Head** selection will have a grid line pattern of one-quarter inch scale with a level head at one end of the line. The **Story Level** selection uses a solid line instead of a grid line pattern, with a level head at one end of the line. The **Story Level – no head** selection uses a solid line without a level head at the end of the level line. Drawing the new level can be done by using either the **Line** or **Pick Lines** tools on the **Draw** panel of the **Place Level** tab. Using the **Line** tool, the user can draw a line to create the level. This is done by clicking a start point and an end point to draw the line.

The **Options Bar** while creating a new level with the **Line** tool shows a few options including a **Make Plan View** check box, a **Plan View Types** button, and an **Offset** text box. The **Make Plan View** check box determines whether or not a corresponding plan view will be created along with the new level created in elevation view. It is strongly recommended to create the plan view that corresponds with the new level in the elevation view. This plan view is necessary to model structure and view the new level in plan. The **Plan View Types** button should not need much use with Revit Structure, as the default and only option is for the **Structural Plan** view. This plan view type is the only type that a structural engineer would be concerned with creating in their model. The **Offset** text box allows the user to specify a dimension that the new level will be offset from the line that the user draws. The level will be created either above or below the drawn line by entering the dimension as a positive or negative value.

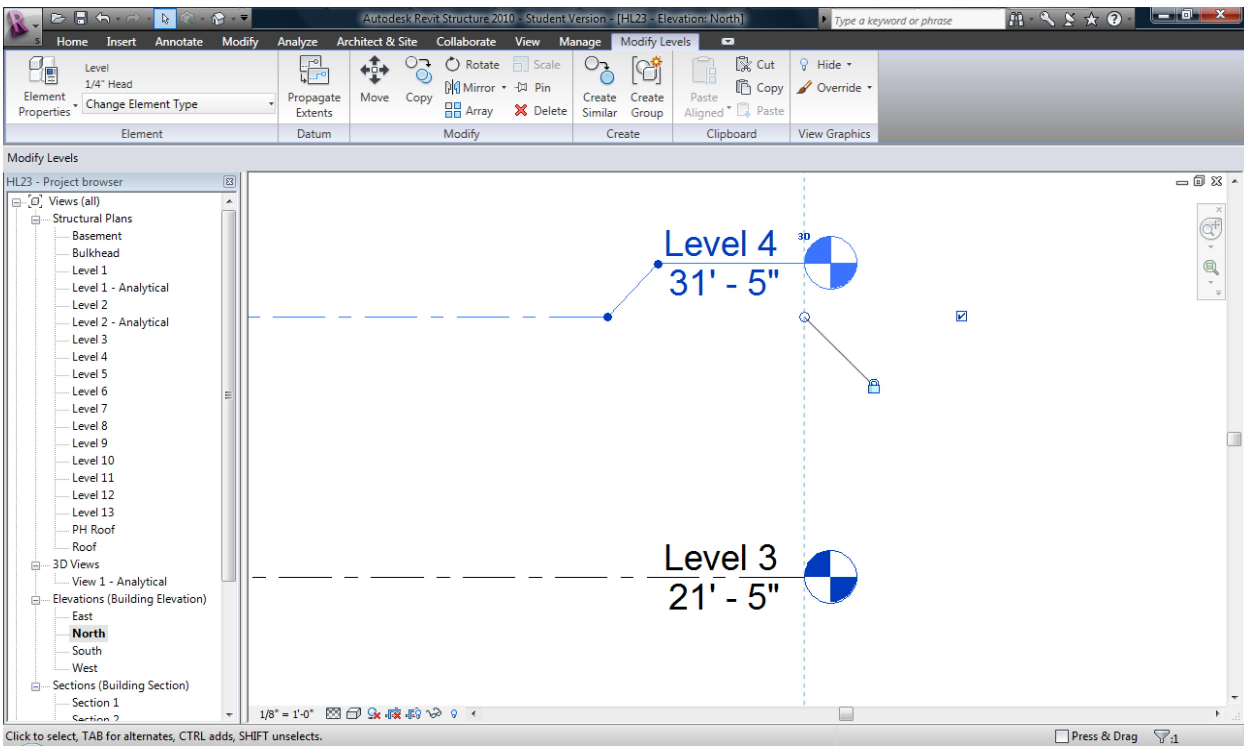
To create a new level using the **Pick Lines** tool, the user can take an existing level line and draw a new parallel line with a specified offset to the selected existing level. After clicking on the **Pick Lines** tool on the **Draw** panel of the **Place Level** tab, the user needs to enter an offset dimension with which to create the new level. Note that while using the **Pick Lines** tool, the user will not be able to select a line until a non-zero dimension is specified. The **Options Bar** contains the same choices as when creating a new level using the **Line** tool, and the **Offset** text box works in the same way. Once the offset is entered, the user can click on the existing level with which to use to create a new level. When this has been done, a new level will be created with an elevation difference from the existing level elevation that matches the offset dimension that was entered. Using the **Pick Lines** tool is the recommended procedure to create new levels for a structure.

After the level lines have been created, the user has the ability to change the length of the level lines by clicking on the start point or end point of the new level, and dragging that point to the desired new length. Note that the default setting is that doing so will also perform the same operation to the other level lines that line up with this level line. This is due to a locking ability that allows the user to keep the ends of the locked levels in line with each other. When a level line is selected, a lock symbol will be shown at each end of the line. When the symbol is in the locked position, the locking ability is enabled, and when the symbol is in the unlocked position, the locking ability is disabled for that particular level line. The length of the individual level lines can also be modified later as required, by selecting the level line and then clicking and dragging the blue circle at the start or end point of the line. Press the Esc key to complete the task and view the new level. Once the level is created, it is automatically given a name tag and an elevation tag. The name tag depends on the order in which the level line was drawn. The elevation tag will show the level elevation value, based on the height at which the line was drawn.

The level name and elevation tags can be hidden by clicking on the check box that appears next to the start or end of the level line. When the check box is filled, then the tag will appear. The check boxes on both sides of the level line can be filled to reveal the level tag on both sides. When there is no level tag being shown, the check box will just be an empty square.

The user can also create an elbow in the level line if there are two level lines in close proximity of each other. This helps avoid the level tags from overlapping and being unclear. Click on the diagonal "z" or zigzag line near the end of the level line to create the elbow and kink the level line tag away from the other level line. Solid blue dots will appear at the points of the elbow when the level line is selected, and the lines of the elbow can be modified as desired by dragging these dots.

Modifications made to level lines in one elevation view can be applied to other similar elevation views. For instance, if changes are made to the level lines of the North elevation view, these same changes can be made to the South elevation view without the need to perform the same editing actions. After selecting the modified level lines, the user can click on the **Propagate Extents** button on the **Datum** panel of the **Modify Levels** tab. When the **Propagate Datum Extents** window opens, the user can add check marks to the other views to include the modifications. When the user clicks **OK**, the modifications made to the selected level lines will be applied to the marked views of the **Propagate Datum Extents** window.



<u>Figure 4:</u> Level Line Tags with Name and Elevation, Level 4 Shown with an Elbow

Modifying A Level Name

When modifying a level name, the user should zoom in on the level tag to be modified while in the elevation view. Double-clicking on the name of the level will allow the user to modify it in a text box that appears. After entering the new name, Revit will ask to rename the corresponding views. It is strongly recommended to answer **Yes**, as this will rename the structural plan view that corresponds to that level, matching it with the new name given in the elevation view.

It is always recommended to use the same naming convention when referring to the same area or level of a structure. This goes not only for the structural engineer's own drawings between plans, elevations, sections and details, but also between the structural engineer's drawings and the other design consultants' drawings on a project. While different names for the same area may be understood between the design consultants, it must be kept in mind that others who are initially unfamiliar with the project will also need to review and understand the drawings. For instance the contractor, subcontractors, and many others involved in the construction of the structure will need to be able to move from one design consultant's drawing to the next for the same area or level. If the same level is named differently from the architect's drawing to the structural engineer's drawing, then this can create confusion and waste time. Ideally the same standard should be applied for when subcontractors produce shop drawings that need to be reviewed by the design consultants. From the perspective of a design consultant, however, one can only have control over their own drawings, and can only make suggestions or requests regarding the procedure for others.

Modifying A Level Elevation

When modifying a level's elevation, the user should zoom in on the existing level elevation to be modified while in the elevation view. The elevation of a level can be modified in the same way as its name. Double-clicking on the elevation of a floor level will make a text box appear, in which the user can enter the floor elevation. This is one example where having a single model is a true advantage. Since Revit has the ability to communicate information between all of the different drawings of one model, when a level's elevation is changed in elevation view, this change is reflected in all other aspects of the model. Specifically, all elements that are tied to that level, such as slabs, beams, columns and any other structural items that use the level as a reference, will be adjusted accordingly. This saves a significant amount of drafting time, and also helps safeguard the user from forgetting to make a corresponding change. Without this ability within Revit, the user could easily forget to make one of the many required modifications that follow when changing a significant design parameter such as a level elevation.

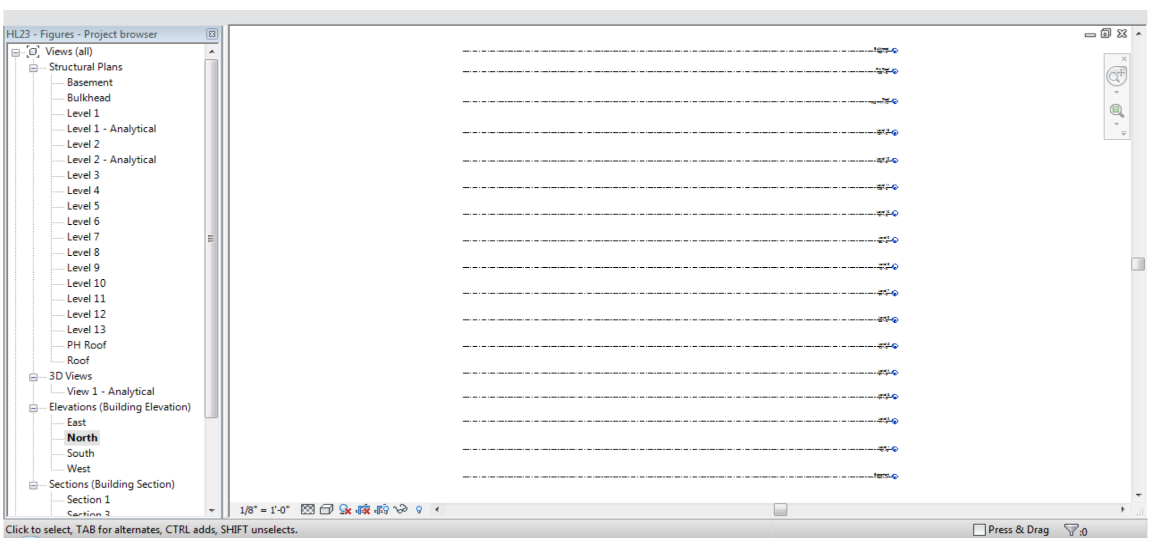


__Figure 5:__ Created Levels For a Structure

7. LINKING OR IMPORTING A DRAWING FILE AND/OR REFERENCE MODEL

Industry experience has shown that the overwhelming majority of a project's electronic drawings are started by the architect, and then are distributed to the other design consultants to apply their portion of the design. In the case of structural engineers, this typically means receiving basic architectural floor plans early on during the schematic phase of a project, and then using these floor plans as a background to overlay the locations of the structural components of the project. As more architectural drawings are created where structure needs to be included, the structural engineer will likely use these drawings as backgrounds, creating their own drawings by referencing these backgrounds. It is important to note the terminology used in Revit for placing drawings into a model.

Revit uses the term **Link** to refer to the process of taking a background drawing or another model and placing it into the current model for reference. For background drawings, this is a similar ability to the external referencing process in AutoCAD. One drawing file, such as an architectural background, can be placed into another drawing within AutoCAD for reference or tracing. In a similar way, that same drawing can be placed into a view within Revit to serve the same purpose. If the drawing is updated in AutoCAD, the most current version will be seen in Revit every time it is refreshed by opening the project or reloading the link. This gives the user the ability to still draft certain parts of a project in AutoCAD, such as details, and then link them to Revit so that the details can be updated in Revit even if the changes are occurring in AutoCAD.

It is important to note that just like external references in AutoCAD, the drawings that are linked cannot be edited from within Revit. To place a drawing into Revit that can be edited, one needs to import that drawing.

The term **Import** in Revit means that the file will be embedded into the model. This is similar to copying a drawing from one file and pasting it into another drawing file in AutoCAD. There will be no connection between the imported drawing and the original drawing after it is imported. If all drafting is planned to be done in Revit, then files can be imported from AutoCAD, and edited within Revit, no longer having a connection to the original AutoCAD drawing. While linking a drawing gives more flexibility to the user in the ability to proceed to draft in either Revit or AutoCAD, this is not without consequence. If a drawing is linked to Revit, it can only be edited in Revit if it is redrawn from within Revit by tracing the linked reference drawing. Otherwise, the detail will need to be updated in AutoCAD as mentioned above. Importing a drawing into Revit, however, allows the user to edit and manipulate the actual imported drawing after exploding it.

It is recommended to link drawings that are going to be used as backgrounds, just as background drawings would be externally referenced into a different AutoCAD drawing. Importing drawings can be useful if taking existing details from AutoCAD to permanently place within Revit, where they can be edited and manipulated. The determination needs to be made whether details will be edited from within Revit or AutoCAD. If it is from within Revit, then any existing details can be imported from

AutoCAD. If it is from within AutoCAD, then the details only need to be linked to the Revit model.

Linking a Revit model, such as an architectural model, can be done for the same purposes as linking an architectural drawing. The architectural model will be displayed for reference, and the user can create structural elements to meet the architectural design. For instance, datum elements such as levels and grid lines from the architectural model will be displayed and can be used to produce the datum elements for the structural model. Similarly, the user can access other information such as slab edges that are shown within the architectural model to create the corresponding structural elements. Like the external reference for a drawing file in AutoCAD, the linked model will be automatically updated to match changes made to the original model file whenever the user reloads the model.

Linking or Importing a CAD Drawing

When linking or importing a drawing, the user should first go to the appropriate view in which the drawing will be linked or imported. If the drawing to be placed in Revit is a floor plan background, then the corresponding structural floor plan should be viewed. To link the drawing file, click on the **Insert** tab of the ribbon, followed by the **Link CAD** button on the **Link** panel. This opens the **Link CAD Formats** window, where the user can select the desired drawing file to link, and can enter the information for how this drawing will be linked. Similarly, when importing a drawing into Revit, the user can click on the **Import CAD** button on the **Import** panel of the **Insert** tab. Now the desired file to import can be selected from the **Import CAD Formats** window, and the user can enter the information for how this drawing will be imported. The parameters for how the drawing will be linked or imported include **Colors**, **Layers**, **Import Units**, and **Positioning**.

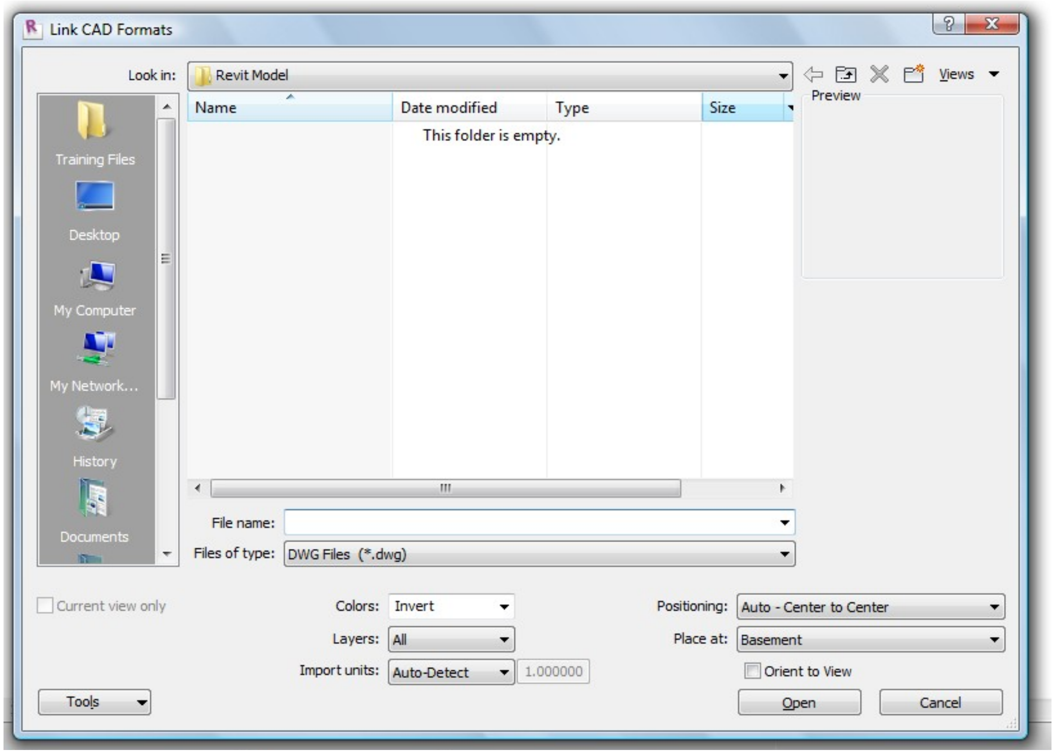


***Figure 6*: The Link and Import CAD Window Options**

The options for the drawing **Colors** include **Invert**, **Preserve**, and **Black and White**. From personal experience using architectural backgrounds with both Revit and AutoCAD, it is ideal for the background drawing to be monotone, so selecting **Black and White** is recommended as the best option. **Invert** can be used if it is desired to have the drawing linked or imported in color. This inverts the original colors so that they can be seen more easily in Revit. The contrasting white background in Revit does not provide the same viewing surface as the black background in AutoCAD, and therefore it is not recommended to use the same colors in the drawing. Selecting **Preserve** will keep the original colors from the linked or imported file.

Below **Colors**, there is the **Layers** option, which allows the user to choose whether the linked or imported drawing will include all of the layers that are in the drawing file, only the visible layers, or only specifically chosen layers. This is done by selecting the appropriate choice of **All**, **Visible**, or **Specify**, respectively. Linking or importing all of the layers gives the user the ability to turn specific layers on and off as desired, and is therefore the safest selection. The layers can be manipulated by going to the **View** tab, and clicking on the **Visibility/Graphics** button of the **Graphics** panel. Once the **Visibility/Graphics Overrides** window opens for the current floor, the user can click on the **Imported Categories** tab. The linked or imported drawing file is shown here, and can be expanded to display all of the file's layers. These layers can be turned on or off by checking or unchecking the box adjacent to the layer name. The linked drawing can also be set to appear lighter in color by enabling the **Halftone** check box at the end of the row for the drawing file.

The manipulation of layers is a very useful tool if the architectural drawing is cluttered with multiple layers that do not need to be viewed all at once. When an architectural drawing is unclear, the user should try to determine which layers are particularly important for structural reference and which may not be necessary.

Architectural layers that usually prove to be important are ones that include the grid lines, slab edges, slab openings, column/wall layouts, and stair/elevator locations. While these are a few examples of items to typically look for, there are likely others that will also provide useful information.

Linking or importing with the **Visible** setting will exclude any of the layers that are turned off or not visible in the original drawing file the last time it was saved. Note that if there are layers that need to be linked or imported with the drawing, these layers must be turned on and the drawing saved in AutoCAD in order to have access to them using the **Visible** setting. Using the **Specify** setting, the ability to select which layers to include is not available until after the file is selected from the **Link CAD Formats** window or the **Import CAD Formats** window. Therefore, all other linking or importing parameters need to be completed before choosing **Layers**. After finishing the parameters, clicking **OK** to choose the selected file, will automatically open up a **Select Layers/Levels to Import/Link** window, where there are check boxes next to the list of available layers from the selected drawing file. The desired layer selections can be specified by placing check marks next to the layers to be included and removing check marks from the layers to not be included. With the layer choices made, the user can click **OK** to link or import the drawing with only those specified layers. Once the drawing has been linked or imported, the layers that were not chosen to be included cannot be turned on later.

The **Import Units** parameter is under the **Layers** option. It is recommended to just keep the **Auto-Detect** default selection, as this will detect the drawing file's units. If the units that the user needs to work in are different from the units of the drawing file, there are also other units provided on the drop-down menu below **Auto-Detect**.

Another drawing parameter is **Positioning**, which determines where the drawing will be placed within the coordinate system of Revit. In general, the same coordinate location should be used as the drawing file, unless the original location was incorrect or is unwanted for a particular reason. Noting from experience, it is ideal to have the same coordinates as the architect and other design consultants, as this will most likely not be the last time drawings are linked to the model.

It is suggested to use the **Auto – Center to Center** option, which places the center of the drawing file in the center of the Revit view. The **Auto – Origin to Origin** option can also be used to place the drawing file into Revit by its origin. This is useful for cases where the drawing file that is being link or imported was originally created in Revit, and therefore has the same origin location. The **Auto – By Shared Coordinates** option is useful if the Revit project and the linked or imported drawing file share the same coordinate system. If not, Revit will align the drawing file's world coordinates with Revit's shared coordinates. This can be used when linking or importing multiple drawing files that need to keep a set relationship to each other.

The remaining selections are manually completed, and the user can place the background drawing using its origin, base point, or center by selecting **Manual – Origin**, **Manual – Base Point**, or **Manual – Center**, respectively. The background drawing will appear after clicking **Open**, and moves along with the mouse arrow until the desired location is clicked. For **Manual – Origin**, the drawing will be moved with the mouse arrow fixed at the drawing file's origin. **Manual – Base Point** will have the drawing fixed to the mouse arrow at the drawing file's base point. **Manual – Center** will have the drawing fixed to the mouse arrow at the center point of the drawing file.

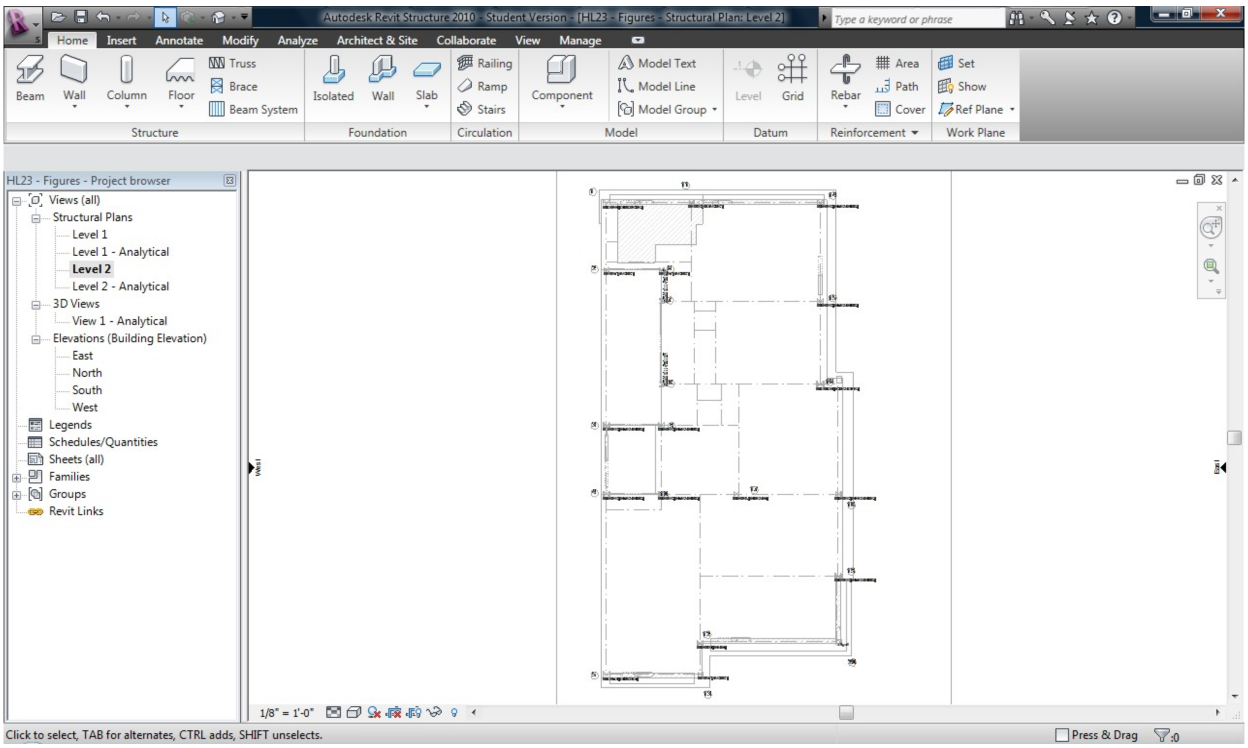


***Figure 7*: A Linked Drawing from AutoCAD**

Linking a Revit Model

The user can begin linking a Revit model for reference by clicking the **Link Revit** button on the **Link** panel of the **Insert** tab. This will open the **Import/Link RVT** window, where the user can search for the model to be linked. Regarding the location where the linked model will be placed within the current model, the user will have the same options as there are when linking a drawing file. See the above explanation of the

Positioning options in the *Linking or Importing a CAD Drawing* subsection for more information regarding the different options for **Positioning**. It is recommended to use the **Auto – Origin to Origin** option for **Positioning**, as this should place the reference model's origin location of the current model. Once the desired location is chosen and the Revit model file to be linked is selected, the user can click the **Open** button to load the reference model.

Once the reference model is linked, the user can adjust visual parameters of the linked model using the option on the **View Control Bar**. Refer to the *View Control Bar* subsection of **THE REVIT USER INTERFACE** section for more information on the buttons of the **View Control Bar**.

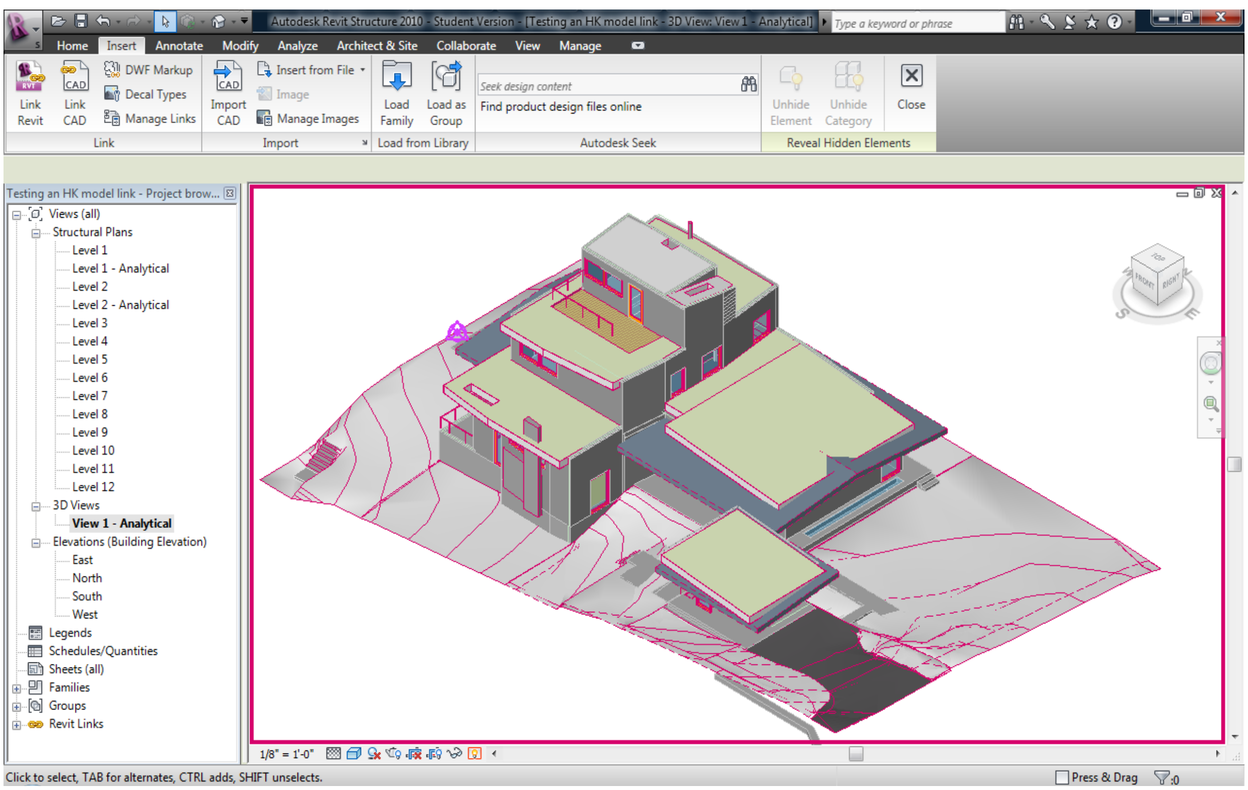


***Figure 8:* A Linked Architectural Revit Model**

Reloading a Linked Drawing File or Model File

The user can reload linked drawing files or models by clicking the **Manage Links** button on the **Link** panel of the **Insert** tab. This opens the **Manage Links** window, where the user can click on a file name under the **Linked File** column to highlight a specific file. With the file highlighted, the user can **Reload**, **Unload**, or **Remove** the link. By clicking on **Reload**, the user will refresh the connection to the linked original file, which updates any changes that were made and saved to the original file to view these changes in the current model. The **Unload** button allows the user to temporarily disable the link between the file and the current model, so that it is not being displayed. The file can be reloaded to reestablish that link. Clicking on the **Remove** button will allow the user to delete the linked file from the current model. To reestablish a link to the removed file, the user would need to link the file again in the same way it was linked originally.

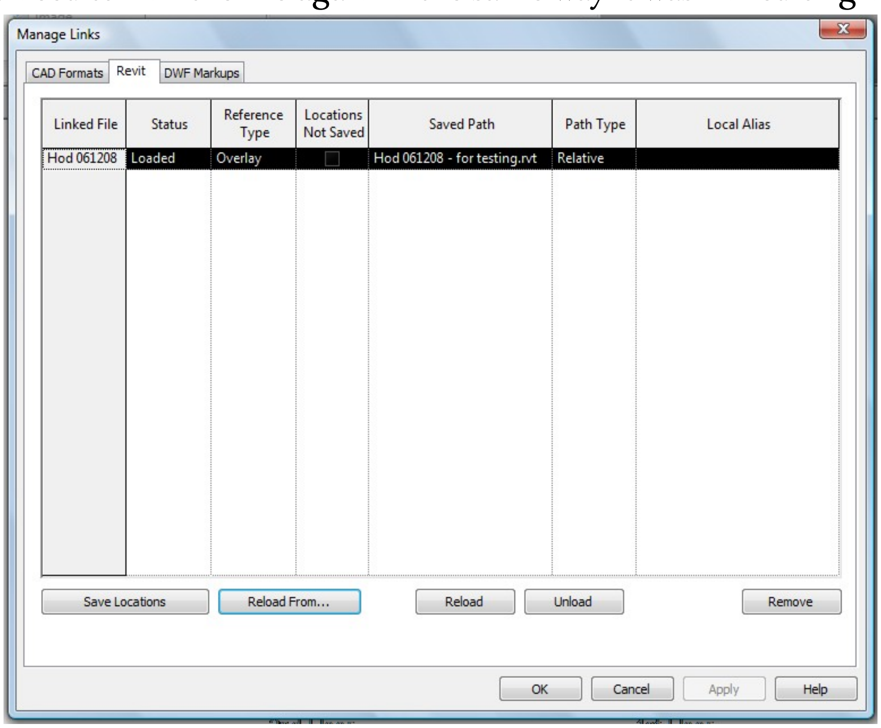


***Figure 9:* The Manage Links Window**

8. GRID SYSTEMS

Depending on the project, grid lines can prove to be the most important reference for locating anything anywhere on a given level of project. If grid lines are not being used, the other commonly used reference system for buildings is coordinate based. Industry experience has shown that most steel structures have been grid based. The majority of concrete structures have been coordinate based, with a few exceptions that were grid based.

The reason for using the different systems with different materials lies in the way that the structures are constructed with those materials and their relative shapes. For instance, the different elements of a steel structure are already fabricated when brought to a job site and have irregular shapes like wide-flange beams, whereas concrete elements need to be formed in place and are typically of simpler geometry. While practically any shape can be made using concrete, in most cases the geometry is straightforward for simplicity and ease of construction. Therefore with concrete structures, edges or corners of an element are located by a single coordinate point, and the size of the element is formed from this point. For steel structures, elements are usually placed along or parallel to reference lines, or at intersections of reference lines. Any elements that do not meet this criteria are typically placed by dimensioning the two ends of the element off of grid lines.

All structural elements are located in this way, as are architectural items and MEP items. The grid or coordinate system is the main way of mapping out a project site so that the design team can communicate real world locations to the construction team. Therefore, a well defined grid system must be developed to call out the locations of the significant elements of a project. Typically every column that falls in some type of pattern capable of being gridded, will form a grid intersection. It is usually a joint effort between the architect and structural engineer to come up with an agreeable grid system that will satisfy both of their locating needs. It is of the utmost importance that the grid lines between design consultants are identical, as any differences will result in field issues and potential conflicts, and possibly even design problems before the conflict reaches the construction phase. One good strategy in drawing grid lines is to trace an architectural background to ensure that the resulting structural grid lines will be the same. In the beginning phase of a project this is typically first done to locate and determine column locations.

The column grid layout determines where the majority of the building's vertical support elements will rest. The architect may attempt a first pass at this layout based upon the design of the spaces within the building. Ultimately the structural engineer, however, will review this layout and will need to design the floor system and columns to determine if the chosen layout is efficient and practical, or at least meets the intent of the project. The engineer will then recommend any necessary changes to improve upon the given layout. While the general layout may be determined early on in a project, grid lines continue to be in flux well throughout the design phase. As more information is known and more decisions are made regarding the architecture of the project, the support elements will need to be adjusted to accommodate this new information.

Adding Grid Lines

In order to guarantee a grid that matches the architect's grid, the most recent architectural background should be linked to the appropriate plan view. See the section on **LINKING OR IMPORTING A DRAWING FILE** for information on how to link a drawing. To create a grid lines, the user can begin by clicking the **Grid** button on the **Datum** panel of the **Home** tab. The element types for grid lines include the **1/4" Bubble**, **1/4" Bubble Custom Gap**, and **1/4" Bubble Gap**. These can be selected from the **Change Element Type** drop-down menu on the **Element** panel of the **Place Grid** tab.

The **1/4" Bubble** is a continuous dashed grid line of one-quarter inch scale with a bubble at the end of the line. The **1/4" Bubble Custom Gap** is a grid line that has a center segment that is a light-colored grid line of one-quarter inch scale, with solid line end segments, and a bubble at the end of the line. The **1/4" Bubble Gap** is a grid line that has only solid line end segments and blank space in place of a center segment, with a bubble at the end of the line.

Other types can be created by accessing the **Type Properties** window for grid lines. This window is found is by clicking on the bottom drop-down portion of the **Element Properties** button, and then selecting **Type Properties**. It can also be reached from the **Instance Properties** window, by clicking on the **Edit Type** button. Once at the **Type Properties** window, the user should select the style of grid line that most closely resembles the desired new grid line. The style is in reference to the line being continuous, having a light-colored line between the ends, or blank space between the ends. The user can click the **Duplicate** button to make a copy of that grid line type for editing. This immediately brings up a **Name** window, in which the new grid line type should be given a name. The new name will now be shown on the **Type** drop-down menu of the **Type Properties** window, and the parameters can be changed to the desired line. Different line weights, colors, and patterns can be chosen for the line segments, as well as specifying the length of the end segment, among a few other options.

The tools available to create grid lines are the **Line** and **Pick Lines** buttons on the **Draw** panel of the **Place Grid** tab. After clicking the **Line** button to create a grid by drawing lines, the user can draw the extents of a line that will become a grid line by clicking on a start point and an end point. The only choice on the **Options Bar** is an **Offset** text box which allows the user to create the grid line offset to the line drawn by a specified dimension that is input into the text box. The **Line** tool can be used to snap to the start and end points of the architectural grid on the background, but this is probably not the most efficient way to replicate an architectural grid.

The **Pick Lines** button allows a user to select the actual grid line that is being viewed on the architectural background, and to create a grid line with the same length and location. After clicking on the **Pick Lines** button, selecting a grid line from the background will automatically create the new grid line. The **Options Bar** when the **Pick Lines** tool is being used displays the same **Offset** text box as when drawing lines. There is also a **Lock** check box, which gives the user the additional option to lock the selected grid line.

This option can also be turned on or off after a grid line is created by clicking on the lock symbol that is shown near the midpoint of the grid line. When a particular grid line is locked, this means that it cannot be moved relative to the other grid lines in the grid system. This allows the user to move the entire grid system at one time if desired. By unlocking the grid line, the user can move a single grid line to a desired location, keeping the remaining grid lines at their current locations. It is recommended to lock all grid lines once they are in the appropriate location, and only unlock them if individual grid lines need to be moved.

When creating a grid, it is strategically best to begin at the start of the grid system, such as at grid line "1" or grid line "A". The reason for this is that the subsequent lines drawn to create grid lines will be named sequentially after the first grid line. Therefore, if the first grid line created is named grid line "1", then the second grid line created will automatically be named grid line "2", and so on. The same is true alphabetically. Therefore, if the first grid line created is named grid line "A", then the second grid line created will automatically be named grid line "B", and so on. The grid tag for the newly created grid line can be edited by clicking on the new grid bubble. An editing box appears so that the new grid tag can be typed in, and chosen by pushing the Enter key.

Once the desired grid tag is specified, the user can begin drawing the remaining grid lines in the order that they will be named. If there is an intermediate grid line that does not have a whole number name (such as 1.5 or A.5), these lines should be skipped until last, so that the whole number grid lines can be created first and named automatically.

After drawing and renaming the grid lines for one direction, the user can edit the length of the grid lines by clicking and dragging the start point or end point of the grid line. Note that the default is to automatically modify other grid lines that line up to match the new line's length. This is because the grid line length is locked with the others, graphically shown by the lock symbol attached to the grid line start or end point. To unlock the grid line so that a single grid line can be edited, the user can click the picture of the lock to display in an unlocked position. Note that this locking ability is different from the locking ability that was available on the **Options Bar** when first selecting the line, and the different locks can be accessed based on their location. The locks for the length of the grid line relative to other grid lines appear near the start or end of the grid line, whereas the lock that prevents a particular grid line from being moved relative to the other grid lines is located near the midpoint of the grid line.

The option to display or hide a particular grid bubble is available by clicking on the check box that appears next to the start or end of the grid line. This ability can be used to switch the side of a grid line from its start to its end, or to show the grid line on both sides. The box will be shown as an empty square on a side where there is no bubble being shown.

An elbow in the grid line can also be created to avoid grid bubbles overlapping in cases where two grid lines may be separated by only a very small distance. The user can do this by clicking on the zigzag line (which looks like a diagonal letter "z") on the grid line near the bubble. Doing this automatically creates an elbow, which is a kink in the end of the grid line. When that grid line is selected, the solid blue dots at the points of the elbow can be dragged to edit the elbow lines.

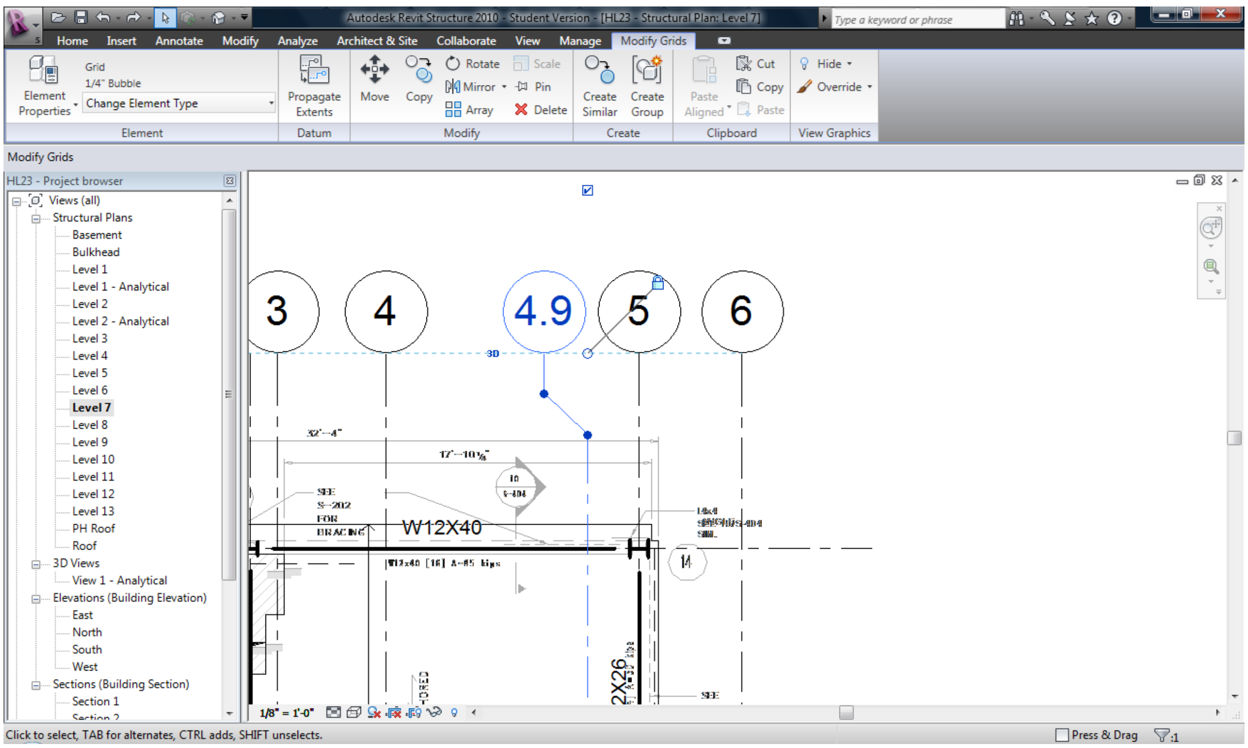


***Figure 10:* Grid Line Bubbles, Grid Line 4.9 Shown with Elbow**

The changes made to grid lines on one level will not be reflected on all levels automatically. Different levels will have different grid line requirements, and all levels may not need the same type of editing. The user can, however, produce the same grid line modifications on multiple levels. After making the necessary changes to grid lines on one level, the user can select the grid lines that were edited and click the **Propagate**

Extents button on the **Datum** panel of the **Modify Grids** tab. This will open a **Propagate Datum Extents** window, where there are check boxes next to the other levels that these changes can be applied to. After marking the desired levels, the user can click **OK** to have the grid line changes be propagated to those levels.

Adding a Curved Grid

A curved grid that matches areas of curved structure will make it easier for the designer to draw and model the curved structure. The curved grid lines can be selected or snapped to when placing curved structural elements. It is ideal to have an architectural background that shows the curved grid, so that it can be snapped to easily. If this is not available, the user can create a curved grid line by drawing it. A curved grid line is started the same way as a straight grid, by clicking the **Grid** button on the **Datum** panel of the **Home** tab. To draw a curved grid, the user can employ one of the two curved drawing tools, which are the **Start-End-Radius Arc** and **Center-Ends Arc** buttons.

The **Start-End-Radius Arc** tool allows the user to click on the start point of the grid line, followed by the end point, followed by a third point which will determine the radius of curvature. This radius can be entered in as a fixed number prior to or while drawing the grid line, by typing in the desired dimension into the **Radius** text box available on the **Options Bar**. Note that a dimension can only be typed in after placing a check in the check box to the left of the word “Radius”. The specified dimension will only be used if the two points chosen for the start and end points are close enough together that it is possible to connect these points with the specified radius. If not, the entry will be ignored, and the user will be able to choose a different radius as if no dimension was entered into the **Radius** text box. There is also an **Offset** text box, which allows the user to specify a dimension with which to offset the grid line from the chosen start, end, and radius points. Only one of the two **Options Bar** text boxes can be used at one time.

The **Center-Ends Arc** tool allows the user to click on the center point of the grid line radius of curvature first, followed by the grid start point and the grid end point. The radius can be entered in ahead of time to restrict the size of the grid line to the desired dimension. This is done in the same place as for the **Start-End-Radius Arc** tool. The **Options Bar** displays the same available options for both tools. Like the **Start-End-Radius Arc** tool, only one of the two options can be used at one time for the **Center-Ends Arc** tool.

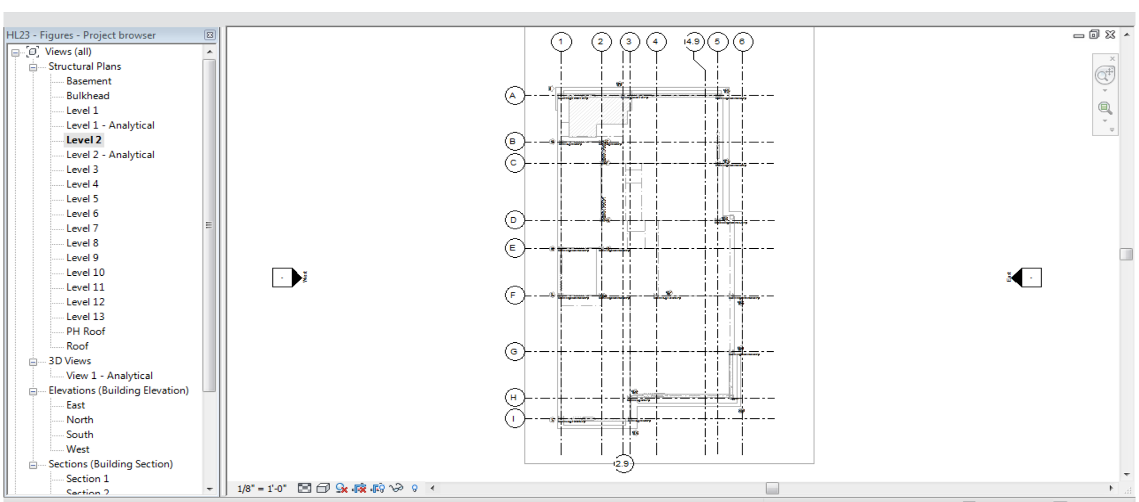


***Figure 11:* Created Grid Lines for a Structure**

9. STRUCTURAL COLUMNS

Structural columns typically provide the majority of vertical support for gravity loads within a structure. This means that columns support the beams and floor systems that connect to them. Shear walls also support gravity loads for their respective tributary areas, but experience with buildings that have decently large footprints has shown that columns account for a larger percentage of the total gravity load.

Columns can be placed vertically in either plan view or 3D view. They can be placed slanted in either section view, elevation view, or 3D view. For laying out the vertical columns of a floor, it is best to start in the plan view. Columns can also be placed one at a time, or in multiple quantities. Placing columns at multiple locations can be a time saving tool, particularly on projects where there are very many columns on each floor.

One of the ways to model multiple columns at once involves placing columns at the intersections of selected grid lines. Much care must be taken, however, to note locations where there are not supposed to be columns at grid line intersections. The columns placed at these locations will need to be deleted or modified as necessary. It is also important to note that the same type or size of column will be placed at all of these locations. This may not seem like a problem in a simplified ideal design, but it is very common for column sizes to change along a grid line. It is also common to have column sizes change or terminate at different levels. Therefore the extents of the column, in addition to the type or size and orientation, must be carefully noted when modeling.

While it is ideal to have the same size columns in as many places as possible for simplicity of design and construction, this is not always the case because of different requirements. For instance an architect may need a few additional inches of space in one direction of a column, but can allow for the extension of the column in the other direction. MEP engineers may request that a column be shifted because of penetrations required for mechanical, electrical, or plumbing systems. Typically almost all columns will be assumed to be the same size at the very early onset of a project until more detailed information is known as the design of a project progresses. Only if there are known instances of column transfers or other special conditions will a separate column size be given at the beginning phase of a project.

Adding Single Columns

When placing columns, it is best to view a floor plan with a background so that the horizontal column locations can be chosen. See the section on **LINKING OR IMPORTING A DRAWING FILE** for information on how to link a drawing. The user can begin adding columns by clicking the **Column** button on the **Structure** panel of the **Home** tab. The type of column can be selected by clicking on the **Change Element Type** drop-down menu, which will display the available column types. If the desired column type is not shown, it can be loaded by clicking the **Load Family** button on the **Detail** panel of the **Place Structural Column** tab.

The **Load Family** button opens the **Load Family** window, in which the user can browse through folders to find the desired column family. The default folder path for columns is within the Imperial Library folder, through the Structural folder, followed by the Columns folder, then the desired material folder. A list of column families that fit the chosen material will be displayed, and a column family can be selected to open a **Specify Types** window. Individual column shapes can be selected here to be made available on the **Change Element Type** drop-down menu.

If a particular column size is not available on the **Specify Types** window, a new type can be created by duplicating an existing type. This may be necessary for specific sizes of concrete columns. The **Duplicate** button to copy an existing column type for editing is found on the **Type Properties** window. This window can be opened by clicking on the bottom of the **Element Properties** button and selecting **Type Properties**. Another way to open the window is by clicking on the top of the **Element Properties** button to open the **Instance Properties** window, and then clicking on the **Type Properties** button adjacent to the **Type** drop-down menu.

With the **Type Properties** window open, the user can select the type of column that most closely resembles the desired type. For instance if the user wants to create a 28” x 28” concrete column, a 24” x 24” concrete square column can be selected here. Then the user can click on the **Duplicate** button, which will open a **Name** window. It is suggested that the user edits the column name to the desired column type, including the material and size in the name. Now the parameters of the column type can be changed on the **Type Properties** window to create the desired column type. To use a concrete rectangular column as an example, after entering the new column type name, the width (b) and height (h) of the column can be edited on the **Type Properties** window to meet the desired column type. Clicking **OK** on the **Type Properties** window will save the column type. If the user opened the **Type Properties** window through the **Instance Properties** window, then the **Instance Properties** window will still be open. The parameters of the column element about to be created can be edited on this window. See the *Modifying Column Parameters* subsection for more information on the **Instance Properties** window for columns.

Once the desired column type is chosen from the **Change Element Type** drop-down menu, the user can begin placing the columns. To place a single column in plan view the user can snap to and click on the desired column location. After a single column is placed, the option to place another column immediately follows. Also after placing a column, the user can rotate it to a different orientation if necessary. When placing a column, the **Options Bar** displays a number of options including a **Tag** check box, a **Rotate After Placement** check box, a **Depth/Height** drop-down menu, a **Floor** drop-down menu, a text box, and (for only concrete columns) a check box for Room Bounding.

Columns that are drawn with the **Tag** check box on will place the column with a tag displaying the column size. For circular concrete columns, the diameter is displayed. For concrete rectangular columns, the two side dimensions are displayed.

For steel columns, the steel member shape designation is displayed.

When the **Rotate After Placement** check box is on, the user has the ability to rotate the column to a different orientation once the column has been placed. Immediately upon placing the column at the desired location, the user is required to determine the column's orientation. There will be the option to place additional columns after the rotation is chosen. Instead of using the **Rotate After Placement** option, there is a quicker way to rotate the column if the desired rotation can be referenced by a line in the background drawing. Pushing the Space bar before clicking to place the column will rotate the column to automatically chosen angles based on lines in the background drawing.

The drop-down menus for the **Depth/Height** and **Floor** of a column will determine on which levels the column will begin and end. Selecting **Depth** on the drop-down menu will give the user the option to indicate (in the adjacent **Floor** drop-down menu) the bottom level that the column will start on, with the current floor being the highest level supported. Selecting **Height** on the drop-down menu will give the user the option to indicate (in the adjacent **Floor** drop-down menu) the highest level that the column will be supporting, with the current floor being the bottom level on which the column will start. Note that depending on which floor the user is drawing the column, there are limitations to these drop-down menus. For instance, when viewing the highest level with the **Height** selection made on the first drop-down menu, the user will not be able to enter a different level for the height of the column because the highest level is already being viewed. The same is true for the lowest level and the **Depth** selection, since the lowest level is already being viewed. The only option in these cases is to select **Disconnected**, which allows the user to enter a cantilever dimension into the text box to the right on the **Options Bar**.

The dimension entered will determine the projection of a disconnected (or cantilevered) column from the current level. For instance if on the lowest level with **Depth** and **Disconnected** selected, and a certain dimension entered, the resulting column will be projecting downward from the lowest level by the specified dimension. The same is true for the **Height** and **Disconnected** selections on the highest floor, which would create a column that cantilevers above the highest floor by the indicated dimension. This tool can be used if a column needs to project beyond the highest slab level.

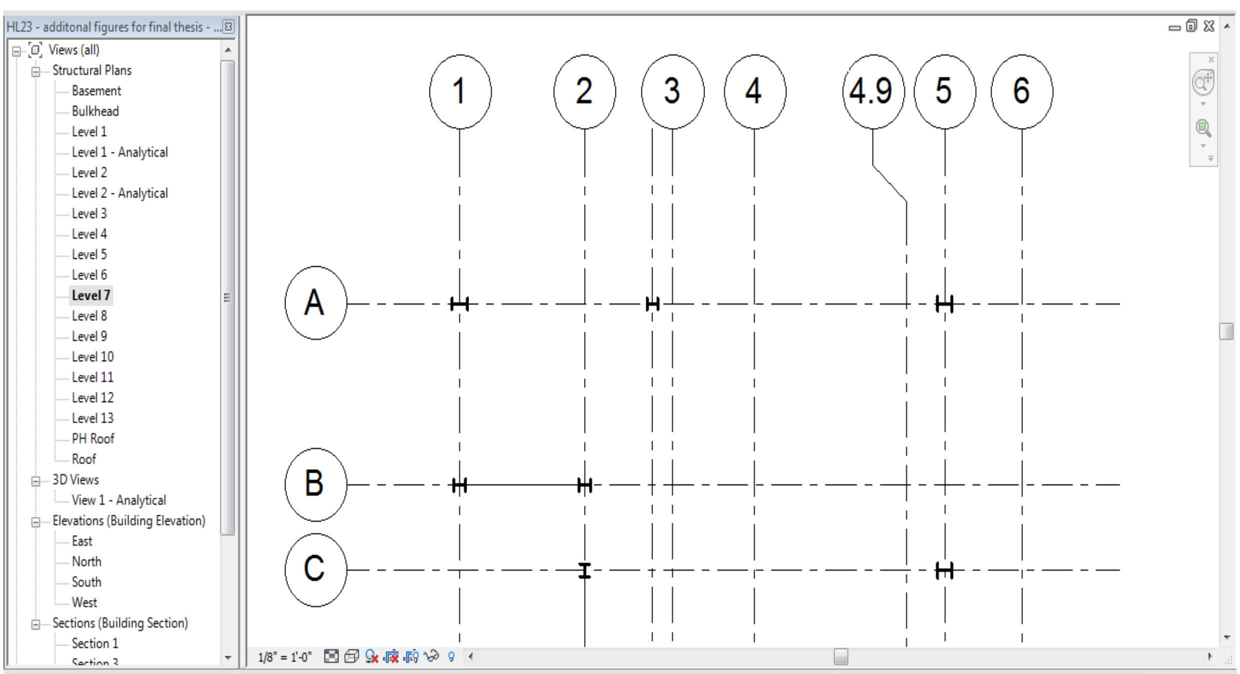


***Figure 12:* Part of a Column Layout in Plan View**

Adding Multiple Columns

To place multiple columns at once, there is a **On Grids** button on the **Multiple** panel of the **Place Structural Column** tab. The **On Grids** button allows the user to select multiple grid lines in order to place columns at the selected grid line intersections. After clicking the **Column** button on the **Structure** panel of the **Home** tab and selecting the desired column type from the **Change Element Type** drop-down menu, the user can click the **On Grids** button to begin selecting grid lines. The **Tag** check box when placing single vertical columns is still available on the **Options Bar**, and can be enabled to tag all of the columns that will be modeled with the **On Grids** button. To select groups of grid lines at one time the user can either make a selection box or hold the Ctrl key down while clicking on individual grid lines to include them in the same selection set. Refer to the subsection on *Selecting Elements* within **THE REVIT USER INTERFACE** section for more information on selecting multiple elements to be used in the same selection set.

When the grid lines are selected, a lightly outlined sketch of where the columns will be modeled is displayed. If the sketch shows the desired result, the user can proceed by clicking on **Finish Selection**. Note that even if a couple of columns are going to be drawn at a location that is not desired, they can be deleted afterward. Drawing a large group of columns at once and then deleting only a few may be less time consuming than drawing individual columns for the entire floor. As mentioned earlier, it is important to note which columns will need to be deleted. The user should use the 3D view to check the columns after modeling.

Another option for adding multiple columns is the **At Columns** button on the **Multiple** panel of the **Place Structural Column** tab. This option presents the ability to place structural columns within architectural columns, by snapping to the center point of the architectural column.

It is important to note when drawing multiple columns at once that the option to input which levels the column will be on is not on the **Options Bar** as it was when drawing individual vertical columns. The user must edit these parameters by clicking the **Element Properties** button, which is described in more detail below within the *Modifying Column Parameters* subsection.

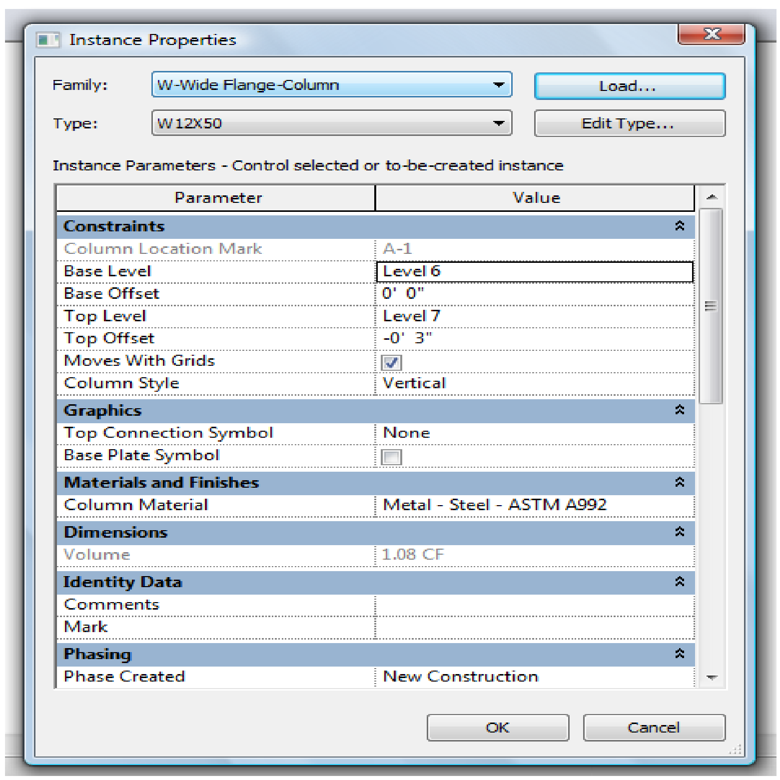


***Figure 13:* The Column Instance Properties Window for a Steel Column**

Modifying Column Parameters

After columns have been drawn, they can be selected either individually or in groups to view their properties. After selecting the desired columns, click on the **Element Properties** button on the **Element** panel of the **Modify Structural Columns** tab, which will open up the **Instance Properties** window. If the drop-down menu below the button was clicked instead, the same **Instance Properties** window can be opened by clicking on **Instance Properties**. This window can be used to modify columns in a few ways, including the type and size of the column, the base and top levels of the column, and the base and top offsets.

The type and size can be changed in a similar manner to the way they were first selected when originally drawing the columns. This will eventually be a necessary parameter to modify as at least some column sizes are inevitably bound to change throughout the course of the project. The base and top levels of the column will define the extents of the column. The user may find it easier to model the upper and lower limits for all of the columns here, as opposed to trying to enter using the **Options Bar** for just the single vertical columns. The **Base Level** chosen specifies the lowest level that the column will be on, and the **Top Level** chosen specifies the highest level that the column will be supporting.

The **Base Offset** and **Top Offset** allow the user to enter a dimension above or below a given level elevation that the column will be cantilevering beyond. For instance if a column is to extend a certain dimension beyond the Roof level, that can be modeled by selecting the Roof as the **Top Level**, and then entering the desired dimension above the Roof as the **Top Offset**. Once the editing is completed, the column should be viewed in 3D to verify that the desired result was achieved.

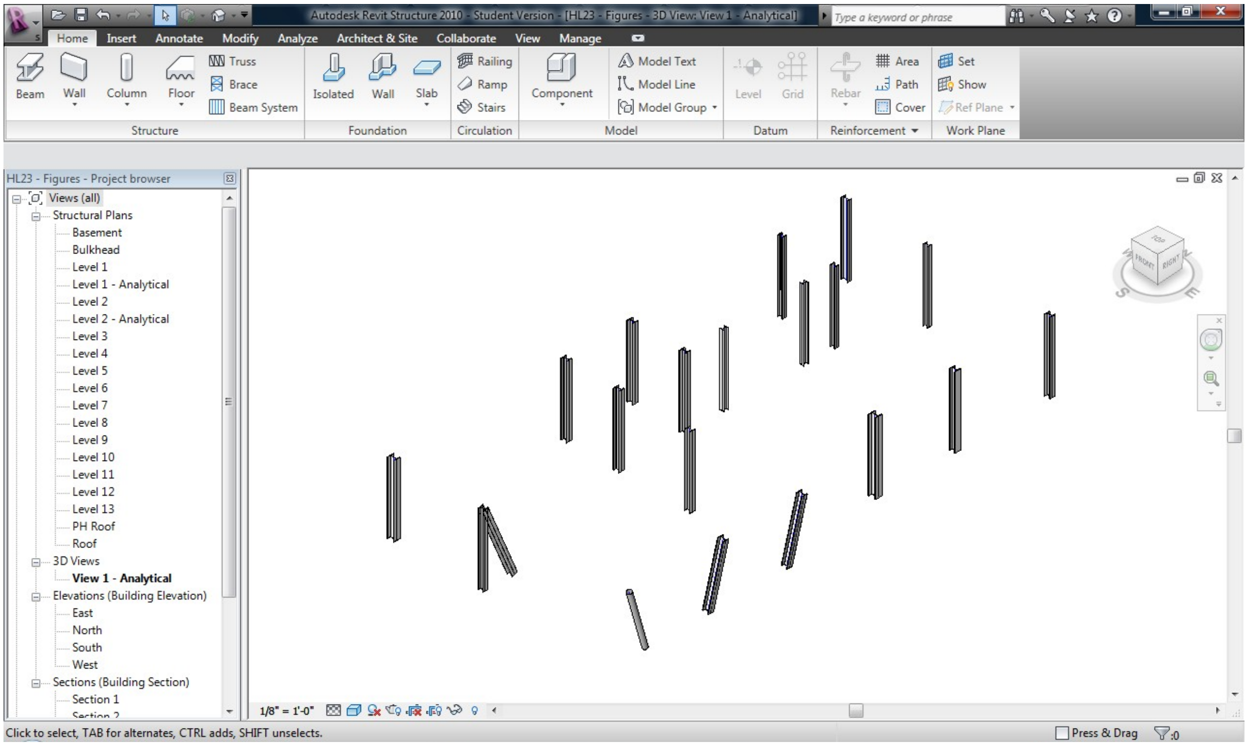


***Figure 14:* A 3D View of the Structural Column Layout for One Level**

10. STRUCTURAL WALLS

Structural walls can provide a structure with the versatile ability to bear both gravity loads and lateral loads. There are often ideal locations to hide these structural walls within thicker architectural partitions, such as around stairwell and elevator shafts within a building. Therefore, the background provided by the architect will be a useful and necessary asset in placing the wall. Similarly to when placing columns, it is best to view a floor plan with a background to locate the walls horizontally. See the section on **LINKING OR IMPORTING A DRAWING FILE** for information on how to link a drawing. Once in the proper view, the user can begin modeling the structural walls.

Adding Structural Walls

To begin placing a structural wall, the user can click the **Wall** button on the **Structure** Panel of the **Home** tab. Once this is done, the wall can be drawn by using one of many options including the **Line**, **Rectangle**, **Inscribed Polygon**, **Circumscribed Polygon**, **Circle**, **Start-End-Radius Arc**, **Center-Ends Arc**, **Tangent End Arc**, **Fillet Arc**, **Pick Lines**, and **Pick Faces** buttons. The desired type of wall can be selected by clicking the **Change Element Type** drop-down menu on the **Element** panel of the **Place Structural Wall** tab.

If the desired wall type or thickness is not available, the closest option should be selected. The properties of this type can now be edited by clicking on the **Element Properties** button. This opens the **Instance Properties** window, where the properties of the wall to be drawn can be edited. Adjacent to the **Type** drop-down menu at the top of the **Instance Properties** window, there is the **Edit Type** button, which allows the user to edit the properties of the chosen type of wall or create a new wall type. To create a new wall type by duplicating and editing the properties of the current type being viewed, the user can click the **Duplicate** button, which is adjacent to the **Type** drop-down menu at the top of the **Type Properties** window. This opens up a **Name** window to name the duplicated wall type. It is suggested to make the name as specific as possible so that the properties of that wall type will be clear when viewing it on the **Change Element Type** drop-down menu. At a minimum, the material type and thickness should be included in the name. Once the appropriate name has been entered, the user can click **OK** to save the name of the wall type.

The **Type Properties** window will still be open, but now the selection on the **Type** drop-down menu has the name of the new wall type just created. The properties of this new wall type are identical to those of the one type was duplicated, so these properties need to be edited. Clicking on the **Edit** button adjacent to the **Structure** parameter will allow the user to edit some of the construction properties of the wall type. This opens the **Edit Assembly** window, where the material and thickness of the wall can input as desired, followed by clicking **OK**. The function of the wall and some other graphical options can be edited here as well, if desired. After clicking **OK** on the **Type Properties** window, the **Instance Properties** window will still remain open. This window can be used to specify the properties of the particular wall element about to be placed. Also, after a wall has been placed, its properties can still be modified by selecting it, and then clicking on the **Element Properties** button to return to the **Instance Properties** window.

The **Instance Properties** window displays parameters such as the **Location Line**, **Base Constraint**, **Base Offset**, **Top Constraint**, **Top Offset**, **Rebar Cover**, and options for the **Analytical Model**. The **Location Line** drop-down menu will determine which line will be used to create the wall. This drop-down menu is also located on the **Options Bar** when drawing the wall, and is explained in further detail below in the *Adding a Structural Wall Using Lines* subsection.

The **Base Constraint** allows the user to select the lowest level that the wall will start on, in the same manner as the **Base Contraint** for column elements. The **Base Offset** allows the user to create a wall that cantilevers beyond the **Base Constraint** level by the input dimension for this parameter. The **Top Constraint** works in the same manner as it does for column elements, and determines the highest level supported by the wall. The **Top Offset** allows the user to create a wall that cantilevers beyond the **Top Constraint** level, in the same way that the **Base Offset** works for the **Base Constraint** level. The **Rebar Cover** is where the user has the option to specify the concrete cover dimension from the rebar to the face of the wall, and is only available for walls that are made of materials containing rebar, such as concrete or masonry.

The **Analytical Model** parameters give the user the ability to manipulate how the analytical model appears. The analytical model can be turned on or off by manipulating the **Enable Analytical Model** check box. The **Horizontal Projection** determines where the vertical lines representing the wall in the analytical model will be shown along the horizontal axes. The **Top Vertical Projection** determines where the horizontal line that represents the top of the wall in the analytical model will be shown. The **Bottom Vertical Projection** does the same for the bottom of the wall in the analytical model. The default option for these projections is **Auto-Detect**, by which Revit determines where the intended connection point is made. The user can change this option for the **Horizontal Projection** to grid lines, or the wall's exterior face, interior face, or center-line. The **Top Vertical Projection** and **Bottom Vertical Projection** can be changed the top or bottom of the wall, respectively, or either one can be changed to one of the levels. These options are useful if the top or bottom of walls are offset from their constraint levels, and the user needs to change the analytical model to match.

Adding Structural Walls Using Lines

To add a structural wall using lines, the user can begin by clicking the **Line** button on the **Draw** panel of the **Place Structural Wall** tab. The **Options Bar** will show a number of options for the wall including the **Depth/Height** drop-down menu, the **Floor** drop-down menu, a text box, the **Location Line** drop-down menu, the **Chain** check box, the **Offset** text box, and the **Radius** check box with a text box.

The **Depth/Height** drop-down menu and **Floor** drop-down menu serve the same purpose as when adding a structural column. By choosing **Depth**, the user can select the lowest floor that the wall will begin on. By choosing **Height**, the user can select the highest floor that will be supported by the wall. In both cases, the current floor serves as the opposite extent. For **Depth**, the current floor will be considered the highest floor supported by the wall. For **Height**, the current floor will be considered the lowest floor that the wall will begin on. If the highest level is being viewed, the **Height** selection can only be used in conjunction with the **Disconnected** option to specify a cantilevered wall that extends beyond the level by the dimension specified in the text box. The same is true for the **Depth** selection when viewing the lowest level, except the cantilever will be extending below the floor in this case.

The **Location Line** drop-down menu provides an option for which wall line the user wants to draw to model the wall. The wall line options include the exterior face, the interior face, or center-line. For example, if the user selects **Finish Face: Exterior** or **Core Face: Exterior** from the menu, the line should be drawn along the exterior face of the wall. With **Finish Face: Interior** or **Core Face: Interior** selected, the line should be drawn along the interior face of the wall. It is important to note which direction to draw the walls when using these selections.

When **Finish Face: Exterior** or **Core Face: Exterior** is selected, the wall will be modeled on the right side of the trace line progress. To clarify, this means that if the user draws the trace line progressing from bottom to top, then the wall will be created on the right side of the trace line. The trace line is the single line that the users draws to model the wall, and can be used to trace a line on an architectural background if one is loaded. If the user draws the trace line from top to bottom, then the wall will be on the left side of the trace line. The opposite is true for **Finish Face: Interior** or **Core Face: Interior**, in which the wall is modeled on the left side of the trace line progress. In this case, the wall will be modeled on the right side of the trace line if the user draws from top to bottom, and on the left side of the trace line if the user draws from bottom to top.

These details can be confusing to remember, but it is also possible to flip the orientation of the wall as the trace line is being drawn. For instance, after the user clicks on the start point to begin tracing the line for the wall, a lightly colored outline of where the wall will be modeled is shown. Before clicking the end point of the wall, the user can push the Space key to flip the orientation of the wall to the other side of the trace line. This eliminates the need to worry about which face needs to be drawn in which way. Other options for the **Location Line** drop-down menu include the **Wall Centerline** and **Core Centerline** selections. These options are straightforward in that the trace line should be drawn along the center-line of the wall.

It is ideal to have an architectural background loaded to trace an existing line to draw the wall, and to verify that the correct thickness of wall is being used. The wall must meet both structural requirements and architectural constraints, such as maximum thickness. If there is a discrepancy between the two requirements then the architect needs to be consulted regarding the thickness of the wall, or the wall will need to be redesigned to fit into the space provided if possible.

The **Chain** check box option is available to draw multiple walls consecutively. When this option is enabled, the endpoint of one wall will automatically be assumed as the start point of the next wall. This is a useful option when drawing walls that are connected, as they might be in a shear wall core.

The **Offset** text box can be used to draw the wall a specified dimension away from the trace line. Note that the **Offset** text box can only be used when the **Radius** check box is disabled. The same rules for wall placement relative to the trace line apply as indicated in the **Location Line** explanation above. The only additional item to note is that when a **Centerline** selection is made for the **Location Line** option, the wall is drawn offset to the centerline on the left side of the trace line progress.

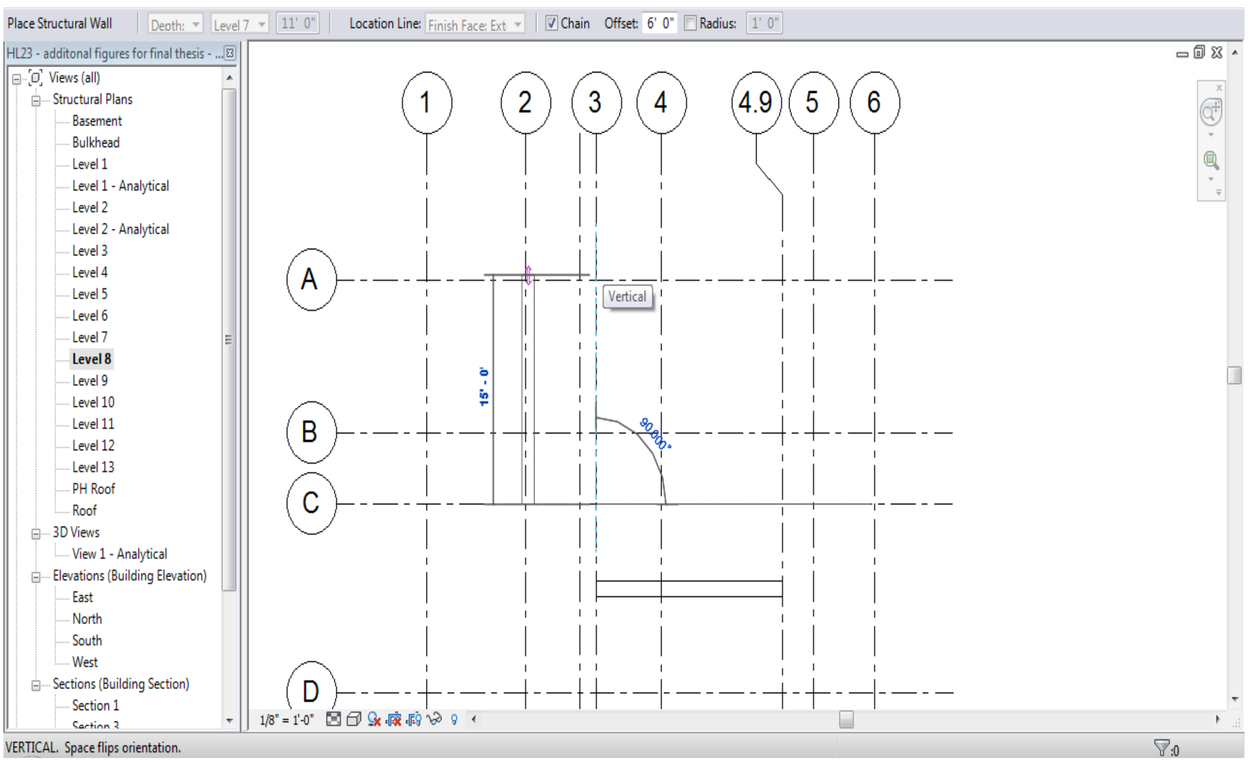


***Figure 15:* A Chain of Structural Walls Being Drawn with an Offset**

If the user has the **Chain** check box on, and draws a wall with an offset, the end point of the trace line is not at the same location as the end point of the wall because the created wall is offset from the trace line. Therefore, the start point of the trace line for the next wall in the chain is still the end of the trace line from the previous wall, and the new wall will have the specified offset from that point. If the next wall in the chain is in a different direction than the previous wall, then the two walls will not be connecting because the new wall will be drawn offset from the new trace line progress. See ***Figure 13*** for an example of a wall being drawn with an offset while the chain check box is enabled.

Adding Structural Walls Using Rectangles

To add a structural wall using a rectangle, the user can click the **Rectangle** button on the **Draw** panel of the **Place Structural Wall** tab. The **Rectangle** drawing tool allows the user to draw a box of walls, which is essentially four walls connected at the corners. Four trace lines are made simultaneously, which can speed up the process of drawing shear wall cores or any wall layouts that have a simple rectangular pattern. The **Options Bar** has the same choices as it does for drawing a wall using lines, with the exception that the **Chain** option is unavailable. The **Location Line** selections work in the same ways that they do for the lines. The suggested method for drawing a set of walls with the **Rectangle** tool is to use the centerline intersections of the walls at their corners.

Adding Structural Walls Using Inscribed Polygons and Circumscribed Polygons

The **Inscribed Polygon** and **Circumscribed Polygon** buttons are available for walls, but it is unlikely that these shapes will be needed on an actual project. They perform in a similar manner to the structural slab buttons of the same type, with an understanding of the **Location Line** option as mentioned above. See the section on **STRUCTURAL FLOORS** and the subsection of *Adding a Structural Floor Using Polygons or Circles* for more specific information regarding these tools. It is recommended, however, to draw any walls that may occur in polygon shapes using the **Line** tool.

Adding Structural Walls Using Circles

If a circular wall pattern is needed, the user can click the **Circle** button on the **Draw** panel of the **Place Structural Wall** tab. The **Location Line** selection on the **Options Bar** is seemingly irrelevant for this tool, as it is easiest to draw the shape using the circle's center point. The **Offset** and **Radius** options work in the same way that they do for placing a circular structural slab. See the section on **STRUCTURAL FLOORS** and the subsection of *Adding a Structural Floor Using Polygons or Circles* for more specific information regarding these tools.

Adding Curved Structural Walls

Curved structural walls can be modeled using the **Start-End-Radius Arc**, **Center-Ends Arc**, **Tangent End Arc**, and **Fillet Arc**. With an understanding of the way the **Location Line** drop-down menu works, these tools perform in a similar manner to the way they do when modeling beams. Therefore, see the section on **BEAMS** and the subsection of *Adding Curved Beams* for more specific information regarding the use of these tools.

Adding Structural Walls Using Pick Lines

The **Pick Lines** button on the **Draw** panel of the **Place Structural Wall** tab allows the user to select a line that will be used to model the wall. The wall will be the same length as the chosen line, and will be placed depending on the selection made from the **Location Line** drop-down menu. Therefore, before picking the centerline of the wall, the user should change the **Location Line** setting to a centerline option.

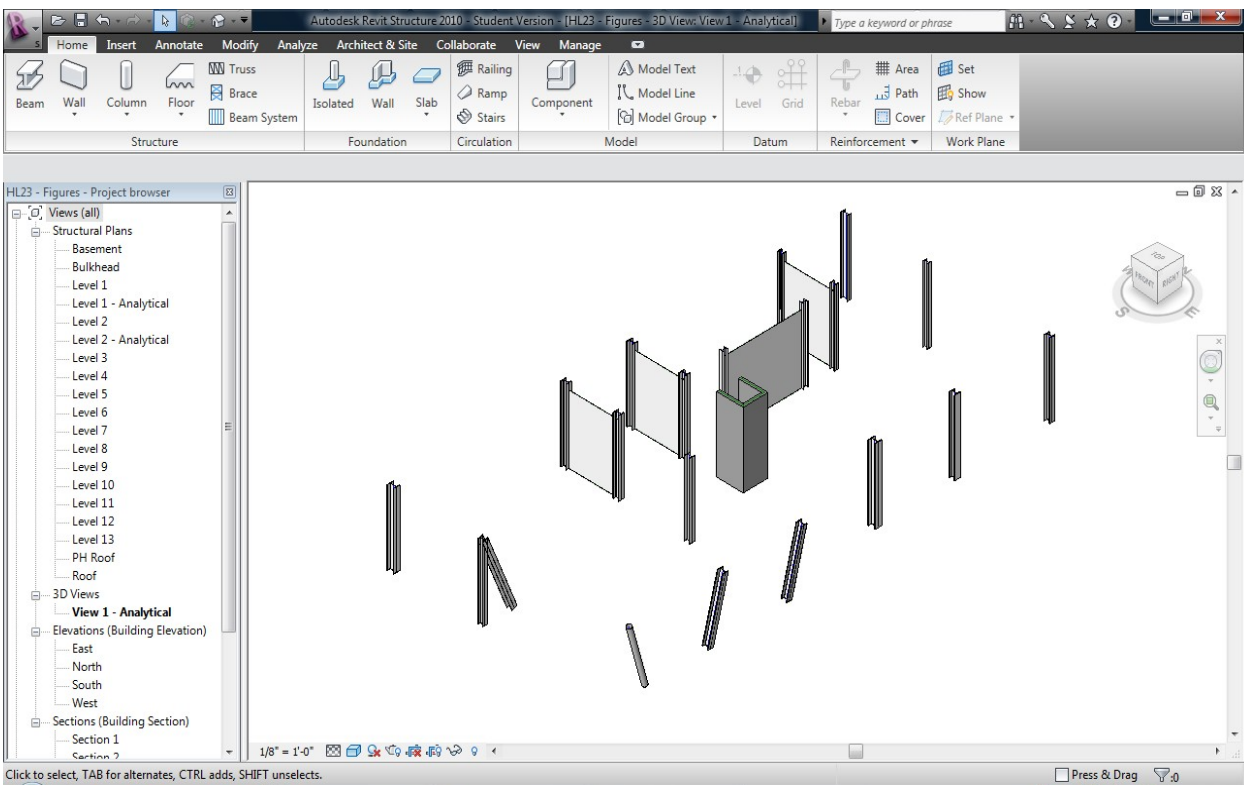


***Figure 16:** A 3D View of the Structural Wall Layout for One Level*

11. BEAMS

Beams make up a significant portion of the floor structure for steel buildings. They may also provide a large amount of support for concrete buildings that are not flat slab construction. Even with flat slab concrete construction, beams are necessary in places where the slab alone cannot support the required loads. The user can model beams by adding a single beam or a network of beams at a time. Adding a single beam span at a time is useful when framing the girders of a floor. The girders are typically considered to be the larger beams connecting the columns of the structure. Other smaller beams frame into these girders to complete the support of the slab.

The user has the option to add a single span beam, a chain of beams across multiple connecting spans, beams in multiple locations along a grid line, or a system of beams for an entire bay. While the methods of adding a single beam at a time are useful for particular complex situations, they are not particularly efficient for simplified construction that has repetitive framing bays. In this case, it would be a tedious process to model each beam individually for every beam on a given floor of a project when many of them will be the same. A structural engineer will have an idea of how to frame out the floor system prior to attempting to model the beams, so the best method of drawing the beams should be determined then. Typically the controlling factor in the layout of a framing plan is the maximum span of the structural floor. If the desired slab on metal deck can only span a certain distance, then beams will need to be placed at a spacing no greater than that maximum distance. Depending on the project geometry, a typical layout of beams may be used repeatedly in multiple bays of a given floor. In this case, the ability to add a system of beams is very useful. In other projects, the geometry may be so irregular that adding a system of beams is not practical.

Modeling beams can be done in either the plan view or 3D view. The grid line option is not available, however, in the 3D view since grid lines are not visible in the 3D view. It is recommended to do all beam drawing in plan view for simplicity and to focus on the framing for the current floor. In order to add beams in the 3D view as mentioned above, the user can enable the **3D Snapping** check box on the **Options Bar** for many of the drawing buttons on the **Draw** panel of the **Place Beam** tab. Turning this box on allows the user to snap to a location along the length of elements. This is generally not suggested, however, since it is difficult to place the beams accurately in the 3D view. The 3D view should generally only be used after drawing the beams to check that they were placed as intended.

Adding a Single Span Straight Beam

To begin adding a single span beam, the user can click the **Beam** button on the **Structure** panel of the **Home** tab. This initiates the **Place Beam** tab, on which the user can select the material, cross-sectional shape, and size of beam to model. These parameters can be chosen from the **Change Element Type** drop-down menu on the **Element** panel of the **Place Beam** tab. If the desired beam type is not displayed on the drop-down menu, the user can load it to make it available. To load a beam type, the user can click the **Load Family** button on the **Detail** panel of the **Place Beam** tab. This opens the **Load Family** window through which the user can browse for the desired beam family. The folder path from the Imperial Library folder is through the Structural and Framing subfolders, where the user can then open the desired material folder. The list of files in the material folder are the families available for the user to load. After clicking on the desired family file, a **Specify Types** window will open so the user can select which particular types from that family will be loaded. This step is skipped if the family is made up very few types, in which case all of the types will be automatically loaded.

If the desired type is not on the **Specify Types** window, as may be the case for a concrete beam of particular dimensions, a new type can be created by duplicating an existing type. The **Duplicate** button to copy an existing beam type for editing is found on the **Type Properties** window. The user can open this window by clicking on the bottom of the **Element Properties** button and selecting **Type Properties**. Another way to open the window is by clicking the top of the **Element Properties** button to open the **Instance Properties** window, where the user can click the **Type Properties** button adjacent to the **Type** drop-down menu.

With the **Type Properties** window open, the user can select the type of beam that most closely resembles the desired type. For instance if the user wants to create a 12” x 30” concrete beam, a 12” x 24” concrete rectangular beam can be selected here. Then the user can click the **Duplicate** button, which will open a **Name** window. It is suggested that the user edits the name to the desired beam type, including the material and size in the name. Now the parameters of the beam type can be changed on the **Type Properties** window to create the desired beam type. To use a concrete beam as an example, after entering the new beam type name, the width (b) and height (h) of the beam can be edited on the **Type Properties** window to meet the desired beam type. Clicking **OK** on the **Type Properties** window will save the beam type. If the user opened the **Type Properties** window through the **Instance Properties** window, then the **Instance Properties** window will still be open, where the parameters of the beam element about to be created can be edited. The window includes parameters such as the beam elevation relative to its referenced level. See the *Editing a Beam Elevation and the Beam Instance Properties Window* subsection for more information on the **Instance Properties** window.

Once the desired beam type is chosen from the **Change Element Type** drop-down menu, the user can draw the shape of the beam span by using the tools on the **Draw** panel of the **Place Beam** tab. The basic span shape that can be drawn is either a straight linear span or a curved span. To add a single linear beam the user can click on the **Line** button to create the beam in the model by clicking on the desired location for the beam starting point and then clicking again on the desired beam ending point. Once this is completed, the user can place additional beams of the same type in other locations by clicking a new starting and ending point. The user can also use the **Pick Lines** button to select an existing line, which will create a beam along this line with the same starting and ending point as the line. This is useful if a background drawing has lines already drawn where the beams will go, but it is generally recommended that the structural engineer draws all of the beams themselves.

Adding a Single Span Curved Beam

There are multiple ways to draw a curved beam. The recommended way to draw a curved beam is to have drawn a curved grid first that can be snapped to when drawing the curved beam. See the **GRID SYSTEMS** section and specifically the *Creating a Curved Grid* subsection for more information on curved grids. One way to draw a curved beam is by clicking the **Start-End-Radius Arc** button on the **Draw** panel of the **Place Beam** tab. With this tool, the user can click the starting point of the beam, the ending point of the beam, and a point along the desired radius of curvature. If a curved grid was created, the point along the desired radius of curvature would be a point along the curved grid.

If a curved grid is not available, however, the user can manually select the radius of curvature by using the temporary dimension shown while selecting the third point of the beam. The user can also enter the radius prior to drawing the beam, by clicking the **Radius** check box on the **Options Bar** for the **Start-End-Radius Arc** button, and then typing in the desired value. This will restrict the radius of curvature to the specified dimension if the start and end points are an appropriate distance away from each other such that it is possible to create a beam with the specified radius. If not, the entered dimension will be ignored.

Another option for modeling a curved beam is the **Center-Ends Arc** button. With this tool, the user can first select the center point for where the radius of curvature will be taken from, followed by the start and end points for the beam. The **Radius** option on the **Options Bar** can also be used with this tool if the **Radius** check box is turned on. In this case, after specifying the desired radius and clicking the center point of the radius of curvature, the user is restricted to picking the start and end points of the beam along a circle of the specified radius.

The **Tangent End Arc** button is another way to model a curved beam. In this case, the user can first select the start point for the beam, followed by the end point, and a determined radius will be automatically chosen. A radius can be entered on the **Options Bar**, as it was for the previously mentioned curved beam tools, allowing the user to specify a particular radius of curvature. This will, however, restrict the locations that the user can choose for the end point of the beam since only certain locations will satisfy the radius requirement. Using the **Radius** option will force the user to choose an end point location that is along a circle of the specified radius. With the **Tangent End Arc** tool, the circle location is determined by the start point of the circle, and not the center point of the circle as it was for the **Center-Ends Arc** tool.

The **Fillet Arc** tool allows the user to create a curved beam at a corner where two beams intersect. First the user can click on one beam that forms the corner, followed by the other beam that forms the corner. The user will now be able to draw a curved beam that intersects these two beams. The radius that is chosen will determine the point at which the curved corner beam will intersect the other beams, and the other beams will be shortened as required. This creates a situation where the first beam will end at a particular point, from which the curved beam will begin, until it intersects the other beam, at which point the curved beam will terminate and the other beam will begin. The same **Options Bar** choices that were available for the other curved drawing tools are also available for the **Fillet Arc** tool.

The **Spline** tool allows the user to create a beam with multiple curves along one span, by selecting control points for the curves of the beam. The user can first enter the spline beam start point, followed by control points for the beam until the desired beam is displayed, when the user can push the Esc key to create the beam. The control points are points that form a zigzagged line that begins at the start point. The spline beam is created by forming curved portions that are tangent to these zigzagged lines.

Adding a Chain of Beams

The user can add a chain of beams across multiple connecting spans by clicking the **Chain** box on the **Options Bar** for many of the drawing tool buttons on the **Draw** panel of the **Place Beam** tab. After clicking the locations of the required points for the first beam, Revit assumes that the specified end point is the start point of the next beam in the chain. This allows the user to skip the step of having to click the required next beam's start point if it can be assumed that it is just the previous beam's end point.

For example when creating a beam using the **Line** button, immediately upon clicking on the end point of the beam, a new beam is initiated with this end point being the new beam's start point. When creating a beam using the **Start-End-Radius Arc** button, immediately upon clicking on the point along the radius of the curved beam, a new beam is initiated with the end point of the previous beam being the next start point. Note that the end point is the second click that was performed, and not the third click which determined the radius of the curve.

Many of the other beam drawing tools also include the **Chain** check box on the **Options Bar**, with the exceptions of the **Center-Ends Arc, Fillet Arc,** and **Pick Lines** buttons. The ability to draw a chain of beams is particularly useful when modeling a number of straight beams that are connected at their ends. Upon drawing the first beam the user can click consecutive endpoints until the chain of beams is completed. It will also save time if there are a number of adjacent curved beams that are connected at their ends.

Adding Multiple Beams Using the On-Grids Button

Another option for drawing beams in a model is to use the **On Grids** button on the **Multiple** panel of the **Place Beam** tab. After clicking the **On Grids** button, the user can click on a grid line in the model, which will show where beams will be placed based upon the supports that are available along that grid line. See the subsection on *Selecting Elements* in **THE REVIT USER INTERFACE** section for suggestions on how to select multiple grid lines. Once the desired beams are displayed on the selected grid lines, the user can click the **Finish Selection** button on the **Multiple Selection** panel of the **Place Beam > On Grid Lines** tab to create the displayed beams.

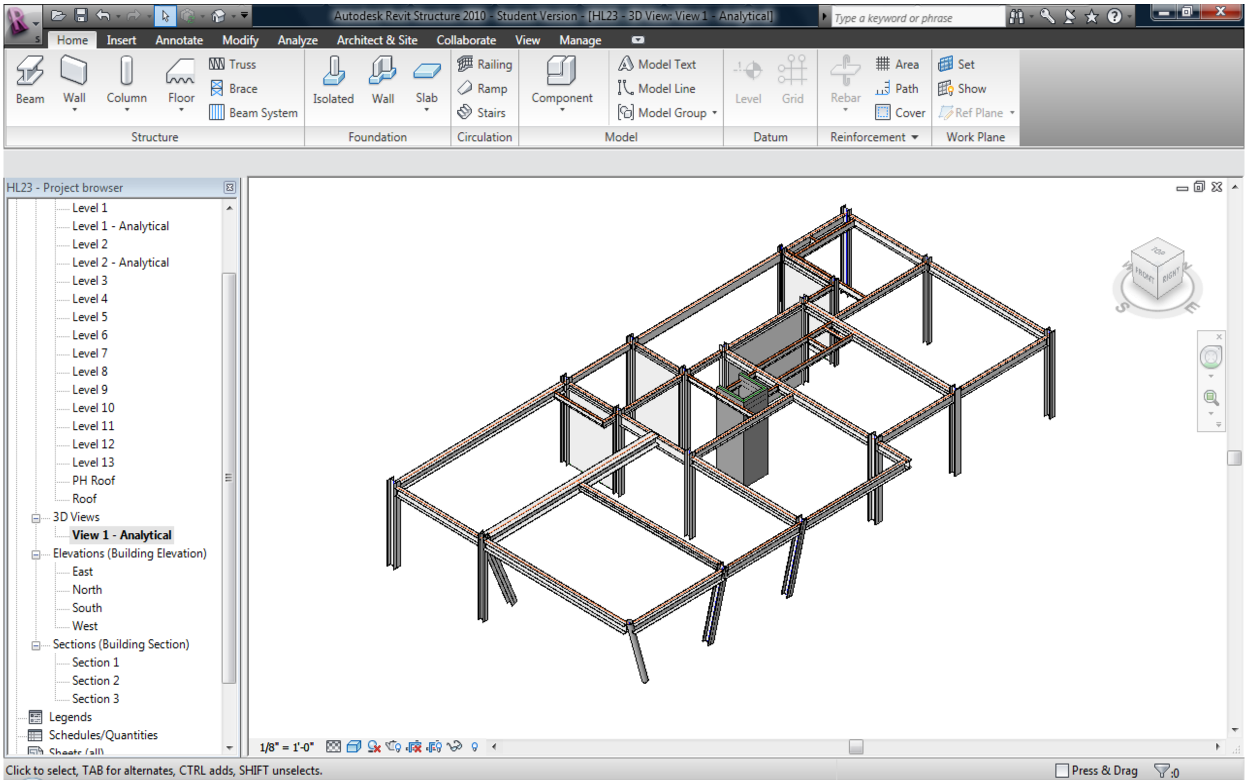


***Figure 17:* A 3D View of the Beam Framing Layout for One Level**

Adding a Beam System

The **Beam System** tool only creates the beams within a framing bay, and not the girders. To begin creating a beam system, the user can click the **Beam System** button on the **Structure** panel of the **Home** tab. This places the view in a sketch mode and opens the **Create Beam System Boundary** tab. The user can choose how to draw the beam system and adjust the properties of the beam system on this tab. The beam system is created by first drawing a boundary within which the beams of the system will be placed. Before drawing the boundary, the setting must be on **Boundary Line** on the

Draw panel of the **Create Beam System Boundary** tab. The user can use the available drawing tools on the **Draw** panel to create the beam system boundary.

Many projects' geometry will be limited to bays that can be drawn with the **Line** or **Rectangle** drawing tools. For the **Line** tool, the user can click on start and end points for individual lines to make up the boundary of the beam system. With the **Rectangle** tool, the user can click on the starting corner and ending corner of a rectangle that will form the boundary of the beam system. All of the drawing tools work in a similar manner to the way they do when creating a boundary for a floor system. See the section on **STRUCTURAL FLOORS** and particularly the individual subsections on each drawing tool for more information about the tools and their us.

After drawing the desired beam system boundary, the user can choose the direction in which the beams of the system will be spanning. To do this, the user can change the setting from **Boundary Line** to **Beam Direction** by clicking on the **Beam Direction** button on the **Draw** panel of the **Create Beam System Boundary** tab. A default direction was already chosen when the beam system boundary was created, and the beam direction is represented by the two lines that are parallel to and on either side of one of the beam system boundary lines. If these lines are spanning in the desired direction, no action needs to be taken to change the direction. To change the direction, the user can select an edge or line from the boundary to use as the new beam direction if the **Pick Lines** tool on the **Draw** panel is selected. The other option is to use the **Line** tool of the **Draw** panel to draw a line in the desired direction for the beams to span.

With the beam system boundary defined and the direction chosen, the properties of the beam system can be modified. The user can click on the **Beam System Properties** on the **Element** panel of the **Place Beam System** tab to open the **Instance Properties** window for the beam system. From this window, the user can enter the beam system elevation, and choose the rules for how the beams in the system will be laid out, the type of beam that will be used, and the level on which the beams will be tagged.

The **Elevation** parameter works the same way as the **z-Direction Offset Value** parameter for beams, which determines the relative elevation to the chosen work plane. Be sure to verify that the work plane shown is the desired level. If it is not, the user can go to the desired plan view in the **Project Browser**. The use can exit the window by clicking **Cancel**, and then exit the command by clicking on the **Cancel Beam System** button on the **Beam System** panel of the **Place Beam System** tab. Now the user can go to the desired plan view in the **Project Browser**, and click the **Beam System** button again. By clicking on the **Beam System Properties** button, the user can check that the work plane now matches the desired level, and can begin specifying the remaining parameters for the beam system.

The **Justification** of the beam system determines how the beams of the system will be justified. For instance, the default is **Center**, which means that the beams will be placed equally spaced from the sides of the boundary. Choosing **Beginning** or **End** instead will justify the beams to one side, with the side chosen having the same spacing as the beam spacing. Depending on the **Layout Rule** and dimension specified to govern the beam spacing, the other side will have the distance that is remaining between the last or first beam and the boundary. Note that the number of beams and/or spacing of beams will be affected by the **Justification** chosen.

The **Layout Rule** determines how the beams in the system are spaced. As previously mentioned, the governing factor for beam spacing is usually the span of the floor type. If this is the case, the user can choose **Maximum Spacing** for the **Layout Rule**, which will limit the spacing of the beams to the dimension specified in the value box adjacent to the **Maximum Spacing** parameter. This parameter will determine the minimum number of beams required to meet the maximum spacing specified, and will space the beams accordingly. Another option for the **Layout Rule** is **Fixed Number**, which allows the user to enter a set number of beams that will be equally spaced within the boundary lines. The same spacing that separates the beams will also be used between the boundary line parallel to the beam direction and the first or last beam in the system.

A third **Layout Rule** option is **Fixed Distance**. This rule places the beams at a specified spacing away from each other, using as many beams as required to fill the beam system boundary. The spacing can be specified in the box next to the **Fixed Spacing** parameter. The last option for the **Layout Rule** is **Clear Spacing**. This rule will place the beams according to a specified dimension using the edge of the beam rather than the beam center-line for the spacing. The **Clear Spacing** dimension entered will be the clear distance between adjacent beams.

The **Beam Type** is another parameter that can be selected on the **Instance Properties** window. This drop-down displays all of the available beam types that were loaded or created. If the desired beam is not available, it can be loaded or created from the **Place Beam** tab by using the **Load Family** button or accessing the **Type Properties** window. See the *Adding a Single Span Straight Beam* subsection for information regarding how to load or create new beam types. Verify that the **Tag New Members In View** parameter on the **Instance Properties** window matches the work plane level, so that the beams are tagged in the proper view. This parameter can be edited to match the work plane if necessary.

After all of the desired parameters for the beam system have been chosen on the **Instance Properties** window, the user can click **OK** to save these properties. Now that the beam system boundary has been drawn, the beam system direction has been chosen, and the beam system properties have been set, the user can model the beam system by clicking on the **Finish Beam System** button on the **Beam System** panel of the **Create Beam System Boundary** tab. After the beam system has been modeled, it is recommended that the user verify that the beams were placed as intended by viewing the plan view of the beam system, and then checking the 3D view.

If the beam system needs to modified, individual beams or the entire system can be selected for editing. Select an individual beam by clicking on the outline of the beam, which will light up in purple when the mouse arrow is over the beam. To select the entire beam system, the user can move the mouse arrow to one of the beam center-lines, which should light up the entire beam system center-line with purple dashed lines. The user can click on the purple dashed lines to select the entire beam system. See **THE REVIT USER INTERFACE** section and *Selecting Elements* subsection for additional information about selecting elements. With the beam or system selected, the user can click modify the beam or beam system from the respective **Modify Structural Framing** or **Modify Structural Beam System** tabs.

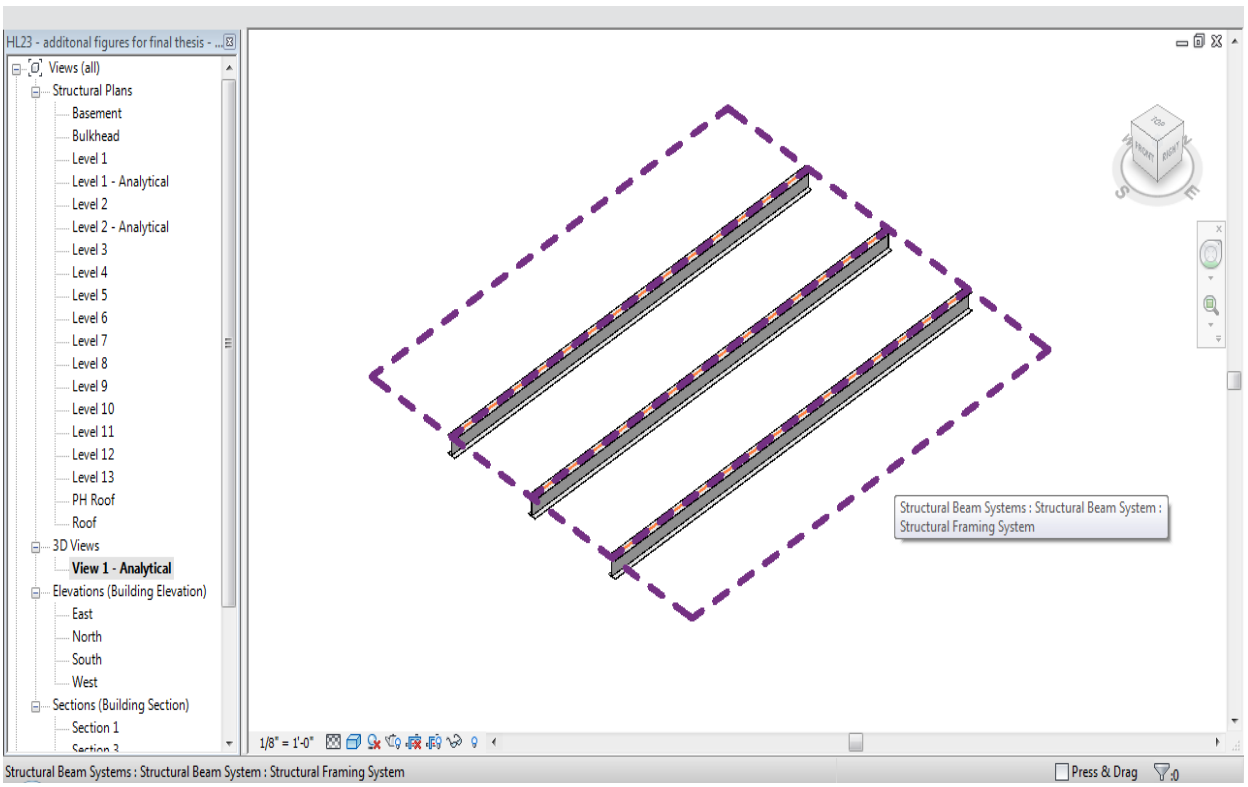


***Figure 18:** A 3D View of a Beam System*

Editing a Beam Elevation and the Beam Instance Properties Window

After drawing a beam, the user can edit the elevation of a beam relative to its level elevation. Some reasons for doing this would be to account for the thickness of the concrete slab for floors consisting of steel framing, or for areas where there is a slab drop to accommodate a different architectural floor system. There are two ways in which a beam's elevation can be offset from its floor level. Both ways are on the **Instance Properties** window of the beam. After selecting the beam or group of beams, the user can click the **Element Properties** button on the **Element** panel of the **Modify Structural Framing** tab. This will open the **Instance Properties** window.

The first method to modify the beam elevation is by changing the **z-Direction Justification** to **Other**, which allows the user to specify a **z-Direction Offset Value**. This value will offset the elevation of the selected beams relative to the floor level elevation. Note that the default condition is that the z-direction is the vertical axis, and that positive is up. Therefore, in order to offset the top of beam elevation lower, the user can enter the **z-Direction Offset Value** as a negative dimension.

Another method to modify the beam elevation is by changing the **Start Level Offset** and **End Level Offset** in the **Instance Properties** window. By entering the same value into both parameters, the beam will be raised or lowered by the same dimension. This ability also allows the user to slope the beam by entering a different value for the **Start Level Offset** than the **End Level Offset**. This may is useful for when the architect requires a sloped structure, such as for roofs or any structure that requires drainage by gravity. By sloping the structure a small amount, the force of gravity will bring liquid towards the drain. If the drain is in the middle of the floor or room, this slope could be resolved by only creating a small amount of slope in the concrete topping. There are instances, however, such as on a rooftop play yard, where the entire framing may need to be sloped in one direction towards a larger trench drain.

Another parameter on the **Instance Properties** window that is of interest to the user is the material of the beam. The material can be changed by clicking on the value box adjacent to the **Beam Material** parameter, and then clicking on the ellipsis (…) button. This button will open the **Material** window with a list of all of the available material types. The user should become familiarized with the **Instance Properties** window for the different types of beams (and other elements) to learn the parameters that can be modified there.

Adding a Truss

Trusses are a possible solution for structures that require very long spans to create large open spaces, such as gymnasiums. A truss is modeled similarly to a beam in Revit. To begin modeling a truss, the user can click the **Truss** button on the **Structure** panel of the **Home** tab. The user can select the desired type of truss by clicking on the **Change Element Type** drop-down menu, which offers the loaded types of trusses that are available, such as a Pratt Truss or Howe Flat Truss. If the desired truss is not listed, the user can load the other truss families that are available in the program by using the **Load Family** button on the **Detail** panel of the **Place Truss** tab. See the *Adding a Single Span Straight Beam* for more information on loading a family. In this case, truss families can be loaded, as opposed to framing families.

If the desired truss is also not available through the **Load Family** button, a truss type can also be duplicated and edited to create a new truss type on the **Type Properties** window. The user can open the **Type Properties** window by clicking on the bottom portion of the **Element Properties** button on the **Element** panel of the **Place Truss** tab, and then selecting **Type Properties**. On the **Type Properties**, the user can change the parameters of a truss type. The user can also create a copy of a truss type to edit by clicking on the **Duplicate** button. After naming the new truss type, the parameters of the new truss type can be specified in the **Type Properties** window. The parameters can be specified for either new truss types or existing truss types prior to drawing the truss. These include the **Structural Framing Type** for the **Top Chord, Bottom Chord, Vertical Web,** and **Diagonal Web**. The user can also edit these member sizes by clicking on the individual members of the truss after it is created.

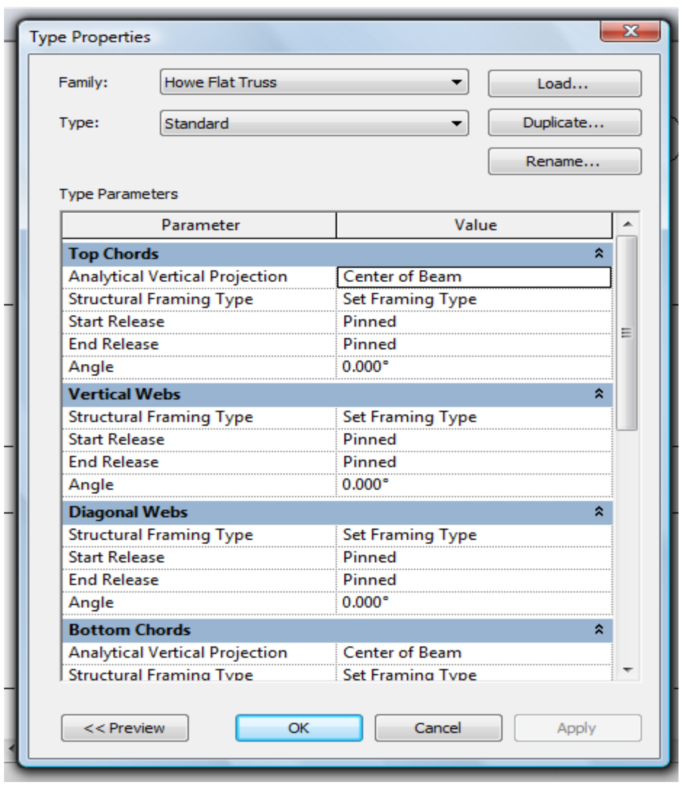


***Figure 19:* The Type Properties Window for a Truss**

After the type of truss is selected, the user can either draw lines or select lines to model the truss. To model by drawing lines, the user can click the **Line** button on the **Draw** panel of the **Place Truss** tab. Similarly to when drawing a beam span, the user can click on a start point to begin the span of the truss, and an end point to end the span of the truss. The other option of modeling the truss by selecting lines is initiated by clicking the **Pick Lines** button on the **Draw** panel of the **Place Truss** tab. The user can select a line to use as the truss span. If a line is not already drawn to meet the desired truss span, this tool is not as useful as drawing lines to create the truss. With either tool, however, a line that is longer or shorter than the desired truss span can be selected and shortened or elongated to meet the required truss span. This is done by clicking and dragging the solid blue dots that are at the ends of the truss to the desired start point and end point. It is generally recommended to draw lines to create the truss with the **Line** tool, as this typically requires the least amount of steps. The **Pick Lines** tool will only be quicker if a line that has the desired truss span is already available and can be selected.

The **Options Bar** for the two different tools are similar. Both have a **Placement Plane** drop-down menu and a **Tag** check box. The **Placement Plane** drop-down menu works in the same way as it does for a beam. The user can select which level to place the top of the truss, regardless of whether or not that level is currently being viewed. The **Tag** check box is on by default, which means that a rectangular box that can be used to name the truss will appear adjacent to the truss. This box will not appear if the

Tag check box option is disabled. To enter information in the box, the user can click on the perimeter of the box to select it, and then click on the interior of the box to enter the name of the truss.

Two options that differ between which tool is used to create the truss are the **Chain** check box and **Lock** check box. The **Chain** check box only appears when creating a truss by drawing lines with the **Line** button. When the **Chain** check box is enabled, it allows the user to draw a connected chain of trusses, with the last end point clicked automatically becoming the start point of the next truss. The **Lock** check box only appears when creating a truss by selecting lines with the **Pick Lines** button. The **Lock** check box functions similarly to the way it does when selecting lines for creating a grid. If the check box is on, the selected line is locked and cannot by dragged to a new location separately from the rest of the structure. If the check box is off, then the line is unlocked and can be dragged independently from the surrounding structure. The two lock conditions are displayed by a picture of a lock at the midspan of the line in either the locked or unlocked position.

It is suggested that the user set up a framing elevation to view the truss in elevation for modifying and checking. This can be done by clicking on the bottom portion of the **Elevation** button on the **Create** panel of the **View** tab. The user can place a framing elevation symbol by clicking on a reference plane such as a grid line. This symbol will determine the direction for the elevation view. When the framing elevation symbol is placed, a framing elevation view will be added to the **Project Browser** under the category of **Elevations (Framing Elevation)**.

The parameters of a truss can be modified by accessing the **Instance Properties** window. The user can click on the truss to select it, and then click the **Element Properties** button on the **Element** panel of the **Modify Structural Trusses** tab. The **Instance Properties** window displays a large amount of information for the truss, including the **Start Level Offset** and **End Level Offset**, the **Bearing Chord**, the **Truss Height**, and **Max Panel Width** among others. The user can change these parameters to meet the desired truss design. It is recommended that the truss is viewed in either the 3D view or the elevation view when modifying its parameters.

The **Start Level Offset** and **End Level Offset** offer the user the ability to raise or lower the elevation of the truss relative to the level elevation. This parameter can be used to allow space for the structural slab and architectural floor construction that will supported by the truss. The **Bearing Chord** setting determines whether the top chord or the bottom chord of the truss will be used to bear on the support members. Note that the default **Bearing Chord** selection is the **Bottom Chord**, which means that the truss will be modeled with the bottom chord at the selected level. Switching the **Bearing Chord** selection to **Top Chord** will model the truss with the top chord at the selected level. This option should match the intended bearing connection for the actual truss. If **Bottom Chord** is selected for the **Bearing Chord** parameter, the truss can still be modeled graphically with the top chord in plane with the rest of the framing for that level by modifying the **Start Level Offset** and **End Level Offset** appropriately.

The **Truss Height** is the dimension from the center-line of the top chord of the truss to the center-line of the bottom chord of the truss. This parameter is important to note when designing and modeling the truss to meet architectural height restrictions. For instance, if the truss is required over a gymnasium, the architect will require a certain amount of clearance below the truss for activities such as playing basketball. The structural engineer must consider the fact that the **Truss Height** parameter entered into Revit does not include the dimension from the center-line of the chords to the top or bottom of the chord members.

The **Max Panel Width** allows the user to change the number of panels of the truss by giving maximum dimensions to the panel width. Note that there are limitations to the number of panels this parameter allows. The user can, however, select individual truss members to modify their geometry by unpinning them. This is done by clicking on the pin graphic that appears next to the selected member. These truss components can then be stretched or shortened and moved around to obtain the desired truss.

It is also important to note that the depending on the truss type selected, vertical truss members may be modeled at the support points. These vertical members may be necessary if the truss is being supported by beams, but that is an unlikely structural situation. The truss is more likely to be supported by vertical elements such as columns, in which case these end vertical truss members are unnecessary and need to be deleted. The user can do this by clicking on the truss member to select it. It is recommended to view the truss in 3D to delete these members. The user should be sure to select only the vertical truss member to be deleted and not the entire truss. The entire truss will be selected if a dashed outline of the truss member center-lines is shown when the mouse arrow is over the element about the be clicked. An outline of just a single truss member will be shown if only that member will be selected. The user can try placing the mouse arrow over the edges of a single member to select only the truss member as opposed to the entire truss. In order to select the entire truss, it is recommended that the user tries placing the mouse arrow over a point where truss members intersect. The user can check the outline shown as a guide before making the selection. See **THE REVIT USER INTERFACE** section and the *Selecting Elements* subsection for suggestions on selecting elements.

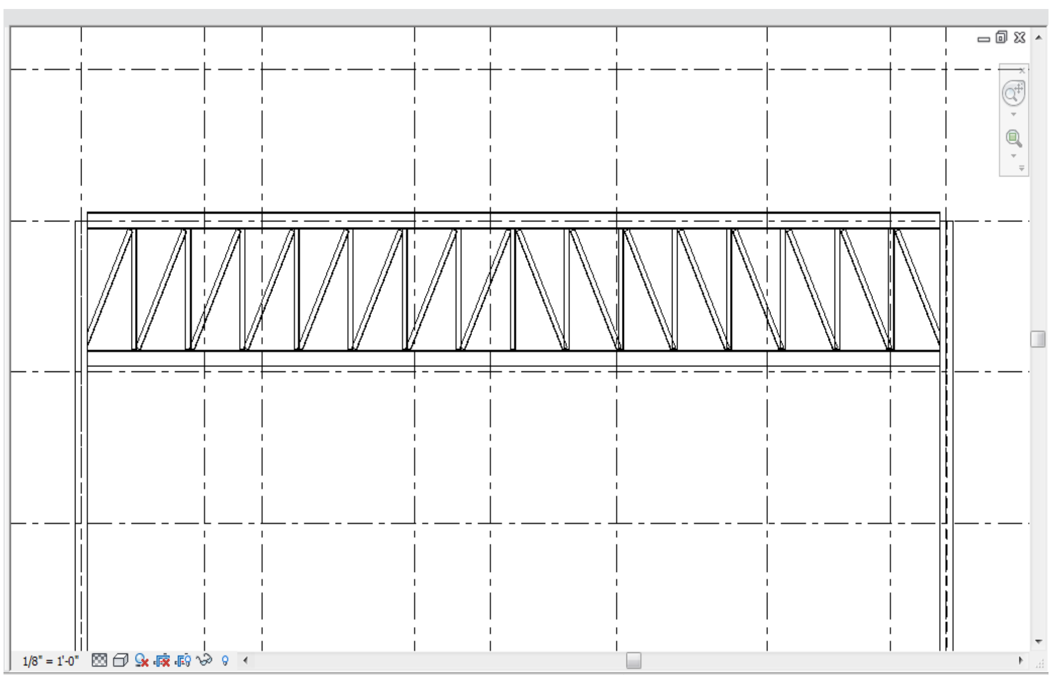


***Figure 20:* Elevation of A Truss**

12. BRACES

Braces are generally diagonal structural members, typically made of steel, that are used to withstand lateral loads. Like shear walls in concrete buildings, braces are usually part of a steel structure's lateral system that will carry the lateral load between different levels of a structure in order to bring the load to the foundation. Depending on the structures that they are a part of, braces are found in both large and small sizes. Braces are commonly used in structures ranging from incredibly tall steel skyscrapers to simple small dunnages for mechanical equipment. The ideal view to add braces to a structure is either an elevation view of the exterior of the structure, or a section view that allows the user to see an elevation of the interior of the structure.

Adding Braces

After the ideal view is set to model the brace, the user can click on the **Brace** button on the **Structure** panel of the **Home** tab. The first time this is done for a given view, it automatically opens up a **Work Plane** window. With this window, the user can specify a work plane for the brace by selecting a grid line or level from the drop-down menu adjacent to the **Name** option. This is the suggested method to specify a work plane.

It is also possible to use the **Pick A Plane** option, which after clicking **OK**, allows the user to click on a line in the model to use as a reference plane. This is useful if the user needs to pick a plane that is not an existing grid line. The views must be switched, however, from different sides of the elevation when selecting the plane and when drawing the brace. For instance, the view when selecting a plane should be perpendicular to the plane that the brace will be drawn in. If the work plane needs to be respecified, the user can open a new **Work Plane** window by creating a new section and clicking on the **Brace** button in the new section view.

After the work plane is specified, the member type should be selected prior to drawing the brace. The user can access the currently loaded members by clicking on the **Change Element Type** drop-down menu on the **Element** panel of the **Place Brace** tab. This list of members will be the same list that was available when placing beams into the model. Therefore, the user can follow the same procedure for loading new members. See the **BEAMS** section and the *Adding a Single Span Straight Beam* subsection for information regarding creating or loading framing types.

With the desired type selected, the user can draw the brace with the only tool available, which is the **Line** button on the **Draw** panel of the **Place Brace** tab. The user can snap to the desired start point and end point for the line. Although it should be done after modeling all types of elements, it is particularly important to view braces in the 3D view.

If the brace does not appear as desired, it can be modified. The first step is to select the brace by clicking on it. With the brace selected, the **Modify Structural Framing** tab opens up with ways that the user can modify the brace. The member type can be changed by clicking on the **Change Element Type** drop-down menu, and selecting the desired type. The brace can also be moved, copied, rotated, mirrored, arrayed, pinned or deleted, with the options on the **Modify** panel. If the start or end points of the brace need to be adjusted, the user can modify these points by clicking and dragging the solid blue dots that appear at the brace's ends. Other parameters of the brace can be edited by opening the **Instance Properties** window, which can be accessed by clicking the **Element Properties** button on the **Element** panel of the **Modify Structural Framing** tab. These parameters will be similar to those for beams, since both are considered structural framing. See the section on **BEAMS** for more information regarding the **Instance Properties** window for structural framing elements.

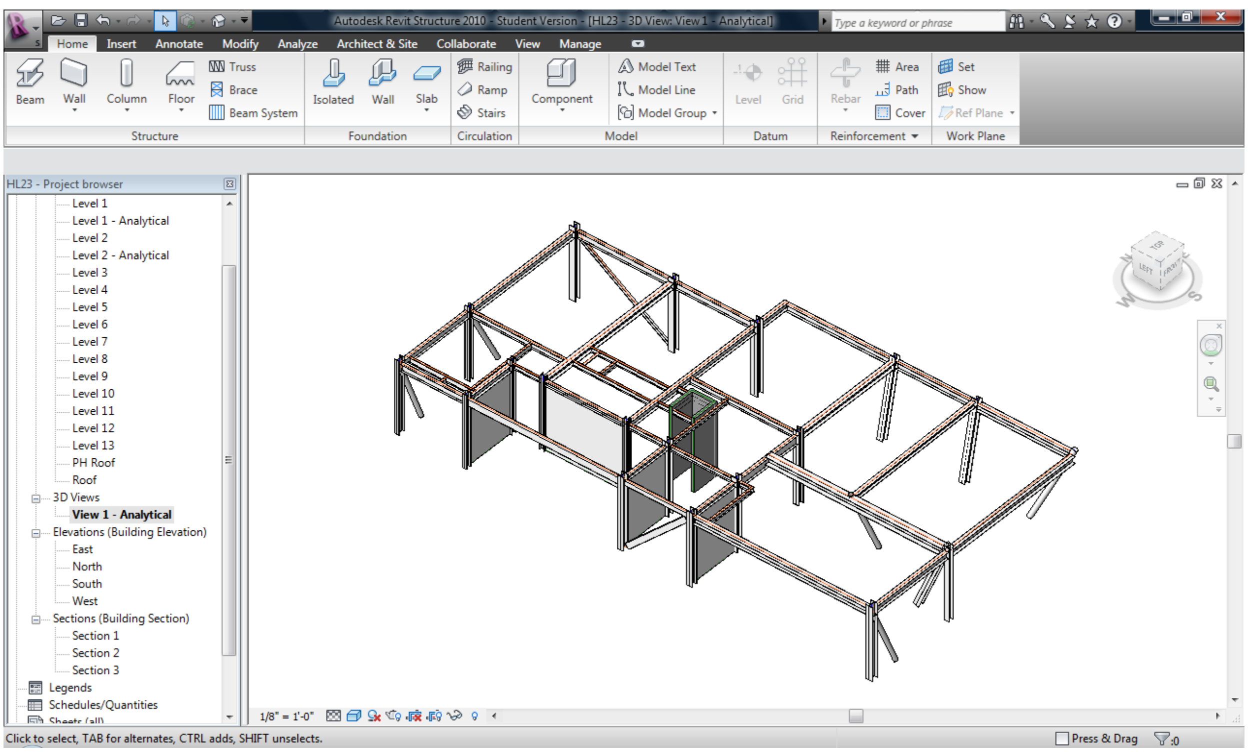


Figure 21: ***Brace Layout for One Level***

13. STRUCTURAL FLOORS

It is recommended when drawing the outline of a structural floor to have a background indicating the architect's required edge of slab. An architect will often issue slab edge drawings as a separate series of plan drawings to clearly show all of the dimensions and required information to build the extents of the slab. The suggested view to draw structural floors is the plan view. After importing an architectural background and adjusting the necessary visual options to view it properly, the user can begin to draw the boundary of the structural floor by snapping to the architectural background in the appropriate locations. See the section on **LINKING/IMPORTING A DRAWING FILE** and the subsection on *Linking or Importing a CAD Drawing* for information on loading an architectural background.

In certain buildings, different levels of the structure will often have similar or even identical structural floor types and boundaries. After one floor is modeled, it may be more efficient for the user to copy this floor to other similar levels, rather than creating a new structural floor for every level. The floor type and See the *Adding a Foundation Slab By Copying A Floor Above* subsection of the **STRUCTURAL FOUNDATIONS** section for information on how to copy and edit a structural floor from one level to another.

Adding a Structural Floor

To start drawing the structural floor, the user can click on the **Floor** button on the

Structure panel of the **Home** tab. This places the model in a sketching mode, where the user can begin outlining the extents of the slab. The sketch mode allows the user to create a floor boundary, and modify this boundary before modeling the floor. Prior to drawing the floor boundary, the user should input the desired floor type on the **Instance Properties** window. This window is opened by clicking on the **Floor Properties** button of the **Element** panel on the **Create Floor Boundary** tab. The user can select the type of floor here, along with which level it will be applied to, the height offset from the level, if the floor is a structural floor, and other properties.

If the desired floor is not available, it can be created. To do this, the user can select the floor type from the **Type** drop-down menu that most closely resembles the desired floor. Now the user can click on the **Edit Type** button, adjacent to the drop-down menu. After the **Type Properties** window opens, the user can edit or duplicate a floor type to create a new one. By clicking the **Duplicate** button, the user can make a copy of the selected floor type for editing. This immediately opens a **Name** window in which the user can enter the name of the new floor type. It is suggested that the name includes as much information as possible to easily identify the floor type later on in the drop-down menus. The floor thickness and material are recommended details to include. For instance, if the floor is a concrete slab on metal deck, then both the thickness of the concrete topping and the thickness of the metal deck can be included in the name. After clicking **OK** to save the entered name, the **Type Properties** window will show the new floor type name with the old floor type properties. The user can edit the structure of the floor system by clicking the **Edit** button adjacent to the **Structure** parameter. When the **Edit Assembly** window opens, the user can specify the thickness of the concrete and the thickness of the metal deck. To save these changes and the new floor type, the user can click **OK** on both the **Edit Assembly** window and the **Type Properties** window.

The **Instance Properties** window allows the user to edit some of the floor parameters for the floor element that will be drawn. The user can select the **Level** that the floor will be created on. The user can also enter the **Height Offset From Level**, to change the elevation of the slab relative to the elevation of the level. This parameter is useful when different areas of the floor are at different elevations. A slab depression could be required for different types of architectural flooring materials. Certain types of tiles, dance floor systems, and other architectural flooring systems require a deeper amount of space, resulting in a need for a depression in the structural slab to keep the finished floor at the correct elevation.

Other parameters that can be changed through the **Instance Properties** window include the **Structural**, **Rebar Cover**, and **Analytical Model** settings. The **Structural** determines whether or not the floor is a structural element. All of the floors created in Revit Structure by the structural engineer are most likely going to be structural elements. The **Rebar Cover** allows the user to input the concrete cover to the slab reinforcement. The **Analytical Model** parameter of **Vertical Projection** determines where the lines that represent the floor in the analytical model will be shown. It is recommended to just leave the **Auto-Detect** option for this parameter.

Once the desired floor type is selected and its are parameters edited, the user can begin to draw the floor boundary by clicking on one of the buttons of the **Draw** panel on the **Create Floor Boundary** tab. Note that the **Boundary Line** button must be selected in order to draw the floor boundary. When the user has **Boundary Line** selected, the lines drawn will be an outline of the floor, and the space within these drawn boundaries will make up the structural floor.

If there are multiple floor types or a slab depression on a particular level, it is suggested that the user treats these different floor elements. Therefore, the user can draw a boundary around the different floor types or the slab depression to exclude from the other floor elements. When a completed boundary of lines is made within another boundary of lines, the area within the interior boundary will be excluded from the exterior boundary area. If necessary, the user can switch between the drawing tools while in the sketch mode to draw different slab edge shapes. See the subsections below for information on the individual drawing tools found on the **Draw** panel.

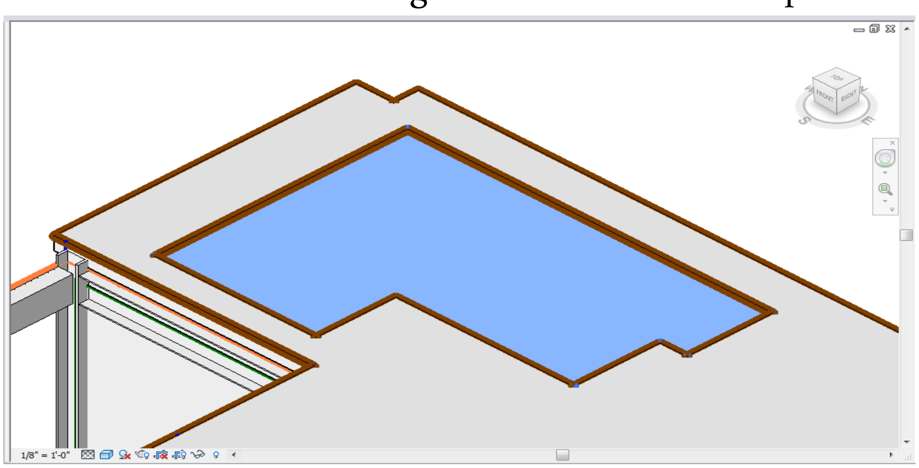


Figure 22: ***The 3D View of a Slab Depression Within a Structural Floor***

Adding a Structural Floor Using Lines

The user can being by clicking on the **Line** button on the **Draw** panel of the **Create Floor Boundary** tab. Openings in the middle of the slab for items such as elevators, stairs, or mechanical equipment can be drawn in later. Therefore the user can disregard opening elements at this point in time. See the *Adding a One Floor Vertical Opening* and *Adding a One or Multiple Floor Shaft Opening* subsections of the **OPENINGS** section for information regarding modeling opening elements in structural floors. After the **Line** button is clicked, the user can begin drawing the floor boundary by clicking on points along the boundary. If an architectural background is loaded, the user can snap to points on the background to trace the edge of slab.

Adding A Structural Floor Using Rectangles

If the slab shape is as simple as a plain rectangle, the user can draw the floor boundary using the **Rectangle** button. This tool allows the user to draw a rectangular boundary by clicking two of the diagonal points that form the corners of the rectangle.

Adding a Curved Portion of a Structural Floor

If the architectural background is showing a curved slab edge, the user can accommodate this by using some of the drawing buttons on the **Draw** panel to match the curve. These buttons include **Center-Ends Arc, Tangent End Arc, Fillet Arc,** and **Spline**. See the *Adding a Single Span Curved Beam* subsection of the **ADDING BEAMS** section for information on using these drawing tools.

Adding a Structural Floor Using Polygons or Circles

The **Inscribed Polygon**, **Circumscribed Polygon**, and **Circle** buttons allow the user to draw either a polygon with a specified number of sides, or a circle. First, the user can click on the center point of where they would like the polygon or circle placed, and then can enter the desired number of **Sides** (only for a polygon), **Offset**, or **Radius** (only after enabling the **Radius** check box). These options are found on the **Options Bar** after the user has clicked on one of these drawing buttons. The **Offset** option will allow the user to draw the polygon or circle beyond the point clicked to determine the size of the radius. This is useful if the user needs to draw an element beyond the farthest snapping point available by known distance. The **Offset** option can only be used if the **Radius** check box is disabled. If the **Radius** check box is enabled, then the user can enter the desired radius directly from that box. For **Inscribed Polygons**, the radius is considered the distance from the center point of the polygon to one of the corners of its sides. For **Circumscribed Polygons**, the radius is considered the distance from the center point of the polygon to the center of one of the sides. For **Circles**, the radius is the radius of the circle.

Adding a Structural Floor Using Ellipses or Partial Ellipses

The **Ellipse** and **Partial Ellipse** buttons are similar to the **Circle** button, except that a radius cannot be entered. After clicking the **Ellipse** button, the user can first click the desired center point, and then click the desired distance from the center point to the end of the ellipse along its major axis (also known as the transverse radius or major radius). Next, the user can click on the desired distance from the center point to the end of the ellipse along its minor axis (also known as the conjugate radius or minor radius). To draw a partial ellipse, the user can begin by clicking on the **Partial Ellipse** button. A partial ellipse is drawn in a similar manner to an ellipse, except that the first point clicked is one of the ellipse ends, and the second point clicked determines the distance from the first point to the other ellipse end along the major axis. This distance is also known as the transverse diameter. The third point clicked determines the distance from the center point to the end of the ellipse along its minor axis. This distance is also known as the transverse radius. Once these points have been entered, a half of an ellipse is drawn with those distances. If the **Chain** box is checked while drawing a partial ellipse, then the user can draw another partial ellipse, with the second point clicked on the first partial ellipse assumed to be the first point of the next partial ellipse.

Adding a Structural Floor Using the Pick Buttons

The **Pick Lines**, **Pick Walls**, and **Pick Supports** buttons allow the user to select a line, wall, or support, respectively, that is already drawn to act as the slab boundary. The **Pick Lines** button uses the line selected as part of the slab boundary, as opposed to drawing a line by clicking on points. If an architectural background with a slab edge shown is available, then the user can select the lines representing the edge of slab to create the slab boundary. This will save time over having to draw all of the lines by tracing the architectural background with the **Line** tool.

The **Pick Supports** button allows the user to click on floor supports such as beams or walls, with the center-lines of these supports used as part of the floor boundary. The **Pick Walls** button allows the user to do the same with walls, except instead of the center-lines of the walls being used as part of the slab boundary, now the user can select one of the wall faces as part of the slab boundary. If the wrong wall face is clicked, the user can switch the wall faces by clicking on the small graphical arrows pointing in different directions at the midpoint of the wall. The option to offset the slab by a specified distance is available on the **Options Bar** for the **Pick Lines**, **Pick Supports**, and **Pick Walls** buttons. The **Offset** ability will draw the floor boundary line a specified distance beyond the chosen line, or center-line of the chosen support or wall.

Editing and Finishing the Boundary of a Structural Floor

The user can delete, move, extend, or otherwise modify the lines drawn before finishing by using the buttons shown on the **Edit** panel of the **Create Floor Boundary** tab. The **Modify** button on the **Selection** panel can also be used to modify the lines. For instance, the arrows to switch wall faces after using the **Pick Walls** button can be viewed by selecting the wall after having clicked on the **Modify** button. Many different options will appear on the **Options Bar**, depending on the element selected. Once the floor boundary is finalized, the user can click on the **Finish Floor** button on the **Floor panel** of the **Create Floor Boundary** tab.

The floor type chosen will be modeled using the boundary lines that were drawn. If the wrong floor type was chosen, the user can click the **Modify** button of the **Selection Panel**. After the floor is selected, the user can click the **Change Element Type** button on the **Element** panel of the **Modify Floors** tab. The **Instance Properties** window can also be accessed on the **Element** panel, by clicking on the **Element Properties** button. If the bottom part of the **Element Properties** button was clicked, the user can select **Instance Properties** to open the same window. Even after the floor has been modeled, the user can make modifications to the floor boundary by clicking on the **Edit Boundary** button on the **Edit** panel of the **Modify Floors** tab. After all of the desired changes are made and the floor is completed, the user should view the floor in 3D to verify that it was modeled as intended.

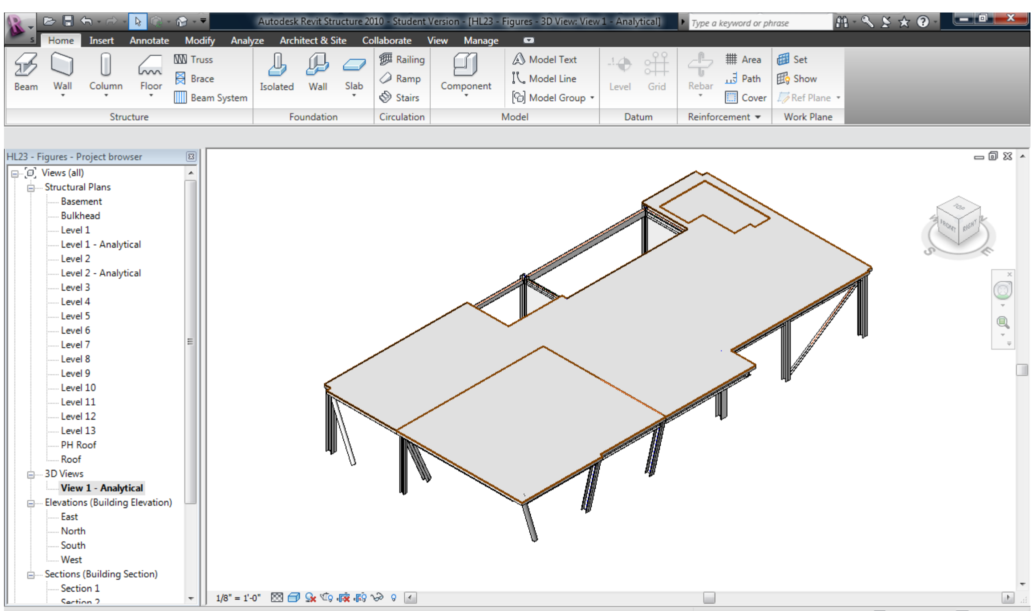


Figure 23: ***A 3D View of the Structural Floor for One Level***

14. OPENINGS

Openings are an inevitable necessity in structural slabs, and are needed for multiple reasons. These reasons include architectural design, MEP design, and also occasionally construction logistics. Architectural reasons include the necessity of having stair and elevator openings for access between floors. Other architectural reasons include openings for items such as skylights, loft spaces, or double height areas among others. MEP penetrations are needed to run the mechanical, electrical, and plumbing systems between floors via ducts, pipes, conduit or any other material used to transport the MEP services throughout a building. Logistical construction penetrations could include a temporary opening for a hoist, crane, or concrete pump for which there may not be room outside of the building extents. This is not uncommon, particularly in the New York City area, where space and land are limited. Developers are interested in using all available land area for the final product and not for temporary construction requirements.

Other types of openings include openings in walls, or through beams. Wall openings are necessary through shear walls for door access to the stairwells and elevators that are often located within the shear core of a building. Beam openings are necessary in cases where there are deep beams that conflict with MEP equipment that is running at a certain elevation. It is not always possible to run the MEP systems below the floor structure, particularly in places where the structure is very deep, so penetrations through beams are not uncommon.

It is recommended to use a plan view to model openings in a structural floor. After a floor has been modeled, the user can link different backgrounds that were provided by either the architect or MEP engineer to show the location and extent of openings on a given floor. If a background is not currently being displayed, the user can follow the procedure in the *Linking or Importing a CAD Drawing* subsection of the **LINKING OR IMPORTING A DRAWING FILE** section. Once the proper background is being viewed, the user can begin to model the openings. The types of openings are shown on the **Opening** panel of the **Modify** tab. These options include the **Vertical**, **Shaft**, **Wall**, and **By Face** buttons.

The **Vertical** button is used to model an opening on a particular floor that is not likely to occur on multiple floors in the same location. This may be a good option for a unique architectural opening, or smaller MEP openings that will either change size or location on different floors. While the **Vertical** opening tool is useful for single floor openings, there are also openings that will penetrate multiple floors while remaining the same size.

Openings of this type often include elevator and stair openings, or temporary construction openings. These openings create vertical shafts throughout the building and can be modeled using the **Shaft** button. Note that it is important to verify with the designer of the opening that the opening does not change size, shape, or location between floors. The opening designer would be the architect, MEP engineer, or construction manager, depending on the reason for the opening. There are reasons why a shaft opening might change between levels.

One reason is the requirement of different size stair landings, where different floor to floor heights will require different opening sizes. Since the riser height of each step is restricted, the stair may require a greater or less number of steps between adjacent floors. These conditions should be reviewed with the consultant responsible for the opening to ensure that the structural engineer has a proper understanding of what is required.

The **By Face** button allows the user to create an opening that is cut perpendicular to a selected face. This is useful when the user needs to model an opening on a sloped surface. This is how the user can model openings in beams. The **By Face** button can be used on floor slabs, but is not as easy to use as the **Vertical** or **Shaft** opening buttons. After modeling openings, the user should check the 3D view to ensure that the openings were placed in the desired locations.

Adding a One Floor Vertical Opening

To model a one floor vertical opening, the user can click the **Vertical** button. The **Status Bar** will tell the user to select a floor, roof, ceiling, or soffit in which to create the opening. The user can select the desired floor by clicking on the floor boundary when it is highlighted. See the subsection on *Selecting Elements* within **THE REVIT USER INTERFACE** section for more information on selecting elements. After the element is selected, the user can begin to model the boundary of the opening in a similar manner to the way the floor was modeled. The user can click on points along the opening perimeter until a complete boundary is made. The same options as were given for modeling the structural floor are also available for creating an opening. These options include the **Line**, **Rectangle**, **Inscribed Polygon**, **Circumscribed Polygon**, **Circle**, **Start-End-Radius Arc**, **Center-Ends Arc**, **Tangent End Arc**, **Fillet Arc**, **Spline**, **Ellipse**, **Partial Ellipse**, and **Pick Lines** tools. Refer the related subsections of the **STRUCTURAL FLOORS** and **BEAMS** sections for more information on using those drawing tools. After the boundary is created and modified as necessary, the user can click on **Finish Opening** to model the opening.

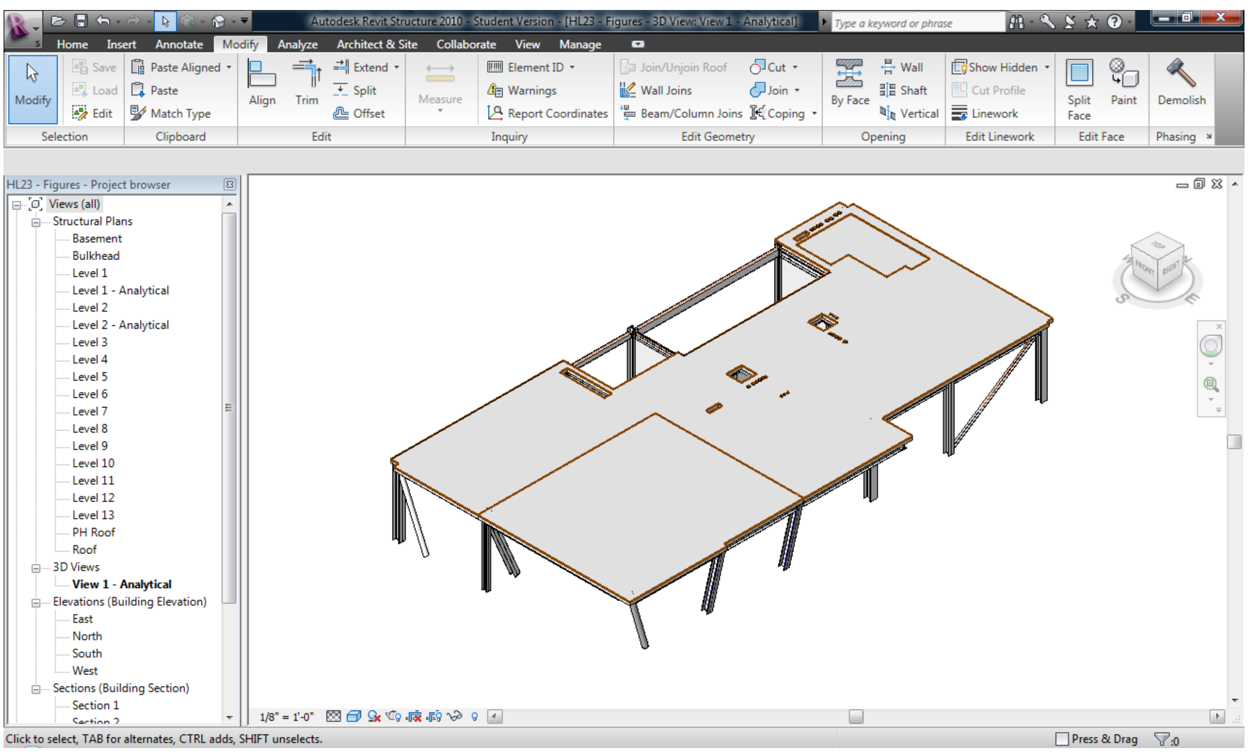


***Figure 24:** A 3D View of the Vertical Openings in a Structural Floor*

Adding a One or Multiple Floor Shaft Opening

To model the shaft opening, the user can click on the **Shaft** button on the **Opening** panel of the **Modify** tab. Now the user can access the **Shaft Opening Properties** tool on the **Element** panel of the **Create Shaft Opening Sketch** tab. This will open the **Instance Properties** window, where the user can select the **Base Constraint** and **Top Constraint**. The **Base Constraint** defines the bottom floor to have the opening, whereas the **Top Constraint** defines the top floor to have the opening. All floors between these two constraints will also have the same opening, creating the shaft. Once these parameters have been set, the user can begin sketching the opening in the same way the boundary for a vertical opening or structural floor is drawn. Upon completion of the desired boundary, the user can click **Finish Opening** to model the opening.

Adding a Wall Opening

It is recommended to user the 3D view to place the opening on the correct wall, and to use an elevation view of the wall to modify the opening. If there is not already an elevation view of this location, a section view can be created or temporarily moved to this location to modify the opening. After viewing the wall in 3D, the user can begin placing a wall opening by clicking the **Wall Opening** button on the **Opening** panel of the **Modify** tab. The user can draw the opening on the correct wall and in the correct plane, disregarding the opening location and size for the moment. To do this, the user can select the wall to create an opening in by clicking somewhere along the outline of the wall while the outline is highlighted. Now the user can begin placing the opening by drawing a rectangle somewhere within the extents of the wall. The first click begins drawing the rectangular opening, and the second click determines the diagonal dimension of the rectangle. Now that the opening is created, it is suggested that the user views an elevation of the wall, where the size and location of the opening can be modified.

After clicking on the opening to select it, dimensions showing its size and location will be displayed. Clicking on these dimensions allows the user to edit them to locate the opening relative to the edges of the wall, or enter the size of the opening. The location can also be modified by placing the mouse over the edge of the opening to highlight it, and then clicking and dragging the opening. The size can also be modified by clicking and dragging the arrows that are shown on each of the opening edges. These arrows allow the user to stretch that opening edge to the desired place.

Adding an Opening By Face

The user can begin modeling the opening by clicking the **By Face** button on the **Opening** panel of the **Modify** tab. When the user places the mouse arrow over a potential element to select, a green box will highlight the plane that the opening will be cut in. If the desired plane is not shown initially, moving the mouse arrow up or down along the height of the member will change the plane. The user can click to select the plane, and can begin sketching the extents of the opening. This is similar to the way an opening is drawn in a floor. The user can draw lines to create a boundary for the opening with the ability to modify these lines before modeling the opening. Upon completing the desired opening, the user can click **Finish Opening** to have the opening modeled.

An opening can be placed in a steel beam web for items such as pipe penetrations using the **Opening By Face** tool. It is recommended that the beam penetration is placed in the 3D view. Then a section or framing elevation view can be used to locate and modify the opening size. This is similar to the case of placing an opening in a wall, described in the *Adding a Wall Opening* subsection.

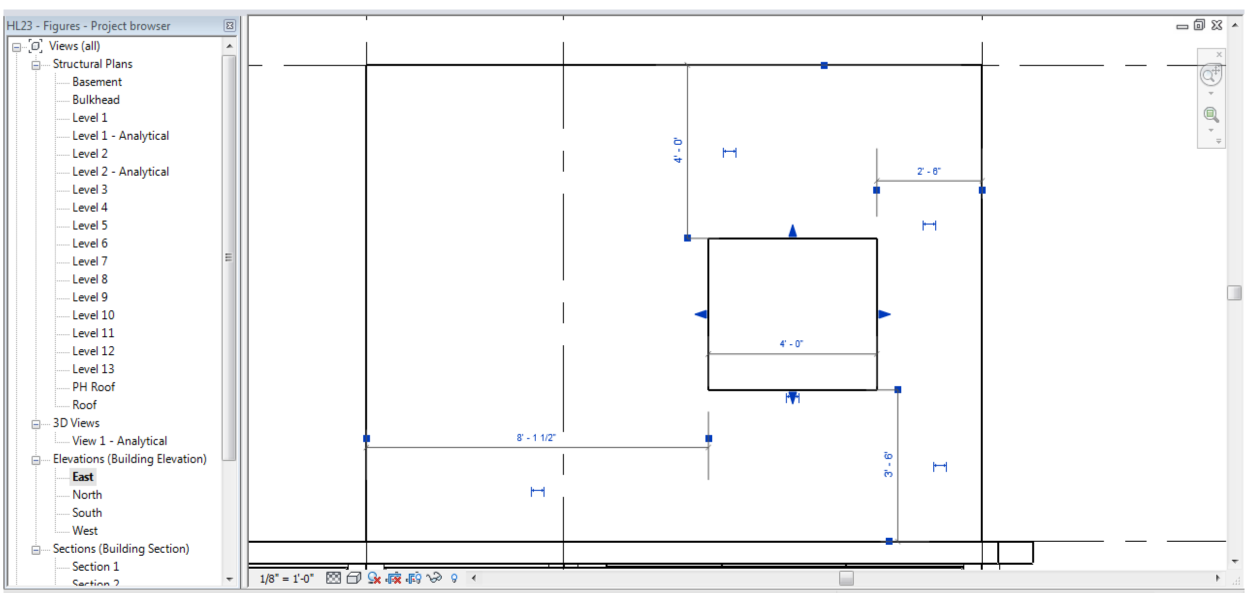


***Figure 25:* A Wall Elevation View Showing the Editable Dimensions of an Opening**

15. STRUCTURAL FOUNDATIONS

Structural Foundations design is a vital part of the design process that needs to be submitted earlier than the remaining parts of structure. A working foundation drawing set must be sent to the Department of Buildings early in the design process in order to get the paperwork started to expedite construction. While this may seem to make sense since a building is built from the bottom up, it is counter-intuitive from a design standpoint. A building must be designed from the superstructure down because the gravity and lateral loads needed to design the foundation are obtained while designing the superstructure.

The gravity loads include the worst case of all of the dead and live loads that a building will experience in its lifetime. The lateral loads include the worst case of all of the wind or seismic loads that a building is expected to possibly experience in its lifetime according to applicable governing codes. The loads used to design the foundation are a cumulative sum of these loads from following a load path from the top of the building down to the foundation. While these loads can be estimated based on a number of assumptions in order to begin an approximate foundation design, there are unknowns early in the project. Ultimately, the foundation design becomes an iterative process as more information is known and more accurate loads are obtained.

The foundation system for projects can vary greatly depending on the soil conditions of the particular site where the project will be constructed.

Foundation walls may be required on projects when excavating below the grade level. These walls are modeled, however, in the same way as other structural walls. See the **STRUCTURAL WALLS** section for information on modeling walls. Isolated foundation elements such as pile caps and footings provide the essential means of transferring the loads from a structure to the ground where the soil demands require them. Wall foundations are essentially strip footings that are used to distribute the load from a wall to the ground below. Like the walls that they support, the footings are designed per foot of length. In some cases, a structural mat will be used in lieu of individual footings or pile caps. This is because it is more efficient to simplify the design into one complete thickened slab for certain situations. The design of the foundation should be understood before attempting to model it.

Adding a Foundation Slab By Copying A Floor Above

To model a structural mat by creating a new floor, see the **STRUCTURAL FLOORS** section for related information regarding modeling structural floors. The user can add a foundation slab by the same technique as other structural floors. If there is an architectural background available, the modeling procedure is very similar to modeling the structural floors mentioned earlier. Regardless of whether or not there is a background, however, it is not uncommon for the foundation level plan to be similar to an adjacent floor. This could also be true for upper structural floors, which can be similar or even identical to adjacent floors. Therefore in some instances, it may save time to copy the adjacent floor and modify it, as opposed to having to draw an entirely new floor.

To copy down the floor above, the user can view the floor to be copied in plan view. The floor can be selected by clicking on its perimeter. With the floor now selected, the user can click the **Copy** button on the **Clipboard** panel of the **Modify Floors** tab. Note that this is different from the **Copy** button on the **Modify** panel of the **Modify Floors** tab. It is recommended to use the clipboard copy as opposed to the modify copy because the user will need to switch floor plan views to view the foundation level. Switching views after clicking the **Copy** button on the **Modify** panel will cancel the copy operation. Clicking the **Copy** button on the **Clipboard** panel, however, saves a copy of the selected floor to the clipboard. The the user can click the **Paste Aligned** drop-down menu on the **Clipboard** panel to paste the floor to the desired level by choosing **Select Levels**. A **Select Levels** window will appear showing the available levels onto which the structural floor can be pasted. After the user selects the foundation level, the floor will be automatically pasted to that level. Viewing the model in 3D can confirm that the floor was pasted to the correct location. The user can also go to the foundation level plan view to verify that the floor was pasted, and to begin modifying the floor to make the foundation slab.

It is important to be sure that all structural elements are supported by the foundation. This is a significant way that the level above will be different than the foundation level. For instance, any slab edges that were affected by openings near the perimeter of the floor will need to be modified in order to have the foundation slab support the walls and structure that encloses these areas. The user will need to modify any places where the slab edge is different from the copied floor. This can be done by clicking on the slab edge to select it, and then clicking the **Edit Boundary** button on the **Edit** panel of the **Modify Floors** tab. This places the drawing in the sketch mode that was used to originally create the slab edge. While in the sketch mode, the user can modify the boundary lines that make up the slab edge. The user can delete lines, draw new lines, or stretch existing lines to meet the requirements of the foundation slab. When the lines have been edited to complete the foundation slab boundary, the user can click the **Finish Floor** button on the **Floor** panel of the **Modify Floors** tab to model the new floor boundary.

The user can edit the structural floor properties to meet the requirements for the foundation slab. The foundation slab will likely have a different floor thickness than the structural slab it was copied from, and this parameter can be modified through the **Element** panel. If the desired type and thickness of floor already exists for the foundation slab, then the user can select the foundation slab and change its type on the **Change Element Type** drop-down menu. This menu is found on the **Element** panel of the **Modify Floors** tab. If the desired thickness is not available, however, the user can create it on the **Type Properties** window.

The user can access the **Type Properties** window by clicking on the floor to select it, then clicking on the **Element Properties** drop-down menu on the **Element** panel, followed by the **Type Properties** option. The desired type of floor can be chosen from the **Type** drop-down menu, disregarding the thickness. In order to create a new thickness of the floor type selected, the user can click on the **Duplicate** button adjacent to the **Type** drop-down menu. When the **Name** window opens, the user can name the new floor type. It is recommended to include the floor thickness and floor type within the name. The user can click **OK** after the new floor type has been named, which will show the new type on the **Type** drop-down menu. Note that the thickness, however, will still show the value from the floor type that was duplicated. The user can modify this by clicking on the **Edit** button of the **Structure** parameter, and then typing in the desired thickness for the **Structure** function under the **Thickness** column on the **Edit Assembly** window. Clicking **OK** will update the thickness on the **Type Properties** window. Clicking **OK** on the **Type Properties** window will give the modeled floor the properties of the new floor type. It is recommended that the user view the foundation level in 3D to verify that the modified slab edge, floor type, and thickness are shown as desired.

There is a **Slab** button on the **Foundation** panel of the **Home** tab. This button opens the same **Create Floor Boundary** tab as the **Floor** button on the **Structure** panel of the **Home** tab. Foundation slab types can be selected or created using either button. If the bottom portion of the **Slab** or **Floor** button is clicked, it will display a drop-down menu with the option to place a **Slab Edge**. This allows the user to create a thickened portion at the slab edge. A haunch such as this in the slab can be used at the ends of slabs on grade, where the slabs can be tied to foundation elements below. In other instances, this haunch can serve as a thickened element to transfer load to the soil below.

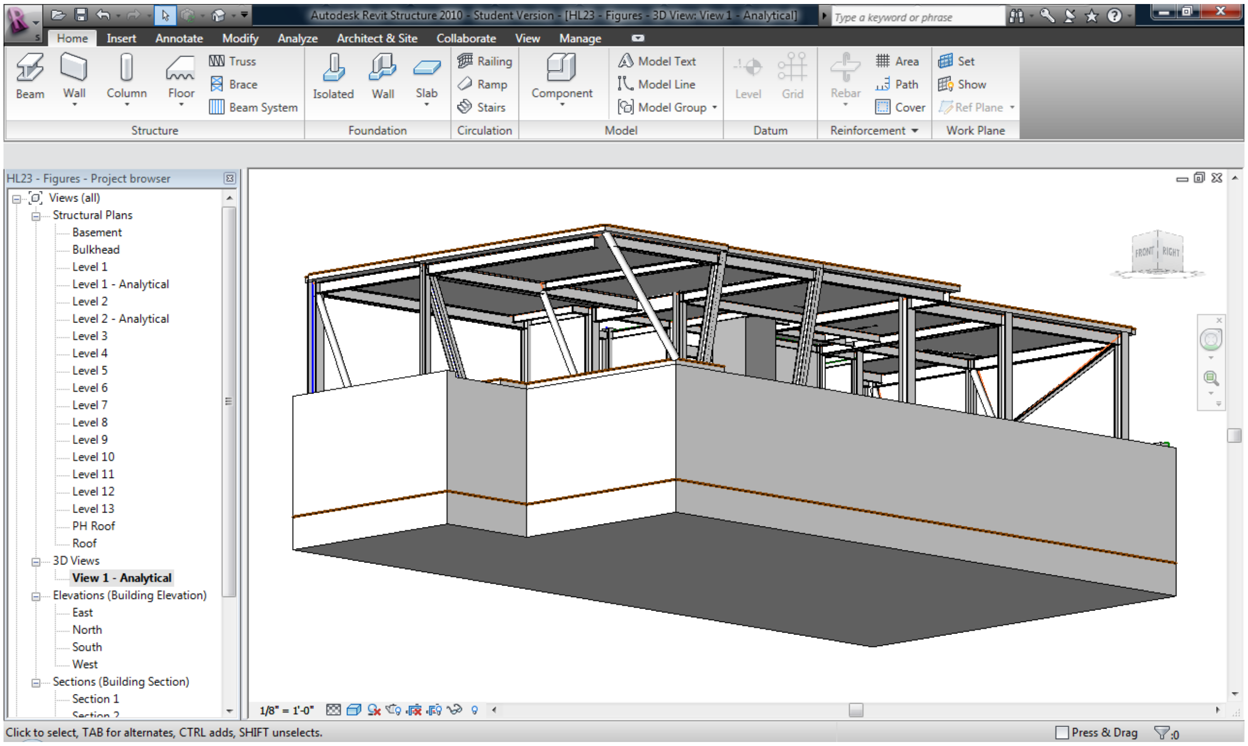


Figure 26: ***A 3D View of a Mat Foundation and Foundation Walls***

Adding Isolated Foundation Elements

It is recommended to create isolated foundation elements in plan view. The user can click on the **Isolated** button of the **Foundation** panel on the **Home** tab to begin modeling isolated foundation elements. The list of available isolated foundation elements can be viewed by clicking on the **Change Element Type** drop-down menu on the **Element** panel of the **Place Isolated Foundation** tab.

If the desired type is not on the list, it can either be loaded or created. There are numerous pile cap layouts that can be loaded by clicking on the **Load Family** button on the **Model** panel of the **Place Isolated Foundation** tab. After locating the foundation folder, there will be a list of various specific and generic pile cap layouts, as well as different types of piles. If the desired size is not available, the user can create it by accessing the **Type Properties** window. This window can be opened by clicking on the drop-down portion of the **Element Properties** button. It can also be found by opening the **Instance Properties** window by clicking on the **Element Properties** button and then clicking on the **Edit Type** button. The user can clicking on the **Duplicate** button to rename a copy of the foundation element. This copy can be edited by entering the desired dimensions on the **Type Properties** window. When the user clicks **OK**, the edited dimensions and new isolated foundation element type will be saved.

Once the desired type is selected, the user can begin placing the elements into the model. The elements are drawn by clicking on a single point, which will be the center-point of the element. The **Option Bar** only has one option for placing isolated foundation elements. This option is the **Rotate After Placement** check box. When this check box is enabled, the user will automatically have the ability to rotate the element immediately after placing it. After selecting the desired rotation by clicking at the appropriate angle, the user will have the opportunity to place additional elements. With the check box disabled, the user can click on new locations for additional foundation elements after clicking on a location for the first element. The user also has the ability to rotate an element prior to placement by pushing the Space bar, which can be quicker than using the **Rotate After Placement** option.

It is structurally ideal to have the center of the foundation element line up with the center of a column. This is not always possible, however, requiring some foundation elements to be moved after placement. After clicking on the foundation element to select it, the user can click the **Move** button on the **Modify** panel of the **Modify Structural Foundations** tab. Now the user can move the element by clicking on a base point with which to move the element, and then clicking on a new location for the base point.

The properties of isolated foundation elements can be modified through the **Instance Properties** window. This window is found by selecting one or more isolated foundation elements, and then clicking the **Element Properties** button on the **Element** panel of the **Modify Structural Foundations** tab. The user can change parameters such as the element's **Type**, **Level**, **Offset**, **Material** and **Structural** properties on the **Instance Properties** window.

The **Type** drop-down menu allows the user to change the type of isolated foundation element. The **Level** constraint allows the user to change the level that the isolated foundation is modeled on. The **Offset** constraint determines the dimension by which the top of the isolated foundation element will be offset from the specified level. The **Offset** parameter will become an important property to monitor as the project progresses. All of the foundation elements will probably not start at or remain at the same elevation. It is common for there to be different offsets due to constraints from the architect, MEP engineer, geotechnical engineer or other design consultants. Different level foundations can be required for basement storage areas, electrical vaults, elevator pits, underslab drainage systems and other conditions that affect individual isolated foundations. Note that the **Offset** parameter may need to be changed as more specific information is known about the design of the lowest level. The **Material** parameter allows the user to change the material of the isolated foundation element. The **Rebar Cover** parameter allows the user to input the desired concrete cover from the rebar to the edge of the isolated foundation element.

Adding Wall Foundations

To draw a wall foundation, it is recommended to start in the plan view showing the wall that the foundation will be supporting. Once in the appropriate view, the user can click on the **Wall** button on the **Foundation** panel of the **Home** tab to begin modeling. The user can select the desired type of **Wall Foundation** on the **Change Element Type** drop-down menu of the **Place Wall Foundation** tab. The two default types of wall foundations are a bearing footing and a retaining footing of specific size.

If the desired size is not one these types, it can be created by clicking on the **Element Properties** button to access the **Instance Properties** window. The user can select the footing that is the correct type except for its dimensions from the **Type** drop-down menu. Clicking on the **Edit Type** button will make this type appear in the **Type** drop-down menu on the **Type Properties** window, where a new size of this type can be created by clicking on the **Duplicate** button. After renaming the footing to include the desired size in the new name and clicking **OK**, the new dimensions can be entered next to the appropriate dimension parameters. Clicking **OK** on the **Type Properties** window saves the new type and size.

After selecting the desired type of footing, the user can select the wall to place the new foundation under by clicking on a wall. This automatically models the new footing as the foundation element for the wall along its center-line. The wall can be offset from the wall center-line by changing one of its parameters on the **Instance Properties** window. The first parameter under **Constraints** is the **Eccentricity** parameter, where the user can enter a dimension with which to offset the foundation from the wall. Note that the dimension entry will be ignored if it will result in the wall not resting on top of the foundation.

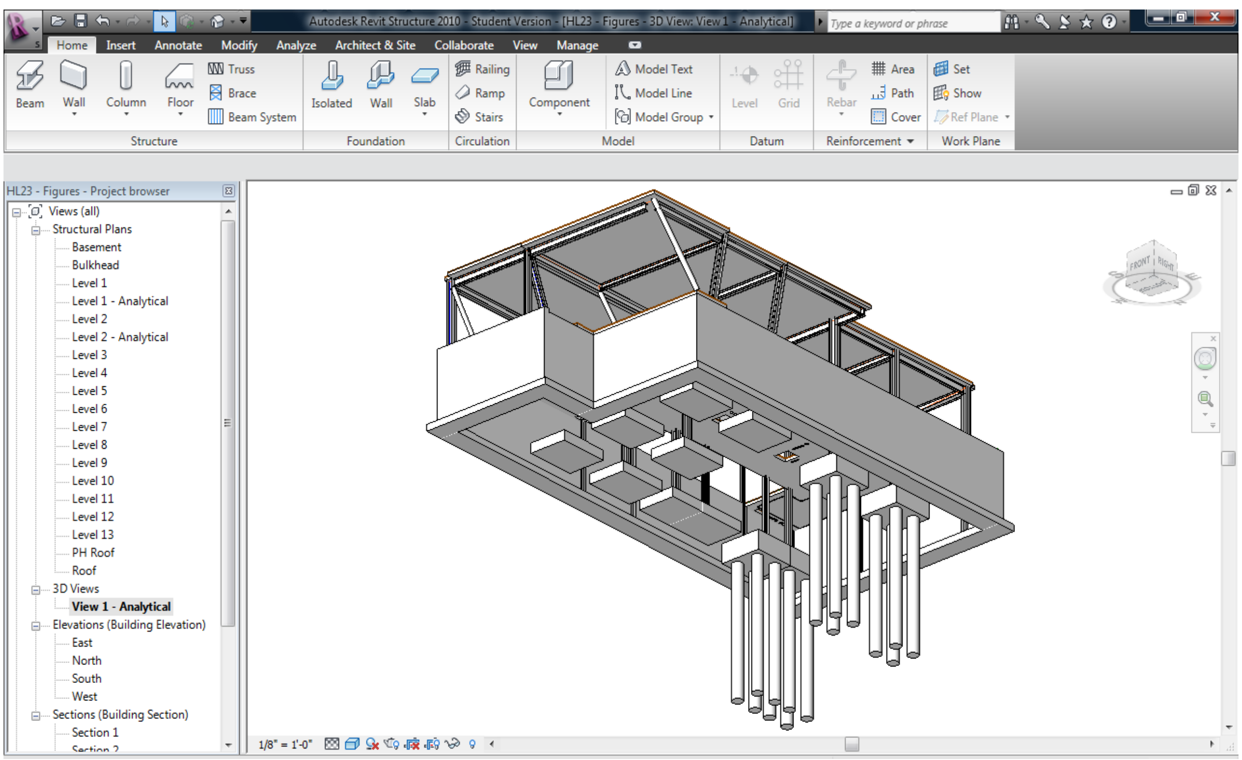


***Figure 27:* A 3D View of Isolated Foundation Elements**

16. REVIT STRUCTURE EXTENSIONS

Autodesk released a series of extensions for Revit Structure that allows the user to broaden the program's capacity in particular areas of interest to structural engineers. These areas include extensions for a range of abilities from structural analysis and design to construction documentation and reinforcement modeling. The Revit extensions specifically for Revit Structure 2010 has applications for structural detailing, structural analysis, reinforcement of structural elements, and steel connection modelers among others. The structural analysis extension has the ability of exporting a structural model into Autodesk Robot Structural Analysis.

Robot Structural Analysis allows structural engineers to perform analysis on structures through a series of analysis capabilities to evaluate the behavior of a structure under different types of loading. The program contains finite element capabilities to mesh the structure for analysis, as well as design codes and the capacity to design reinforced concrete and steel members. After a model has been developed in Revit Structure, it can be analyzed and designed in Robot Structural Analysis, and the designed structural model can then be returned to Revit Structure with the updated designed members. Robot Structural Analysis can also export the results into Autodesk's structural detailing software to create fabrication drawings from the design. While these are only a few examples, the interoperability of programs like these from a single model allows the user to explore the true benefits of BIM programs.

17. REVIT ON REAL CONSTRUCTION PROJECTS

The use of Revit was discussed with multiple engineers from a reputable structural engineering firm. Some insight was gained from these engineers' experience with using Revit on real construction projects within the New York metropolitan area. The following includes a summary of the opinions of these engineers based on their personal experiences.

General Comments From a Structural Engineering Standpoint

The material choice for the building structure typically has a direct impact on the usefulness of the program for structural engineers. For concrete buildings, the biggest advantage of using Revit for a structural engineer is mostly only as a tool to coordinate amongst the design team. Particularly in New York, it seems that the industry's concrete contractors do not have an interest in using Revit. It is worth noting that the time requirement to build a Revit model for a concrete building is typically less than that for a steel building.

If the structure of a steel building is more complex, however, Revit becomes more useful as a tool for the structural engineer. Revit's three-dimensional model assists in the observation of the complex geometries of the project. It seems that the steel subcontractors in the local industry commonly use TEKLA 3D models for their purposes, as opposed to Revit. From experience with viewing a TEKLA 3D model, it was extremely helpful in understanding the complexity of structure that occurred at different locations. These aspects of the project were easily overlooked without the help of the 3D model. Using the model to locate and understand these conditions provided valuable insight into how to design, detail, and ultimately construct the varying conditions. This is a testament to usefulness of a 3D model, such as one produced through Revit or TEKLA.

Regardless of the material type, it was recommended to start the model as late in the design phase as possible. The reason for this is so the structural engineer has as much information as possible when starting the model. Since it is very time consuming to build a model as compared to producing traditional AutoCAD plans and details, the goal is to avoid the many changes that are inevitable at the earliest stages of the design. As a rule of thumb, it was suggested to not start a Revit model earlier than the Design Development phase of a project. The Schematic Design phase is too early in the process with too many variables for the model to be particularly useful and developed efficiently.

One additional possible use for Revit models is as a marketing tool for the developer. Although this does not apply particularly to Revit Structure, as the structure is often enclosed and hidden, Revit Architecture can be very useful to the developer for this cause. Revit Structure does, however, provide some marketing value to the structural engineering firm. Since Revit Structure can provide varying views and renderings of the project's structure, these can be used when marketing the firm's project experience.

While Revit can be a very useful tool for structural engineers, it is thought that it could have a greater impact on the other fields of the building design industry. For instance, MEP engineers have various layers of mechanical, electrical, and plumbing lines that cross each other in multiple places on a single floor. Using a two-dimensional drawing to display such information is not particularly effective. A Revit model is far more advantageous by having the ability to view the different elevations of these systems in 3D. Having a Revit Structure model to compare to the Revit MEP model is also useful in determining any conflicts that the may occur between the MEP equipment and structural elements of the project. This is an example of how a structural Revit model is helpful for coordinating amongst the other design consultants.

Issues With Using Revit On Real Construction Projects

While Revit is a very useful tool with a lot of potential, it seems from experience that using Revit on real construction projects changes the natural progression of the project. The problem does not seem to stem so much from the structural engineer using Revit Structure, but more from the architect using Revit Architecture. The reason for the changes to a project's development is that Revit Architecture demands a very large quantity of detailed information from an architect in order to create a model. This demand for so much detail changes the order by which architectural information is typically shared.

For both previous and currently ongoing projects, some architects have been caught up in some of the lesser details of a project because these details are necessary to produce the architectural model. In turn, developing these details has affected the flow of information by slowing down the normal progression of architectural design. Without receiving the necessary architectural information to design their aspects of a project, the structural engineer's and other design consultants' production is delayed. These delays inevitably affect the construction schedule of the project by forcing a slower schedule in comparison to the current project development process when only using AutoCAD to produce construction documents.

Revit has many potential connectivity abilities that would allow the user to streamline the process of taking information from one place and using it elsewhere. For instance, the program has the potential to export the structural model into other structural analysis and design programs. Through the use of this process, a structural engineer would be able to greatly reduce the amount of time it currently takes to create a separate model in each of these analysis and design programs. Unfortunately at this point in time, however, it seems that the industry has not yet been able to fully take advantage of abilities such as these. The programs have not been able to connect in the ideal way intended, and there has been a need to develop workarounds for exporting the model.

Another potential connectivity ability would help quicken the process of sharing information between multiple design consultants. For instance, structural engineers could ideally link their models with architectural and MEP models. Doing so would allow the user to automatically update the model to include the most recent changes made to the architectural and MEP models. The respective consultants on current and past projects, however, have not yet been able to link their models in this way. The potential gains from working out the kinks of this ability are immense, as the flow of information among the design team is one of the most important aspects of a project in design.

Multiple users are also potentially able to work on a single model at the same time within Revit. This has been an unresolved issue with AutoCAD in that two different users could not access the same drawing simultaneously to make changes. At best, one user can access the original drawing file, while a different user must work on a copy of the file. The modifications made to the copy then need to be pulled from the copy and placed on the original drawing file. Revit has the potential ability for multiple users to work on a single model file through a series of permissions and releases that restrict which parts of the model different users can modify. While the idea is progressive, it has not proven to be particularly practical on actual projects. It seems that users are constantly required to change the permissions and releases settings that it becomes more burdensome than advantageous for multiple users to simultaneously work on a single model.

The actual tools for producing construction documents within Revit have not yet been fully developed enough for the industry to comfortably use Revit alone in creating construction documents. On previous projects, structural engineers have had to create the final drawing sheets in AutoCAD, as opposed to Revit. The reason for this is that since the model has such an immense amount of information within it. It is difficult or seemingly impossible at times to display only the desired portions of projects within the viewing areas. Even trained expert Revit modelers have had to come up with workarounds for attempting to display the desired information.

A liability issue exists with the prospect of a designer, such as an engineer, handing over a three-dimensional model to a contractor. The liability is in the contractor using information that can be taken from the model that is not under the control of the designer. Similar liability exists with releasing electronic AutoCAD drawings to a contractor. The designers attempt to deal with this issue by using legal verbiage that restricts the contractor from taking electronic information at face value. For example, designers often include a note with their drawings stating that the drawings may not be scaled to retrieve dimensions. Only dimensions that are specifically shown for the elements that have been designed by the designer may be used by the contractor.

The amount of information and liability, however, increases greatly in magnitude with releasing a three-dimensional model because of the larger amount of information that can be extruded from it. While the legal aspects of the liability can potentially be worked out through contract and specification writing, the reliance of the contractor when producing the model will be much more difficult to monitor. If the liability issues can be resolved, however, the three-dimensional model can serve as a significant tool by which all trades involved can improve the efficiency and quality of their work.

Heavy reliance on the three-dimensional model is not only limited to contractors in terms of being a disadvantage of using Revit. Some design consultants have shown a tendency to remove their impact on the coordination required when designing a project by relying on the model too heavily. While the model is a useful tool for determining conflicts within a project, it is neither infallible nor all-knowing. There is an irreplaceable need for a professional to be present and active in finding and solving a project's problems. Every project will present new and different challenges to overcome. Without actively participating in understanding the intricacies of each project and how the different design participants must interact, the designer merely becomes a technical operator of a design tool.

18. CONCLUSIONS AND RECOMMENDATIONS

Revit Structure is a powerful tool that structural engineers can use to develop a three-dimensional model to aid in the design of a project, and develop the documents required to construct a project. In order to maximize the value of using Revit Structure, the user must fully understand the tools and abilities of the program, as well as the process of producing a project through completion. By having a greater understanding for the process by which projects are created, modelers can produce a working model in an efficient manner. While there are disadvantages to using such a tool in the lieu of older more established methods such as producing traditional AutoCAD drawings, there are also many advantages to be gained.

The aforementioned issues cannot be overlooked when intending to use Revit on a real construction project, especially considering that improved coordination is supposed to be one of the program's greatest advantages. Future releases from Autodesk may help fix any bugs or limitations that are preventing users from exploring the full potential of the program. Autodesk is working on particular issues to resolve some of the program's shortcomings. There are intended release dates in the upcoming years for a series of updates that are aimed at remedying the program's pitfalls.

Perhaps some of the issues can also be resolved through more experience and comfort with the program. As the case seems to be based on actual projects employing the program, the users of the program within the industry are not quite there yet. Due to the perpetual state of rush that the industry faces, many users of the program have not yet been able to master the program's abilities. Organizations are, however, aiding in the spread of information that will lead to more productive and efficient use of BIM programs. Books are being released, seminars are being held, and articles are being distributed to promote the advantages and widespread acceptance of BIM programs such as Revit within the industry through smarter use and application. Since this is a relatively new program in terms of its actual use on real construction projects, the same familiarity that has been gained with AutoCAD over decades of use has not yet been experienced with Revit.

With time, the program will be improved by new updates and editions, and the industry will become more familiar with its use. Just as AutoCAD had become the industry standard over hand drawn documents by overcoming the initial obstacles it faced, three-dimensional BIM programs such as Revit will lead the way to a new industry standard by providing unique and remarkable advantages to the construction industry.

REFERENCES

Autodesk. (2007). Maintaining BIM Integrity in the Structural Engineering Office. Retrieved from http://images.autodesk.com/adsk/files/maintaining_bim_integrity_in_structural_engineering_office.pdf

Autodesk. (2007). Revit Structure and BIM. Retrieved from http://images.autodesk.com/adsk/files/revit_structure_and_bim.pdf

Autodesk. (2007). Revit Structure Interoperability with AutoCAD. Retrieved from http://images.autodesk.com/adsk/files/revit_structure_interoperability_with_autocad.pdf

Autodesk. (2009). Revit Structure 2010. Available from http://usa.autodesk.com/adsk/servlet/pc/indexid=5523749&siteID=123112

Autodesk. (2009). Revit Structure 2010 Imperial Tutorials. Retrieved from http://usa.autodesk.com/adsk/servlet/item?siteID=123112&id=13093246&linkID=9243180

Autodesk. (2009). Revit Structure 2010 User's Guide. Retrieved from http://images.autodesk.com/adsk/files/helpstructureenu1.pdf

Autodesk. (2010). Integrating Revit Structure and Robot Structural Analysis Professional. Retrieved from http://images.autodesk.com/adsk/files/linking_revit_structure_models_with_robot_structural_analysis.pdf

Autodesk (2010). Revit Structure Extensions. Retrieved from http://students.autodesk.com/?nd=revit_struc_ext

BIMechanics LLC. (2009) BIM History. Retrieved from http://www.bimechanics.com/bim-history.htm

McGraw Hill Construction. (2009). Transforming Design and Construction to Achieve Greater Industry Productivity. Retrieved from http://images.autodesk.com/adsk/files/mcgrawhill_construction_bim_smartmarket_report_december_2008.pdf

McGraw Hill Construction. (2009). The Business Value of BIM. Retrieved from http://images.autodesk.com/adsk/files/final_2009_bim_smartmarket_report.pdf

San, M. (2005). BIM - What is it, why do I care, and how do I do it? Breaking Down The Walls. Retrieved from http://modocrmadt.blogspot.com/2005/01/bim-what-is-it-why-do-i-care-and-how.html

Srinivas, H. Looking at the Big Picture: Life Cycle Thinking for the Building and Construction Sector. Green Construction. Retrieved from http://www.gdrc.org/uem/green-const/sbc-lca.html