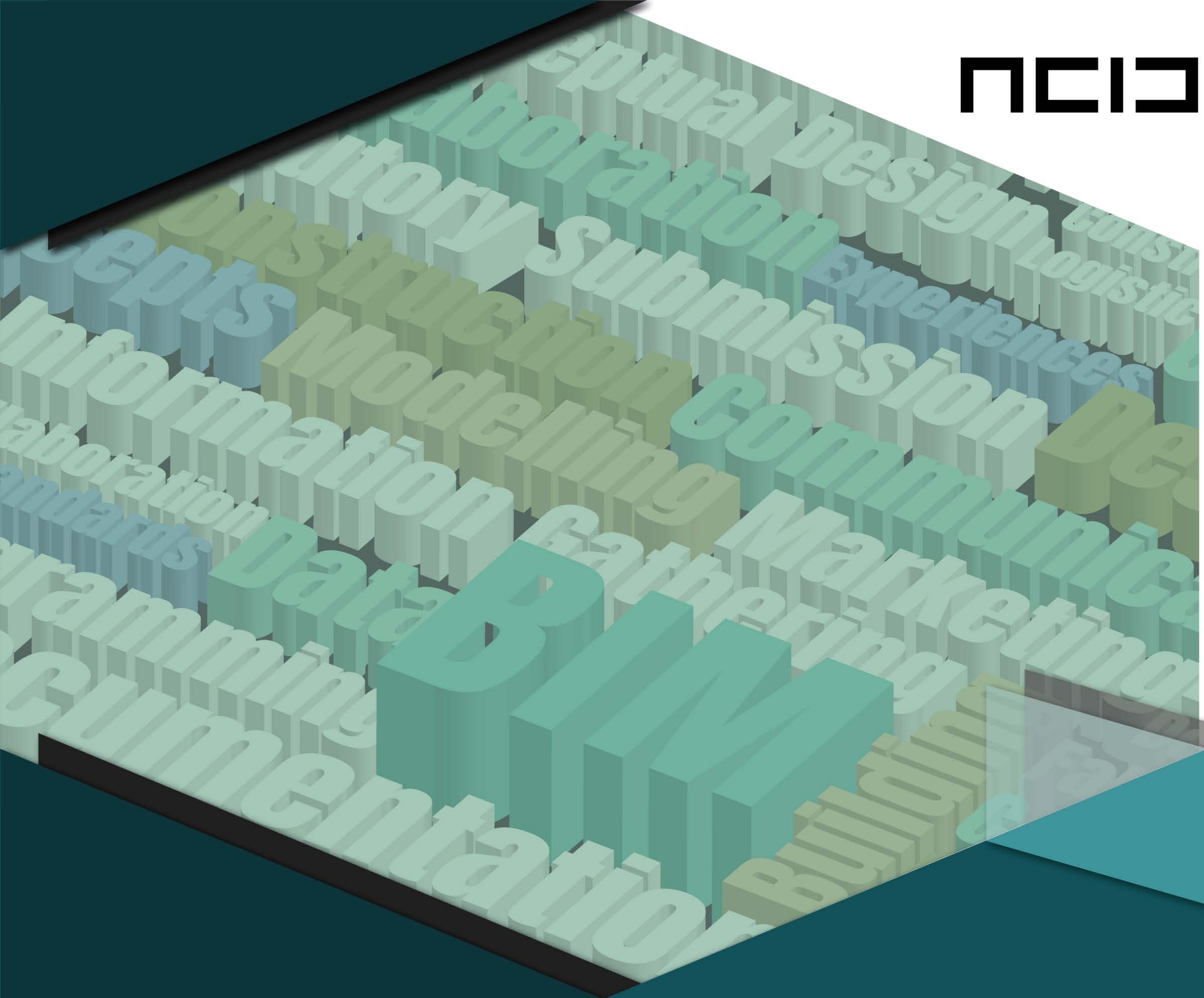


BUILDING INFORMATION MODELLING

Course Notes

Advanced Modelling Course (Structure) - Revit 2018

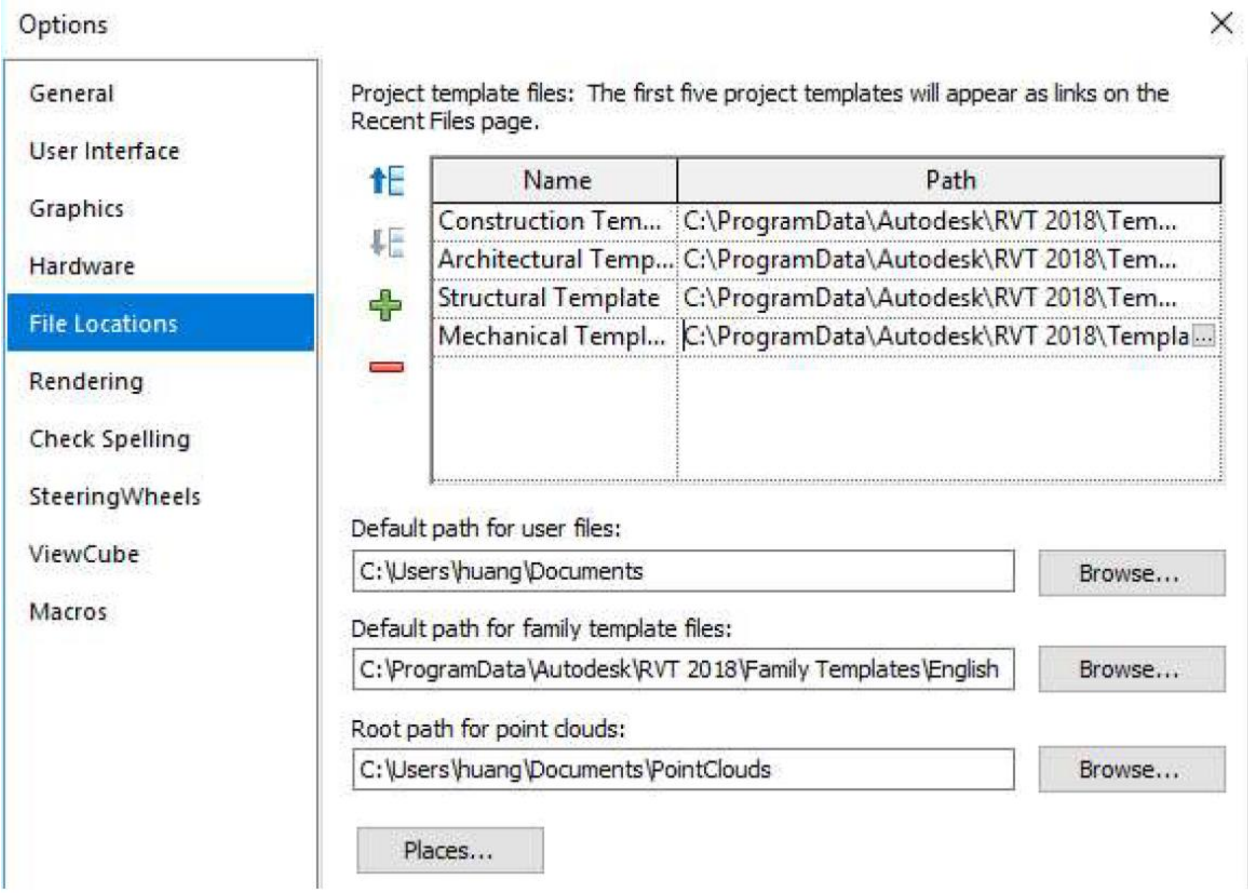
Advanced Construction Information Development Ltd.

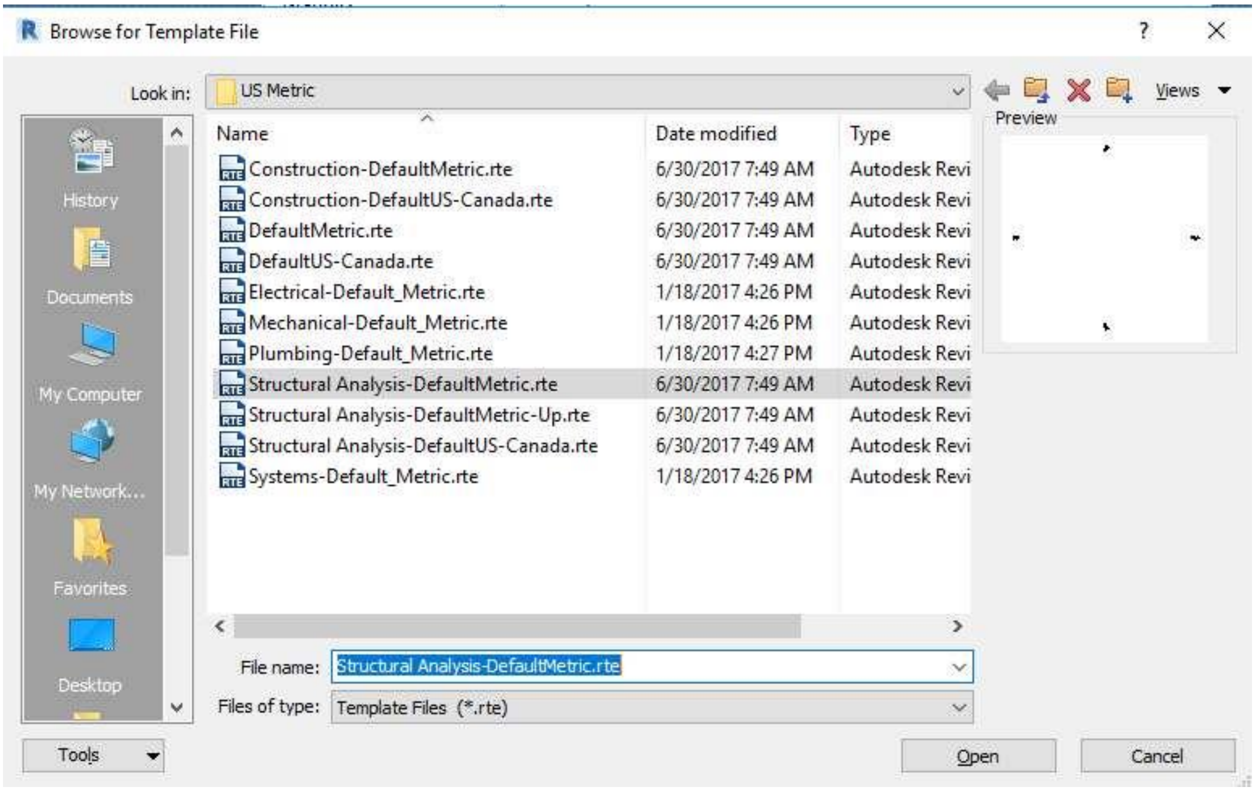
Table of Contents

CHAPTER 1: STRUCTURE PROJECT START UP

1.1. Creating an Architectural Underlay

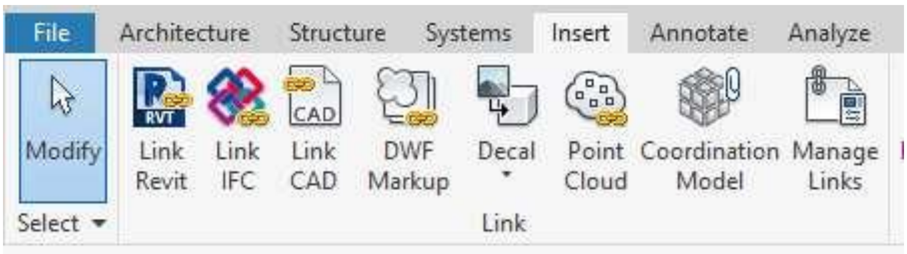
1. Set the location for Default template files in Option



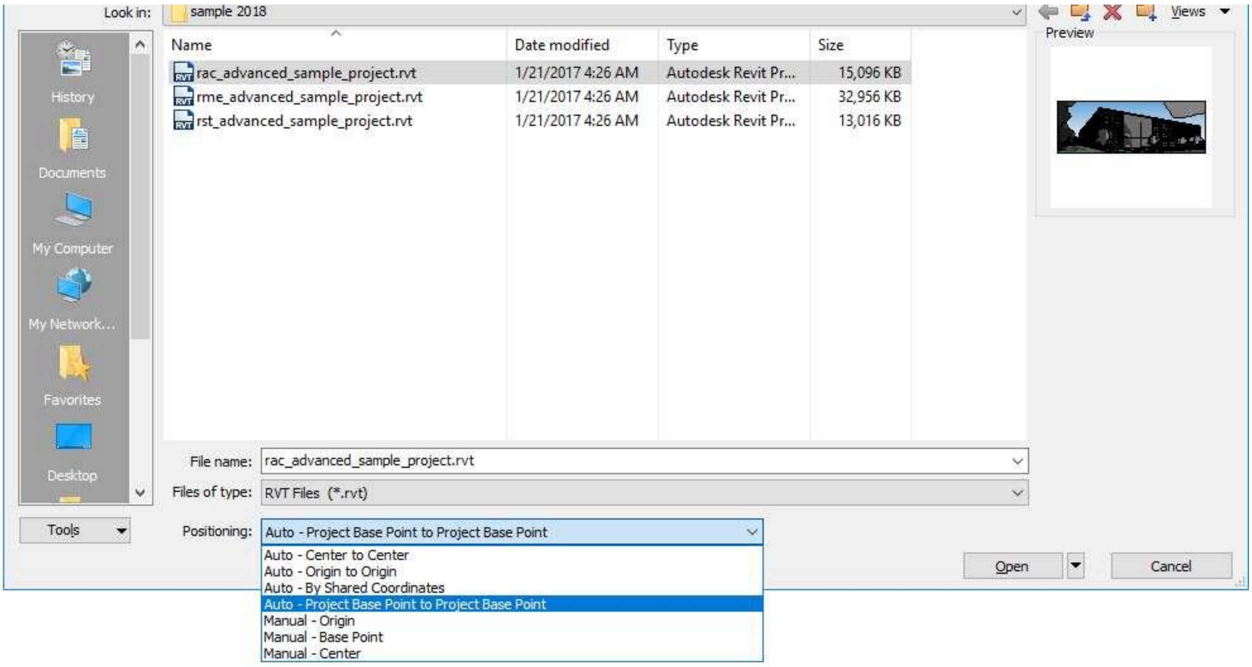


2. Create a New project from Structural Template

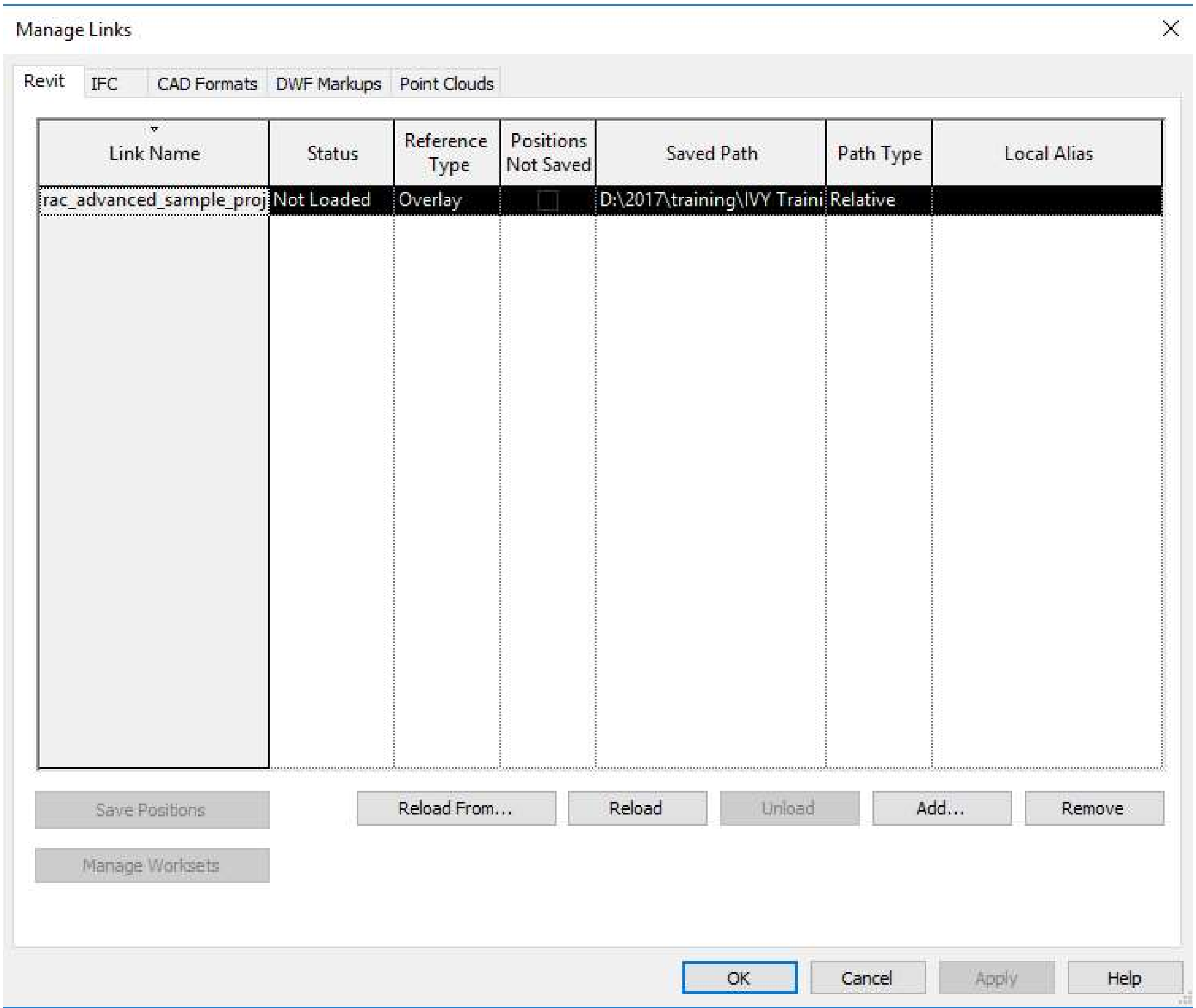
3. Link Architecture Model by Click Link Revit



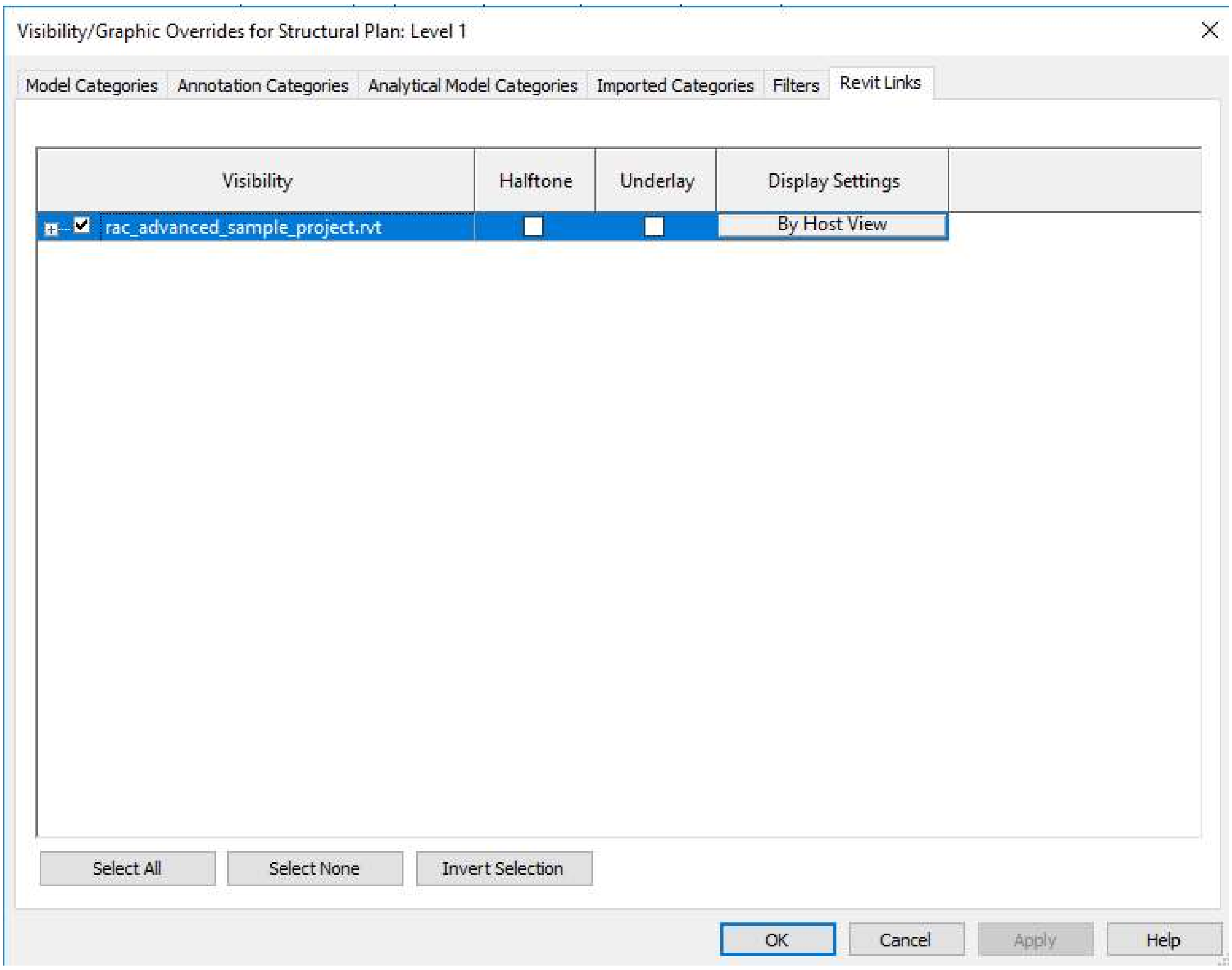
4. Choose Positioning



5. Reload or Unload Links by Manage Links



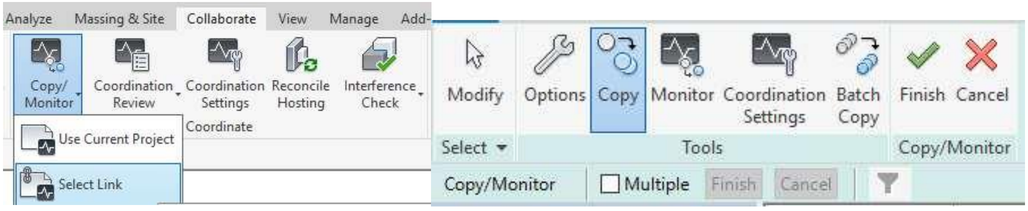
6. Set the Visibility of Linked Models

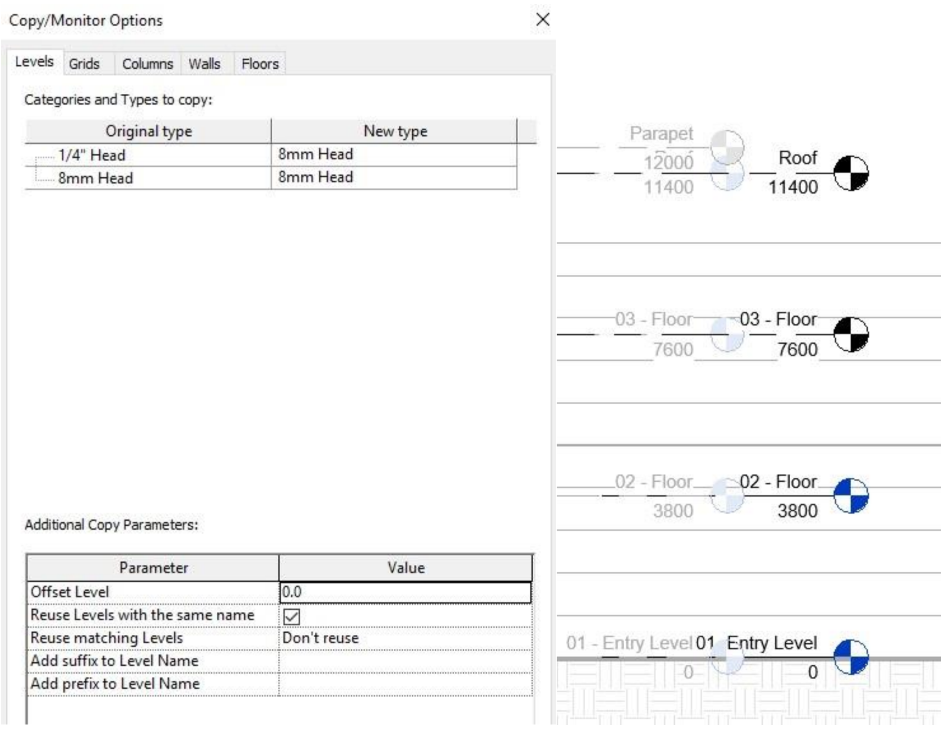


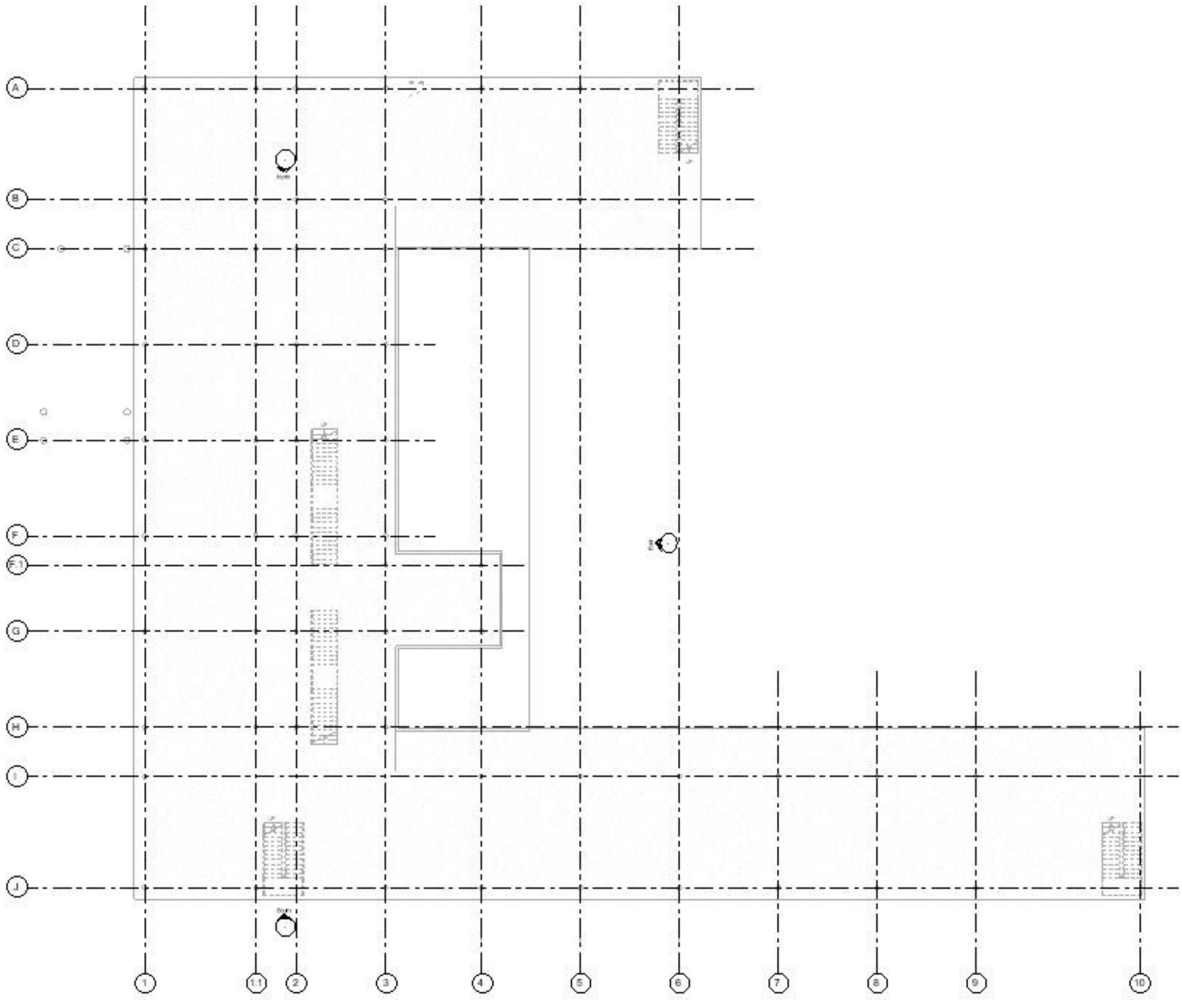
The Revit Links tab in the Visibility/Graphics dialog contains the following columns:

- Visibility: Select the check box to show the linked model in the view, or clear the check box to hide the linked model.
- Halftone: Select the check box to draw the linked model in halftone.
- Underlay: Select the check box to display the linked model as an underlay in the project. The geometry will display in halftone and will not obscure new lines and edges you draw in the project.
- Note: Elements in a model set as an underlay are not eligible for tagging. Optionally, in the Visibility/Graphic Overrides dialog, specify the model as "Halftone" instead of "Underlay" to display the model in halftone and allow tagging of model elements.
- Display Settings: Options to override additional settings for each linked model in the current host view. The button displays the current display setting state (By Host View, By Linked View, or Custom).

7. Copy/ Monitor Levels and Grids







Edit Link Model

To update linked models without closing the current project, you can reload the linked models. Unload linked models to temporarily remove them from the project.

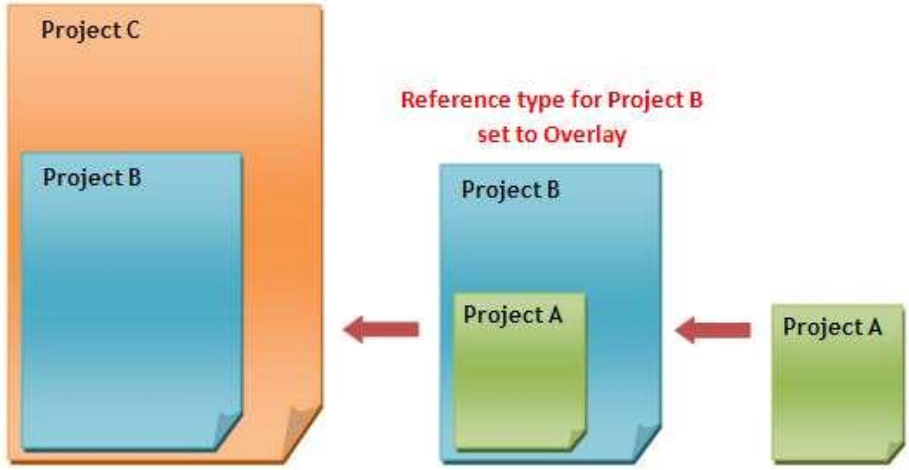
Click Manage tab > Manage Project panel > (Manage Links).

1. In the Manage Links dialog, click the relevant tab.
2. Select the linked model.
3. To unload the selected model, click Unload. Click Yes to confirm.
4. Note: It is not required that you unload a linked model before reloading.
5. To reload the selected model, click Reload.

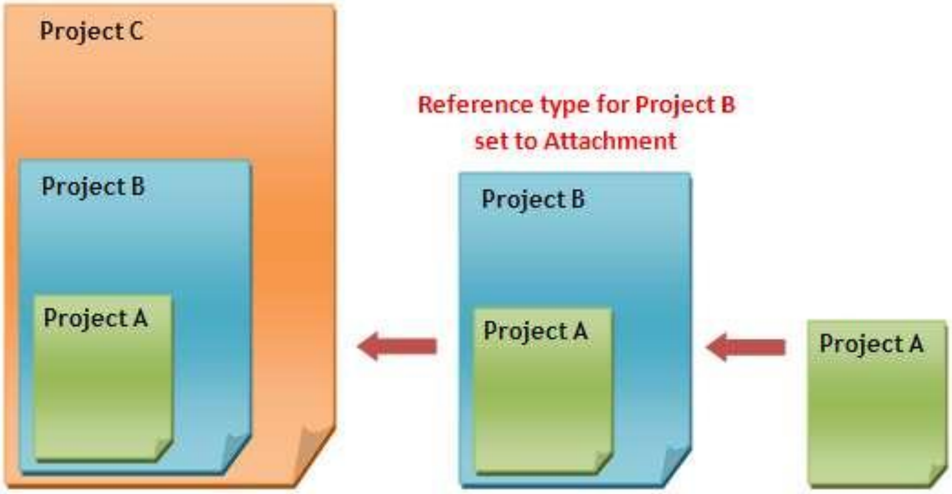
Reference Type

Reference Type has two types: Overlay and Attachment.

When you import a model that contains linked models, the nested links display according to the Reference Type setting in their parent model.

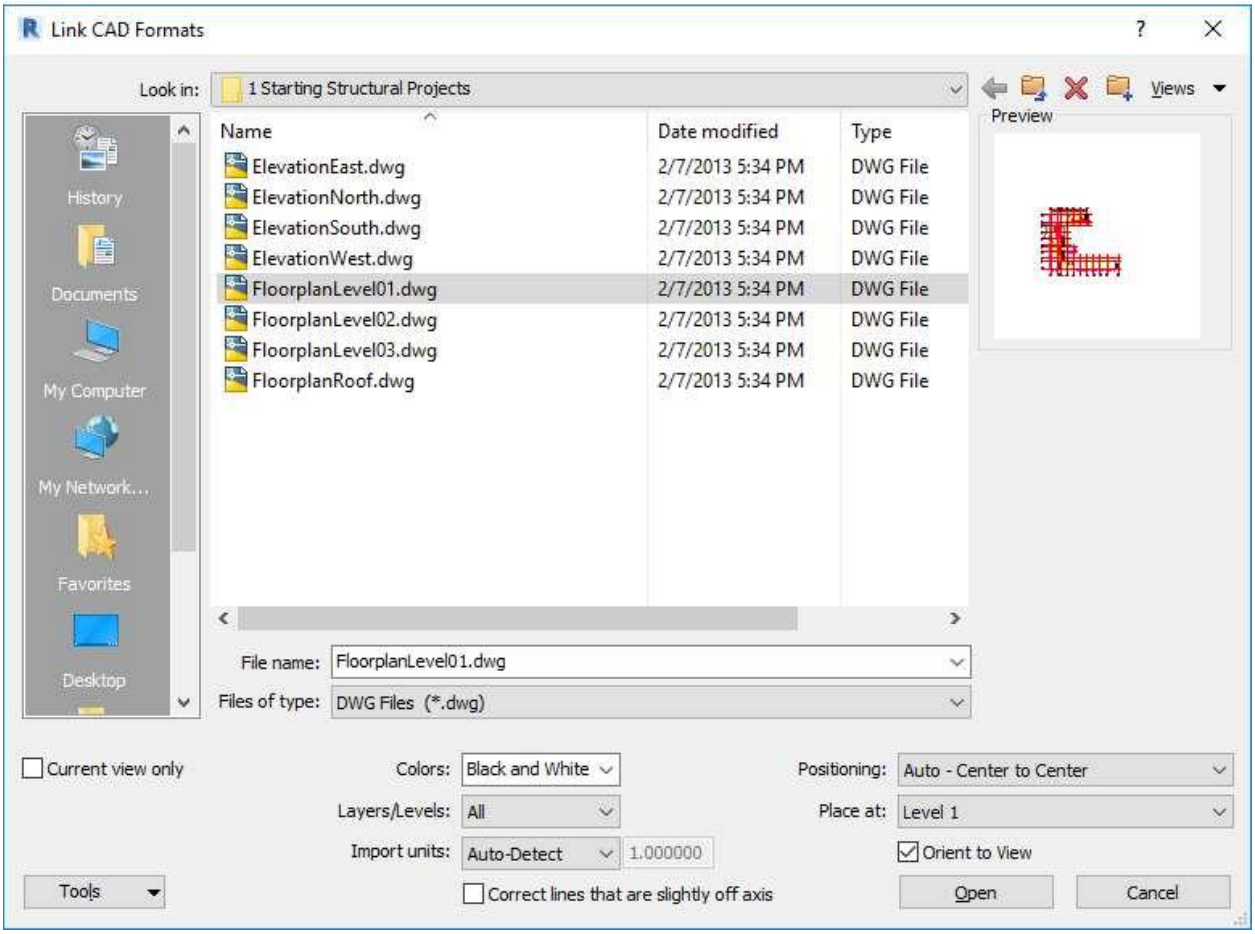


If you change the Reference Type setting for Project B to Attachment, when you import Project B into Project C, the nested link (Project A) displays.

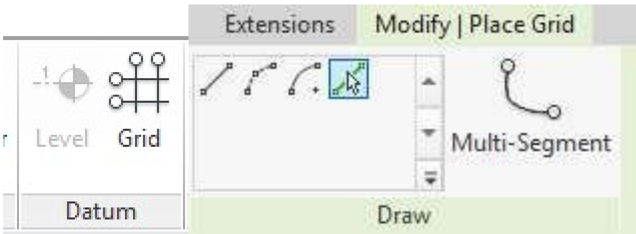


1.2 Importing and Linking CAD files

1. Import Floor plan CAD for Level 1

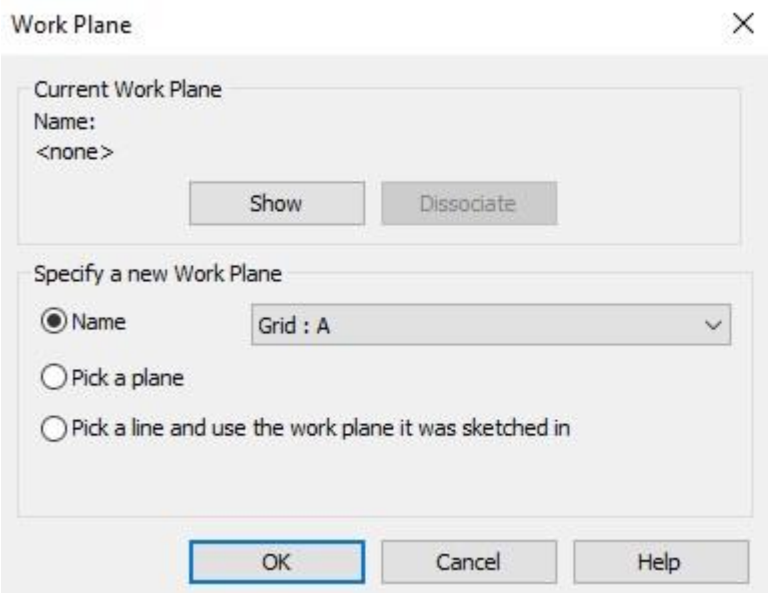


2. Add Grids Line



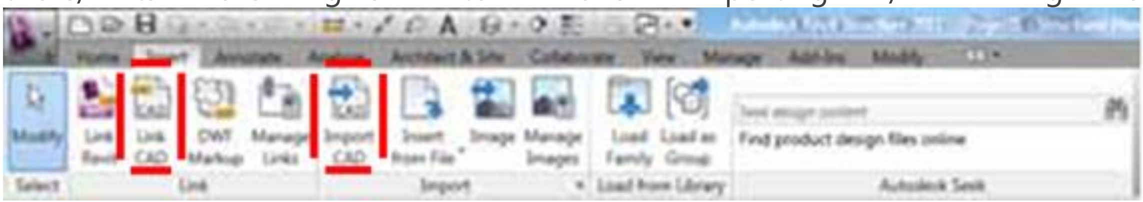
3. Import Floorplan Level02 and Align to Level01

4. Go to the North Elevation
5. Set the Grid A as Work Plane
6. Link ElevationNorth.dwg
7. Align Grid and Level
8. Go to the East Elevation
9. Align CAD to Grid A



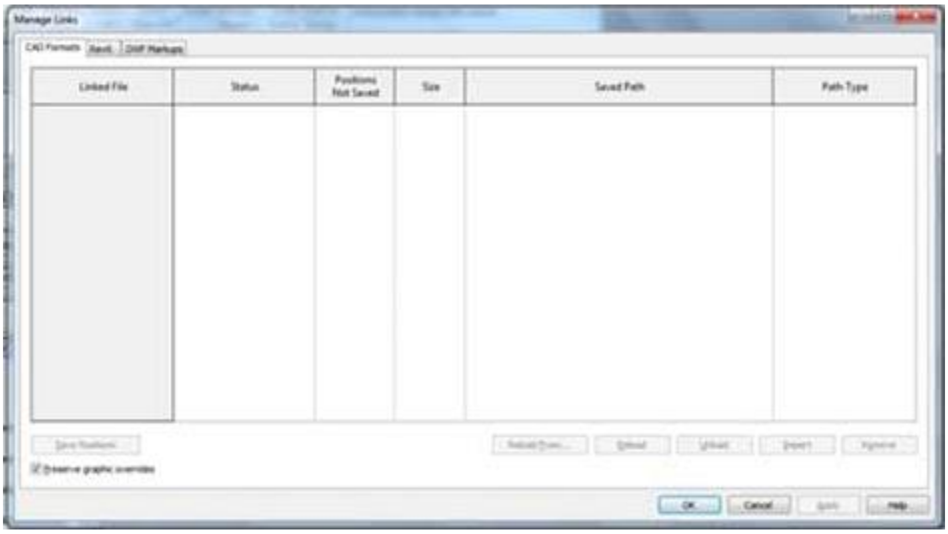
Importing / Linking Drawing Files

Importing / Linking Drawing Files are essential when a project is not 100% in Revit by all disciplines. It is also the normal industry practice that deliverables by some professional services are still in CAD format, such as Topographic Survey Plan. Due to some limitation in Revit for the drawing import/link, users always find difficulties in the process. Here, steps for the Importing / Linking are explained. Also, some tricks/hints are given to make Importing / Linking easy.



To make it simple, first of all, Importing and Linking are basically doing the same thing, i.e. to Import a CAD drawing into Revit to use. The only difference for Linking is that Revit keeps the path of the CAD file such that when the Revit Model is loaded, the CAD file is re-loaded. However, Revit do store the content of the CAD drawing in the Revit Model. In case if the CAD file is not found while it is re-loaded, the CAD drawing can still be seen in the Revit Model.

The benefit of Linking is that Links can be managed. That means, the CAD file can be Reloaded, Unloaded or Removed. The path can also be changed by using Reload From. This is in particular good when users want to remove old imported drawings but forget which view the drawing is added. The CAD drawing can be removed in Manage Links.

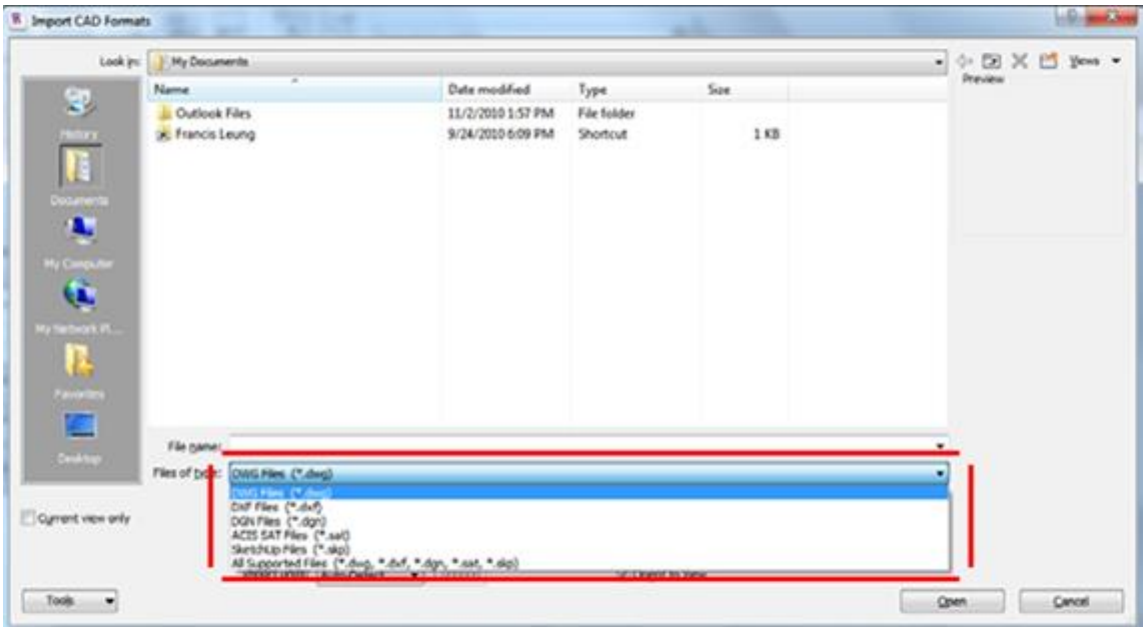


In some cases, users may find it frustrated once the linked file is not found when Revit Model is opened. A warning message comes out to ask if Manage Link is opened to correct or it is ignored. Although it can just be ignored and the drawing can still be used, new users may think there is some problems with the Revit Model or they have done something wrong.

Since Importing and Linking are doing the same thing except the “path”, the description below will only use Importing as reference. Users may do the same for Linking if it is needed.

CAD Formats

Revit accepts different types of CAD formats including “.dwg”, “.dxf”, “.dgn”, “.sat”, “.skp”. Some of these are CAD are in 3D and some are in 2D. Basically, the import for CAD for different CAD formats are same.

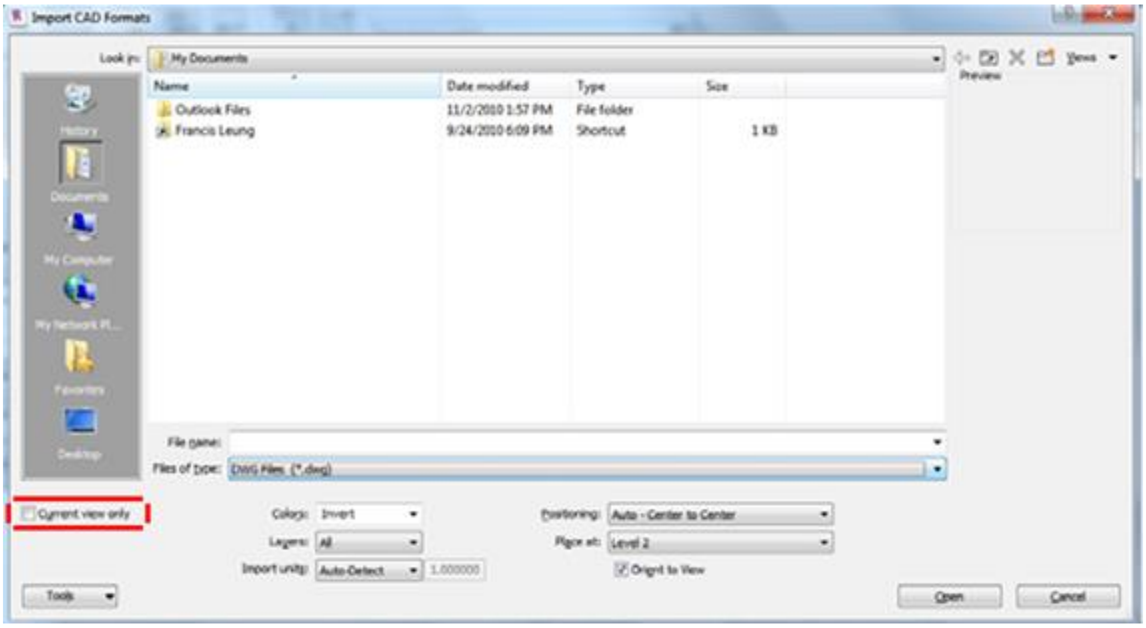


2D / 3D

CAD can be imported as 3D which is similar to Model objects in Revit and can be seen in different views. CAD can also be imported as 2D which is view related. The advantage of importing to 3D is that the drawings can be presented in more than 1 View. It can also be seen in 3D Isometric Views or Camera Perspective Views. The disadvantage of importing to 3D is that it would slow down the operation. The CAD will need to be re-generated when the View is re-generated even if the CAD is out of the View Range.

When the CAD is imported as 2D, the Z-dimension in the CAD will be ignored. That means, even if the CAD is drawn in 3D, it will become real 2D in Revit. For example, if a Topographic Plan with 3D Contour is imported as 2D, the 3D properties will lose.

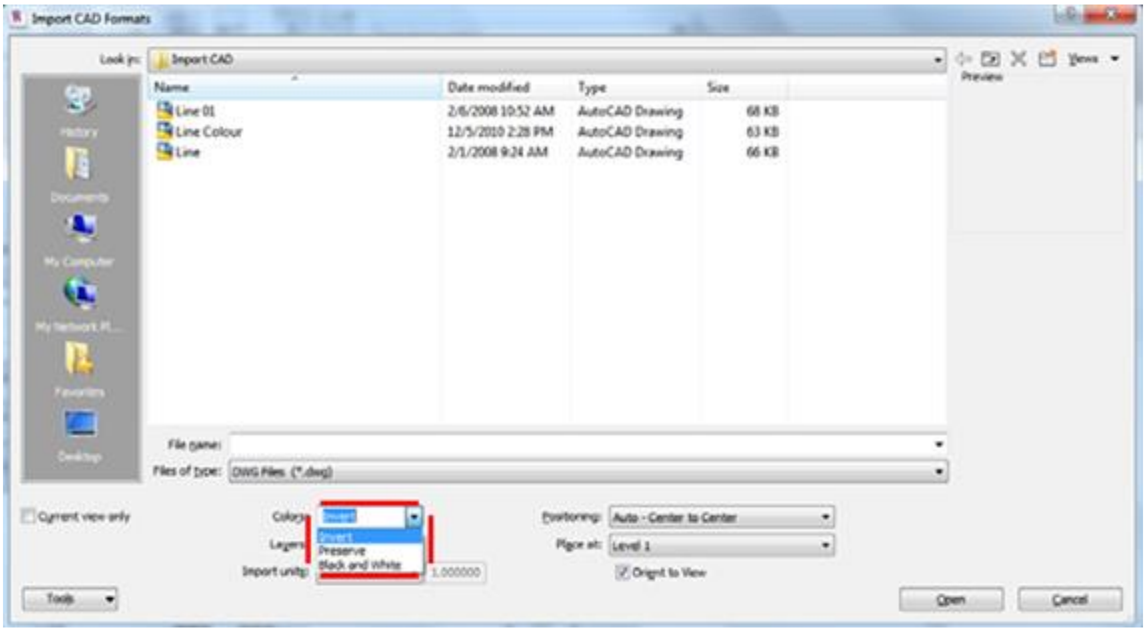
Hints: Most of the drawings should be imported as 2D (Current view only) unless real 3D CAD or Topographic Plan with 3D Contour that will be used for modelling purpose.

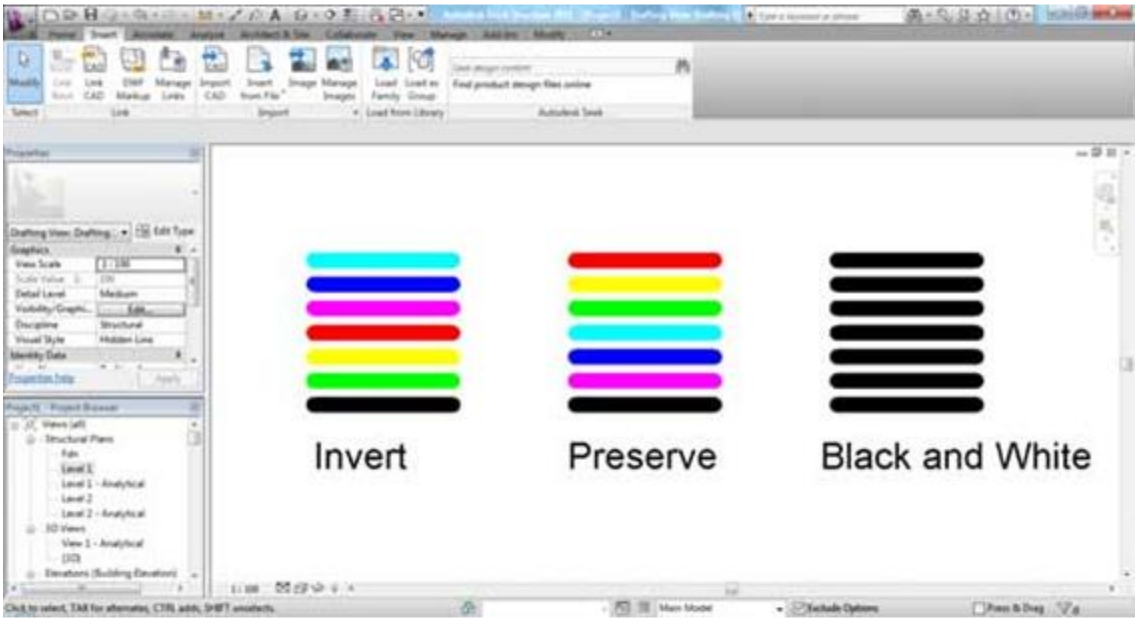


Colors

Colors are colours of the drawings and layers after the CAD is imported. 3 Options are available (Invert, Preserve and Black and White). Invert means the colour will be changed such as Colour 1 (Red) becomes Colour 4 (Cyan). Preserve maintains the same colour in CAD. Black and White will set all colours to black.

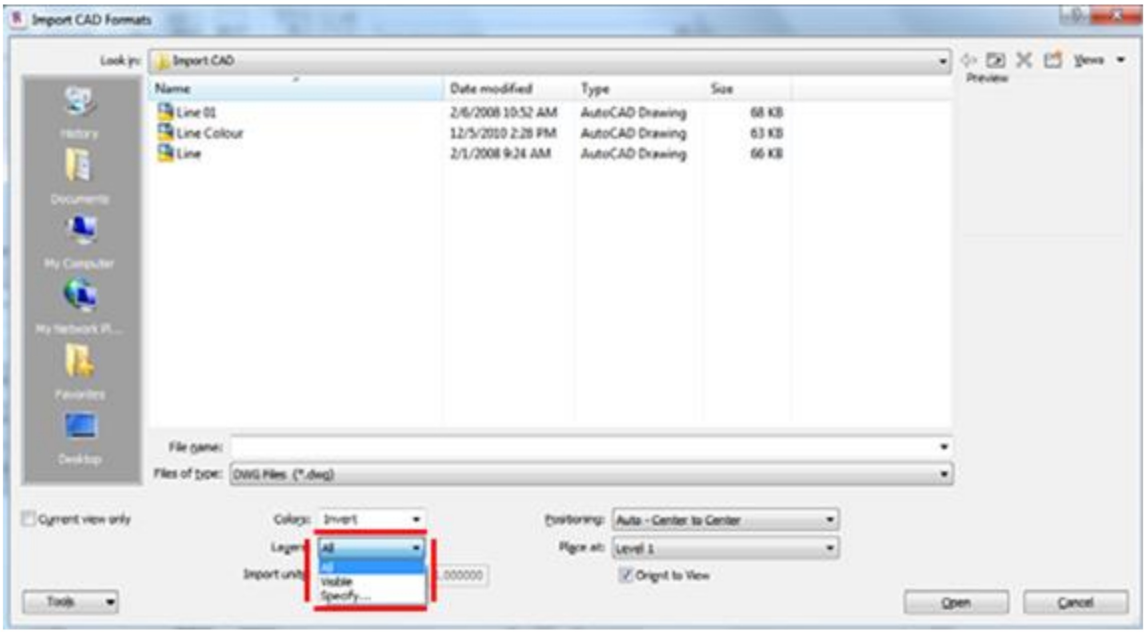
There is no standard about which colour setting should be used. In normal practice, layer colours will be overridden in View Visibility or Object Style. So, it is not having much difference in the colour selection.





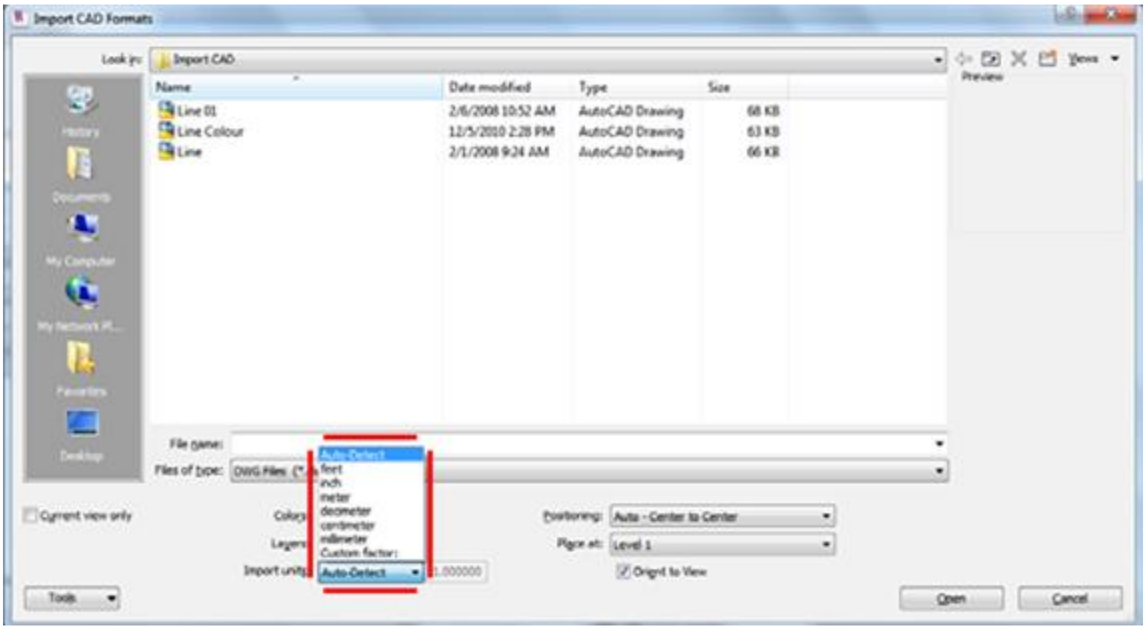
Layers

The CAD can be imported with All, Visible or Specify layers. In case if only some layers are needed from the CAD, Visible and Specify should be chosen to reduce the size of imported CAD. Otherwise, normally All layers are imported. On and off of layers can be controlled in Visibility.

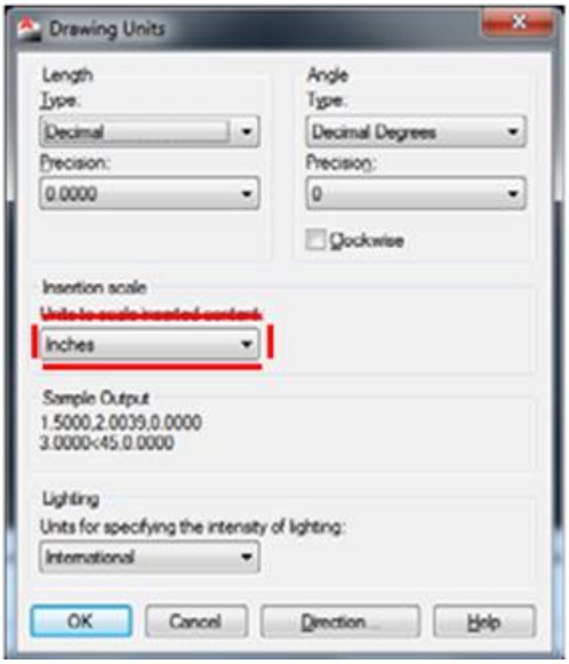


Import units

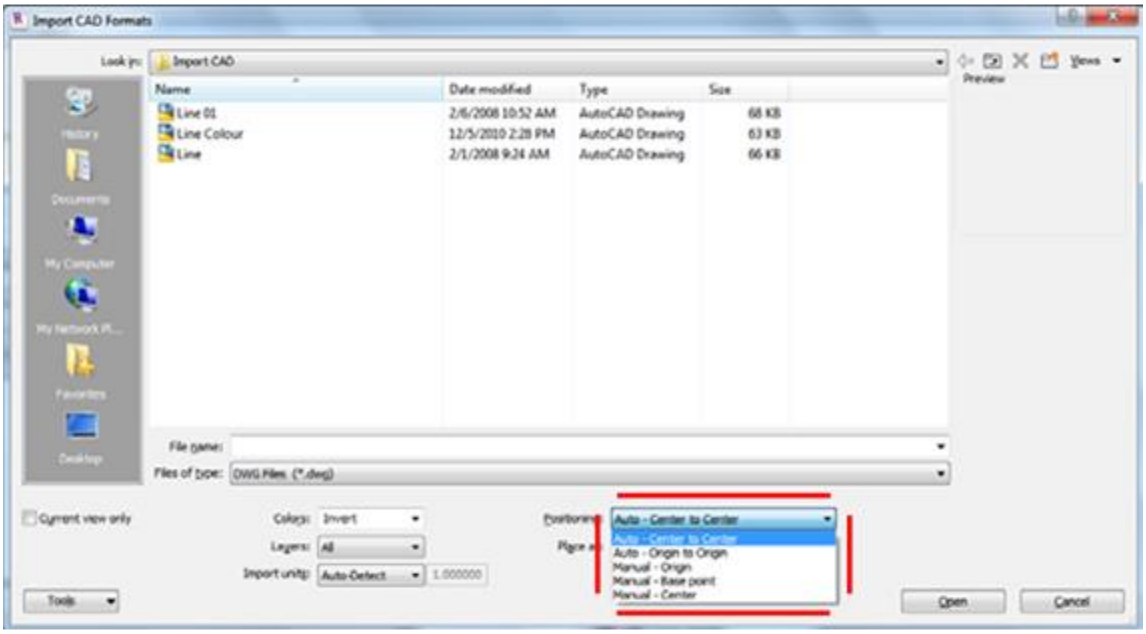
Import Units is used to define the scale of import for the CAD. The default value is Auto-Detect which reads the value of variables in the CAD file.



Say, for example, when AutoCAD drawing is imported, the settings in Units will be read. If the Insertion Scale is “Inches”, the CAD will be imported with a scale that 1 unit in CAD is 1 inch in Revit Model. If Insertion Scale is “Unitless”, “feet” is assumed. There is also an option of “Custom factor:” in the Import Units such that user can input a preferred scale. So, if the drawing scale is known, it is highly recommended to select the right scale instead of just leaving it with “Auto-Detect”.



Positioning



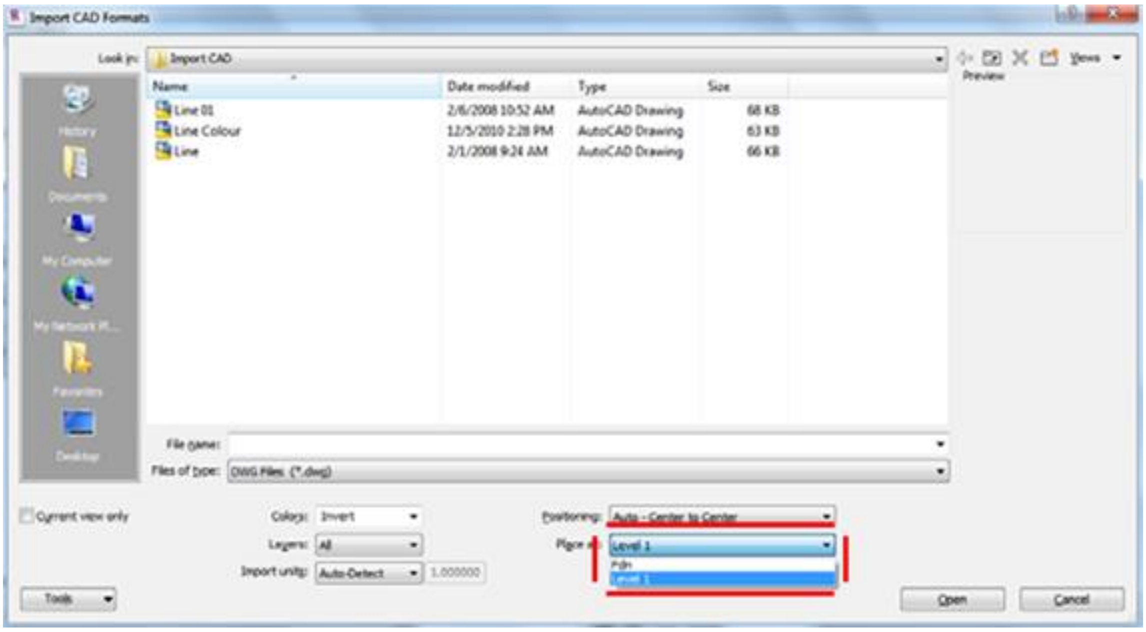
The CAD drawings can be automatically placed to “Centre to Centre” or “Origin to Origin” when it is imported. “Centre to Centre” means the “Centre” of CAD drawing is placed at the “Centre” of the Model on plan. Since the “Centre” of Model counts all 3D objects like Reference Plane, it may not be easily control the coordinates of the CAD in relation to the Model. However, the CAD can be easily found after it is imported.

“Origin to Origin” means the Origin (0, 0, 0) in CAD is placed at the Origin of the Revit Model. Users may note that it may not be easy to locate the Origin of a Revit Model. Also, there is a practice of using True Coordinates in CAD. So, the origin in CAD may be far away from the drawing content

The CAD can also be positioned manually using Origin, Base Point and Centre. Base Point is the value of variable “Insbase” in AutoCAD drawing. With these selections, the CAD will follow the cursor before the insertion point is picked

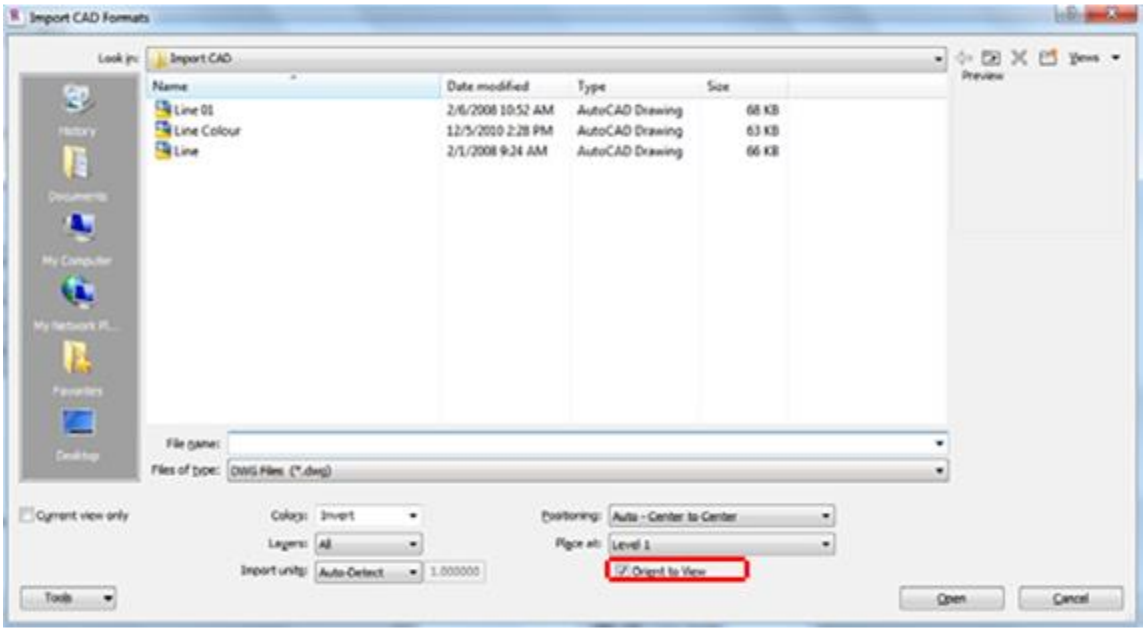
Place at

Place at is a choice of placing the CAD at the current view of the lowest level in the Revit Model. That means the “Z=0” in CAD is placed at the selected level. In case if the CAD is imported to Elevation or Section, only the lowest level can be selected. After the CAD is imported, the drawing can be moved vertically in Elevation or Section.



Orient to View

When Orient to View is ticked, the CAD X-Y plane will be oriented to the View. For example, if the CAD is imported to a Plan View, the X-Y plane is the CAD is oriented to the X-Y plane of the Model. If the CAD is imported to an Elevation, the Y-direction in the CAD is oriented to Z in the model. If this is un-ticked, X-Y plane in CAD will always be imported to X-Y plane in the Model.



Tricks / Hints

In the older version of Revit, there is a limitation of 2 miles for the CAD import. This limitation is relaxed in 2011 version. Only a message will appear when the CAD extent is over 33km to re- confirm whether it should be continued. People may say it is quite rare that the CAD drawing covers an extent of over 33km. However, it did happen when the CAD is defined in True Coordinate (and even in mm). The coordinates in Hong Kong will therefore over 800km. Of course, if there are no objects near the origin, the coordinate system will not cause trouble to the drawing extent. However, in most of the case, Block, Hatch or XRef in CAD may be mistakenly imported with some definitions remained near the origin. This will then result with a large extent.

Difference between Link & Import

Linking CAD means the CAD file has a relative relationship with Revit project, when CAD drawing is updated, the linked CAD drawing in Revit can be updated simultaneously.

Importing CAD file means the CAD drawing was pasted it directly into the Revit model, you can define the visibility of CAD file of its layers. And you can set the visibility of CAD drawing in different Revit views. In order to maintain the CAD drawing even when the Revit model was relocated, import tool is suggested.

CHAPTER 2 - STRUCTURAL COLUMNS

Structural columns are loadable families in Revit. Create/Customize structural column family will be taught later in the course. In this section, you will learn

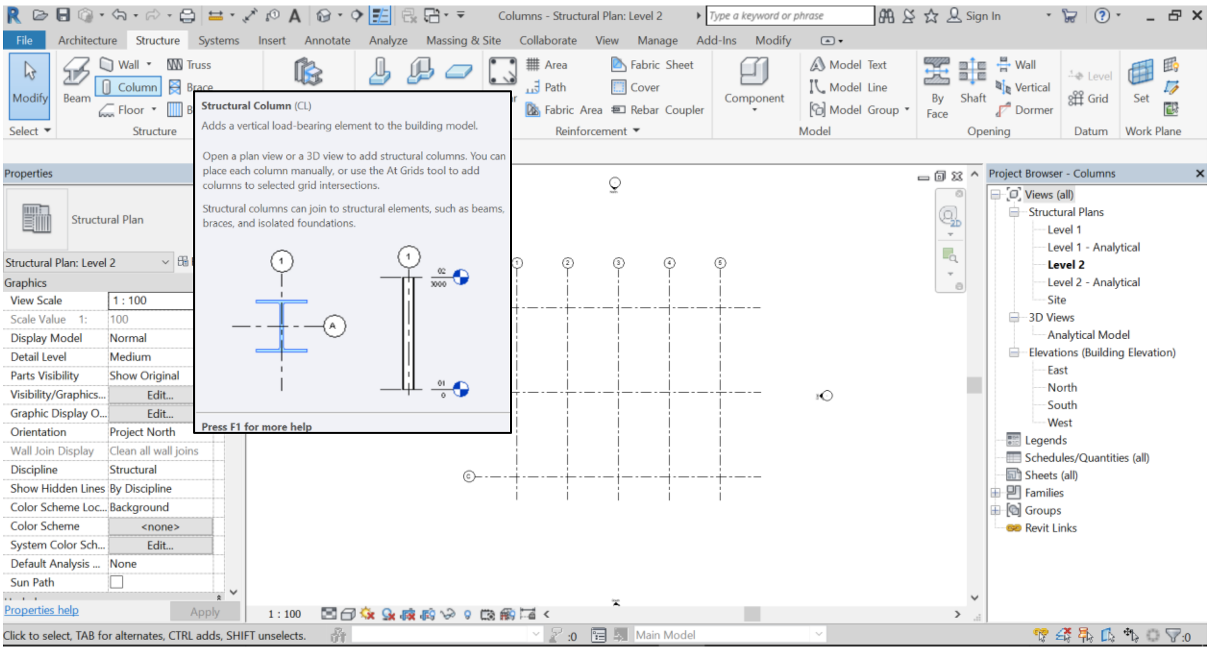
- How to add columns (vertical and slanted).
- Adjust location of a column using temporary dimensions.
- Options in column placement.
- How to create column type by duplicating existing type.
- How to modify mechanical properties of a column.

2.1 Adding Columns with default settings

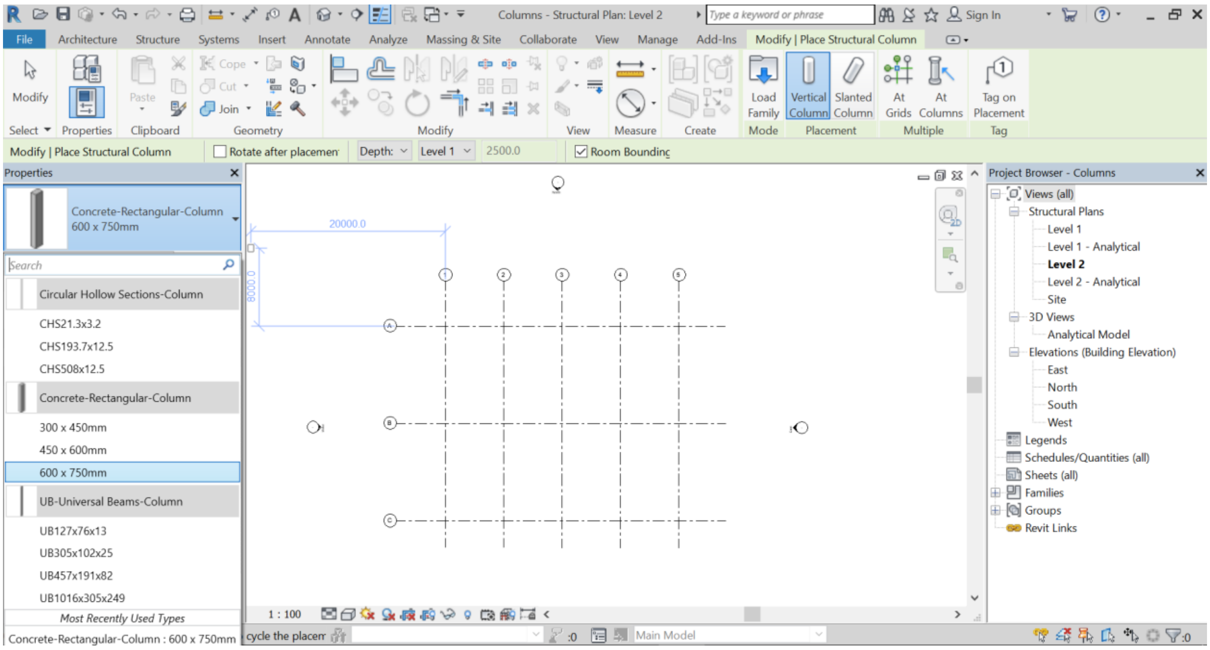
To add structural columns in Revit,

1 Open from the course folder the "Columns.rvt" file.

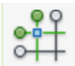
2 Click Structural Tab ➢➢ Column in Structure panel.



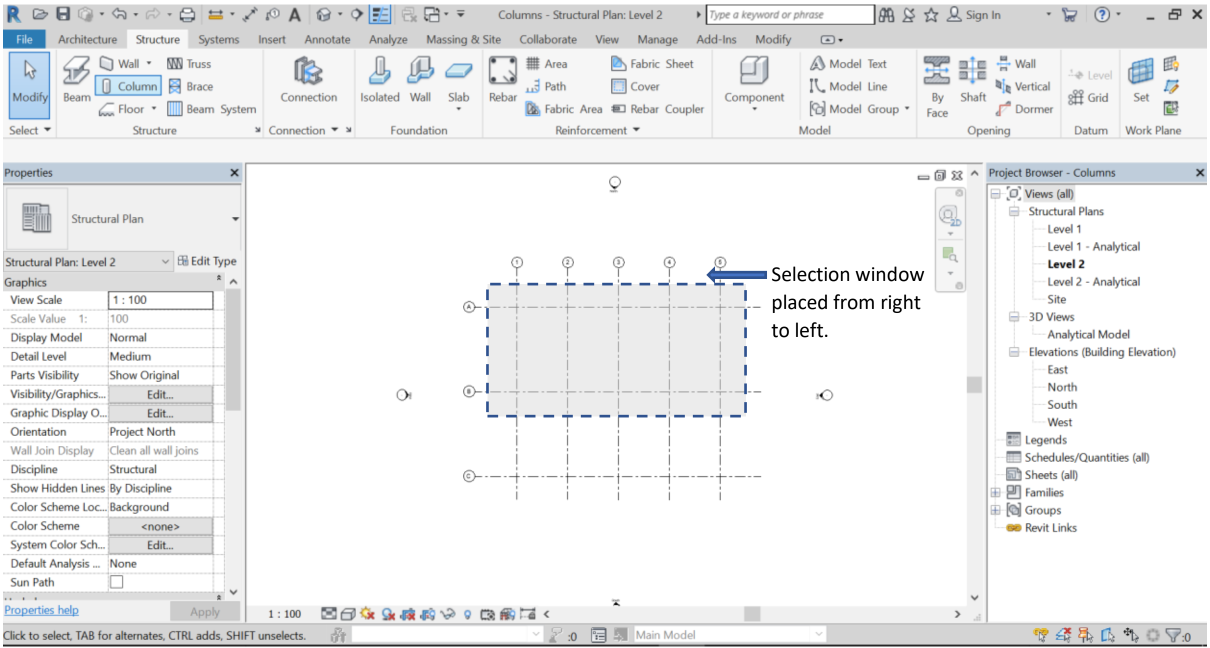
3 In the Properties Palette, select "600 x 750mm" type from Concrete-Rectangular-Column.



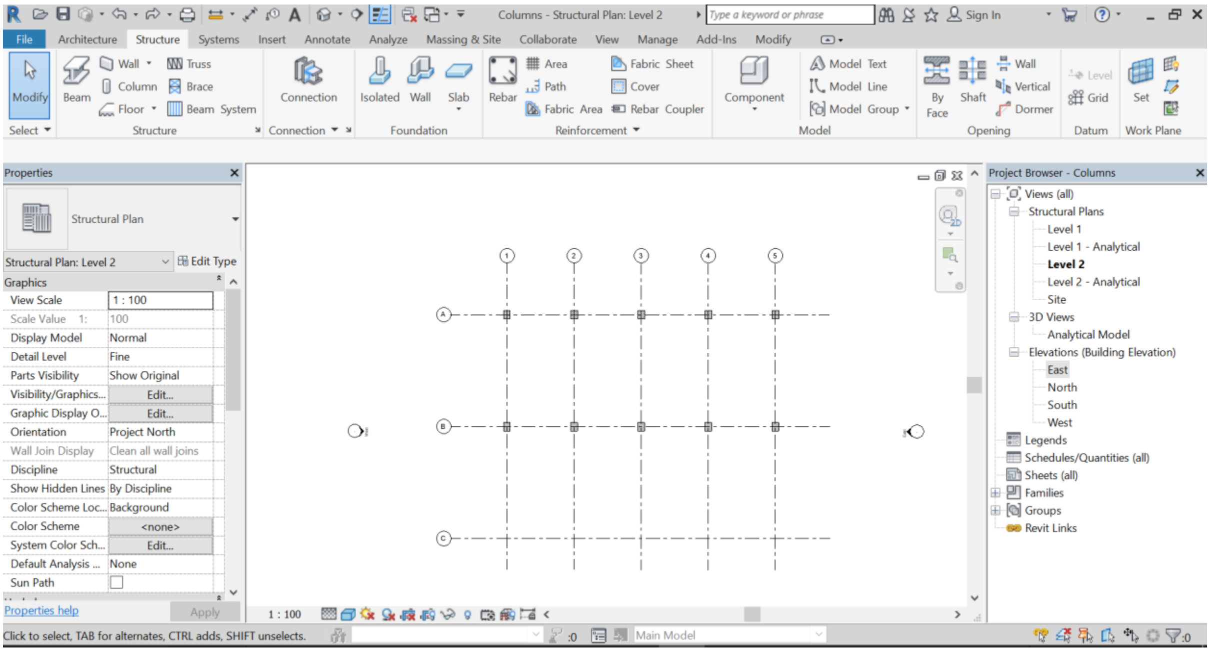
4 Keep all options unchanged.

5 Select “At Grids”  from the ribbon to place columns at grids.

6 Select Grids 1-5 and A-B.

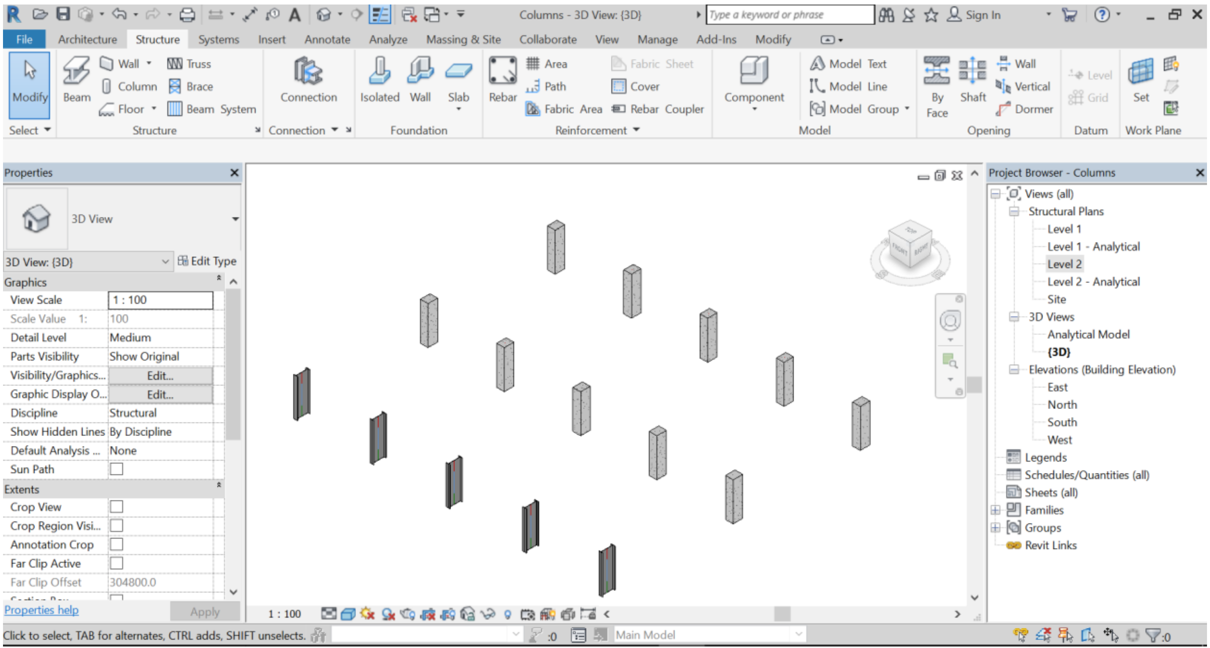


7 Click “Finish”  and “esc” a few times to complete placing the columns.



8 Place column type “UB1016x305x249” at intersections of Grids 1-5 and A.

9 Click  to view the columns placed in 3D.

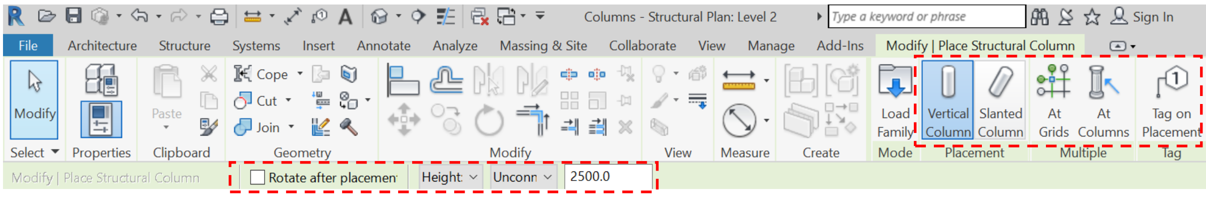


2.2 Options in Column Placement

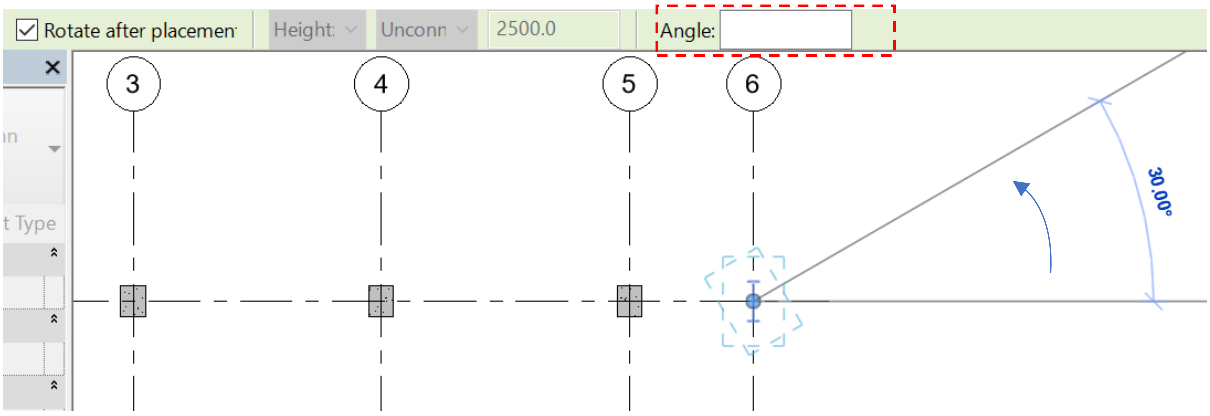
There are many options available to help you to place columns in Revit. Many of the options are self-explanatory. You can hover over the option and see its description and press F1 for more help. Under the contextual ribbon tab, the options for column placement are:

- Vertical Column / Slanted Column.
- Multiple “At Grids” / “At (Architectural) Column”.
- Tag on Placement

- Rotate after placement.
- Depth / Height and (Connected to) Specific level / Unconnected.



For vertical column, if you have checked the option "Rotate after placement", then you can either pick the angle on screen or enter it directly after you picked the column location.



2.3Adding Slanted Columns

A Slanted Column is a column that is not vertical. To add slanted columns in Revit,

1 Open from the course folder the "Slanted Column.rvt" file.

2 Click Structural Tab ➢ Column in Structural panel.

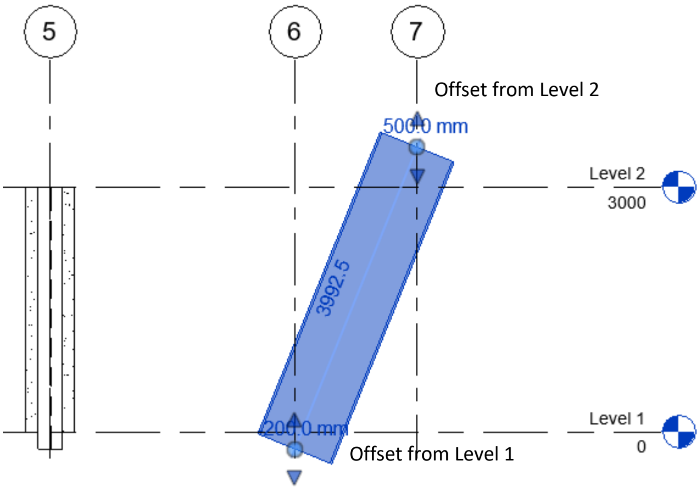
3 Click Slanted Column  on the contextual ribbon.

4 Change the offset of the 1st Click to -200 (mm) and that of the 2nd Click to 500 (mm).

5 Pick the intersection of Grids 6 and A. This is the 1st point of the slanted column and defines its base point.

6 Pick the intersection of Grids 7 and A. This is the 2nd point of the slanted column and defines its top point.

7 Add similar slanted columns at intersection of Grids 6 and B, and Grids 6 and C.



Notes *(from Autodesk Knowledge Network)*:
The following general rules apply to the procedures for placing slanted columns.

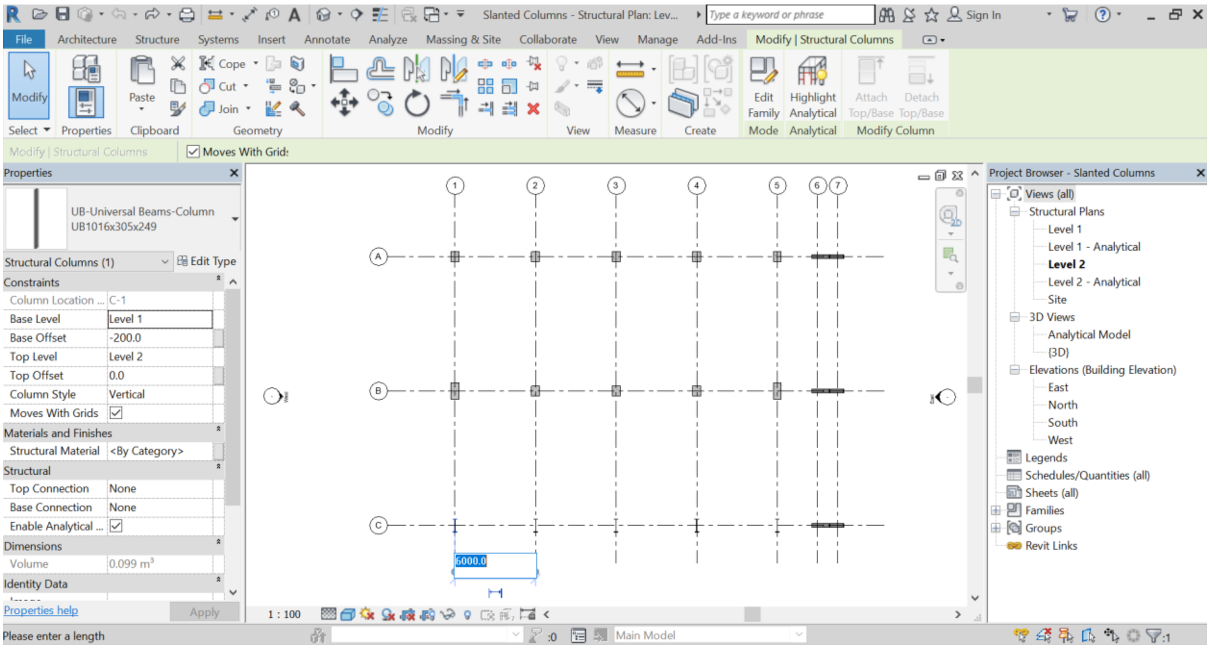
- When placing slanted columns, the top of the column will always have a higher elevation than its base. When you place the column, the endpoint at the higher elevation is the top, and the lower elevation is the base. Once defined, the top may not be set below the base.
- When placed in a 3D view, your 1st Click and 2nd Click settings define the associated levels and offset of the column. When placed in an elevation or cross section, the endpoints associate with their nearest level. The distance between the endpoint and the elevation are the offset by default.
- If 3D Snapping is disabled, snap references for elements on the current work plane display, as well as typical temporary dimensions. When you place columns with 3D snapping enabled, the 1st Click and 2nd Click level settings are used if a snap reference is not found or utilized.
- Slanted columns do not appear in graphical column schedules. Columns in a slanted state will not display element properties related to graphical column schedules, such as Column Location Mark.
- The Copy/Monitor tool does not apply to slanted columns.

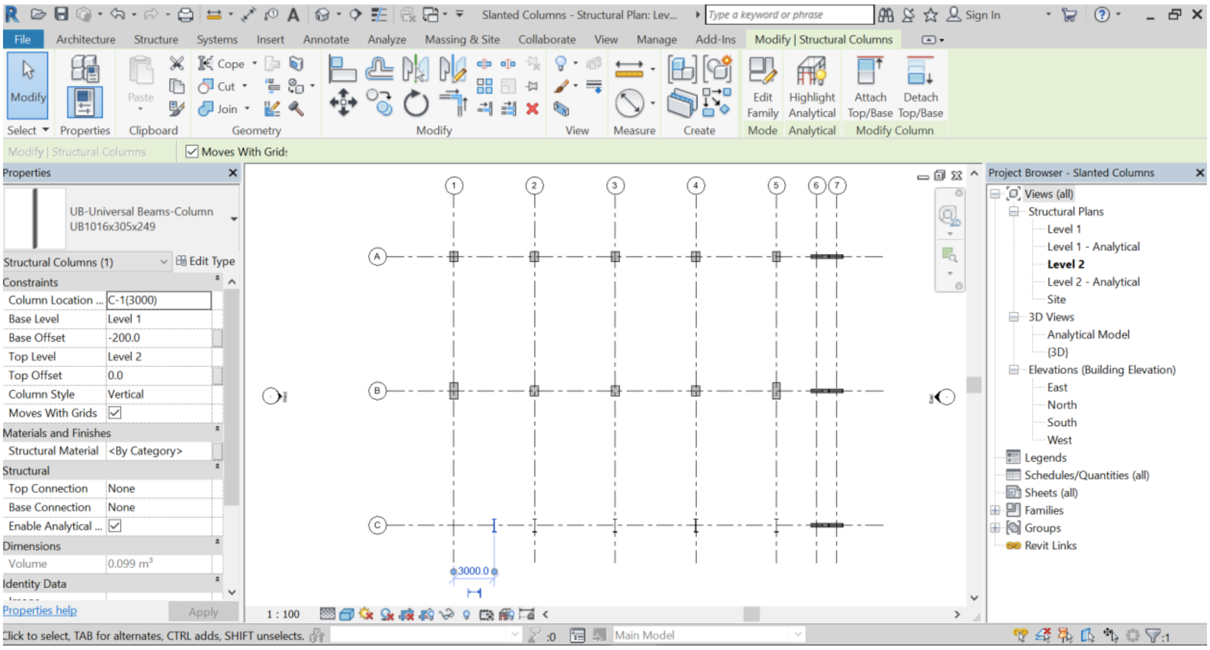
2.4Adjust Location of Columns using Temporary Dimensions

There are many ways to adjust the location of a column. One way is to change its temporary dimension value.

1 Select the column at Grid 1-C.

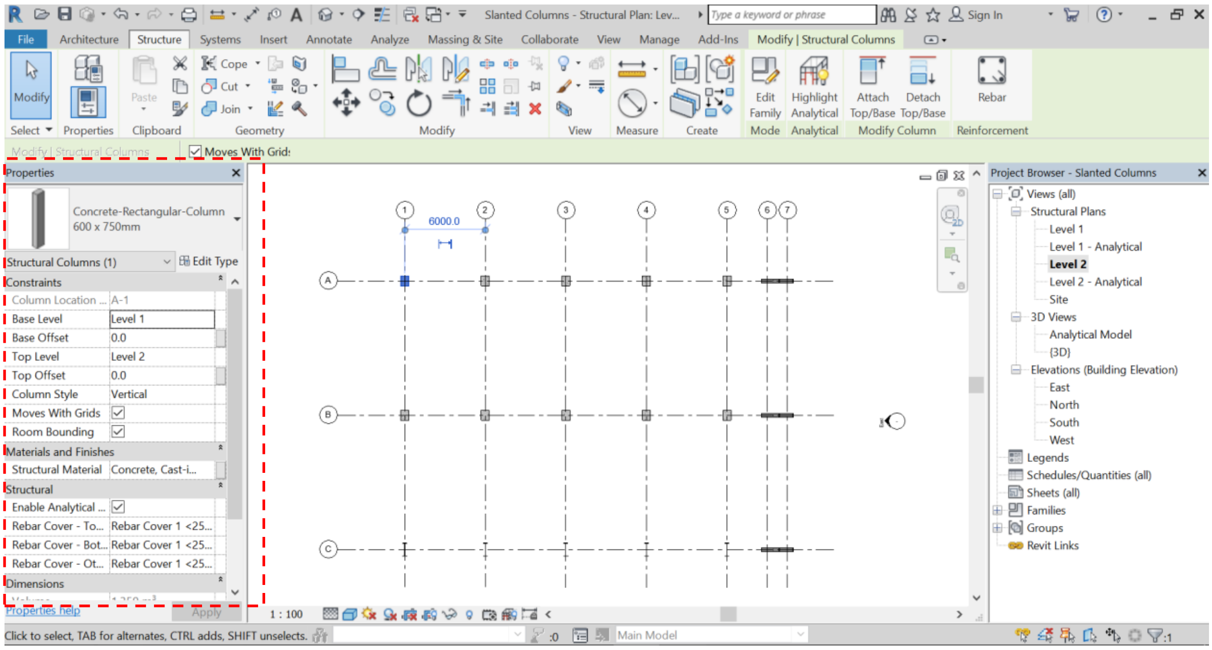
2 Click on the temporary dimension “6000” and change it to “3000”. Press “enter” and “esc”.





2.5Create a New Column Type “600 x 1200mm” by Duplicating an Existing Type

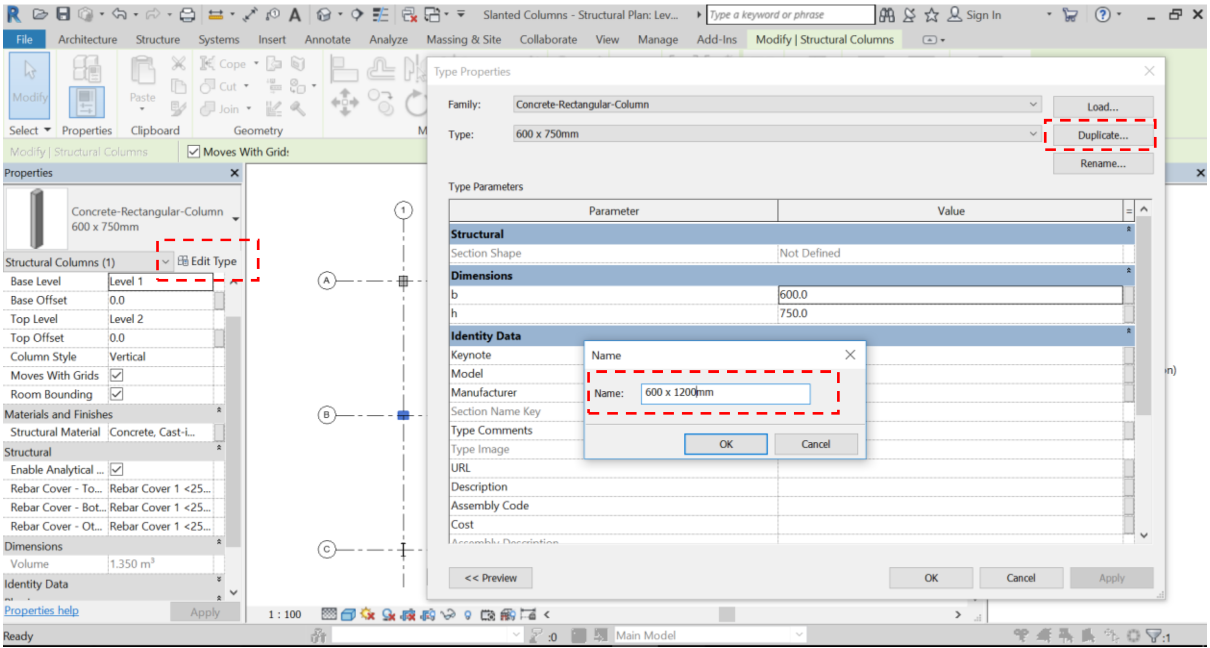
You can manage/modify the properties of a column from its Properties Palette.



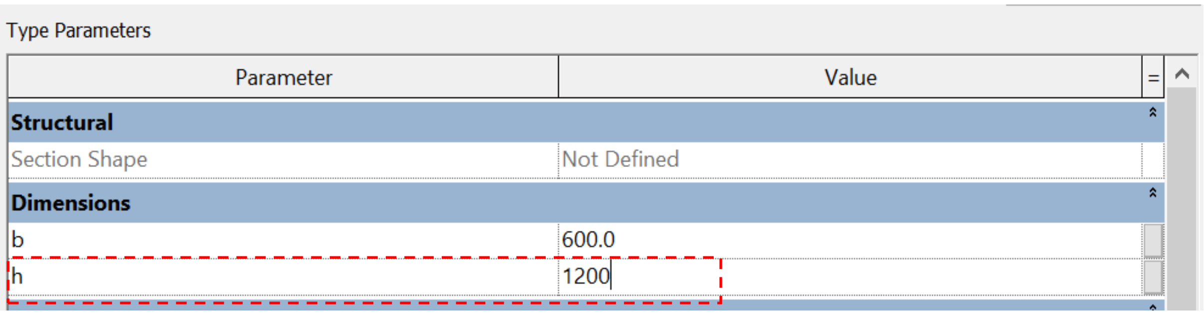
1 Select the column at Grid 1-B.

2 Click "Edit Type" from the Properties Palette.

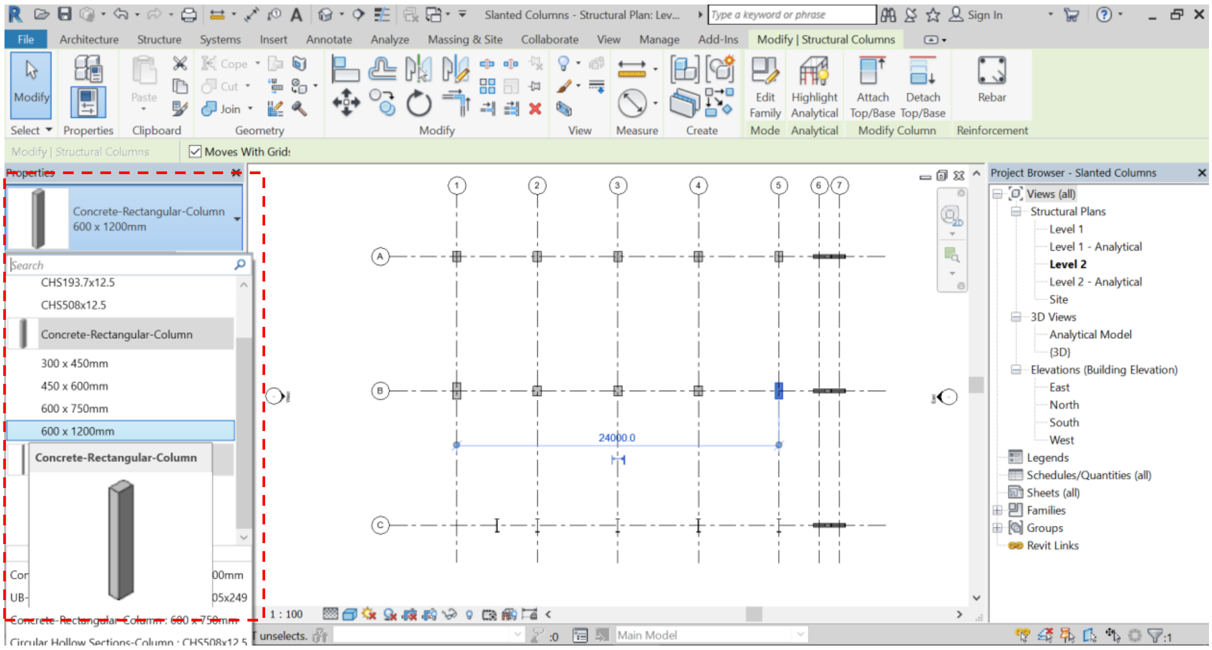
3 Click "Duplicate" and change the "Name" to "600 x 1200mm". Click OK.



4 Change the Value of "h" to 1200. Click OK.



5 Now select the column at Grid 5-B. Select Type “600 x 1200mm” from the Type Selector and press “esc”.



2.6 Modify the Mechanical Properties of a Column

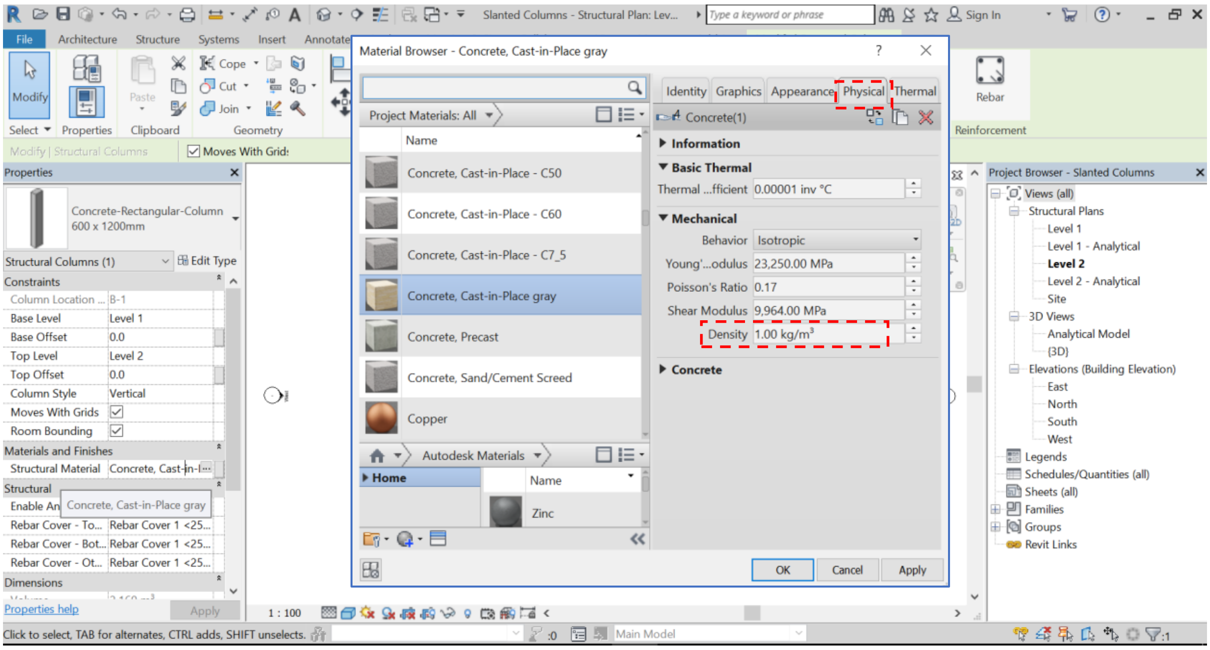
1 Select the column at Grid 1-B.

2 To bring up the Material Browser, click on the builder button in structural material from its properties palette.

2 Select the “Physical tab” and change the Density to 1 kg/m³. Click OK and press “esc”.

You have just changed the density of the material “Concrete, Cast-in-Place gray” and it will be the same for all columns having this material in the same project.

You can also duplicate the material by right click at it in the Material Browser and give it a new name to avoid overwriting the original material.



2.7 Detail Description of Structural Column Instance Properties

(source: Autodesk Knowledge Network)

https://knowledge.autodesk.com/support/revit-products/learn-explore/caas/CloudHelp/cloudhelp/2018/ENU/Revit-Model/files/GUID-5EBA5016-4E9C-4131-BD5B-36FC99A92846-htm.html

Name	Description
Constraints	
Column Location Mark	The coordinate location of a vertical column on the project grid.
Base Level	Constraining level of the column base.
Base Offset	Offset of base from base level.
Top Level	Constraining level of the column top.
Top Offset	Offset of top from top level.
Moves With Grids	Changes the vertical column constraint to the grid. See Lock Columns to a Grid.
Move Top With Grids	Constrains the top endpoint of the slanted column to the grid.
Move Base With Grids	Constrains the base endpoint of the slanted column to the grid.
Top Cut Style	Either Perpendicular, Horizontal, or Vertical. Specifies the cut style at the top of the column when unattached to a reference or element.
Top Extension	Offset of the top of the column when unattached to a reference or element.
Base Cut Style	Either Perpendicular, Horizontal, or Vertical. Specifies the cut style at the base of the column when unattached to a reference or element.
Base Extension	Offset of the base of the column when unattached to a reference or element.
Column Style	Either Vertical, Slanted - End Point Driven, or Slanted - Angle Driven. Specifies the slant style of the column which enables type specific modification tools. See Slanted Column Manual Adjustment Tools.
Base Geometry Alignment	Either Location Line, Top of beam, Bottom of Beam, or Center of Beam. Specifies the working point along an attached beam to which the centerline of a slanted column adjusts at its base.

Name	Description
Top Geometry Alignment	Either Location Line, Top of beam, Bottom of Beam, or Center of Beam. Specifies the working point along an attached beam to which the centerline of a slanted column adjusts at its top.
Base is Attached	Read-only parameter specifying that the base of the column is mid-joined to a beam or attached to a structural floor or roof.
Attachment Justification At Base	Either Minimum Intersection, Intersect Column Midline, Maximum Intersection, or Tangent. Specifies the degree of cutback or tangential justification at the base of the column at the join.
Offset From Attachment At Base	Offset of the base of the column from the mid-joined beam or attached elements.
Top is Attached	Read-only parameter specifying that the top of the column is mid-joined to a beam or attached to a structural floor or roof.
Attachment Justification At Top	Either Minimum Intersection, Intersect Column Midline, Maximum Intersection, or Tangent. Specifies the degree of cutback or tangential justification at the top of the column at the join.
Offset From Attachment At Top	Offset of the top of the column from the mid-joined beam or attached elements.
Room Bounding	Changes column constraint to room-bounding conditions.
Materials and Finishes	
Structural Material	For more information, see Change the Physical Properties of a Material.
Structural	
Top Connection	Applies to steel columns only. Turns on the visibility of a moment or shear connection symbol. The symbols are visible only in elevations and cuts parallel to the main axis of the column in coarse view.
Base Connection	Applies to steel columns only. Turns on the visibility of the base plate symbol. The symbols are visible only in elevations and cuts parallel to the main axis of the column in coarse view.
Enable Analytical Model	Displays the analytical model and includes it in analytical calculations. Selected by default. See Disable an Analytical Model.
Top Attachment Type	Either Distance or Ratio. Determines if the top join of a slanted column is measured as a distance or a ratio of the length of the beam.
Top Attachment Distance	Distance from the referenced end of the top attached beam to the join location on the slanted column.
Top Attachment Ratio	The ratio of distance from the referenced end of the attached beam to the top join location on the slanted column to the total length of the beam.
Top Attachment Reference End	Either Start or End. Specifies the end of the top attached beam from which to calculate distance or ratio.
Base Attachment Type	Either Distance or Ratio. Determines if the base join of a slanted column is measured as a distance or a ratio of the length of the beam.
Base Attachment Distance	Distance from the referenced end of the base attached beam to the join location on the slanted column.
Base Attachment Ratio	The ratio of distance from the referenced end of the attached beam to the base join location on the slanted column to the total length of the beam.
Base Attachment Reference End	Either Start or End. Specifies the end of the base attached beam from which to calculate distance or ratio.
Rebar Cover - Top Face	Applies to concrete columns only. Sets the rebar cover distance from the column top face.
Rebar Cover - Bottom Face	Applies to concrete columns only. Sets the rebar cover distance from the column bottom face.
Rebar Cover - Other Faces	Applies to concrete columns only. Sets the rebar cover distance from the column to other element faces.

Name	Description
Estimated Reinforcement Volume	Specifies the estimated reinforcement volume of the selected element. This is a read-only parameter that only displays when rebar has been placed.
Dimensions	
Volume	Volume of the selected column. A read-only value.
Identity Data	
Comments	User comments.
Mark	A label created for the column. Possible use: shop mark. This value must be unique for each element in a project. Revit warns you if the number is already used but allows you to continue using it. You can see the warning using the Review Warnings tool. For more information, see Review Warning Messages.
Phasing	
Phase Created	Indicates in which phase the column component was created. For more information, see Project Phasing.
Phase Demolished	Indicates in which phase the column component was demolished. For more information, see Project Phasing.

CHAPTER 3 - STRUCTURAL WALLS

Structural walls are loadable families in Revit. You can Create/Customize structural wall family for special use (not to be covered in this course). In this section, you will learn

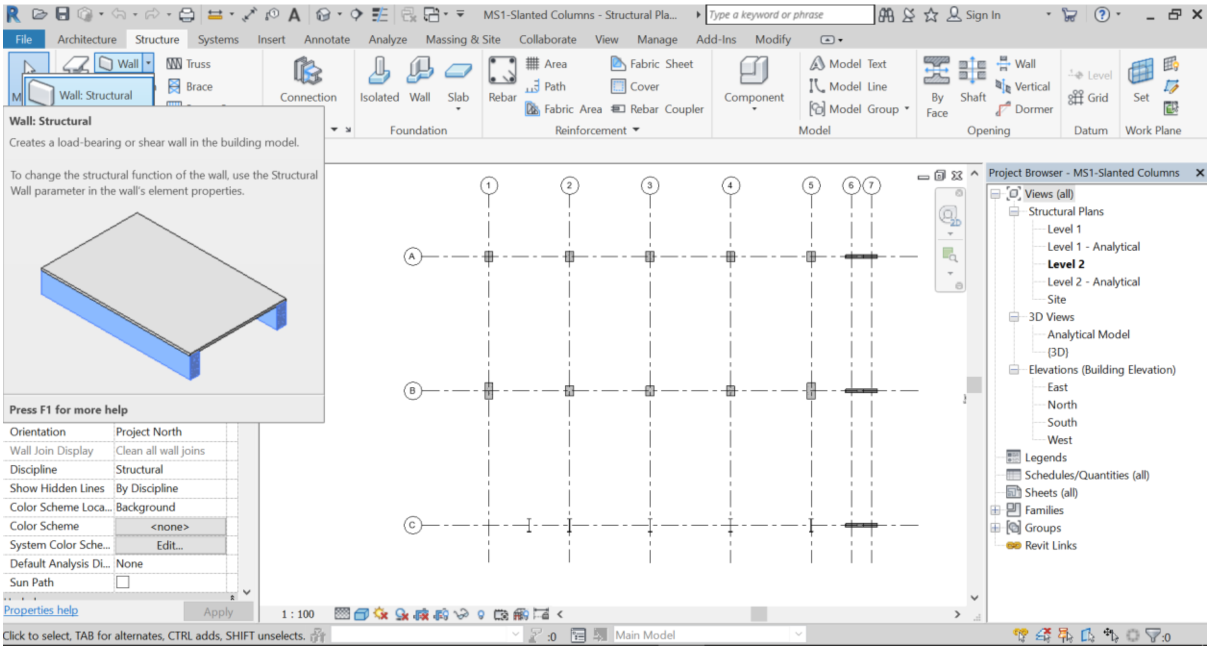
- How to add structural walls.
- Options in wall placement.
- How to add layers to a wall type.

3.1 Adding Walls with default settings

To add structural walls in Revit,

1 Open from the course folder the “Walls.rvt” file.

2 Click Structural Tab ➢ Wall in Structure panel. (The default option is Wall: Structural.)

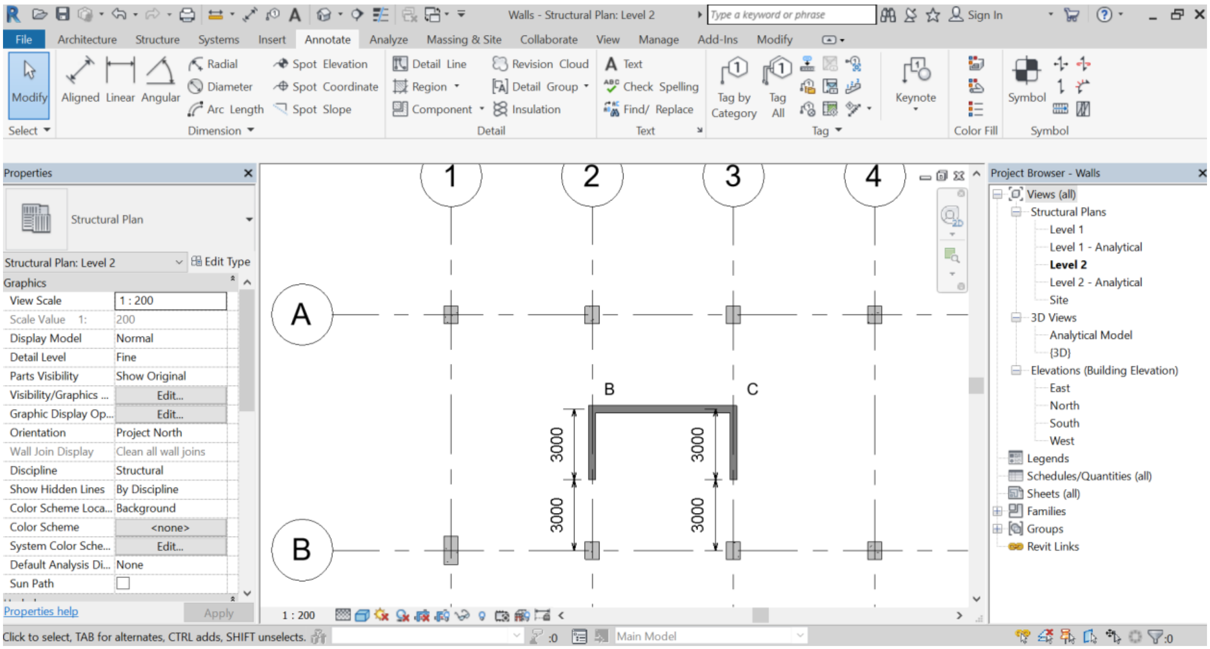


3 In the Properties Palette, select “Generic – 300mm” type from Basic Wall types.

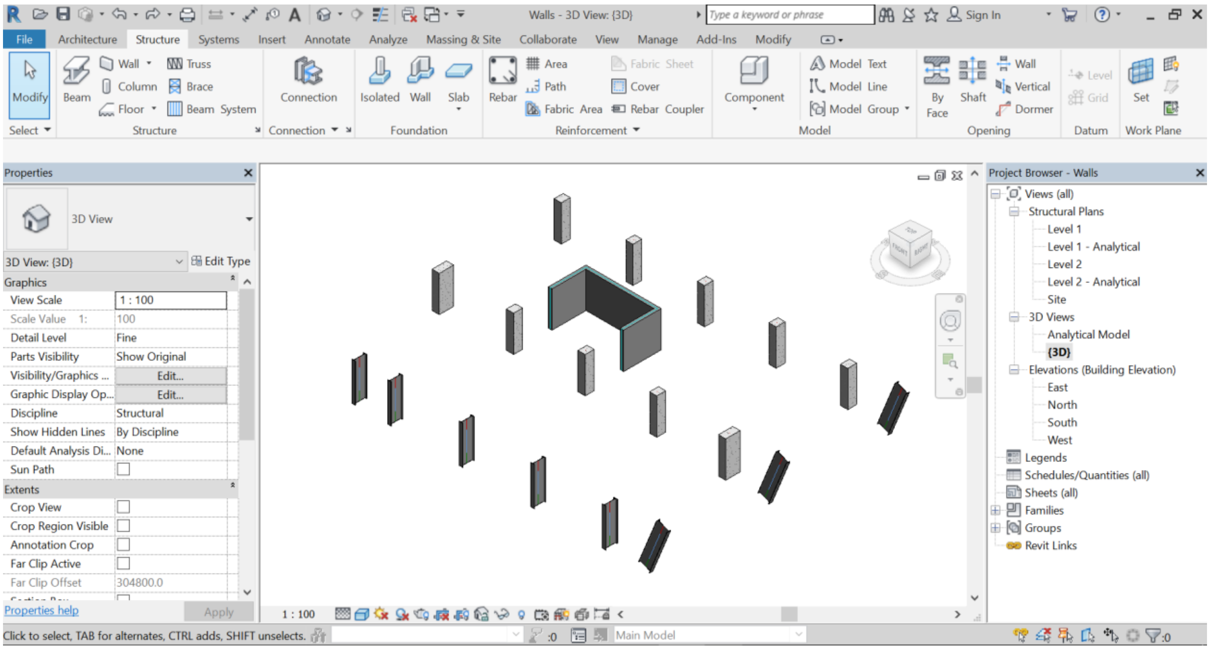
4 Keep all options unchanged.

5 Pick the start point of the wall at Grid-2 and 3000mm up from Grid-B.

6 Continue to pick points B and C as shown below. Press “esc” a few times to complete adding the walls.



7 Click  to view the walls placed in 3D.

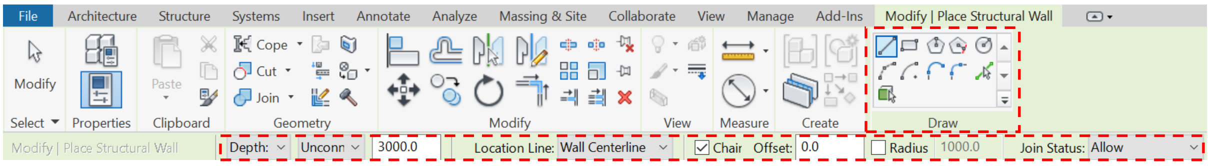


3.2 Options in Wall Placement

There are many options available to help you to place walls in Revit. Many of the options are self-explanatory. You can hover over the option and see its description and press F1 for more help. Under the contextual ribbon tab, the options for wall creation and placement are:

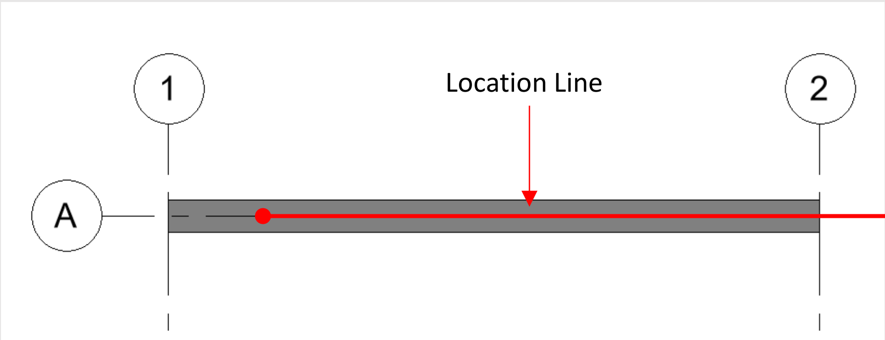
- Draw by: Line / Rectangle / Inscribed Polygon / Circumscribed Polygon / Circle / Start-End-Radius Arc / Center-ends Arc / Tangent End Arc / Fillet Arc / Pick Lines / Pick Faces.
- Depth / Height and (Connected to) Specific Level / Unconnected.
- Location Line: Wall Centerline / Core Centerline / Finish Face: Exterior / Finish Face: Interior / Core Face: Exterior / Core Face: Interior.

- Chair / Offset / Radius
- Join Status: Allow / Disallow

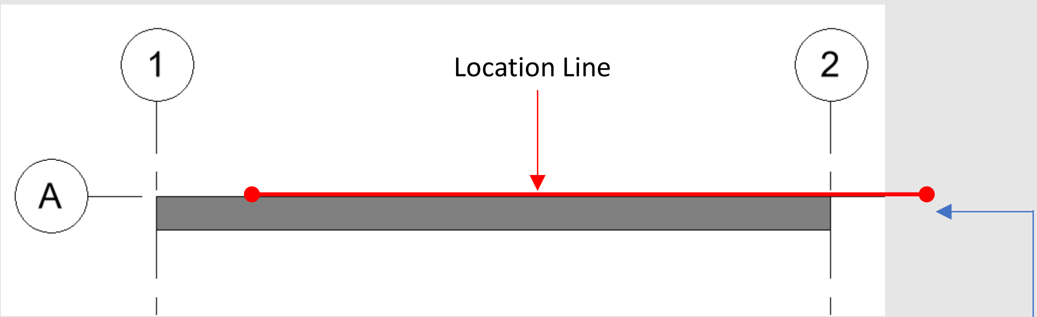


Notes:

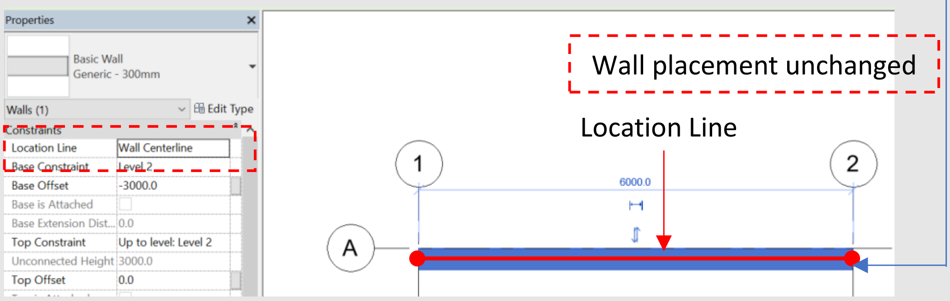
- The Location Line option will affect how the wall is drawn.
- For examples: Wall drawn from Grid-1 to Grid-2 along Grid-A, with Location Line set as "Wall Centerline" will look like



- And wall drawn from Grid-1 to Grid-2 along Grid-A, with Location Line set as "Core Face: Exterior" will look like



-
- Once the wall has been created, changing the Location Line option in the Properties Palette will change the location of the Location Line but not the position of the wall.



3.3Adding (Non-structural) Layers to a Wall Type

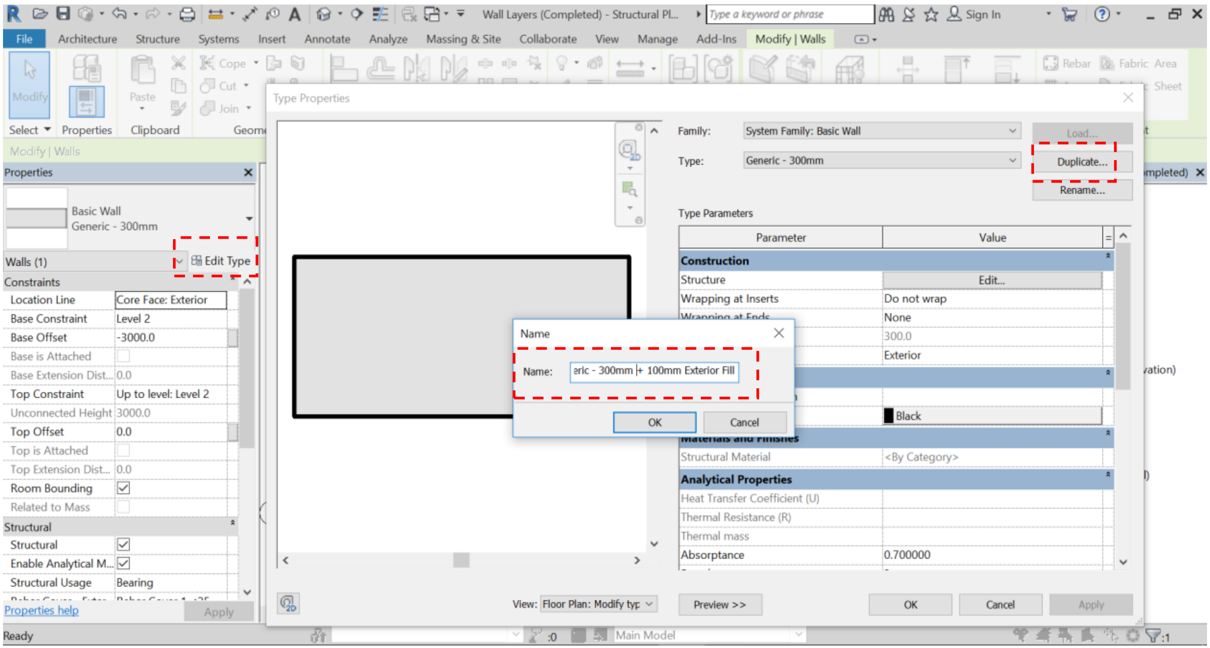
To add layers to a wall type,

1 Open from the course folder the “Wall Layer.rvt” file.

2 Select the horizontal wall.

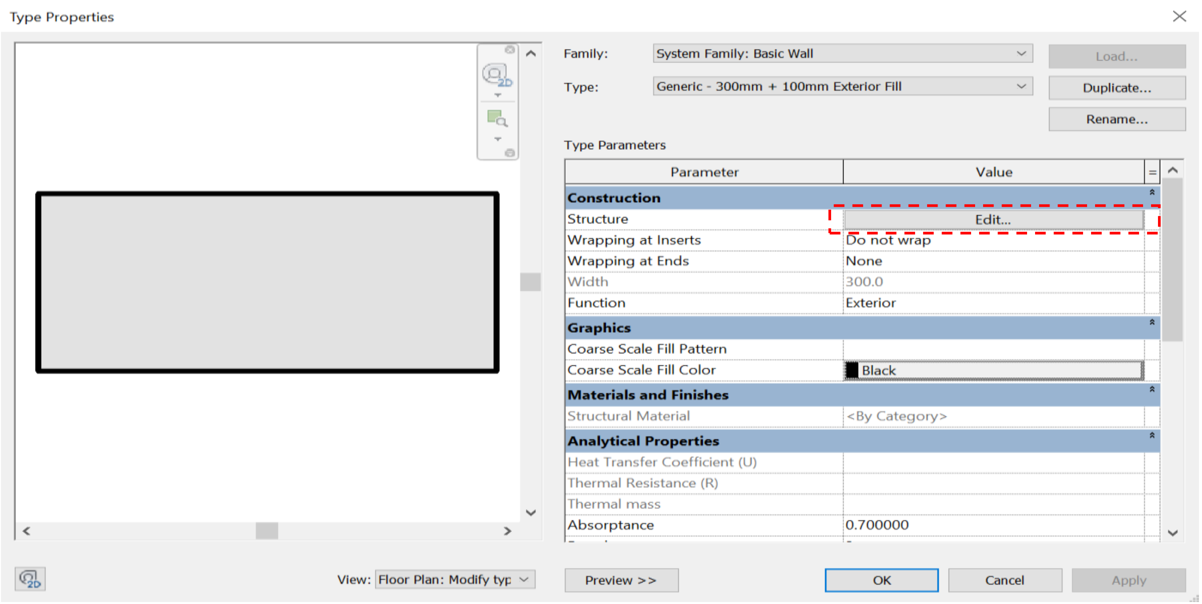
3 Click “Edit Type” from the Properties Palette.

4 Click “Duplicate” and change the “Name” to “Generic 300mm + 100 Exterior Fill”. Click OK.

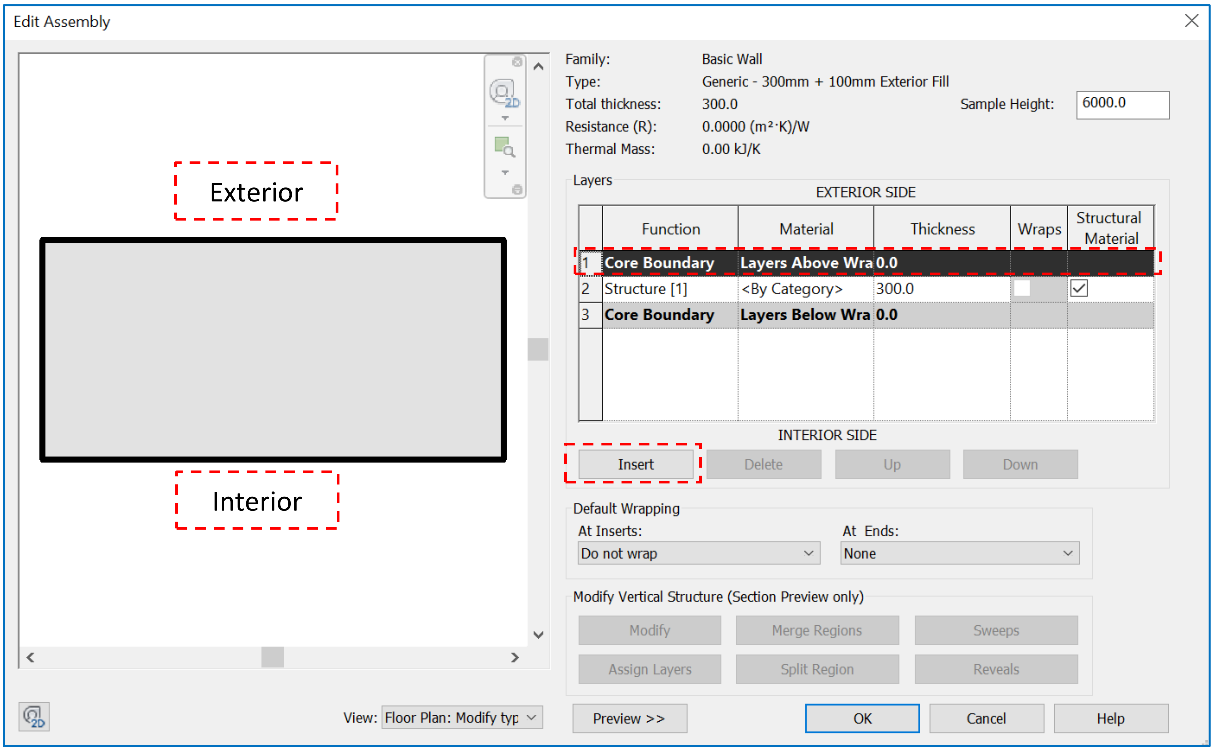


Now add an exterior layer to the wall.

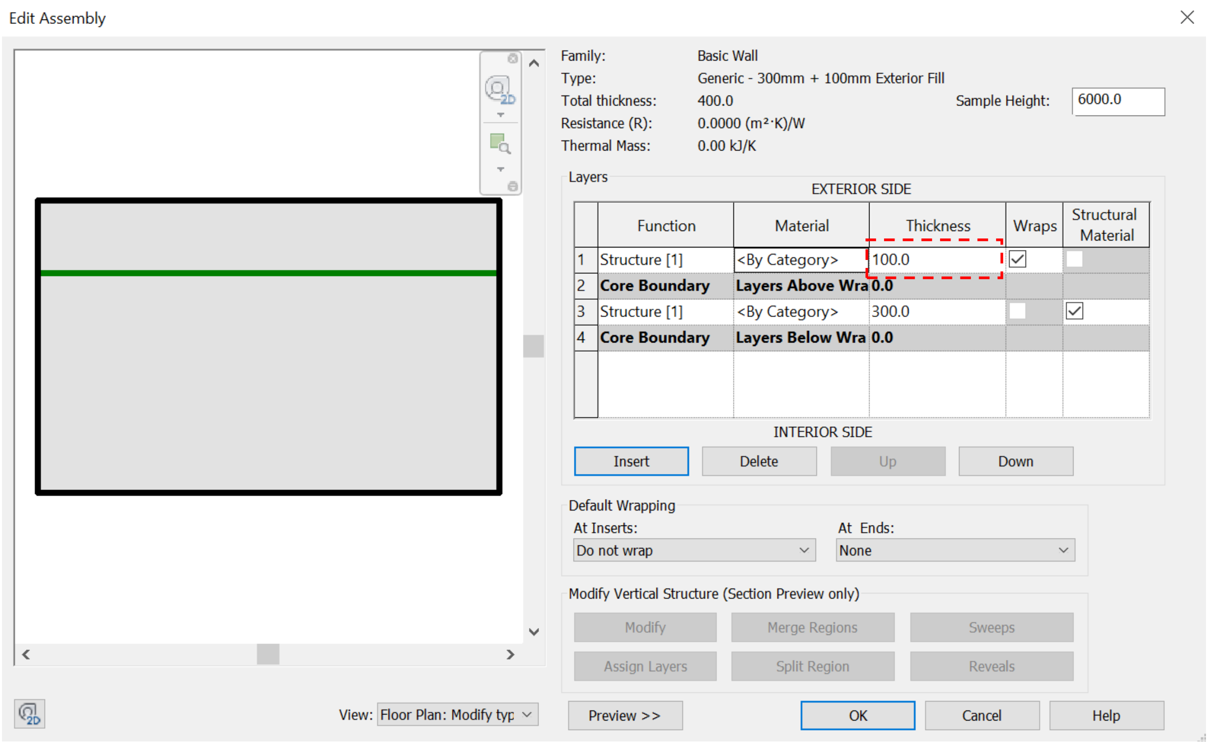
5 Click “Edit” to open the “Edit Assembly” dialogue box.



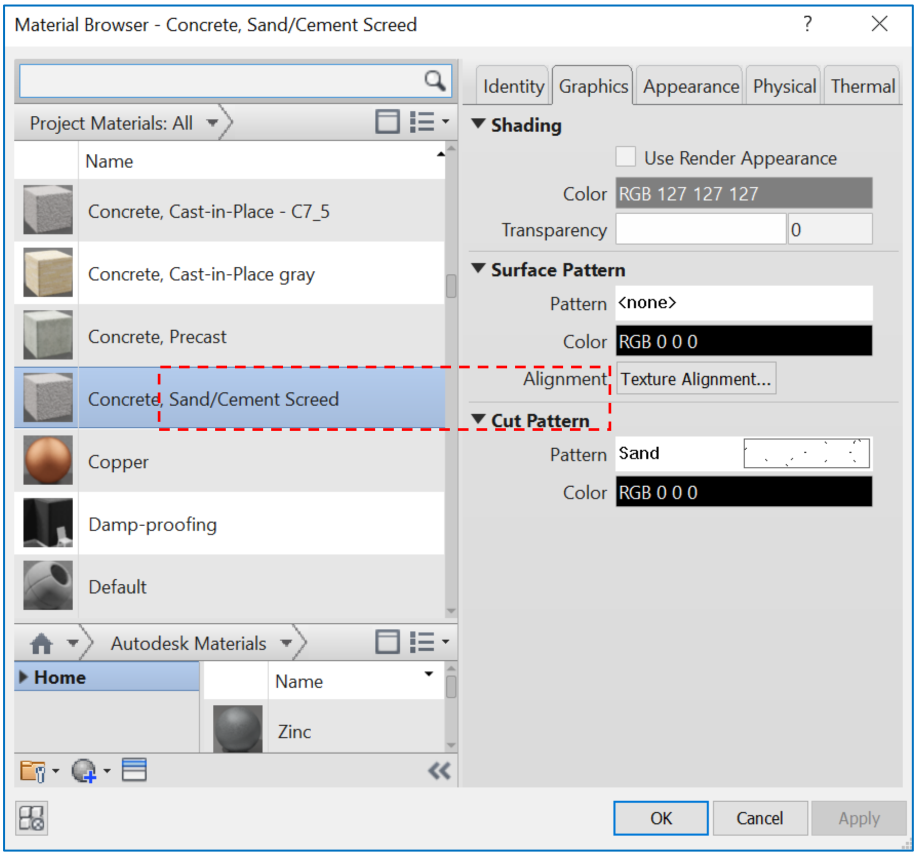
6 Select layer “1” and click “Insert” to insert a layer outside “Core Boundary”.

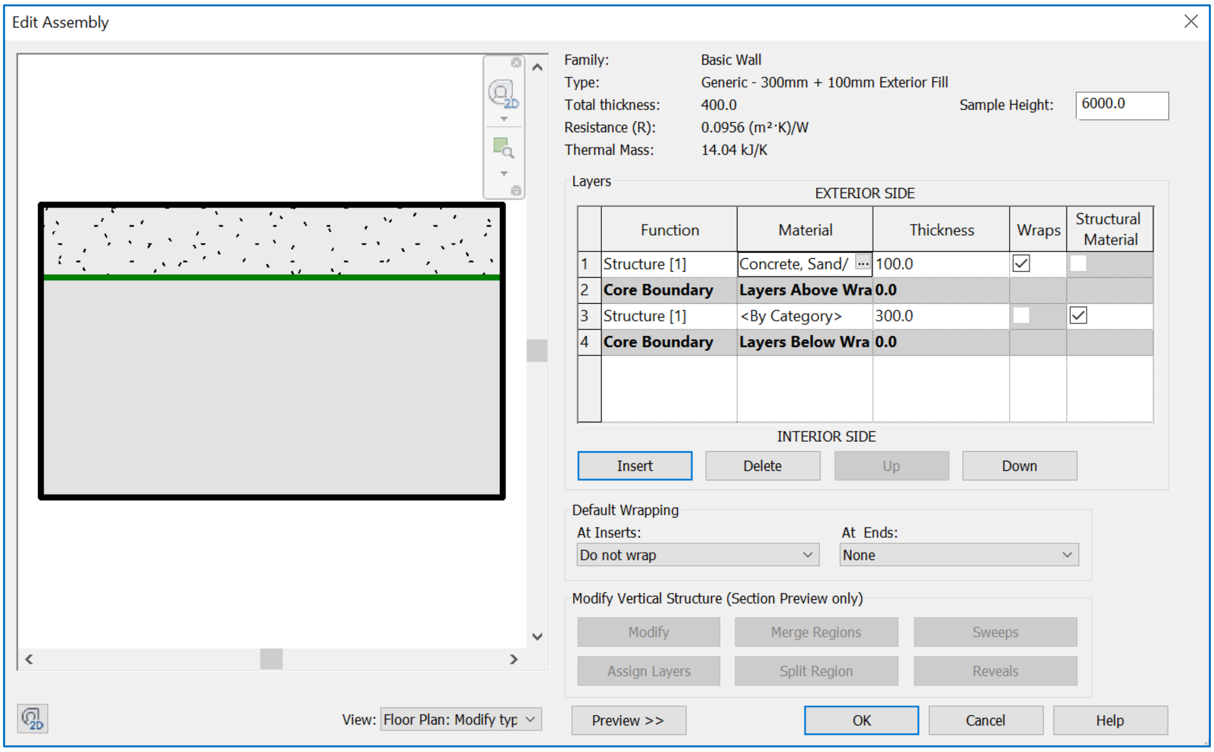


7 Change the thickness of the exterior layer (Layer 1) to 100mm.

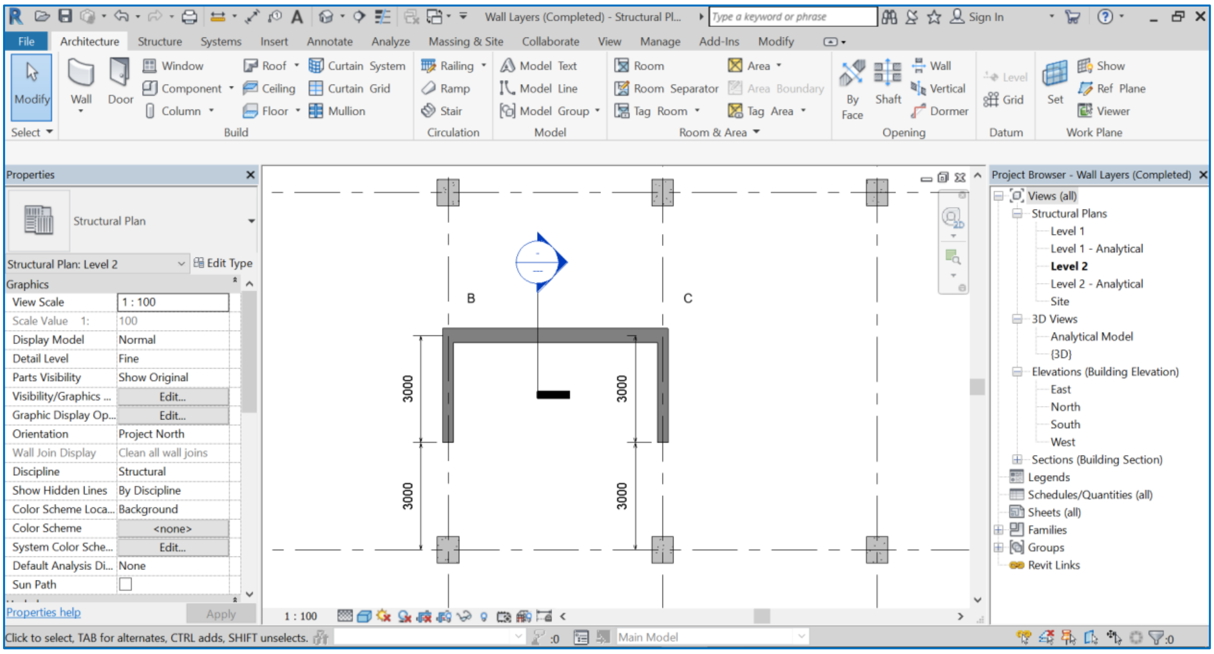


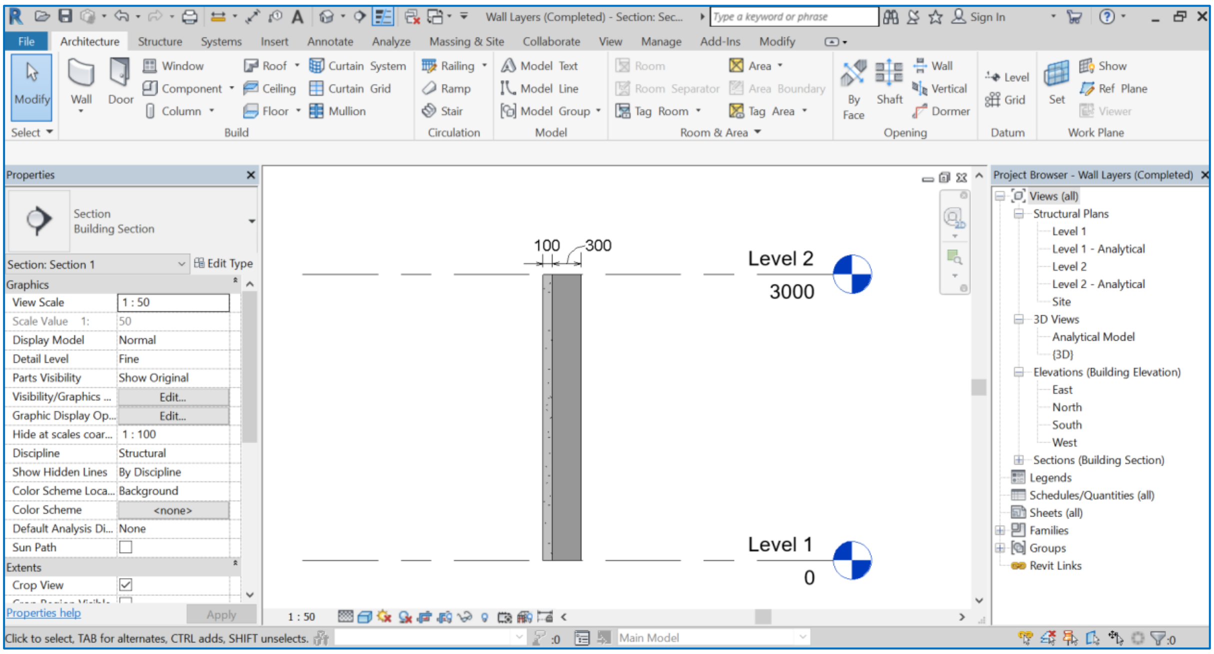
8 Change the “Material” of the exterior layer (Layer 1) to “Concrete, Sand/Cement Screed”.





9 Now add a Section View across the horizontal wall and go to that view to see the added exterior layer.





3.4 Detail Description of Structural Wall Instance Properties

(source: Autodesk Knowledge Network)

https://knowledge.autodesk.com/support/revit-products/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/Revit-Model/files/GUID-F1AD4E4F-3505-4B3F-8096-61BAC86B7ACC-htm.html

Name	Description
Constraints	

Name	Description
Location Line	Specifies the position of the wall with respect to the line sketched in the project elevation. For more information, see Place a Wall. The wall location line remains the same for that wall, even if the type changes.
Base Constraint	Specifies the level from which the wall base is referenced.
Base Offset	Specifies the offset of the base of the wall from its base constraint.
Base is Attached	Indicates whether the base of the wall is attached to another component, such as a structural floor. This is a read-only value.
Base Extension Distance	Indicates the distance you have moved the base of the layers in a wall. See Compound Structure. This parameter is enabled when the layers of a wall are set to extendable.
Top Constraint	The name of the level to which the wall top is set.
Unconnected Height	If top constraint is unconnected, you may set an unconnected height of the wall. This value is read-only if there is a top constraint.
Top Offset	Specifies the offset of the top of the wall from its top constraint; enabled only when the top constraint is set to a level.
Top is Attached	Indicates whether the top of the wall is attached to another component, such as a structural floor. This is a read-only value.
Top Extension Distance	Indicates the distance you have moved the top of the layers in a wall. See Compound Structure. This parameter is enabled when the layers of a wall are set to extendable.
Room Bounding	Indicates whether the wall is part of a room boundary. This parameter is enabled after you place the wall.
Related to Mass	This is a read-only value.
Structural	
Rebar Cover - Exterior Face	Specifies the rebar cover distance from the wall exterior face.
Rebar Cover - Interior Face	Specifies the rebar cover distance from the wall interior face.
Rebar Cover - Other Faces	Specifies the rebar cover distance from the faces of adjacent elements.
Estimated Reinforcement Volume	Specifies the estimated reinforcement volume of the selected element. This is a read-only parameter that only displays when rebar has been placed.
Structural	Specifies the wall as being a structural element able to posses an analytical model.
Enable Analytical Model	Displays the analytical model and includes it in analytical calculations. Selected by default. See Disable an Analytical Model.
Structural Usage	Either Bearing, Shear, or Structural Combined. The structural use of the wall.
Dimensions	

Name	Description
Length	Indicates the length of the wall. This is a read-only value.
Area	Indicates the area of the wall. This is a read-only value.
Volume	Indicates the volume of the wall. This is a read-only value.
Identity Data	
Comments	A field for entering comments about the wall.
Mark	A label created for the wall. This value must be unique for each element in a project. Revit warns you when the number is already used but allows you to continue using it. You can see the warning using the Review Warnings tool. See Review Warning Messages.
Phasing	
Phase Created	Indicates in which phase the wall component was created. See Project Phasing.
Phase Demolished	Indicates in which phase the wall component was demolished. See Project Phasing.

CHAPTER 4 - STRUCTURAL BEAMS (STRUCTURAL FRAMING) and BEAM SYSTEMS

Structural beams are categorized as structural framing in Revit. Usually beams are placed after creating datum (grids / levels) and vertical elements (columns / walls). You can Create/Customize structural beam family for special use (to be covered later in this course). In this section, you will learn

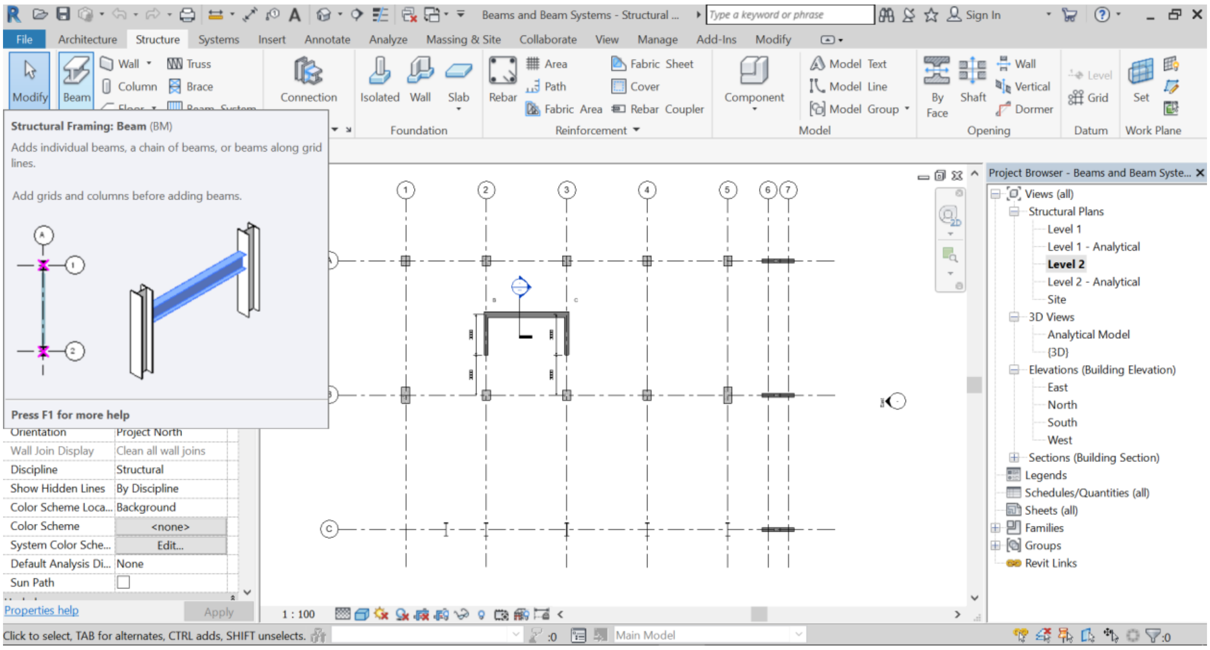
- How to add structural beams.
- Options in beam placement.
- How to add a system of beams (Beam System).
- Options in annotating beams.

4.1 Adding Beams with default settings

To add structural beams in Revit,

1 Open from the course folder the “Beams.rvt” file.

2 Click Structural Tab ➢ Beam in Structure panel.

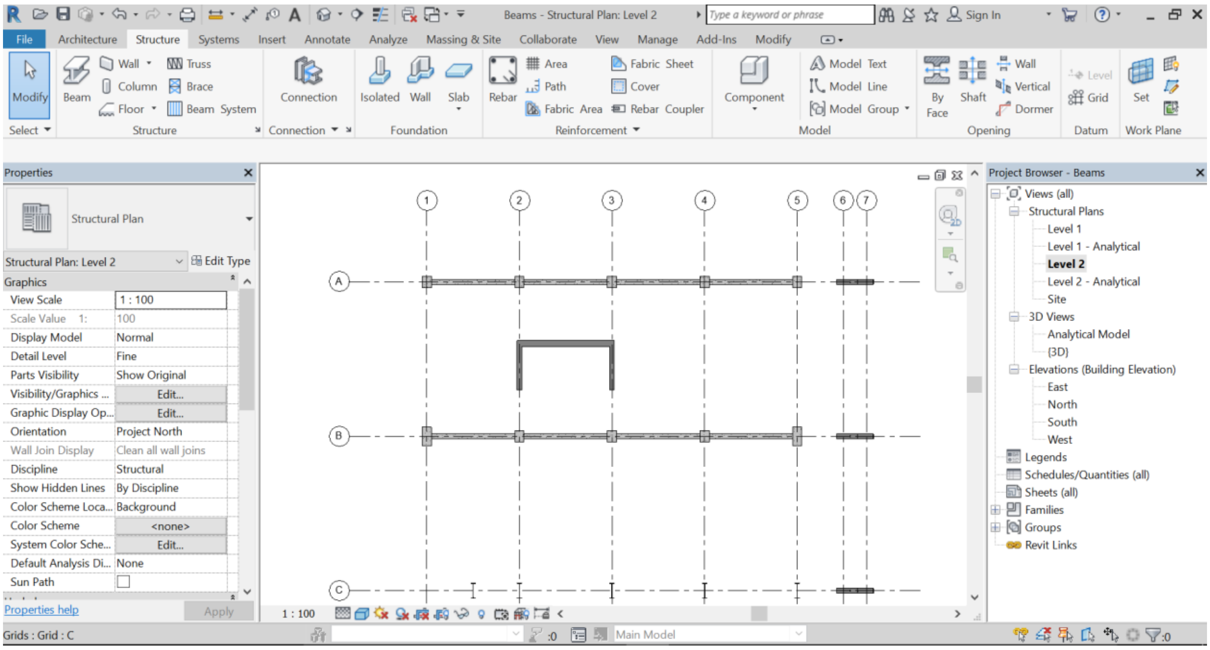


3 In the Properties Palette, select “300 x 600mm” type from Concrete-Rectangular Beam types.

4 Keep all options unchanged.

5 Pick the start point of the beam on the grid intersection Grid-A-1, then pick the grid intersections along Grid-A. After picking the intersection point Grid-5-A then press “esc”.

6 Repeat Step 5 and add “300 x 600mm” beams along Grid-B.



Now, add some steel beams

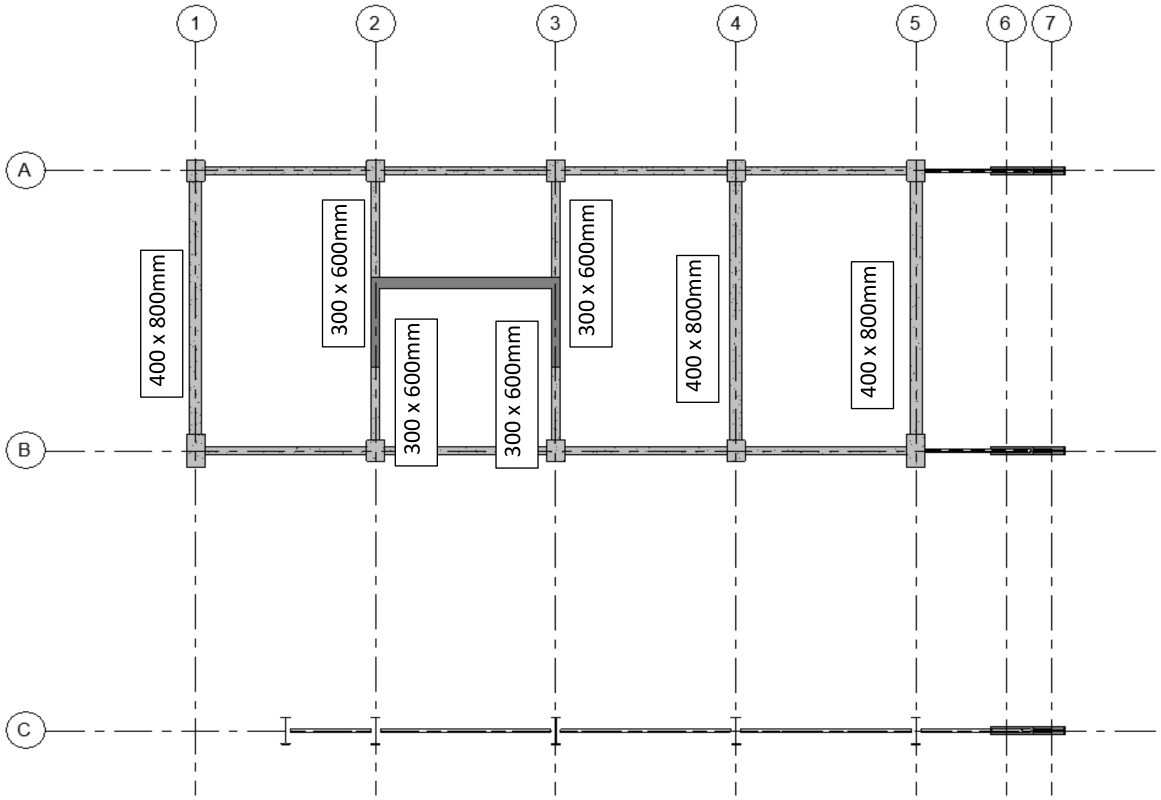
7 In the Properties Palette, select “UB305x127x37” type from UB-Universal Beams types.

8 Pick the intersection point Grid-A-5 and then pick the grid intersection Grid-A-7. Press “esc”.

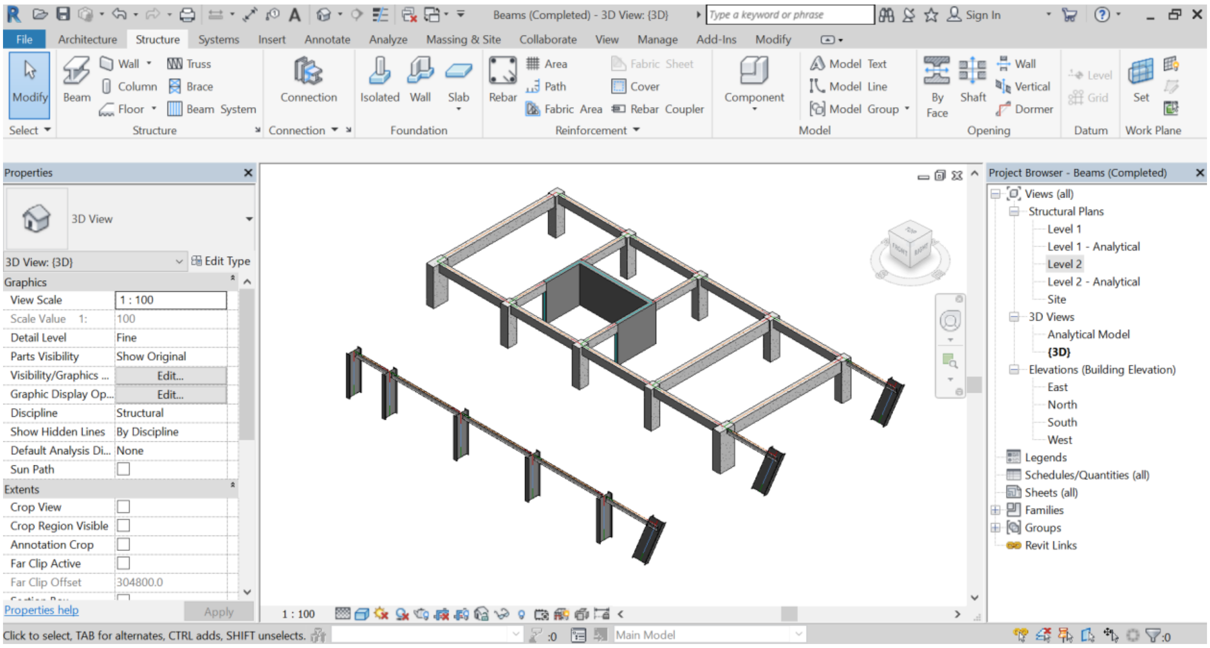
9 Pick the intersection point Grid-B-5 and then pick the grid intersection Grid-B-7. Press “esc”.

10 Pick the column between Grid-1 and Grid-2 on Grid-C. Then pick the grid intersection points along Grid-C (skip grid intersection point Grid-C-6). Press “esc”.

11 Add other beams as shown below.



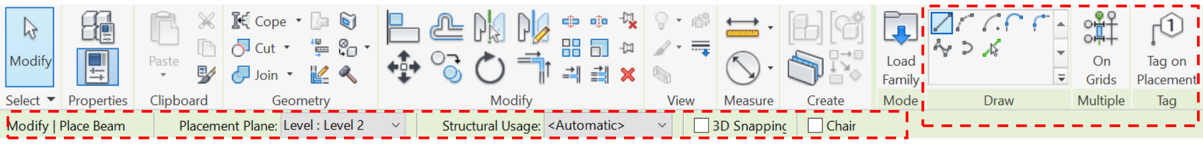
12 Click  to view the beams placed in 3D.



4.2 Options in Beam Placement

There are many options available to help you to place beams in Revit. Many of the options are self-explanatory. You can hover over the option and see its description and press F1 for more help. Under the contextual ribbon tab, the options for beam creation and placement are:

- Draw by: Line / Start-End-Radius Arc / Center-ends Arc / Tangent End Arc / Fillet Arc / Spline / Partial Ellipse / Pick Lines.
- Place beams: On Grids / Tag on Placement.
- Placement Plane: Select an available structural level.
- Structural Usage: <Automatic> / Girder / Horizontal Bracing / Joist / Other / Purlin.
- 3D Snapping / Chair

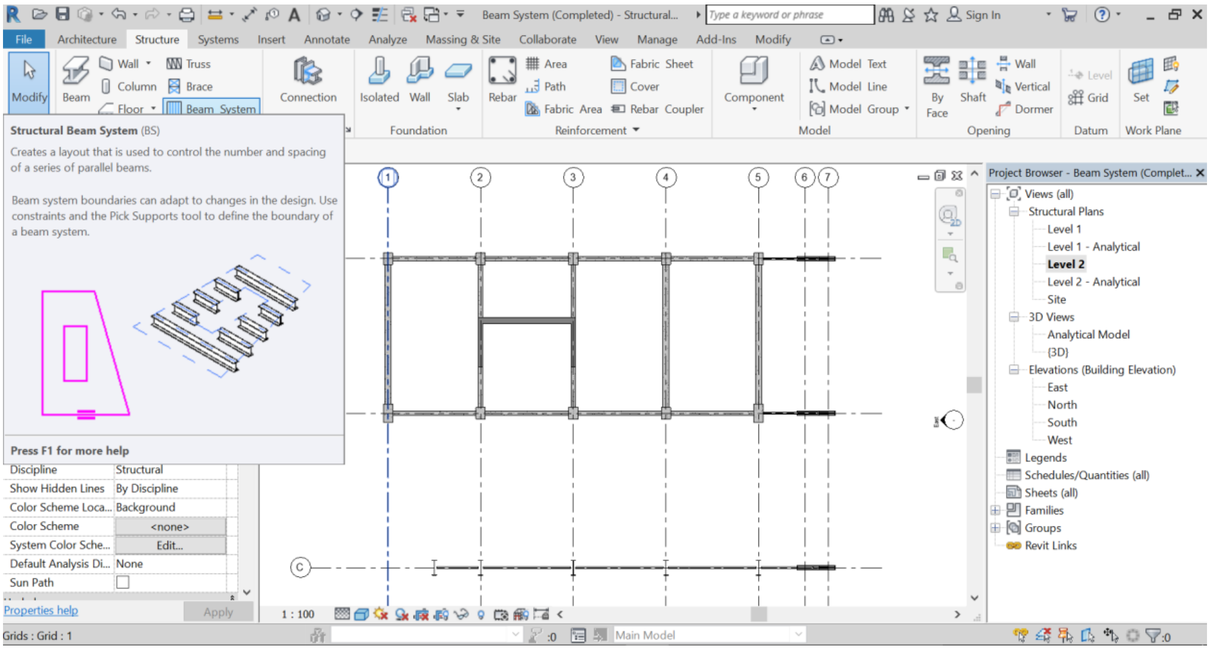


4.3Adding a system of beams (Beam System)

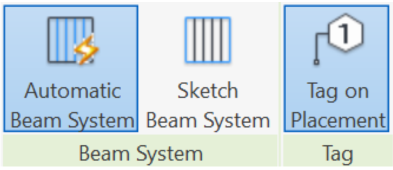
Revit provides an easy way to add a system of beams within a closed boundary of area. It is called Beam System in Revit. To add a system of beams (Beam System),

1 Open from the course folder the “Beam System.rvt” file.

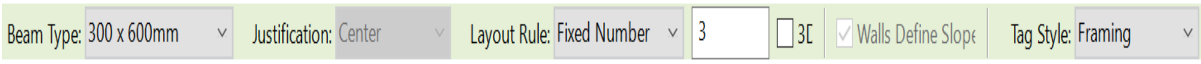
2 Click Structural Tab ➢ Beam System in Structure panel.



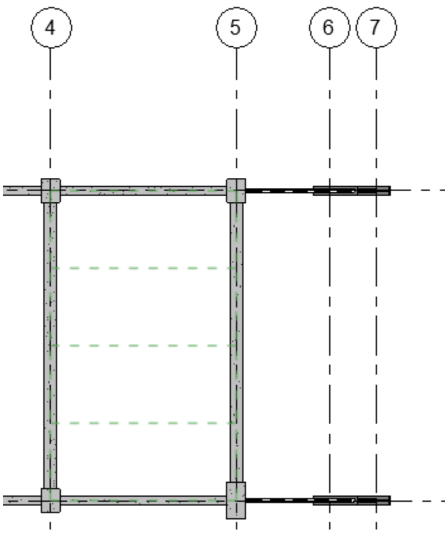
3 There are two ways to add a Beam System. One is Automatic (set as default) and the other is by Sketch where you can draw the boundary of an area to place the system of beams. By default, all beams in the system will be tagged. Click the “Tag on Placement” to change the tag option.

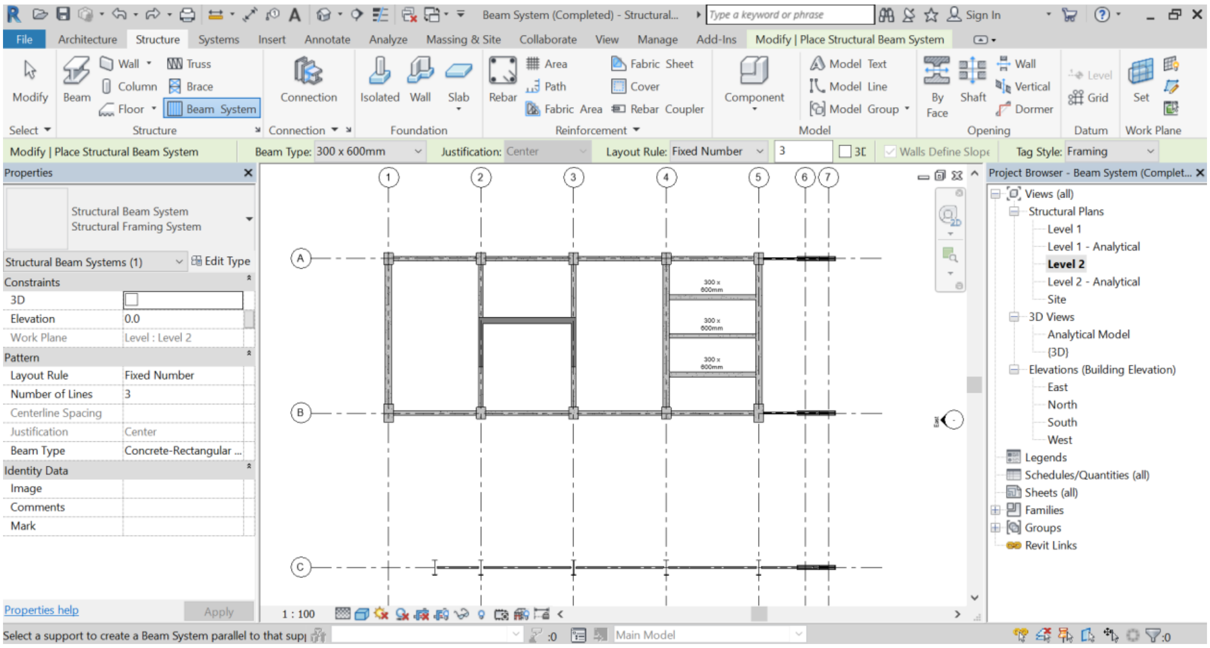


4 Set the Beam Type, Justification, Layout Rule, Tag Style as shown below.

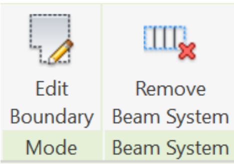


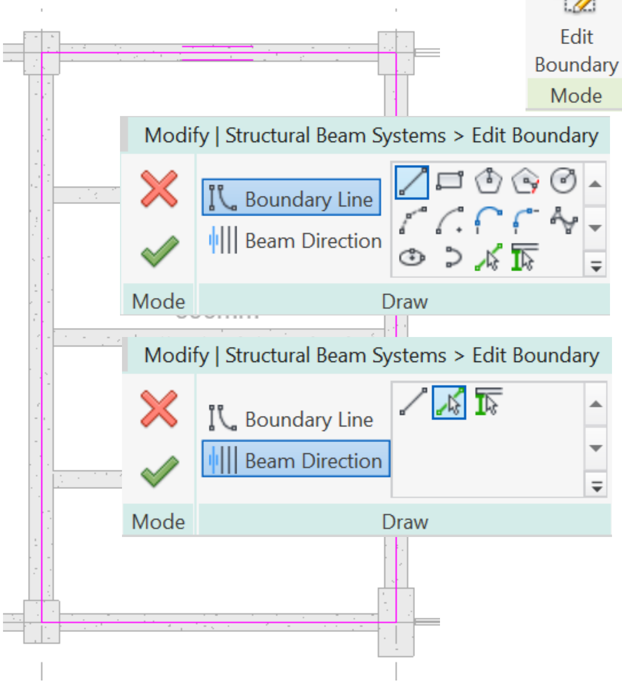
5 Hover the cursor near the beam on Grid-A and between Grid 4 and Grid-5, you will see a tentative beam system displayed. Click to add the indicated beam system.





When you select a Beam System, you can then edit its boundary and the direction of the beams or remove the beams from the Beam System.

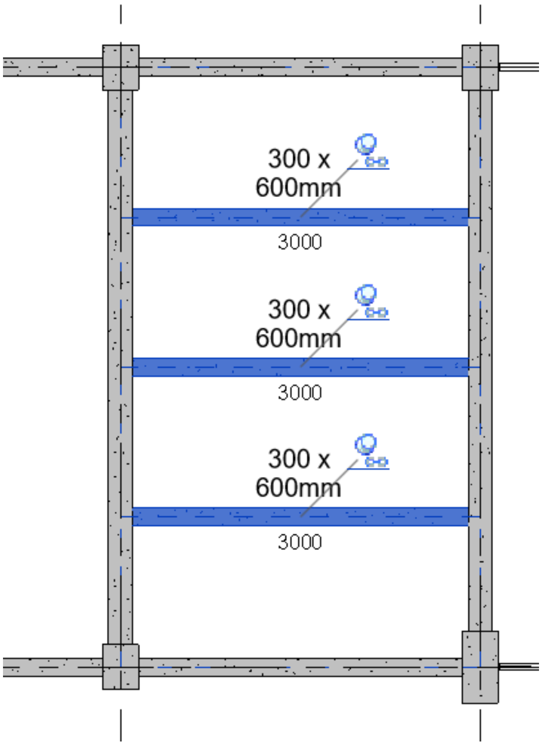




4.4Beam Tag/Annotation

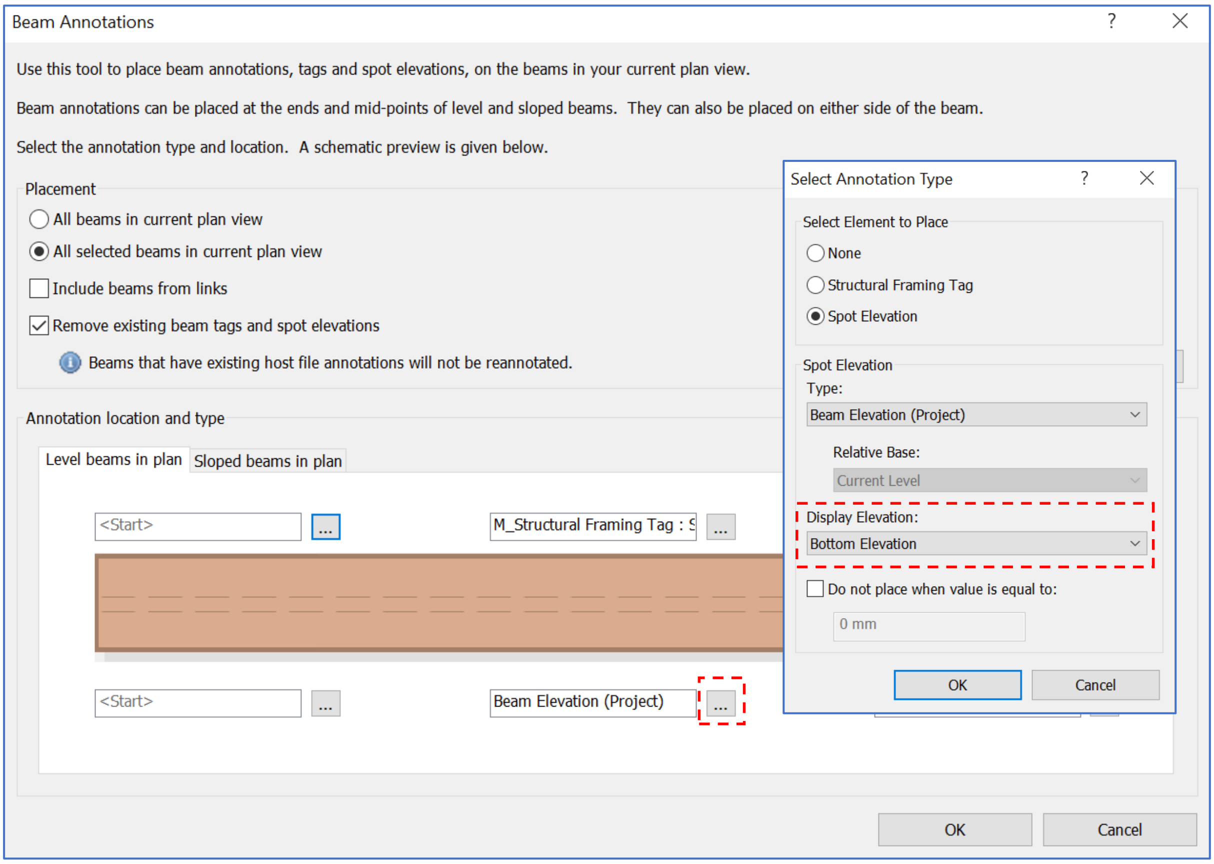
To control how the beams are tagged/annotated,

1 Select the beams in the beam system.

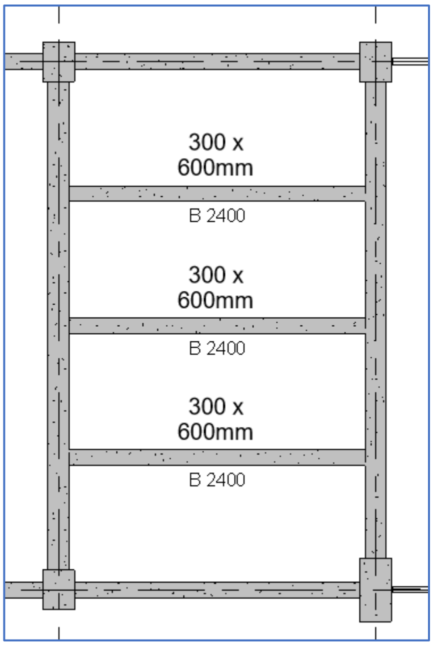


2 Click "Annotation" Tab ➢ Beam Annotations  in Tag panel.

3 Click the builder button of the middle bottom tag and change the Display Elevation to "Bottom Elevation" then click OK.



The annotation of the selected beams will change to



Notes on Beam Annotation:

- The Beam Annotations tool is only available when working in a structural plan or ceiling plan view.
- At least one structural framing tag specified to "Rotate with Component" in their family parameters must be loaded for the tool to be available.
- If no beam is selected, the Beam Annotations tool will tag all beams in the current view.

4.5 Detail Description of Structural Beam Instance Properties

(source: Autodesk Knowledge Network)

https://knowledge.autodesk.com/support/revit-products/learn-explore/caas/CloudHelp/cloudhelp/2016/ENU/Revit-Model/files/GUID-89D4AB0D-4D2E-4391-802E-F8AF478637FC-htm.html

Name	Description
Constraints	
Reference Level	The constraining level. This is a read-only value that is dependent on the work plane on which the beam was placed.
Work Plane	The current plane on which elements are placed. This is a read-only value.
Start Level Offset	The distance of the beam start point from the reference level. Values entered here are reset when the member is pinned. Read-only when pinned.

Name	Description
End Level Offset	The distance of the beam endpoint from the reference level. Values entered here are reset when the member is pinned. Read-only when pinned
Orientation	The beam orientation to the current plane on which elements are placed. This is a read-only value.
Cross-Section Rotation	Controls rotating beams and braces. The angle of rotation is measured from the beam's work plane and the direction of the center reference plane.
Construction (Pre-2014 projects)	
Start Extension	Applies to steel beams only. The dimension between the edge of the start end of the beam and the element to which it is connected. This parameter is only available in projects created in versions prior to Autodesk Revit 2014 containing beam families that have not been updated. It is recommended to use the Geometric Position Start Extension below. Using both is an additive extension.
End Extension	Applies to steel beams only. The dimension between the edge of the finish end of the beam and the element to which it is connected. This parameter is only available in projects created in versions prior to Autodesk Revit 2014 containing beam families that have not been updated. It is recommended to use the Geometric Position Start Extension below. Using both is an additive extension.
Materials and Finishes	
Structural Material	Controls the hidden view display of structural elements. Concrete or Precast will display as hidden. Steel or Wood will be visible when another element is in front of it. Unassigned will not display if hidden by another element. See Change the Physical Properties of a Material.
Structural	
Stick Symbol Location	Applies to steel beams only.
Start Connection	Applies to steel beams only. A moment frame, or cantilever symbol at the start end of a beam.
End Connection	Applies to steel beams only. A moment frame, or cantilever symbol at the finish end of a beam.
Cut Length	The physical length of the beam. This is a read-only value.
Structural Usage	Specifies the usage. Either Girder, Horizontal bracing, Joist, Other, Purlin, or Chord.
Start Attachment Type	Either End Elevation or Distance. Specifies the elevation orientation of the beam. End Elevation maintains the placement level and Distance orients to the join location on a column. See About Adjusting the Beam Elevation at Column Join.

Name	Description
Start Attachment Distance	Specifies the offset of the beam start from the column connection point when Start Attachment Type is set to Distance. See About Adjusting the Beam Elevation at Column Join.
End of Referenced Column	Specifies either the Top or Bottom of the beam from which the Start Attachment Distance is determined. See About Adjusting the Beam Elevation at Column Join.
End Attachment Type	Either End Elevation or Distance. Specifies the elevation orientation of the beam. End Elevation maintains the placement level and Distance orients to the join location on a column. See About Adjusting the Beam Elevation at Column Join.
End Attachment Distance	Specifies the offset of the bean end from the column connection point when End Attachment Type is set to Distance. See About Adjusting the Beam Elevation at Column Join.
End of Referenced Column	Specifies either the Top or Bottom of the beam from which the End Attachment Distance is determined. See About Adjusting the Beam Elevation at Column Join.
Rebar Cover - Top Face	Applies to concrete beams only. The rebar cover distance from the beam top face.
Rebar Cover - Bottom Face	Applies to concrete beams only. The rebar cover distance from the beam bottom face.
Rebar Cover - Other Faces	Applies to concrete beams only. The rebar cover distance from the beam to adjacent element faces.
Estimated Reinforcement Volume	Applies to concrete beams only. Specifies the estimated reinforcement volume of the selected element. This is a read-only parameter that only displays when rebar has been placed.
Camber Size	Applies to steel beams only. The beam camber.
Number of Studs	Applies to steel beams only. The number of studs per beam.
Enable Analytical Model	Displays the analytical model and includes it in analytical calculations. Selected by default. See Disable an Analytical Model.
Dimensions	
Length	The length between beam handles. See About Beam Handles. This is the analytical length of the beam. This is a read-only value.
Volume	The volume of the selected beam. This is a read-only value.
Elevation at Top	Indicates the elevation used for tagging the top of the beam. This is a read-only parameter that reports Varies for sloped planes.
Elevation at Bottom	Indicates the elevation used for tagging the bottom of the beam. This is a read-only parameter that reports Varies for sloped planes.
Identity Data	

Name	Description
Comments	User comments.
Mark	A label created for the beam. Possible use: shop mark. This value must be unique for each element in a project. Revit warns you if the number is already used but allows you to continue using it. You can see the warning using the Review Warnings tool. See Review Warning Messages.
Phasing	
Phase Created	Indicates in which phase the beam component was created. See Project Phasing.
Phase Demolished	Indicates in which phase the beam component was demolished. See Project Phasing.
Geometric Position	
Start Extension	Applies to steel beams only. A dimension that adds beam geometry beyond the start end of the beam.
End Extension	Applies to steel beams only. A dimension that adds beam geometry beyond the finish end of the beam.
Start Join Cutback	Applies to steel beams only. The dimension between the edge of the start end of the beam and the element to which it is connected. Only for the joined start of the element.
End Join Cutback	Applies to steel beams only. The dimension between the edge of the finish end of the beam and the element to which it is connected. Only for the joined end of the element.
yz Justification	Applies to steel beams only. Either Uniform or Independent. Uniform allows setting the same parameters to both start and end of a beam. Independent allows setting different parameters to start and end of a beam.
y Justification	Applies to Uniform justified steel beams only. Specifies the location of the physical geometry with respect to the location line: either Origin, Left, Center, Right.
y Offset Value	Applies to Uniform justified steel beams only. The numeric value that offsets the geometry. The distance between the location line and the characteristic point set in y Justification parameter.
z Justification	Applies to Uniform justified steel beams only. Specifies the location of the physical geometry with respect to the location line: either Origin, Top, Center, Bottom.
z Offset Value	Applies to Uniform justified steel beams only. The distance between the location line and the characteristic point set in z Justification parameter.
Start y Justification	Applies to Independent justified steel beams only. Specifies the location of the physical geometry of the start end of the beam with respect to the location line: either Origin, Left, Center, Right.
Start y Offset Value	Applies to Independent justified steel beams only. The numeric value that offsets the geometry at the start end of the beam. The distance between the location line and the characteristic point set in the Start y Justification parameter.
Start z Justification	Applies to Independent justified steel beams only. Specifies the location of the physical geometry with respect to the location line at the start end of the beam: either Origin, Top, Center, Bottom.

Name	Description
Start z Offset Value	Applies to Independent justified steel beams only. The numeric value that offsets the geometry at the finish end of the beam. The distance between the location line and the characteristic point set in the Start z Justification parameter.
End y Justification	Applies to Independent justified steel beams only. Specifies the location of the physical geometry of the finish end of the beam with respect to the location line: either Origin, Left, Center, Right.
End y Offset Value	Applies to Independent justified steel beams only. The numeric value that offsets the geometry at the start end of the beam. The distance between the location line and the characteristic point set in the End y Justification parameter.
End z Justification	Applies to Independent justified steel beams only. Specifies the location of the physical geometry of the finish end of the beam with respect to the location line: either Origin, Left, Center, Right.
End z Offset Value	Applies to Independent justified steel beams only. The numeric value that offsets the geometry at the start end of the beam. The distance between the location line and the characteristic point set in the End z Justification parameter.
Other	
Start Extension Calculation	Applies to steel beams only. Specifies family parameters; defines maximum distance of start extension parameter. This parameter is only available in projects created in versions prior to Autodesk Revit 2014 containing beam families that have not been updated. This is a read-only value.
End Extension Calculation	Applies to steel beams only. Specifies family parameters; defines maximum distance of end extension parameter. This parameter is only available in projects created in versions prior to Autodesk Revit 2014 containing beam families that have not been updated. This is a read-only value.

CHAPTER 5 - STRUCTURAL FLOORS (SLABS)

Slabs are called Floors in Revit. They are loadable families in Revit. You can Create/Customize structural floor family for special use (to be covered later in this course). In this section, you will learn

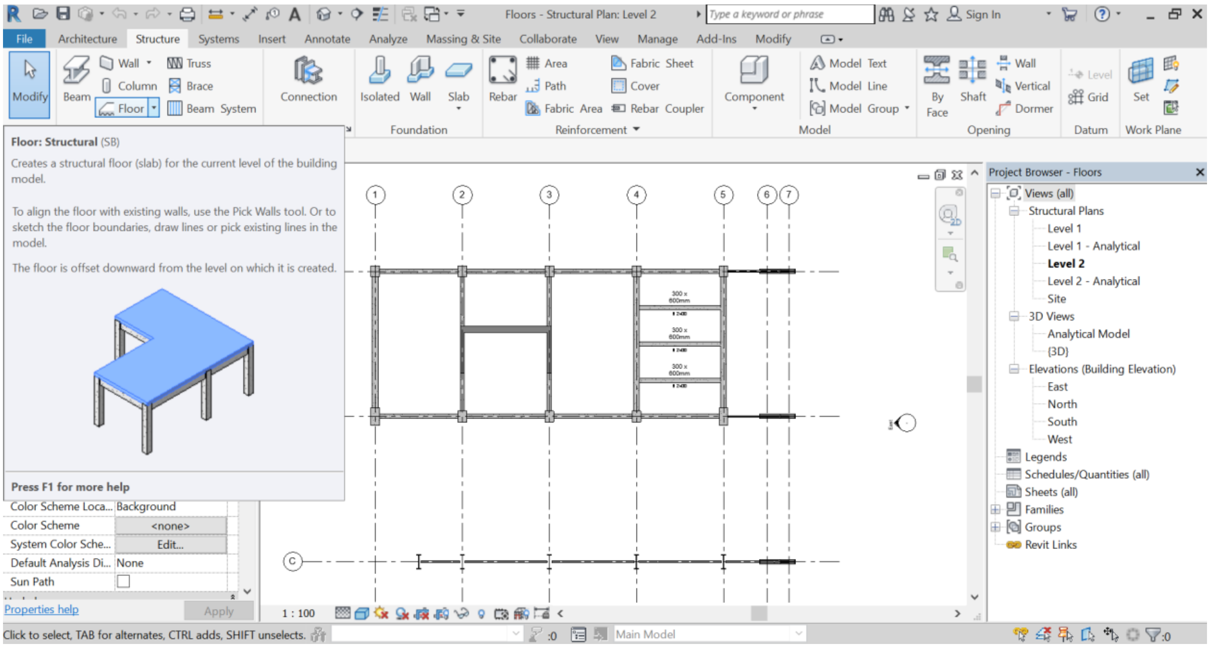
- How to add structural floors.
- Options in floor placement.
- How to edit boundary of a floor.
- Add slabs with metal deck.

5.1 Adding Floors with default settings

To add structural floors in Revit,

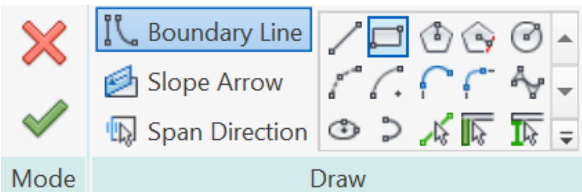
1 Open from the course folder the "Floors.rvt" file.

2 Click Structural Tab ➢ Floor in Structure panel. (The default option is Floor: Structural.)



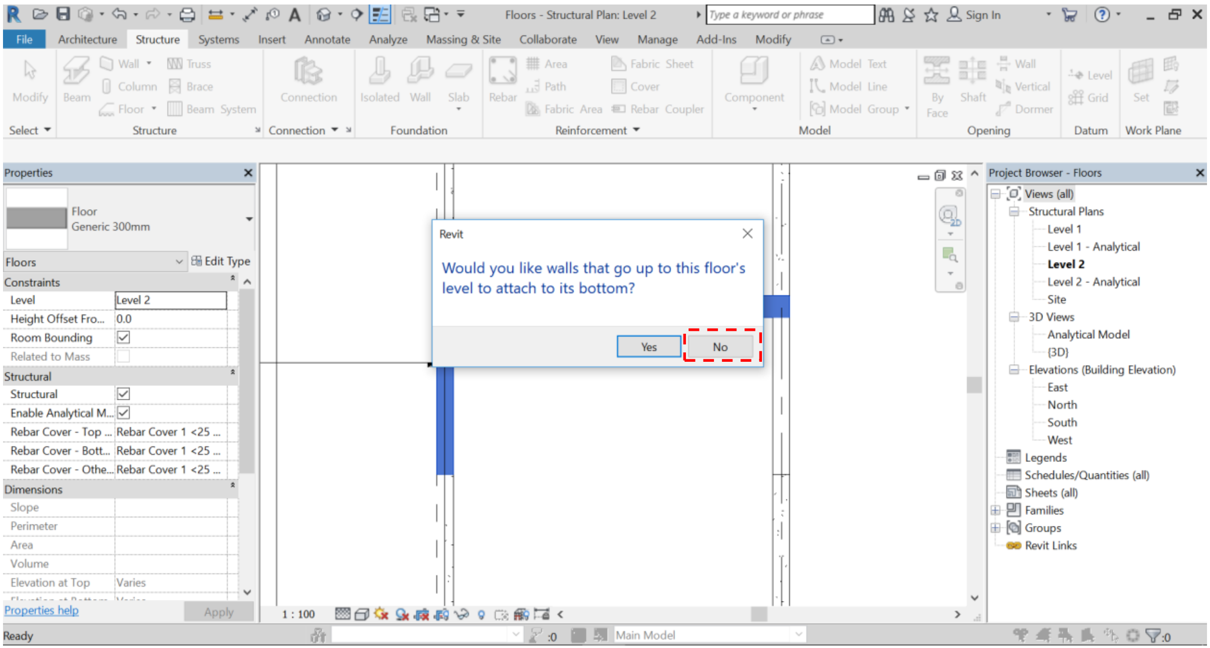
3 In the Properties Palette, select "Generic – 300mm" type from Floor types.

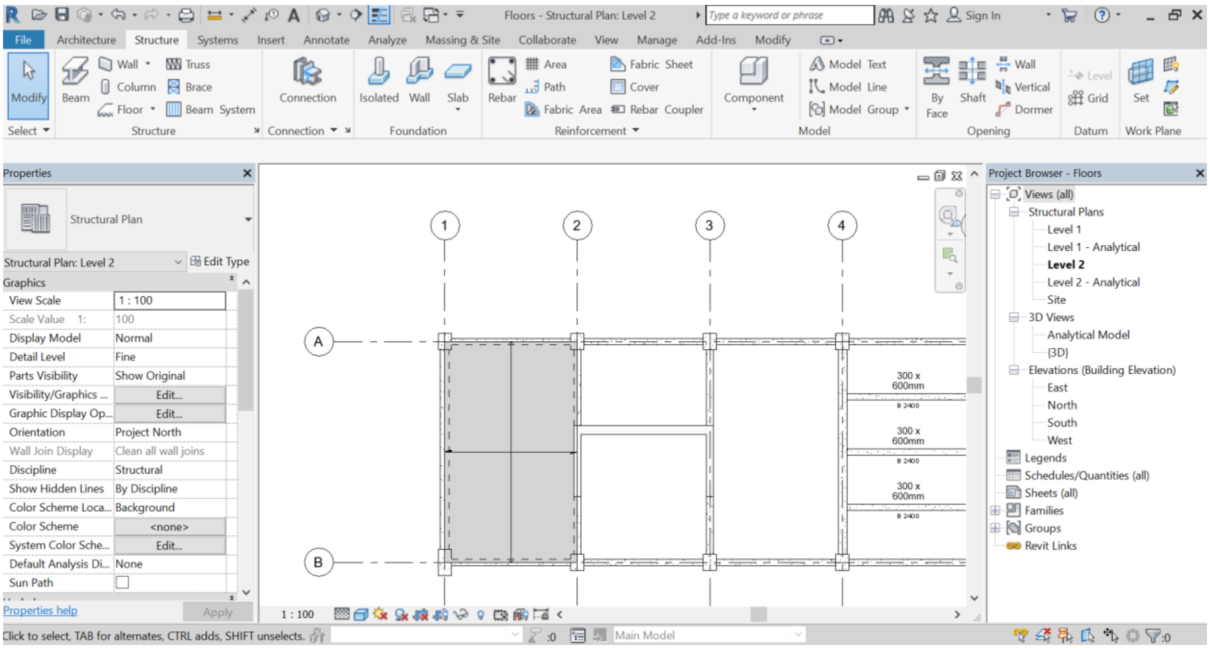
4 Select "Boundary Line" and pick "Rectangle" from the Draw panel. Keep all options unchanged.



5 Pick the first point of the rectangle at Grid-A-1, then pick at Grid-B-2 to finish drawing the rectangle.

6 Click Finish Edit Mode to add the floor. You will be asked "Would you like walls that go up to this floor's level to attach to its (the slab's) bottom". Click "No" such that top of the walls will remain unchanged.

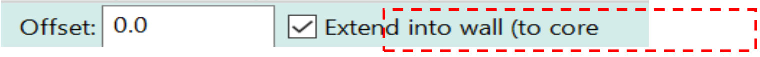


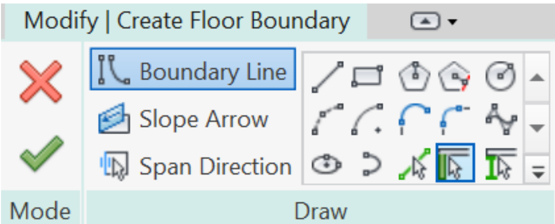


5.2 Options in Creating the Floor Boundary

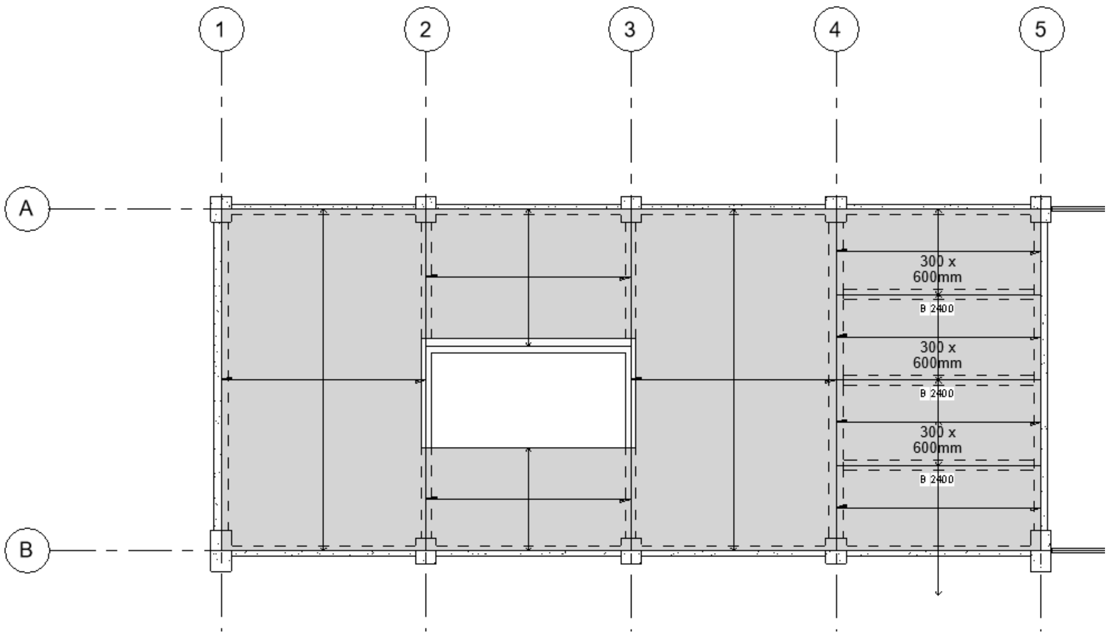
There are many options available to help you to define the boundary of a floors in Revit. Many of the options are self-explanatory. You can hover over the option and see its description and press F1 for more help. Under the contextual ribbon tab, the options for creating floor boundary are:

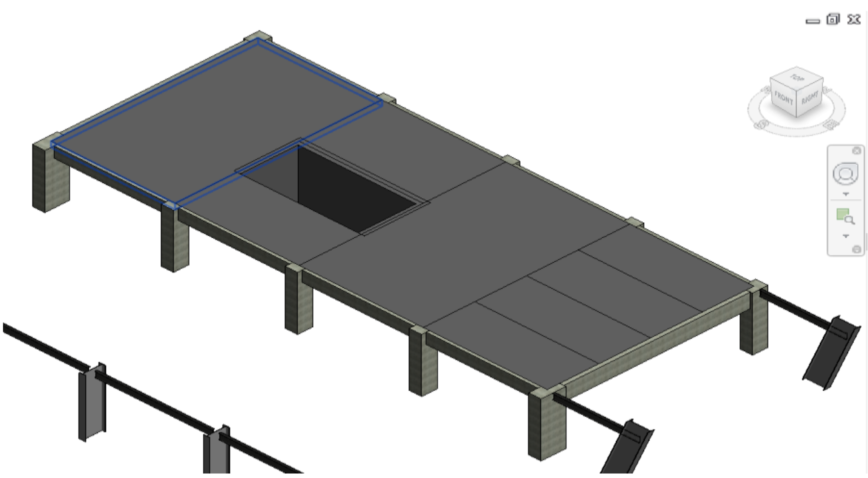
- Draw: Boundary Line / Slope Arrow / Span Direction

- Draw by: Line / Rectangle / Inscribed Polygon / Circumscribed Polygon / Circle / Start-End-Radius Arc / Center-ends Arc / Tangent End Arc / Fillet Arc / Spline / Ellipse / Partial Ellipse / Pick Lines / Pick Walls / Pick Supports.
- Options: Chair / Offset / Radius
- Note: By default, Revit will extend the floor into wall (to core). If you do not want the floor to be extended into wall, uncheck this option before you click on any of the boundary creation method. 



Now, draw all the floors as shown using Rectangle or any method available. All the floors are Generic 300mm. (Note: Floors are created one at a time. You cannot create two or more floors at the same time.)

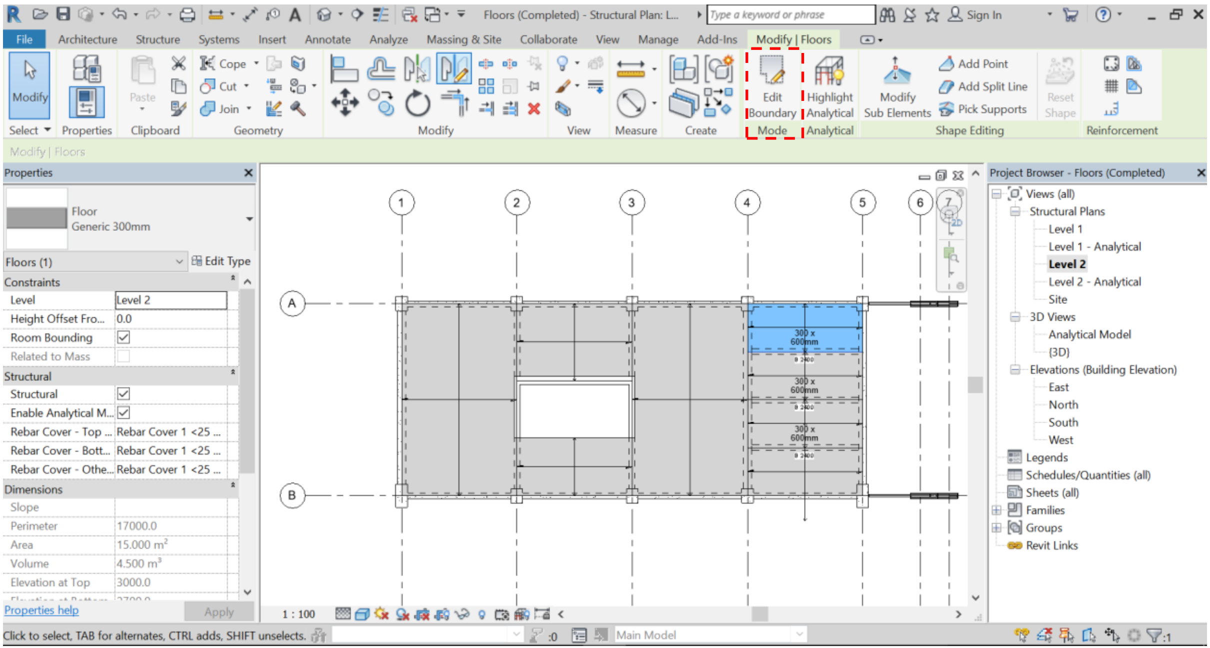




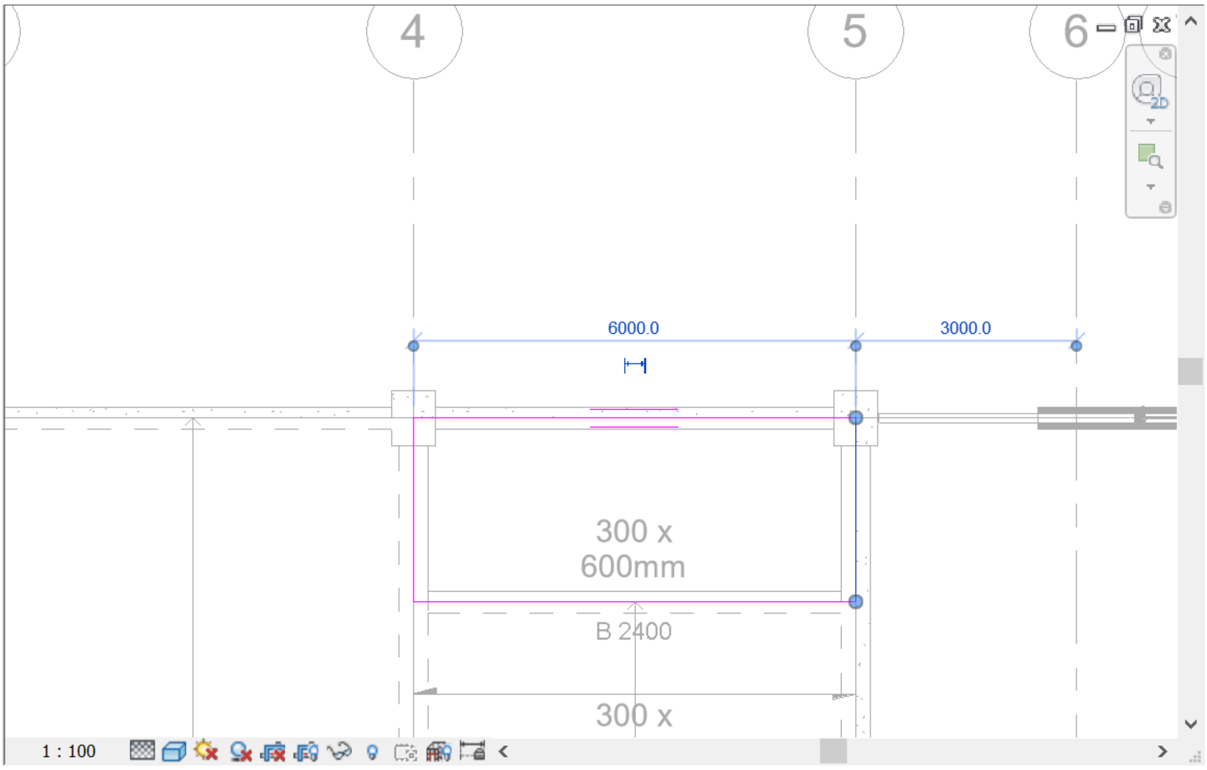
5.3Edit Floor Boundary and Span Direction

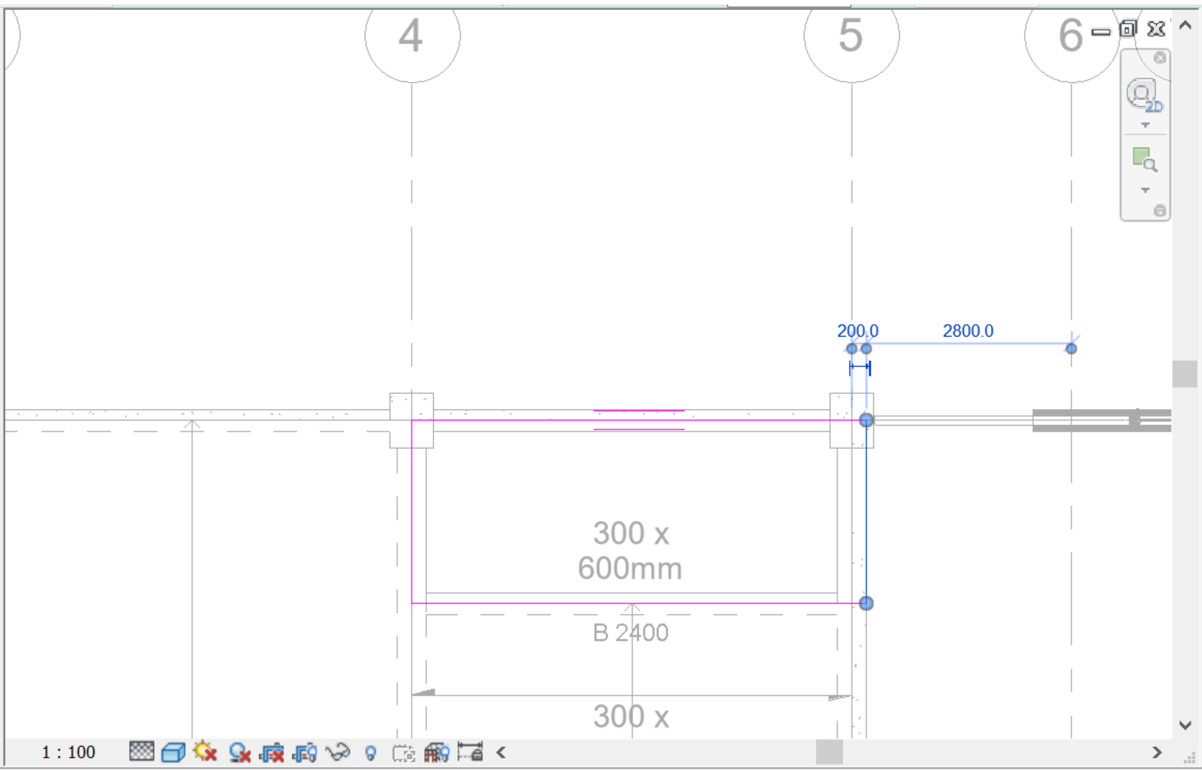
1 Go to Structural Plan / Level 2.

2 Select the top right floor and click Edit Boundary.

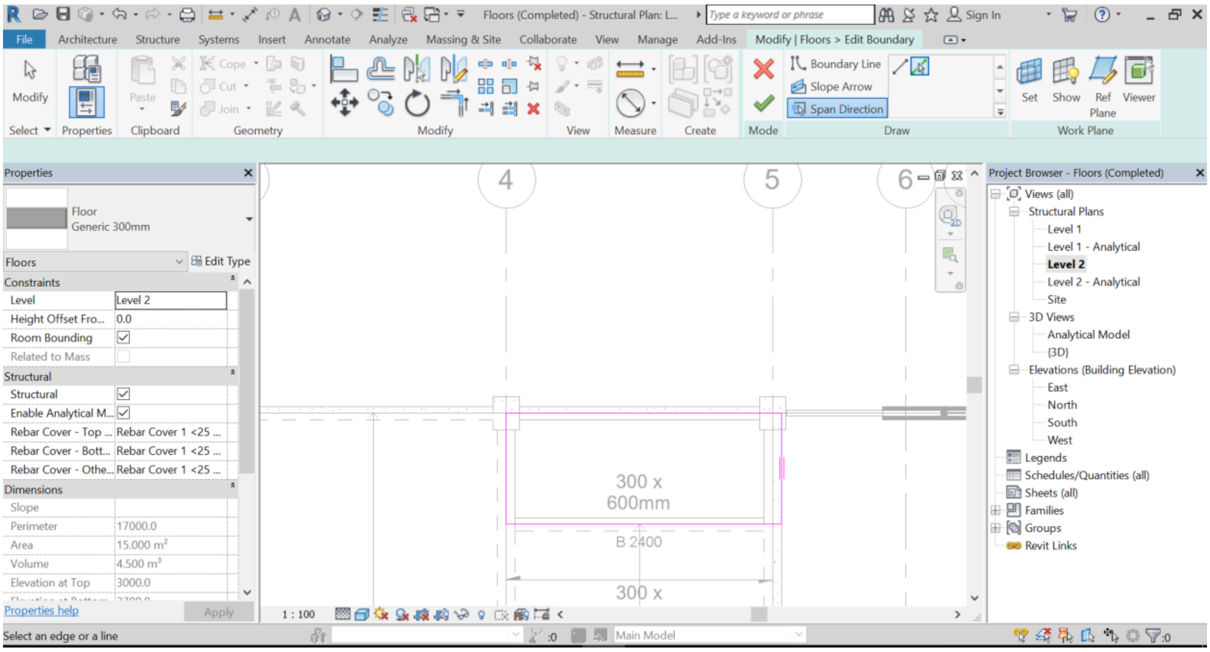


3 Select the short edge on Grid-5.  Move it to the outer edge.

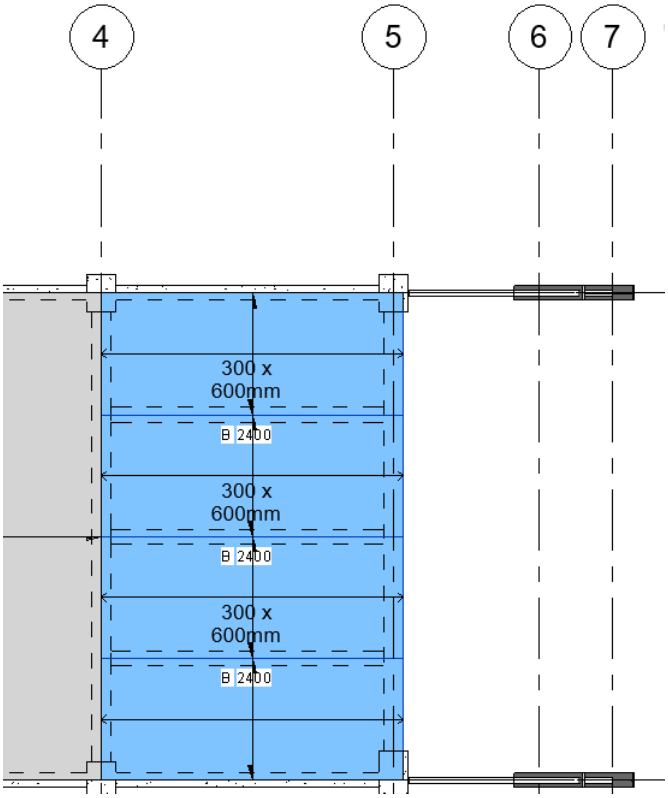




4 Click on Span Direction and pick the edge near Grid-5 to change the span direction to this short span. Click to finish editing.

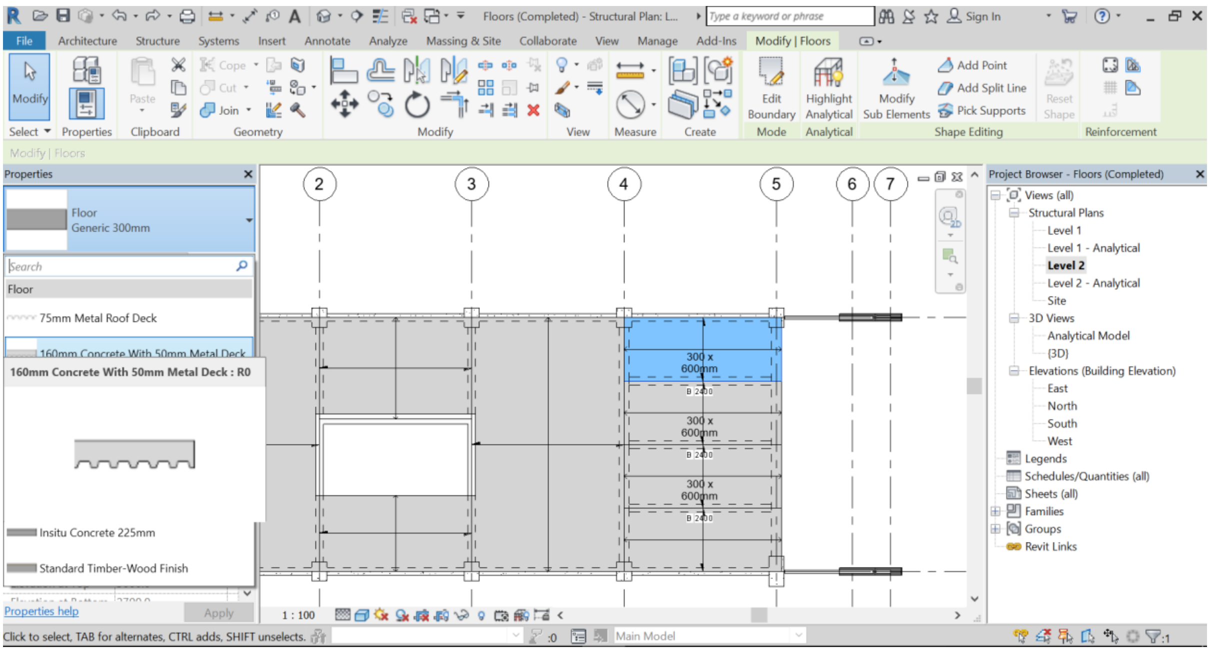


5 Repeat steps 3 and 4 for the other 3 floors along Grid-5.

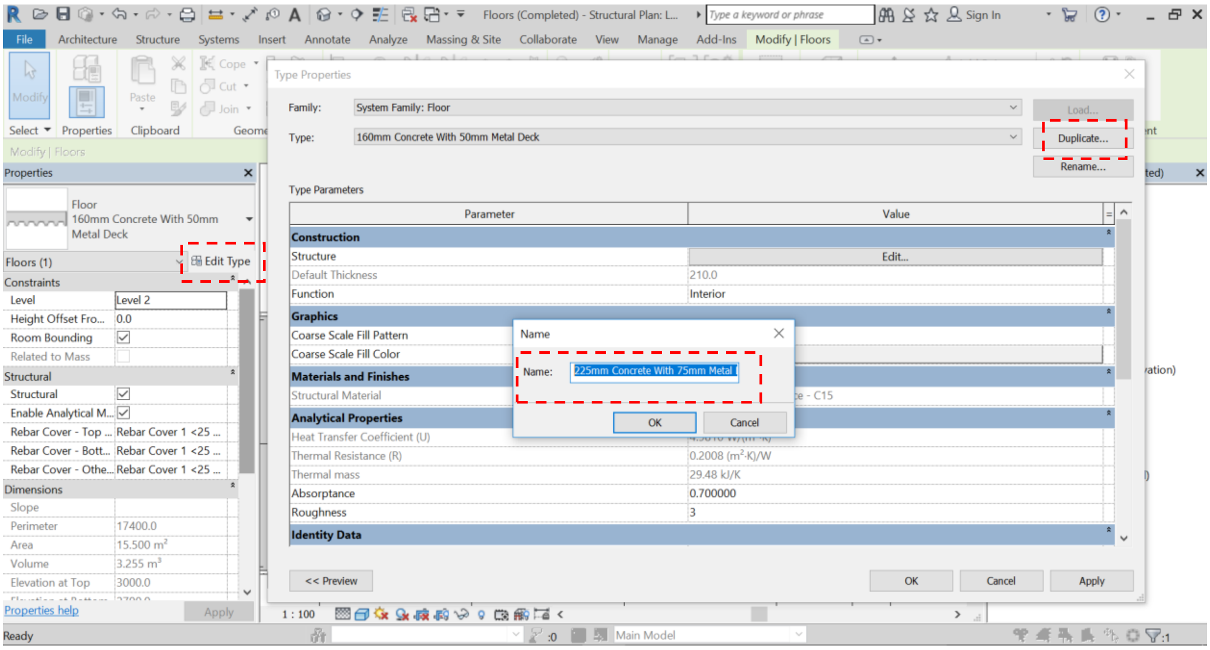


5.4Composite Slab (Slab with Metal Deck)

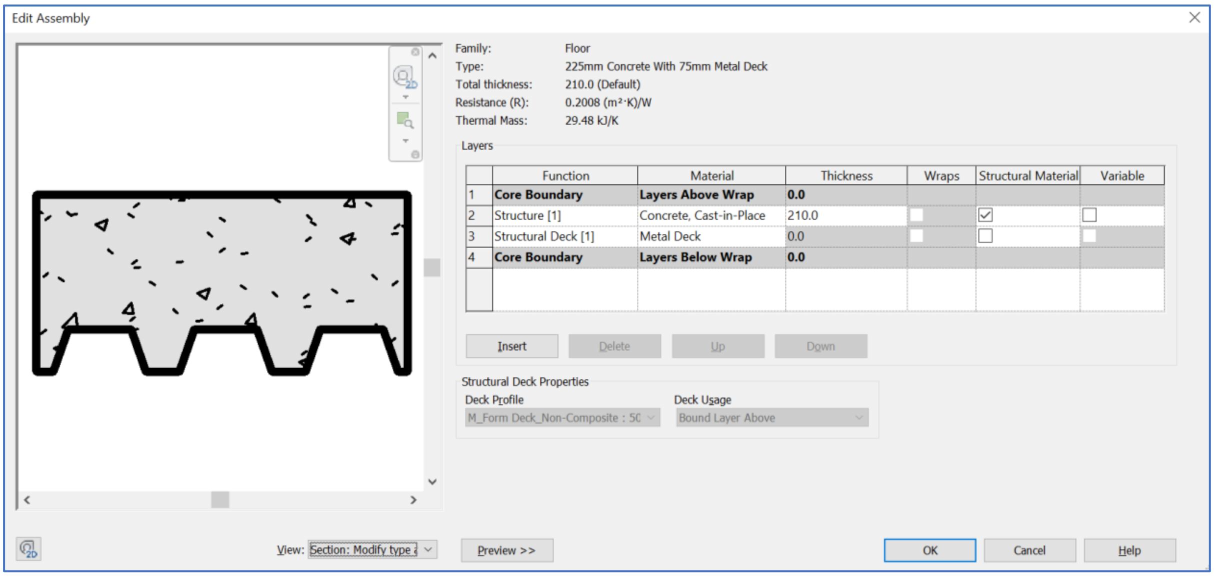
1 Select the top right floor then change its type to “160mm Concrete With 50mm Metal Deck” from the Type Selector.



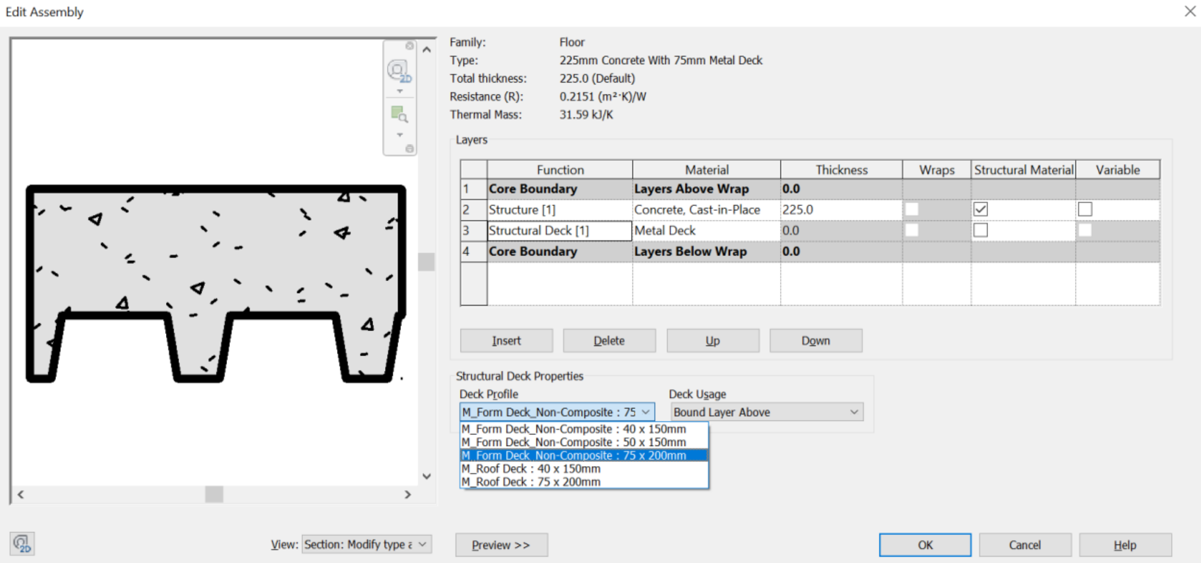
2 Click Edit Type then duplicate. Change the name to “225mm Concrete With 75mm Metal Deck”. Click Ok.



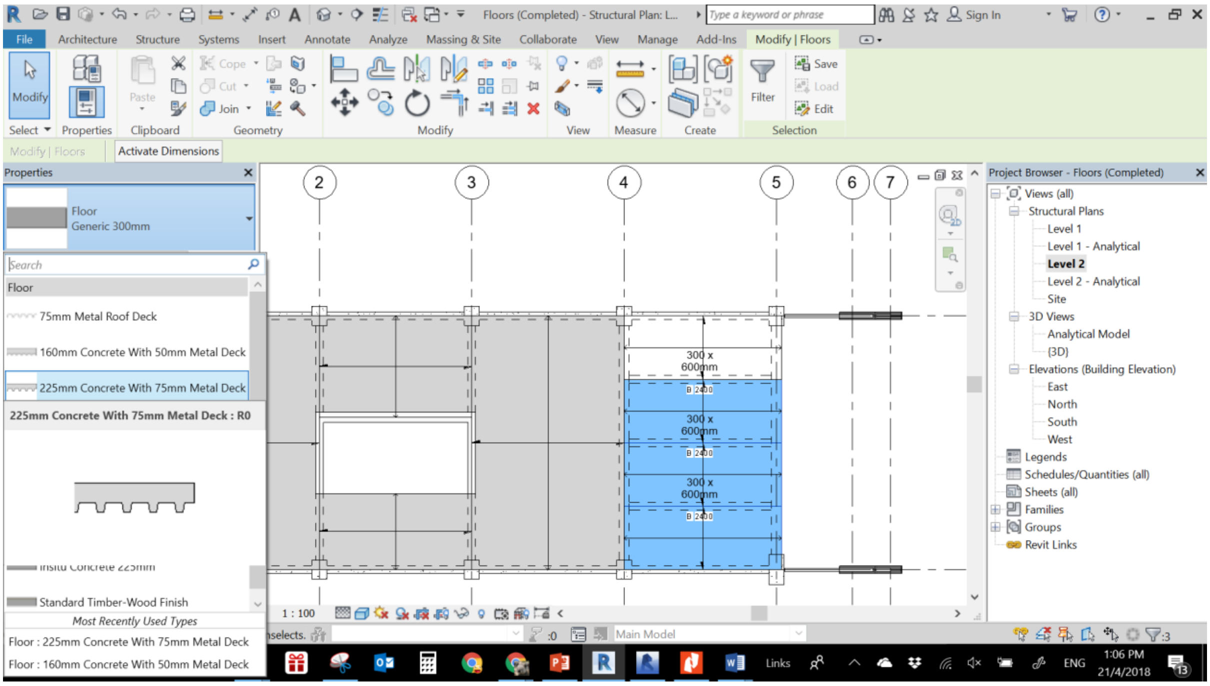
3 Click “Edit” (within the Type Properties dialogue box) to open the “Edit Assembly” dialogue box. Click “Preview” to see the section of the floor type.



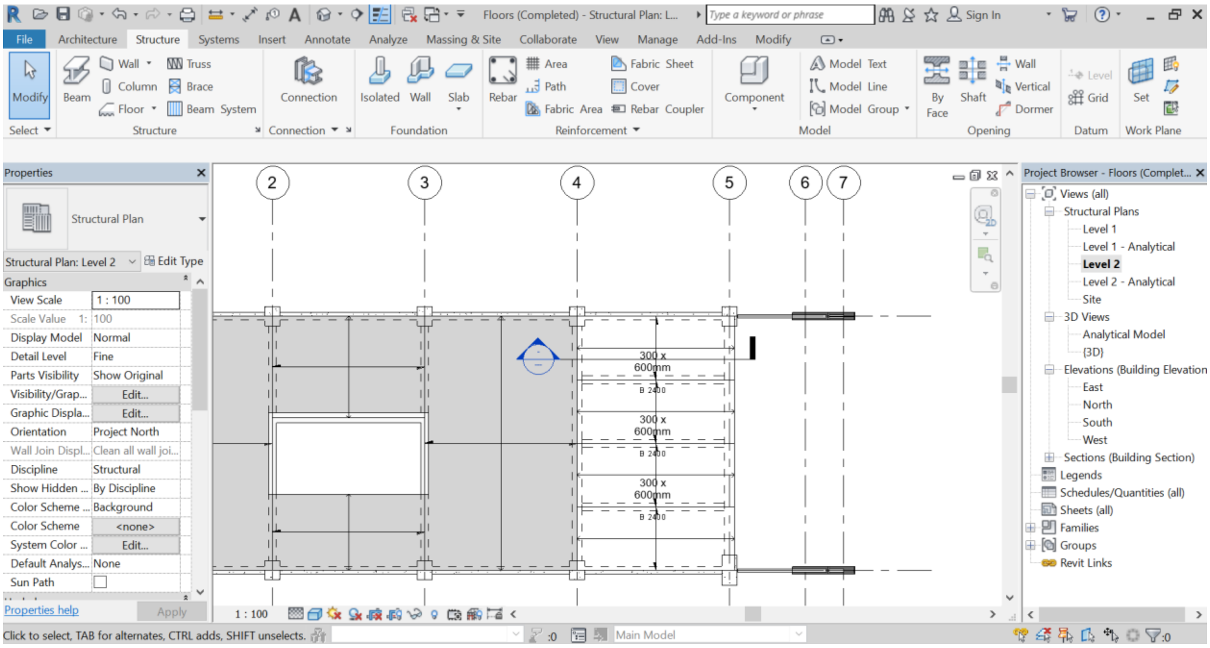
4 Select the Structural Deck layer [1] (Layer 3) and change the Deck Profile to “75 x 200mm”. Also change the thickness of Structure [1] (Layer 2) to 225mm. (Note that 225 is the overall thickness of the floor.) You can customize the deck profile and loaded it in the project for use.

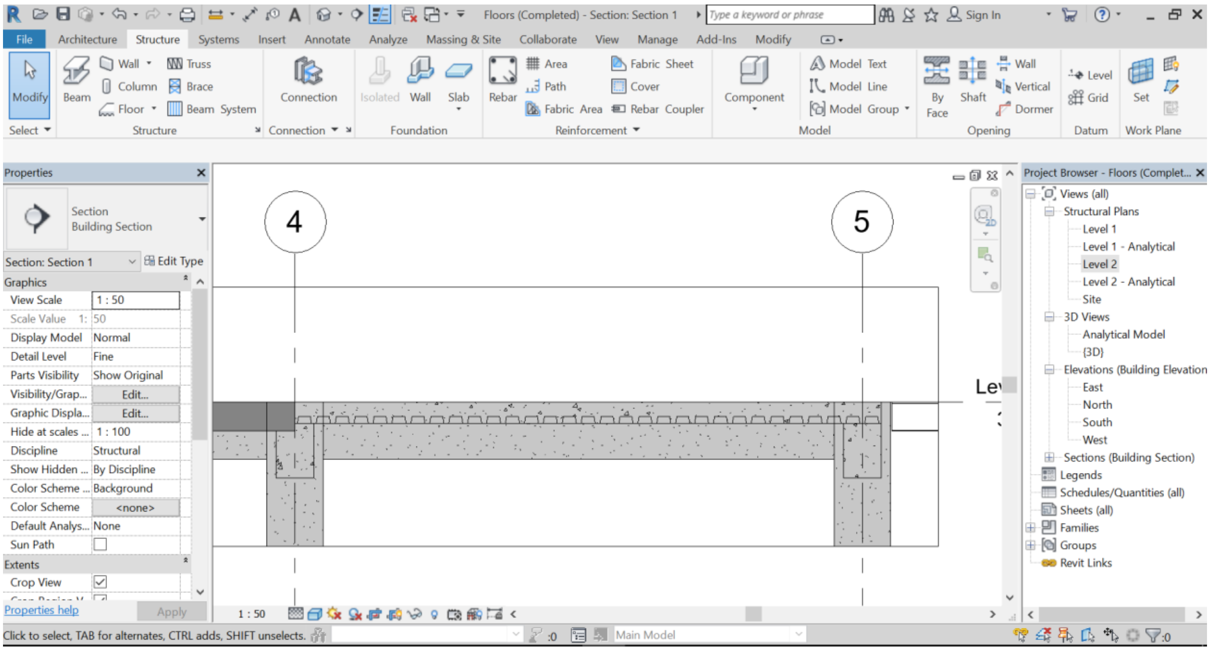


5 Select the other 3 floors along Grid-5 and change their type to “225mm Concrete With 75mm Metal Deck”.



6 Cut a section across the top right floor and then go to this section view to view the section.

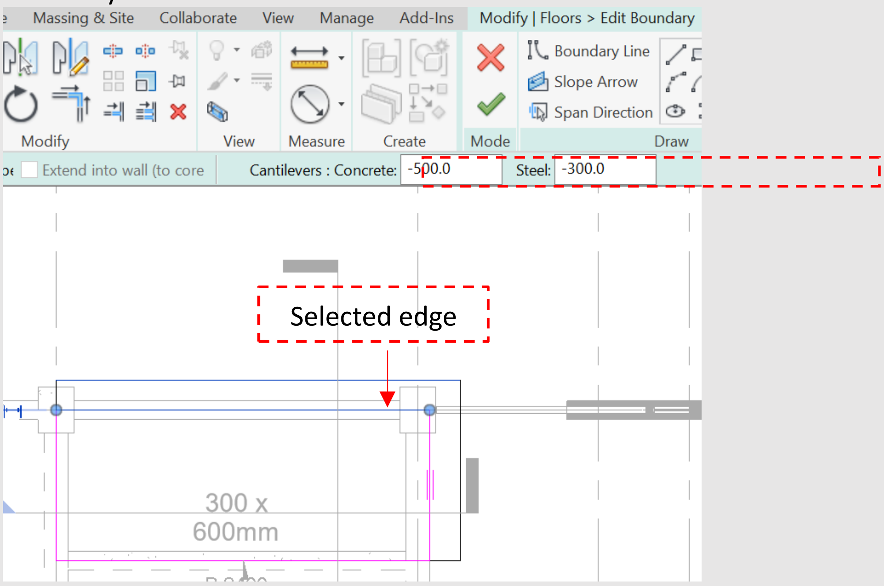


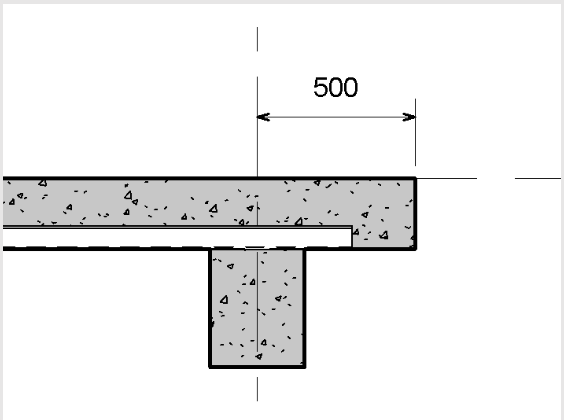


You can create and load a customized metal deck profile for the composite floor. It will be discussed in more detail in the customize families section later.

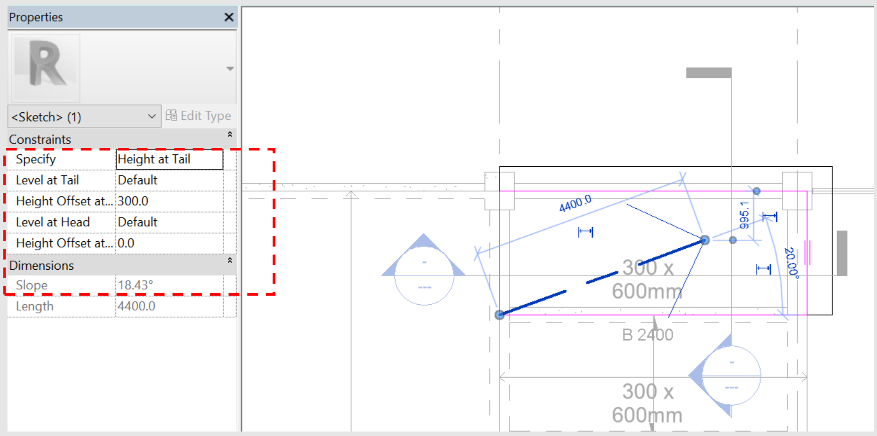
Notes:

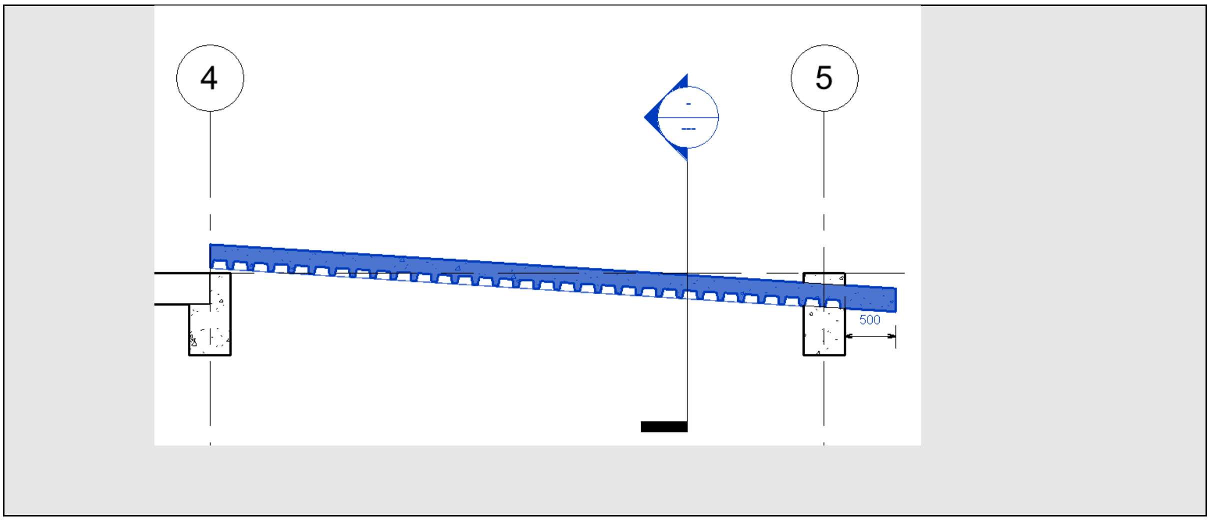
- The amount of cantilever of concrete and metal deck can be adjusted separately for composite floor. They can be set when you select an edge of the floor after entering Edit Boundary mode.





- Steps to create a sloped structural floor:
  1. Draw a Slope Arrow while sketching or editing the floor boundary.
  2. Specify a value for the Offset from Base property for parallel floor sketch lines.
  3. Specify values for the Defines Slope and Slope properties for a single floor sketch line.





5.4 Detail Description of Structural Floor Instance Properties

(source: Autodesk Knowledge Network)

https://knowledge.autodesk.com/support/revit-products/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/Revit-Model/files/GUID-1E6B2B45-1650-481B-AC72-C755FC4BDAC6-htm.html

Name	Description
Constraints	
Location Line	Specifies the position of the floor with respect to the line sketched in the project elevation. For more information, see Place a Floor. The floor location line remains the same for that floor, even if the type changes.
Base Constraint	Specifies the level from which the floor base is referenced.
Base Offset	Specifies the offset of the base of the floor from its base constraint.
Base is Attached	Indicates whether the base of the floor is attached to another component, such as a structural floor. This is a read-only value.
Base Extension Distance	Indicates the distance you have moved the base of the layers in a floor. See Compound Structure. This parameter is enabled when the layers of a floor are set to extendable.
Top Constraint	The name of the level to which the floor top is set.
Unconnected Height	If top constraint is unconnected, you may set an unconnected height of the floor. This value is read-only if there is a top constraint.
Top Offset	Specifies the offset of the top of the floor from its top constraint; enabled only when the top constraint is set to a level.
Top is Attached	Indicates whether the top of the floor is attached to another component, such as a structural floor. This is a read-only value.

Name	Description
Top Extension Distance	Indicates the distance you have moved the top of the layers in a floor. See Compound Structure. This parameter is enabled when the layers of a floor are set to extendable.
Room Bounding	Indicates whether the floor is part of a room boundary. This parameter is enabled after you place the floor.
Related to Mass	This is a read-only value.
Structural	
Rebar Cover - Exterior Face	Specifies the rebar cover distance from the floor exterior face.
Rebar Cover - Interior Face	Specifies the rebar cover distance from the floor interior face.
Rebar Cover - Other Faces	Specifies the rebar cover distance from the faces of adjacent elements.
Estimated Reinforcement Volume	Specifies the estimated reinforcement volume of the selected element. This is a read-only parameter that only displays when rebar has been placed.
Structural	Specifies the floor as being a structural element able to posses an analytical model.
Enable Analytical Model	Displays the analytical model and includes it in analytical calculations. Selected by default. See Disable an Analytical Model.
Structural Usage	Either Bearing, Shear, or Structural Combined. The structural use of the floor.
Dimensions	
Length	Indicates the length of the floor. This is a read-only value.
Area	Indicates the area of the floor. This is a read-only value.
Volume	Indicates the volume of the floor. This is a read-only value.
Identity Data	
Comments	A field for entering comments about the floor.
Mark	A label created for the floor. This value must be unique for each element in a project. Revit warns you when the number is already used but allows you to continue using it. You can see the warning using the Review Warnings tool. See Review Warning Messages.
Phasing	
Phase Created	Indicates in which phase the floor component was created. See Project Phasing.
Phase Demolished	Indicates in which phase the floor component was demolished. See Project Phasing.

Table of Contents

CHAPTER 6 - STRUCTURAL FOUNDATIONS (FOUNDATIONS)

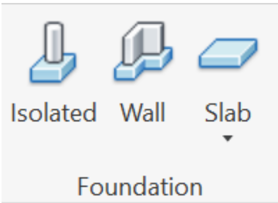
Foundations are loadable families in Revit. You can Create/Customize structural Foundation family for special use (not to be covered in this course). In this section, you will learn

- Different foundations (Isolated / Wall Foundations and Foundation Slabs) in Revit and how to add them.
- Create step footings.

6.1 Different Foundations in Revit

You can create foundations for the building model using the following three foundations in Revit:

(1) Isolated Foundations / (2) Wall Foundations / (3) Foundation Slabs



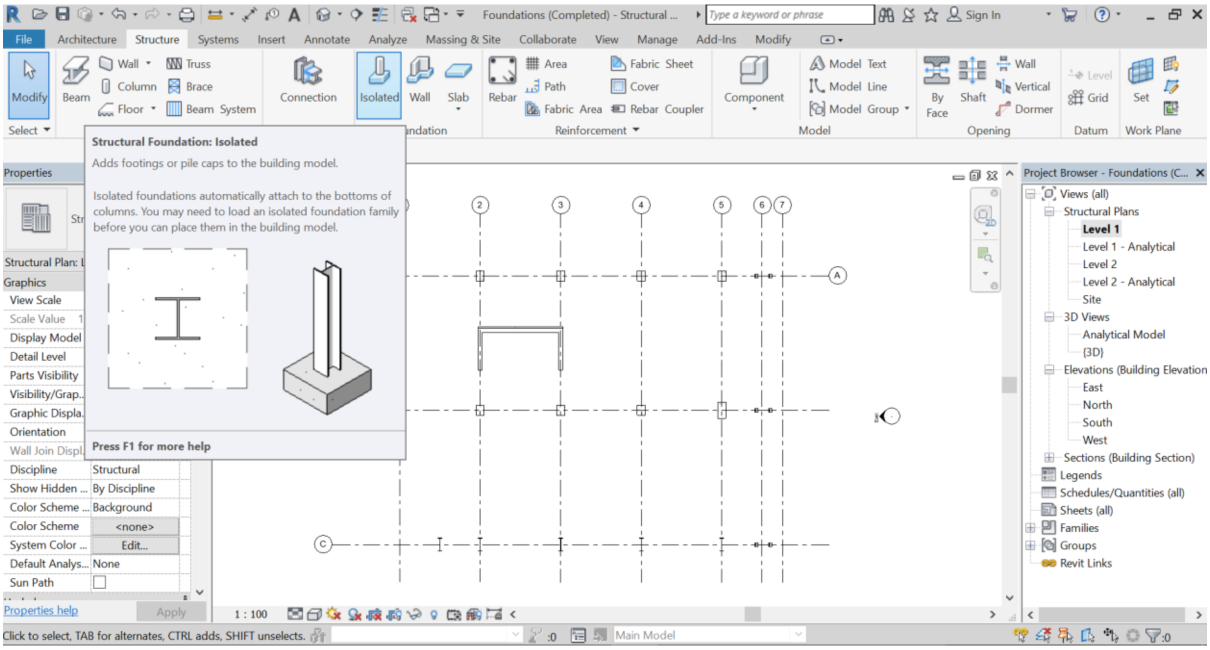
- Use Isolated Foundation to place a footing at the bottom of a structural column.
- Use Wall Foundation to place a footing at the bottom of a structural wall.
- Use Foundation Slab to sketch a foundation in the model.

6.2 Adding Isolated Foundations with default settings

To add a footing underneath a structural column in Revit,

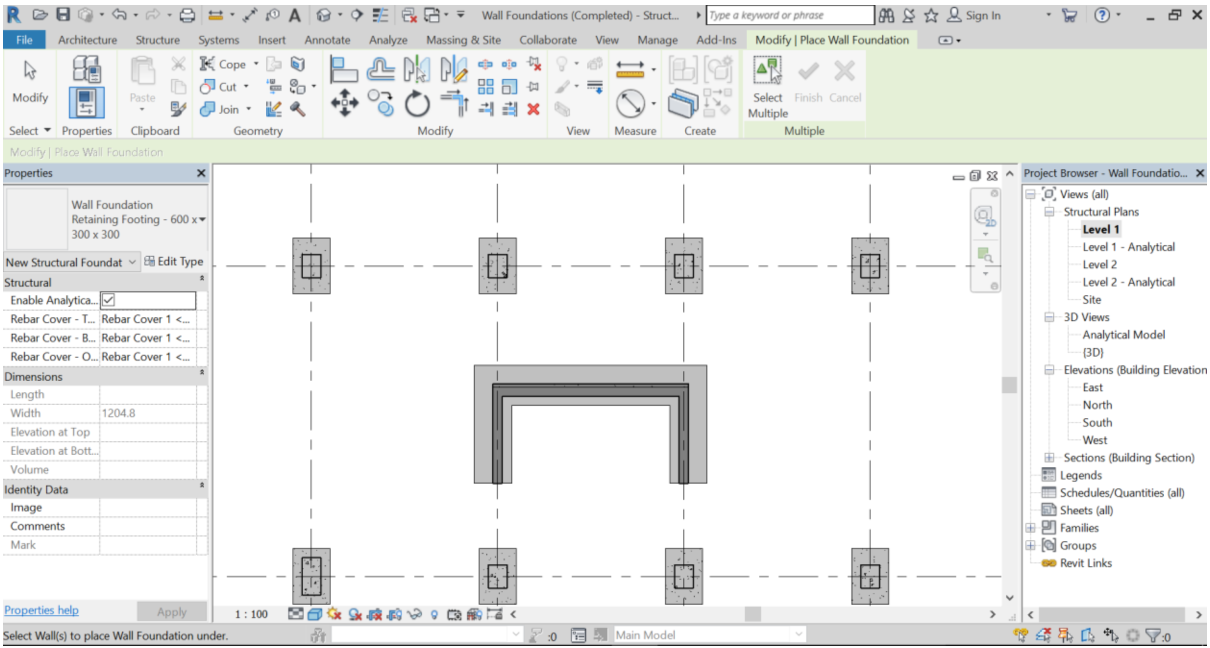
1 Open from the course folder the “Foundations.rvt” file and go to Level 1 (Structural Plan).

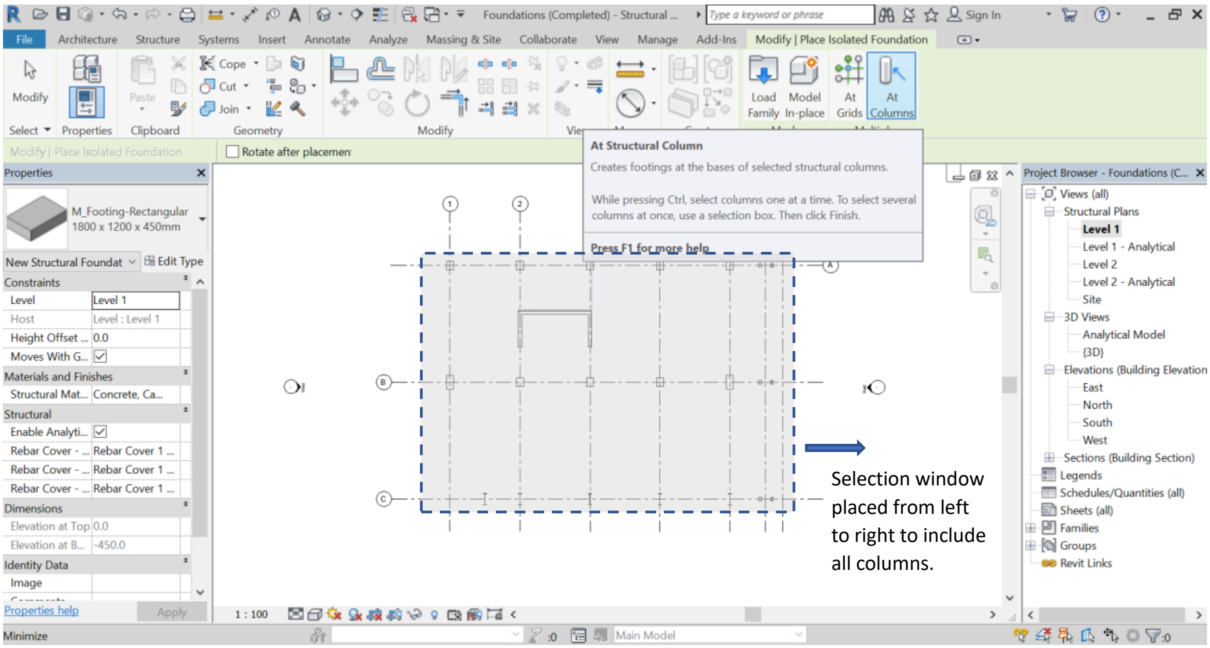
2 Click Structural Tab ➢ Isolated (Foundations) in Foundation panel.



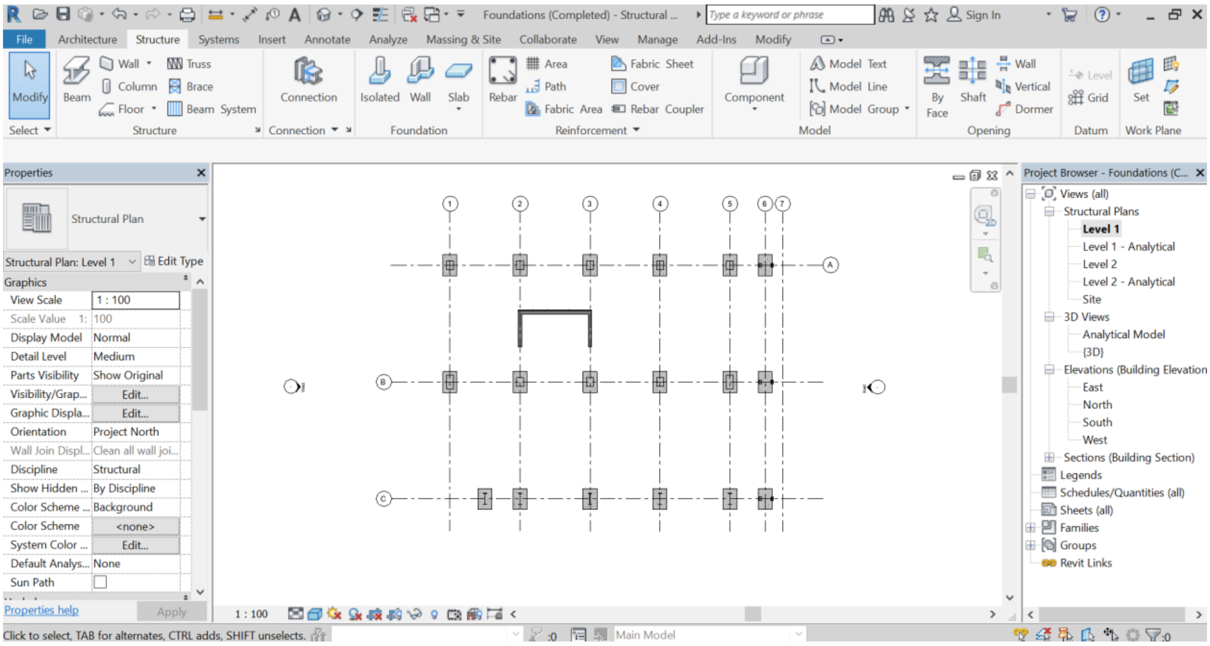
3 In the Properties Palette, select “1800 x 1200 x 450mm” type from M-Footing-Rectangular types.

4 Select “At Columns” from the Multiple panel then pick all the columns. Keep all options unchanged.





5 Click Finish Edit Mode to add the Foundations.

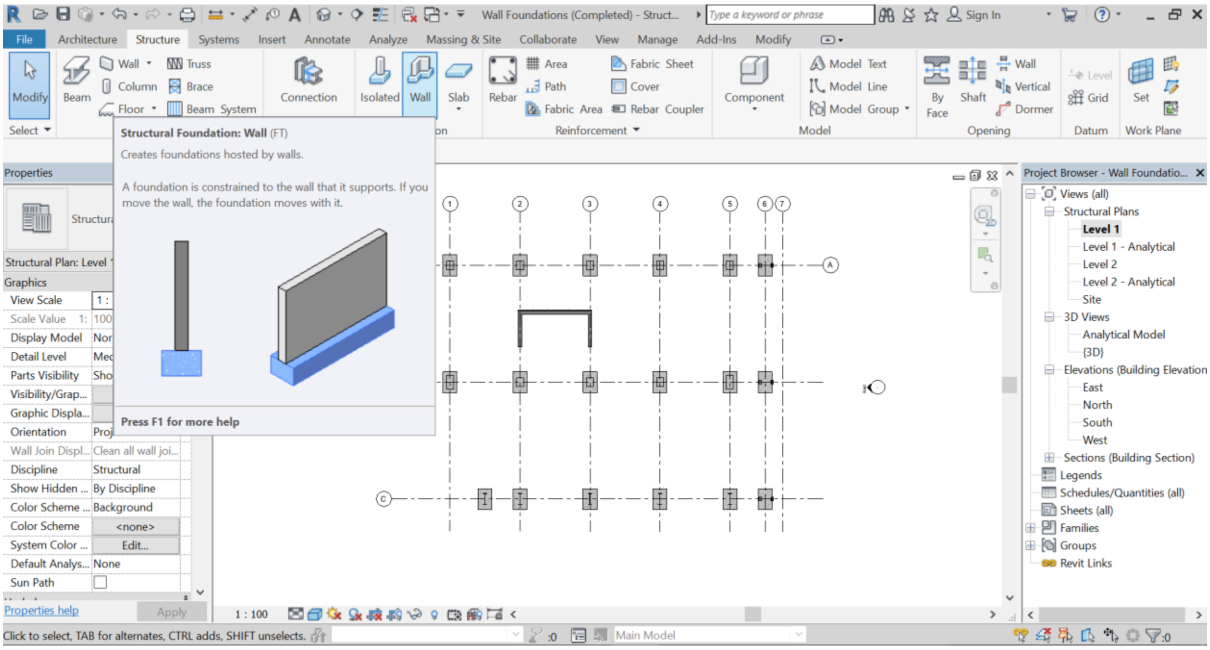


6.3 Adding Wall Foundations in Revit

To add footings underneath structural walls in Revit,

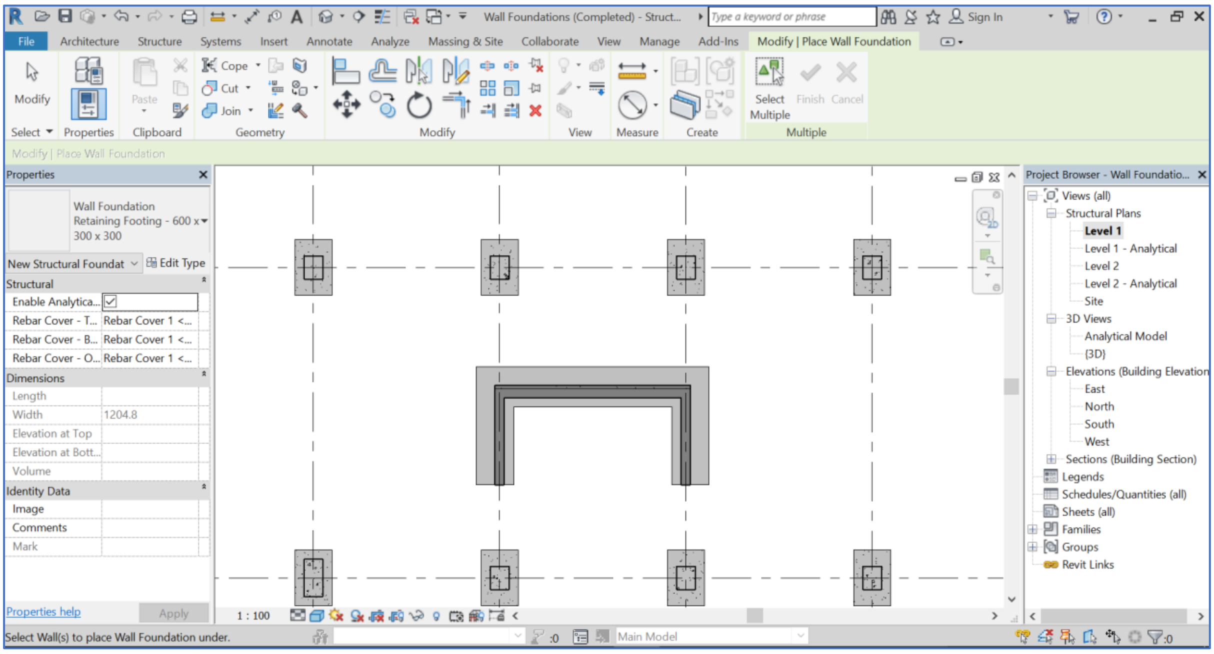
1 Open from the course folder the "Wall Foundations.rvt" file and go to Level 1 (Structural Plan).

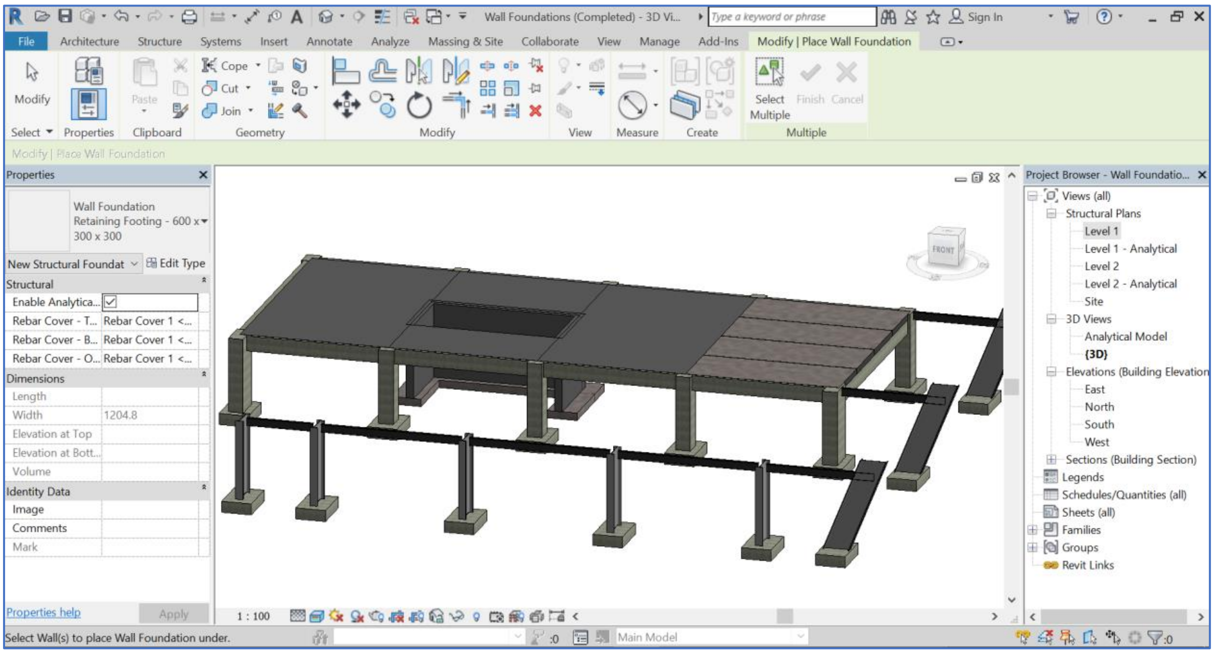
2 Click Structural Tab ➢ Wall (Foundations) in Foundation panel.



3 In the Properties Palette, select “Retaining Footing – 600 x 300 x 300” type from Wall Foundation types.

4 Select all walls in the view. Keep all options unchanged.



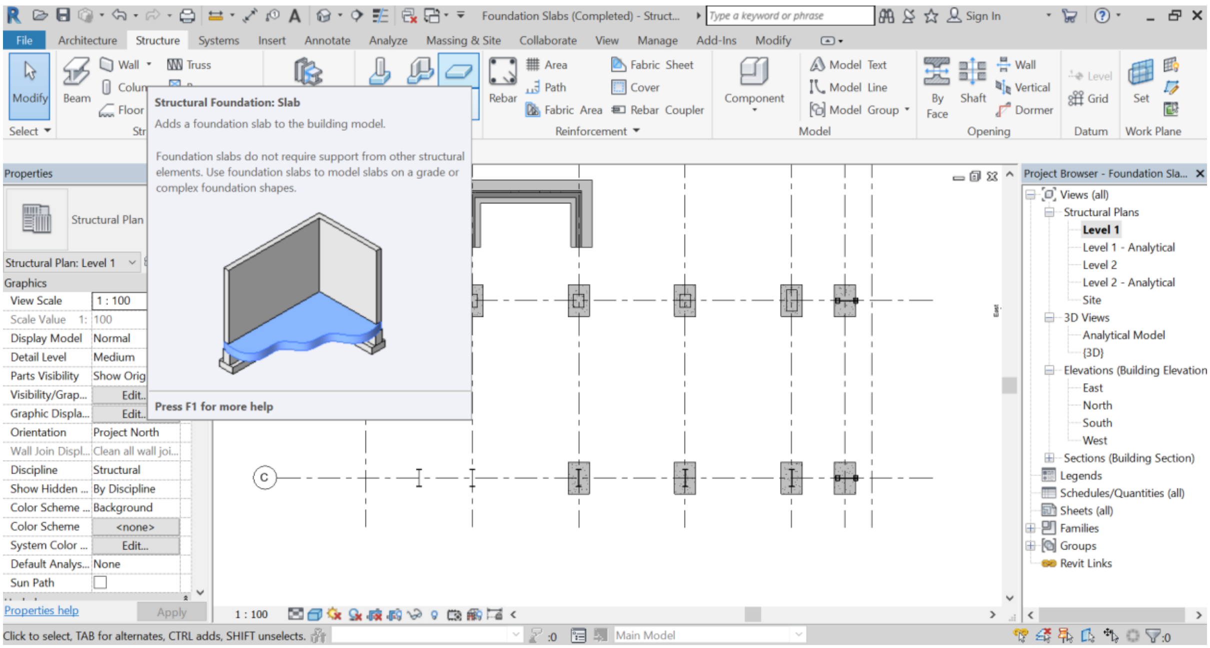


6.4Adding a Footing (Foundation Slab) under 2 Columns

To add a footing underneath 2 (or more) structural columns in Revit,

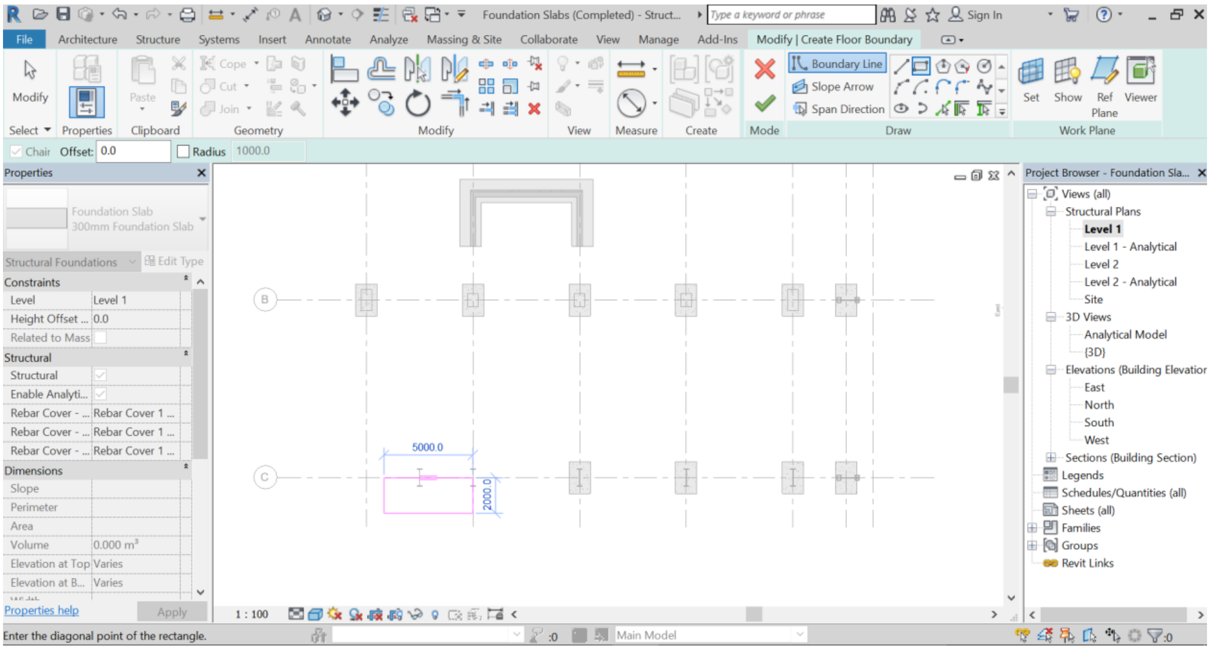
1 Open from the course folder the “Foundation Slabs.rvt” file and go to Level 1 (Structural Plan).

2 Click Structural Tab ➢ Slab in Foundation panel. (The default option is “Structural Foundation: Slab”.)



3 In the Properties Palette, select “300mm Foundation Slab” type from Foundation Slab types.

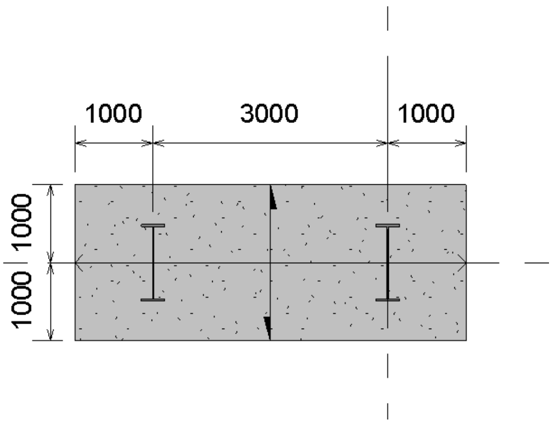
4 Create a 5000 x 2000mm rectangular footing at Grid-B-2 at shown below. Keep all options unchanged.

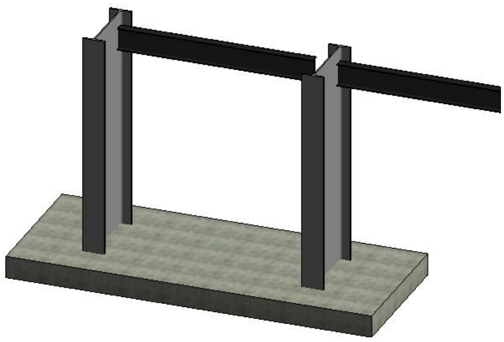


5 Select Span Direction Span Direction and pick the shorter edge along Grid-2 to change the span direction.

6 Click Finish Edit Mode to add the footing.

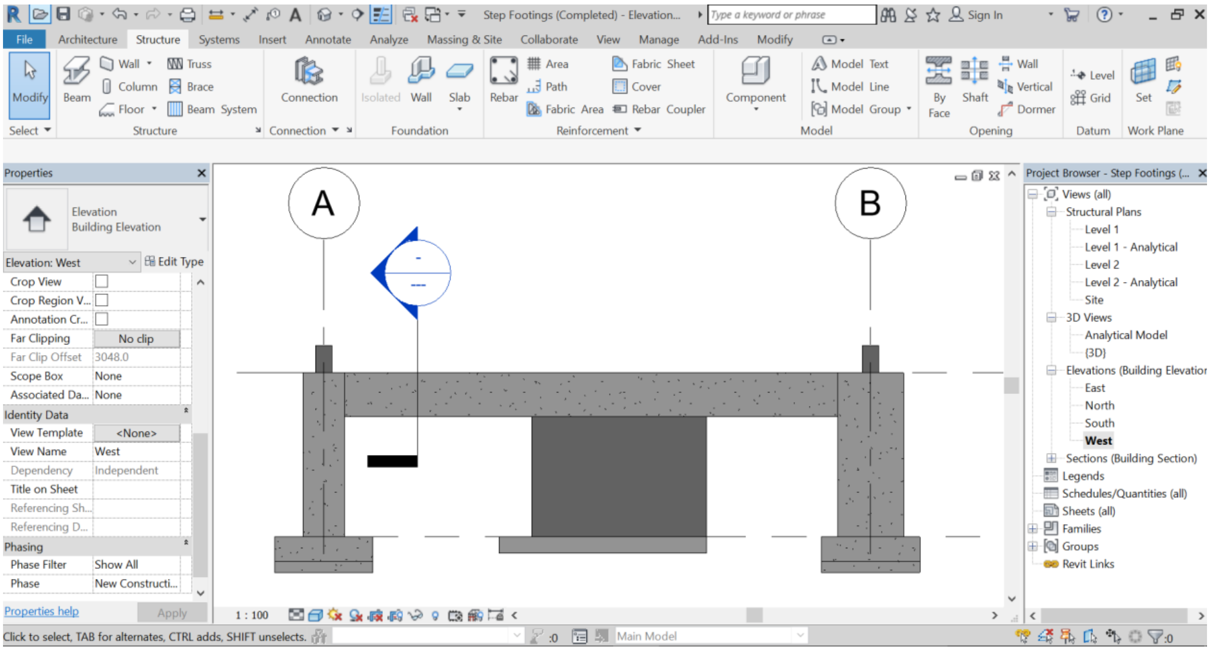
7 Move the footing to the location as shown below.

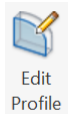




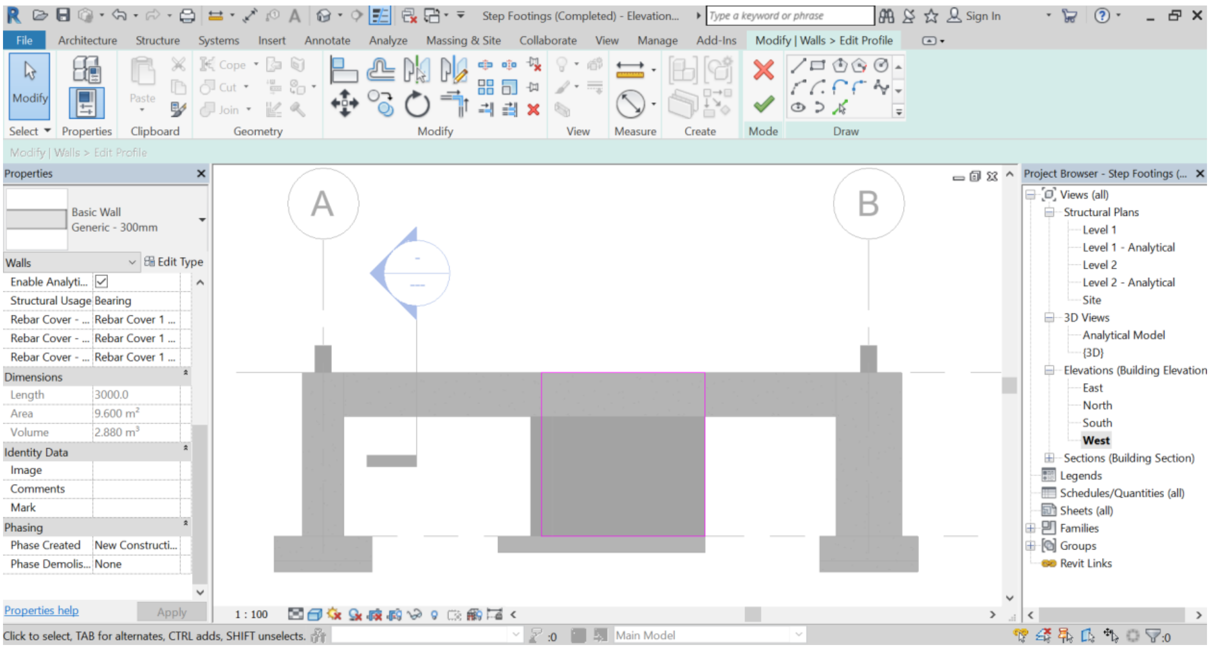
6.5Creating a Step Footing (Under structural walls)

1 Open from the course folder the “Step Footings.rvt” file and go to West Elevation.



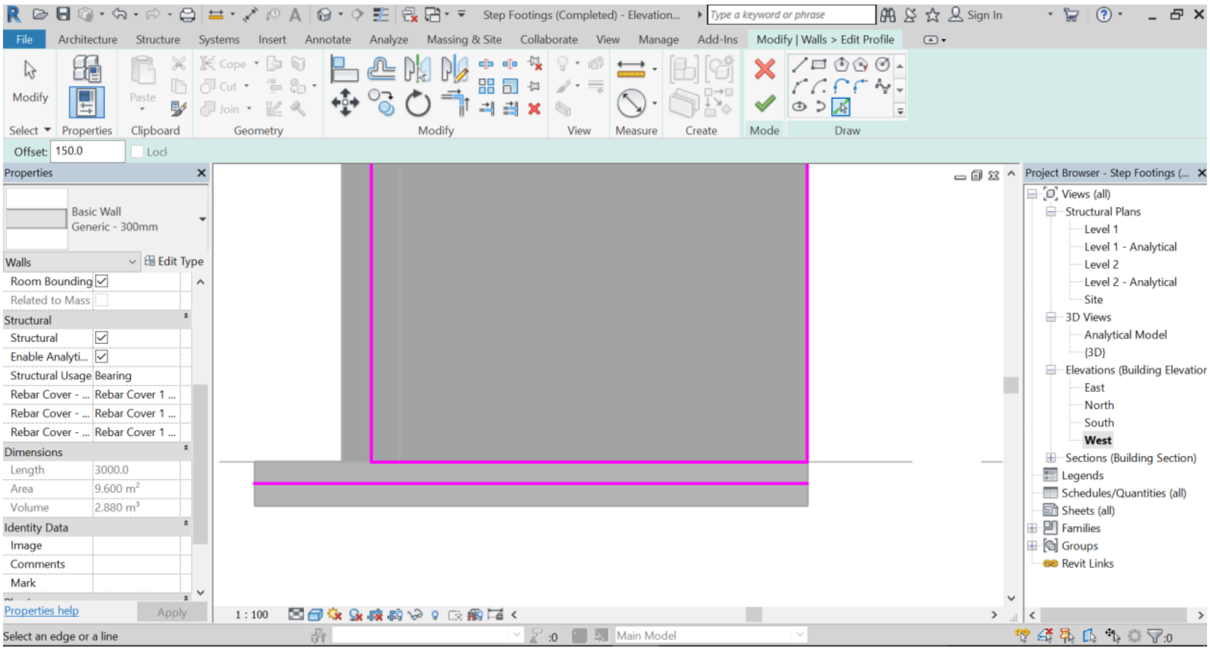
2 Select the wall facing the west elevation. Then click Edit Profile 

.

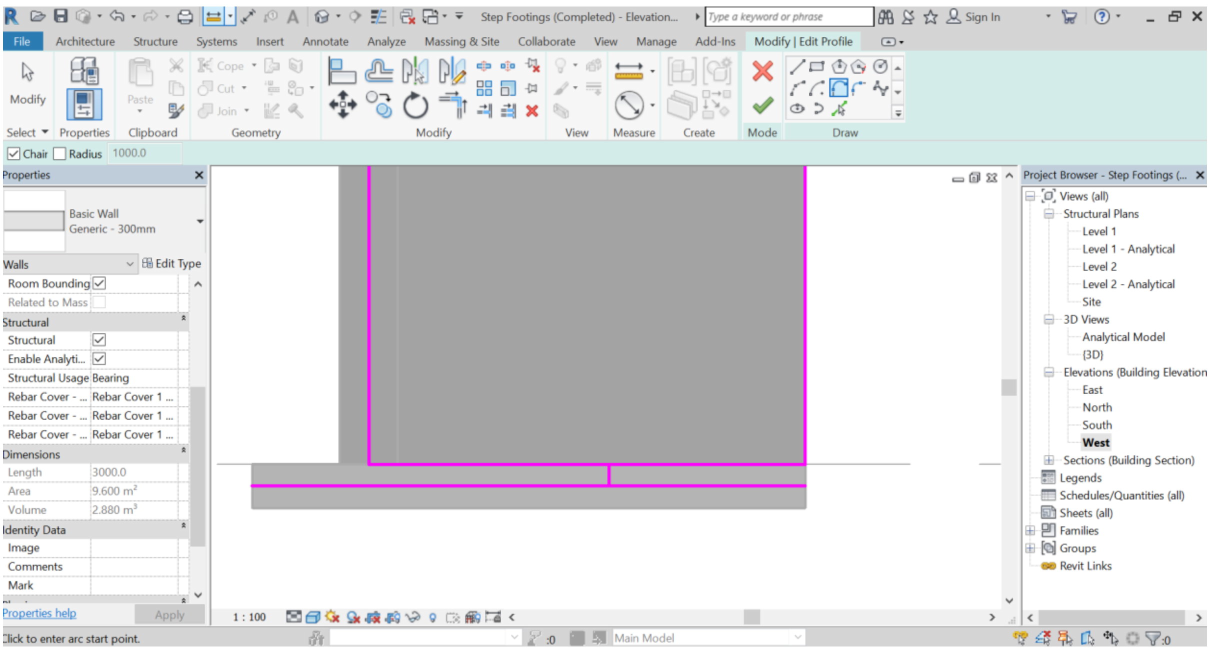


3 Use “Pick Lines”  to draw the profile and set Offset to 150mm Offset: 150.0 .

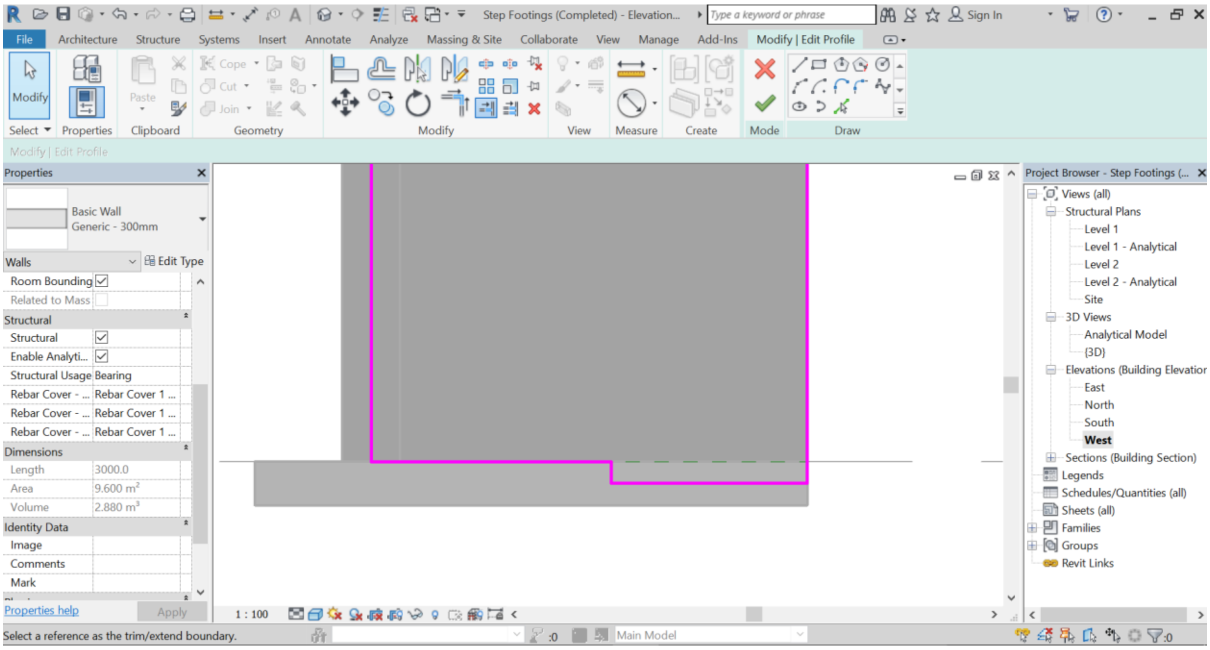
4 Hover near the bottom of the wall from beneath and click to draw a profile line.



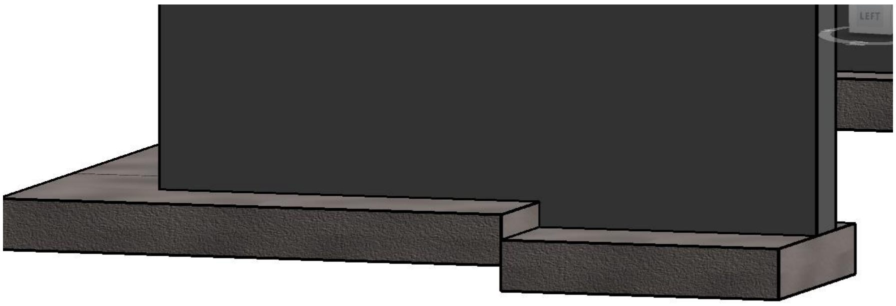
5 Draw a line from the mid-point of the wall bottom to the line below it.



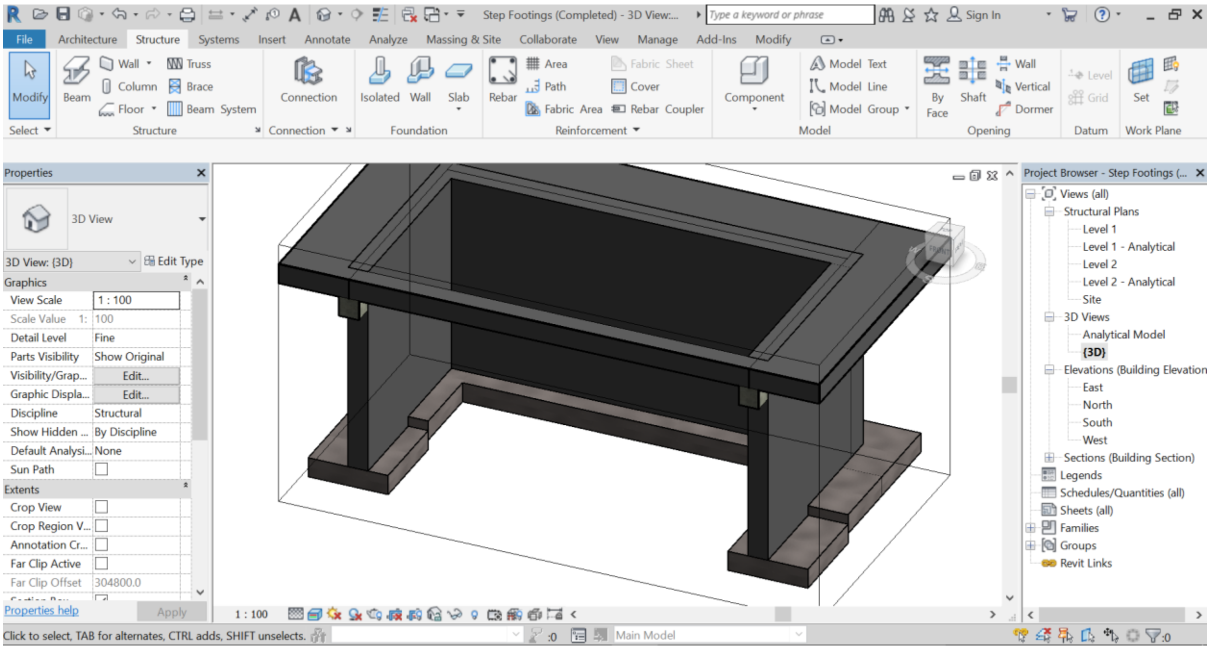
6 Trim the profile using the modify tool (Trim/Extend Single Element) to form a closed profile of the wall. (Remember to click the portion of the line you want to retain.)



7 Click Finish Edit Mode  to finish editing.



8 Create similar step footing for the wall facing south.

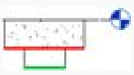


6.6 Detail Description of Isolated Foundation Instance Properties

(source: Autodesk Knowledge Network)

https://knowledge.autodesk.com/support/revit-products/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/Revit-Model/files/GUID-F7483668-8348-41EC-AB54-ED74BBA6D4E6-htm.html

Name	Description
Constraints	
Level	The level to which the slab is constrained.
Host	The level to which the host of the isolated slab is constrained.
Offset	Specifies the elevation of the top of the isolated foundation relative to its level.
Moves With Grids	Changes the column constraint to grid.
Materials and Finishes	
Structural Material	Specifies a material for the structure of the element. This information can be included in schedules. Click the value box to open the Material Browser. See Materials.
Structural	
Enable Analytical Model	Displays the analytical model and includes it in analytical calculations. Selected by default. See Disable an Analytical Model.

Name	Description
Rebar Cover - Top Face	Specifies the distance of the rebar cover from the element top face.
Rebar Cover - Bottom Face	Specifies the distance of the rebar cover from the element bottom face.
Rebar Cover - Other Faces	Specifies the distance of the rebar cover from the element to adjacent element faces.
Estimated Reinforcement Volume	Specifies the estimated reinforcement volume of the selected element. This is a read-only parameter that only displays when rebar has been placed.
Dimensions	
Elevation at Top	Indicates the elevation used for tagging the top of the foundation.
Elevation at Bottom	Indicates the elevation used for tagging the bottom of the foundation. This parameter is affected by the family parameter Cap (see Specify Family Category and Parameters). The following image shows the difference between the Cap parameter enabled (in red) and disabled (in green).  The Elevation at Bottom measurement is taken from this setting. This is a read-only parameter.
Identity Data	
Comments	A field for entering comments about the wall.
Mark	A label created for the element. This value must be unique for each element in a project. Revit warns you if the number is already used but allows you to continue using it. You can see the warning using the Review Warnings tool. See Review Warning Messages.
Phasing	
Phase Created	Indicates in which phase the component was created. See Project Phasing.
Phase Demolished	Indicates in which phase the component was demolished. See Project Phasing.

6.7 Detail Description of Foundation Slab Instance Properties

(source: Autodesk Knowledge Network)

https://knowledge.autodesk.com/support/revit-products/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/Revit-Model/files/GUID-C14650C3-F139-4401-B2CD-2D1E8CD8FD89-htm.html

Name	Description
Constraints	
Level	Specifies the level that the slab is constrained to.
Height Offset From Level	Specifies the elevation of the top of the slab relative to the constrained level of the slab.
Related to Mass	This is a read-only value.
Structural	
Structural	If checked, Slab has an Analytical Model and is exported to analysis packages.
Enable Analytical Model	Displays the analytical model and includes it in analytical calculations. Selected by default. See Disable an Analytical Model.
Rebar Cover - Top Face	The rebar cover distance from the foundation slab top face. S See Allow Stirrup and Tie Rebar to Snap to the Exterior....
Rebar Cover - Bottom Face	The rebar cover distance from the foundation slab bottom face. S See Allow Stirrup and Tie Rebar to Snap to the Exterior....
Rebar Cover - Other Faces	The rebar cover distance from the foundation slab to adjacent element faces. S See Allow Stirrup and Tie Rebar to Snap to the Exterior....
Estimated Reinforcement Volume	Specifies the estimated reinforcement volume of the selected element. This is a read-only parameter that only displays when rebar has been placed.
Dimensions	
Slope	Specifies the slope of the slab from the horizontal plane.
Perimeter	Indicates the slab perimeter. This is a read-only parameter.
Area	Indicates the area of the slab. This is a read-only parameter.
Volume	Indicates the volume of the slab. This is a read-only parameter.
Width	Indicates the width of the slab, if rectangular. This is a read-only parameter.

Name	Description
Length	Indicates the length of the slab, if rectangular. This is a read-only parameter.
Elevation at Top	Indicates the elevation used for tagging the top of the foundation. This is a read-only parameter.
Elevation at Bottom	Indicates the elevation used for tagging the bottom of the foundation. This is a read-only parameter.
Thickness	Indicates the thickness of the slab. This is a read-only parameter.
Identity Data	
Comments	A field for placing general comments about the slab.
Mark	A label created for the slab. Possible use: shop mark. This value must be unique for each element in a project. Revit warns you when the number value is already used but allows you to continue using it. You can see the warning using the Review Warnings tool. See Review Warning Messages.
Phasing	
Phase Created	Indicates in which phase the slab component was created. See Project Phasing.
Phase Demolished	Indicates in which phase the slab component was demolished. See Project Phasing.

6.8 Detail Description of Wall Foundation Instance Properties

(source: Autodesk Knowledge Network)

https://knowledge.autodesk.com/support/revit-products/learn-explore/caas/CloudHelp/cloudhelp/2016/ENU/Revit-Model/files/GUID-615AB37C-044C-4104-85BC-7FA8E29A5AE0-htm.html

Name	Description
Constraints	
Eccentricity	**Bearing walls only.** Specifies the offset from the wall center line to the bearing foundation center line.
Structural	
Enable Analytical Model	Displays the analytical model and includes it in analytical calculations. Selected by default. See Disable an Analytical Model.
Rebar Cover - Top Face	Specifies the rebar cover distance from the wall top face.

Name	Description
Rebar Cover - Bottom Face	Specifies the rebar cover distance from the wall bottom face.
Rebar Cover - Other Faces	Specifies the rebar cover distance from the wall to adjacent element faces.
Estimated Reinforcement Volume	Specifies the estimated reinforcement volume of the selected element. This is a read-only parameter that only displays when rebar has been placed.
Dimensions	
Length	Indicates the overall length. This is a read-only parameter.
Width	Indicates the overall width. This is a read-only parameter.
Volume	Indicates the volume of the foundation. This is a read-only parameter.
Elevation at Top	Indicates the elevation used for tagging the top of the foundation. This is a read-only parameter that reports Varies for sloped planes.
Elevation at Bottom	Indicates the elevation used for tagging the bottom of the foundation. This is a read-only parameter that reports Varies for sloped planes.
Identity Data	
Comments	A field for placing general comments about the foundation type.
Mark	A label created for the foundation. This value must be unique for each element in a project. Revit warns you when the number value is already used but allows you to continue using it. You can see the warning using the Review Warnings tool. See Review Warning Messages.
Phasing	
Phase Created	Indicates in which phase the foundation component was created. See Project Phasing.
Phase Demolished	Indicates in which phase the foundation component was demolished. See Project Phasing.

CHAPTER 7 - TRUSSES AND STEEL CONNECTIONS

Revit include a library of trusses and steel connections tool for your structural model. You can Create/Customize truss family for special use (not to be covered in this course). In this section, you will learn

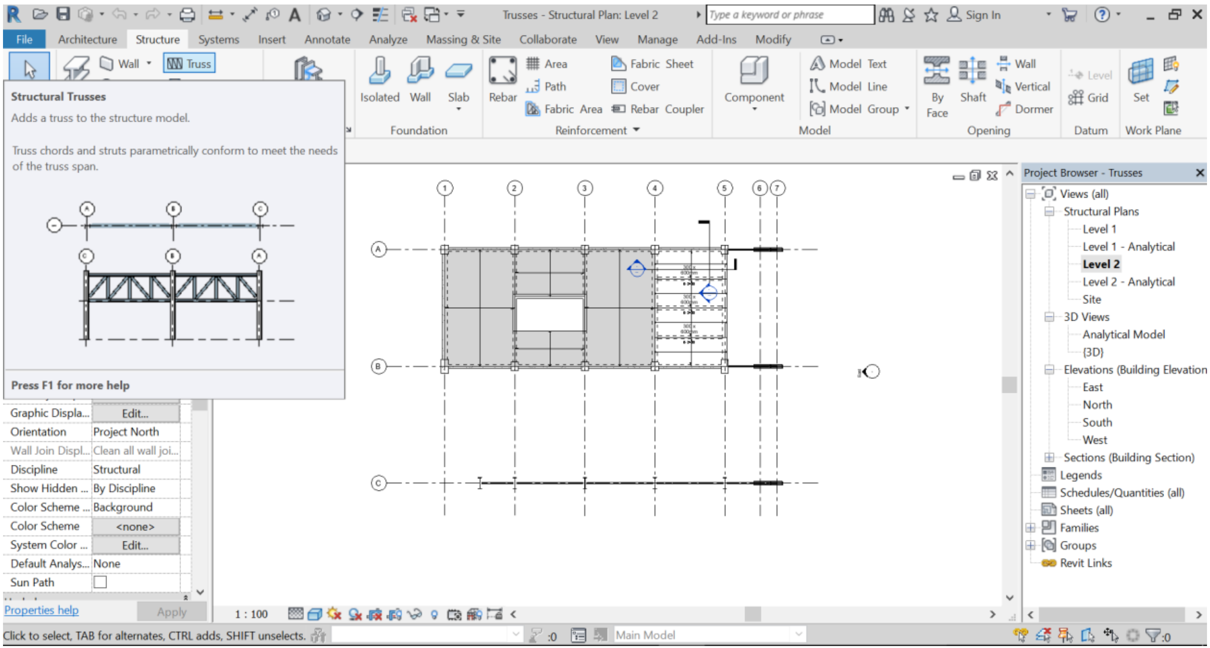
- How to add a Howe Flat Truss in Revit.
- How to edit Top/Bottom chords of a truss.
- Coping diagonal members of a truss.

7.1 Adding a Howe Flat Truss with default settings

To add a Howe Flat truss in Revit,

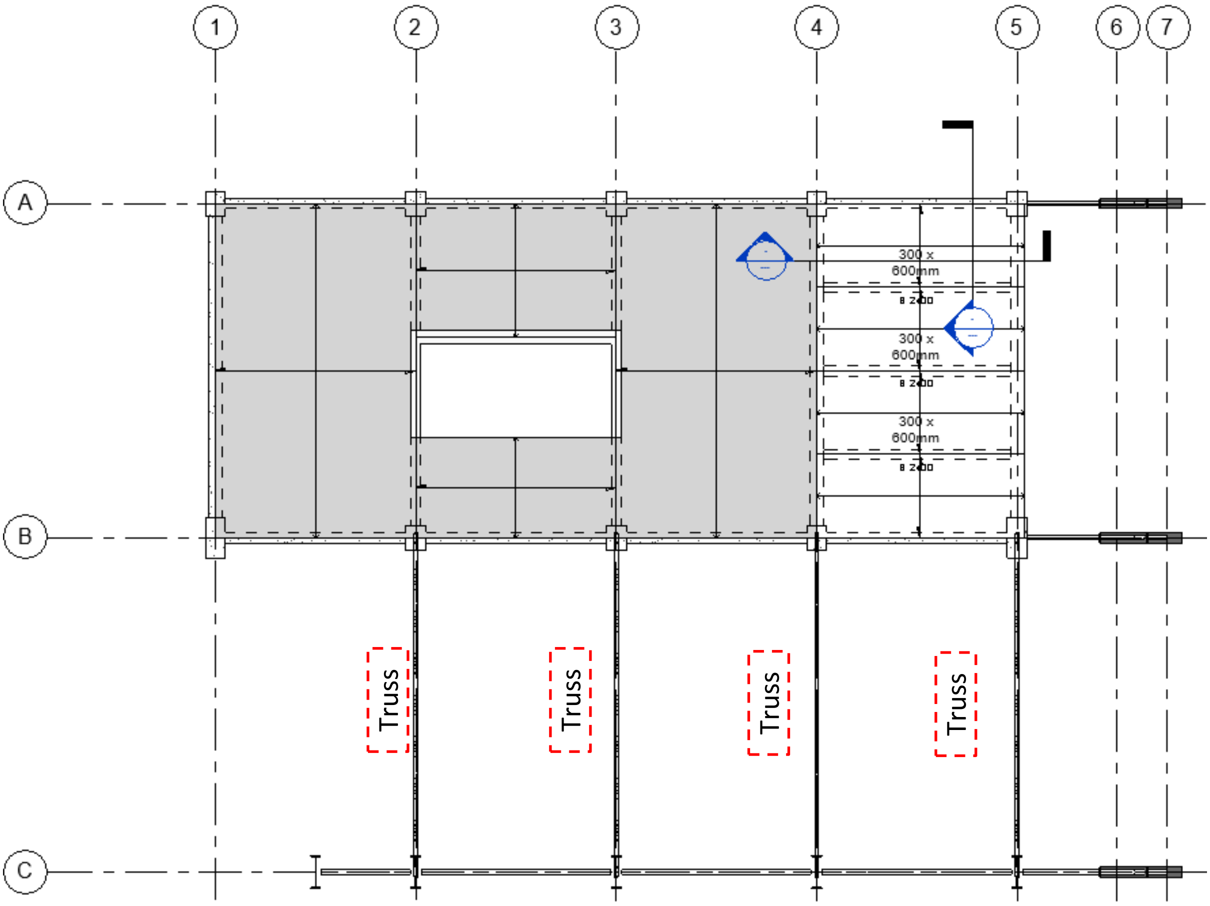
1 Open from the course folder the “Trusses.rvt” file and go to Level 2 (Structural Plan).

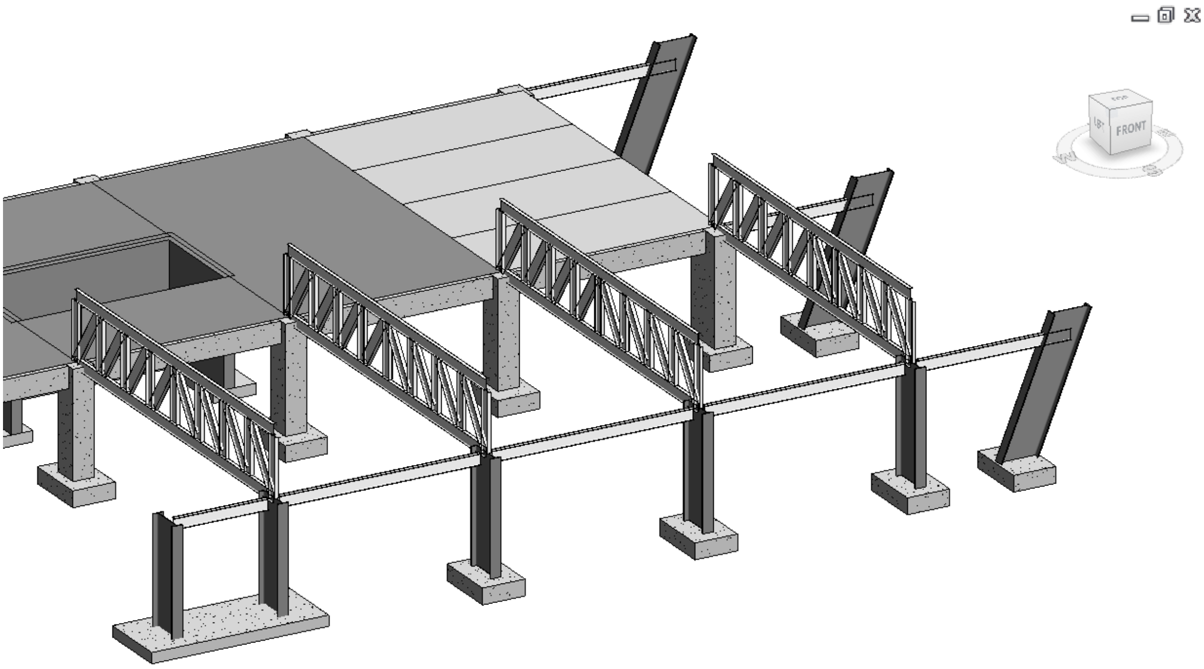
2 Click Structural Tab ➢ Truss in Structure panel.



3 In the Properties Palette, select “Standard” type from Howe Flat Truss types.

4 Keep all options unchanged. Place the first truss by picking the column at Grid-B-2 and then the column at Grid-C-2. Place other Howe Flat trusses at Level 2 as shown below.


1
2
3
4
5
6
7
A
B
C
300 x 600mm
300 x 600mm
300 x 600mm
Truss
Truss
Truss
Truss

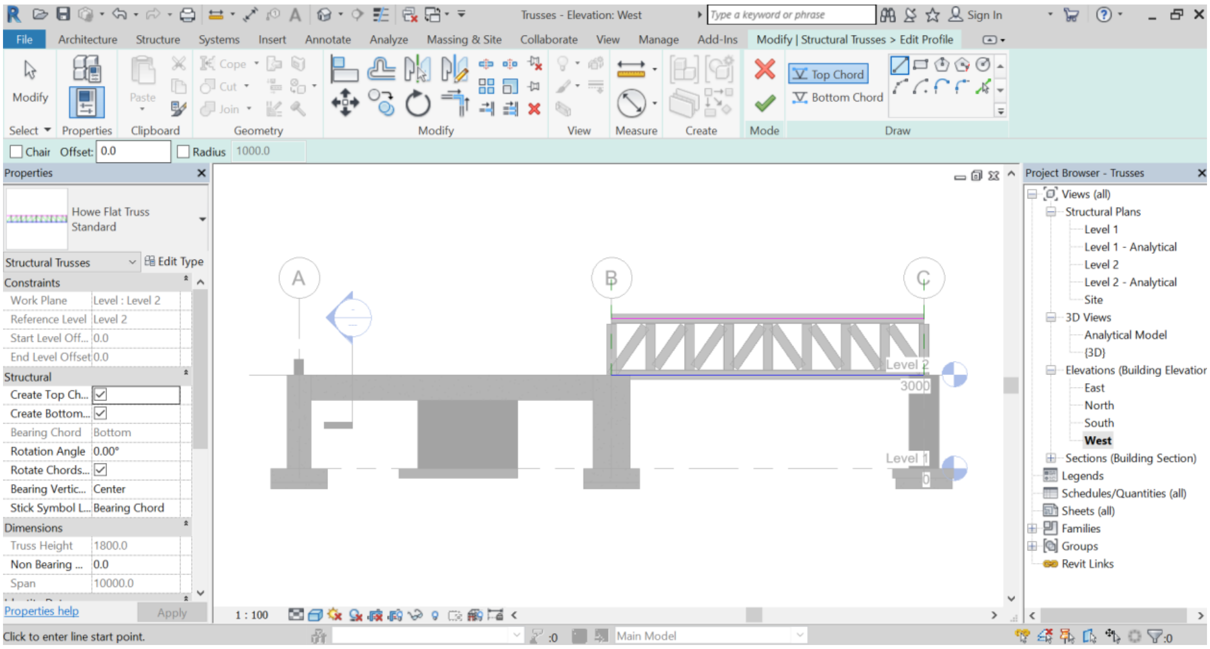

FRONT

7.2 Edit Top/Bottom Chord of a Truss

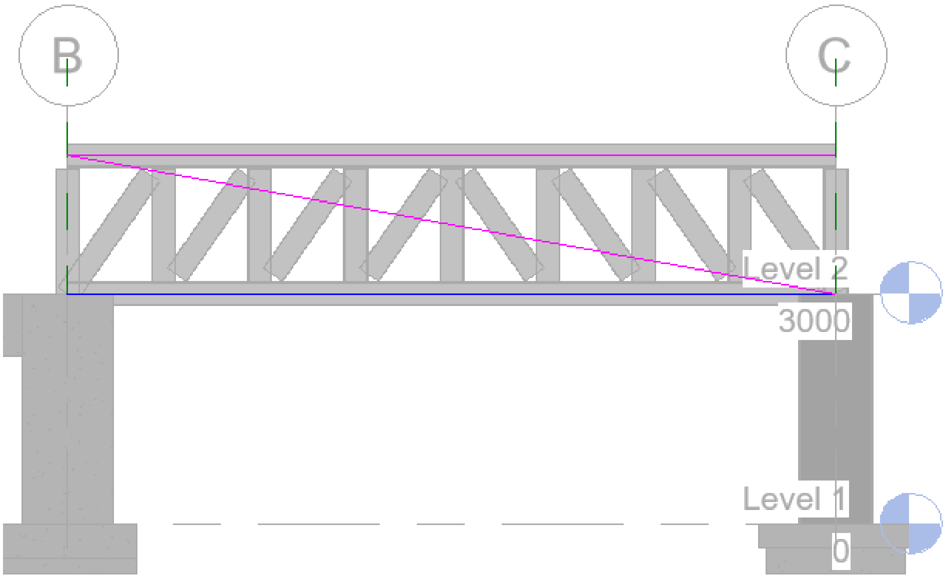
To edit the top/bottom chord of a truss in Revit,

1 Go to West Elevation and select the truss.

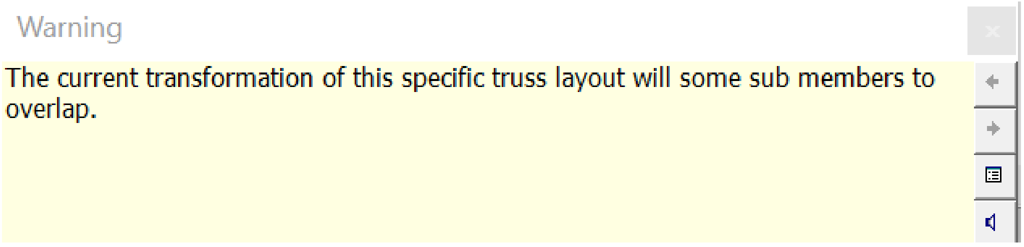
2 Click Edit Profile  and select Top Chord.

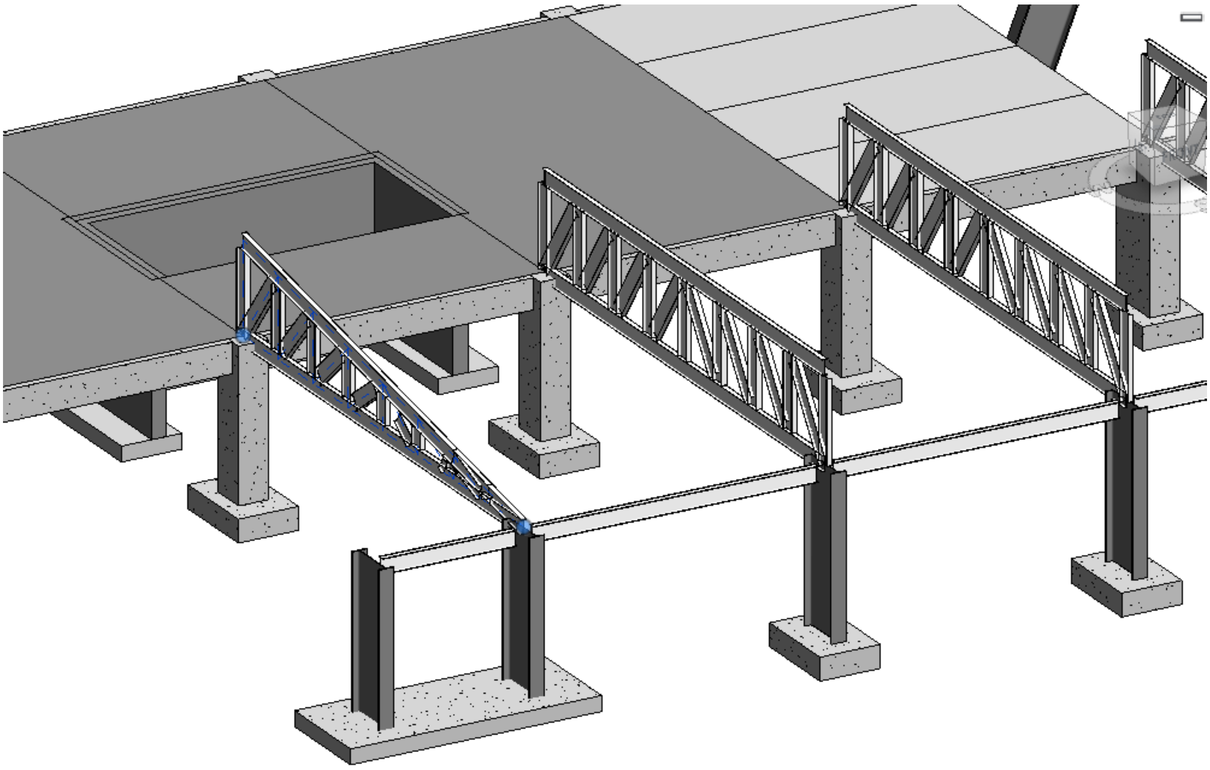


3 Draw a new Top Chord from top of truss at Grid-B to bottom of truss at Grid-C. Press “esc” twice.

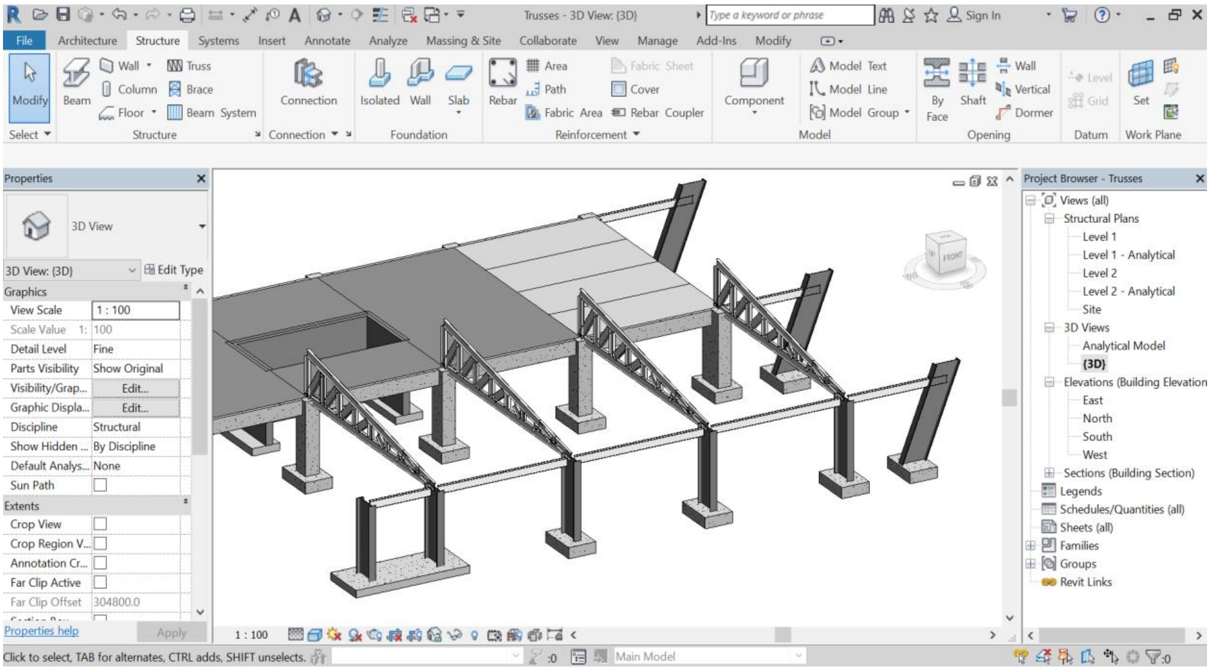


4 Select the original top chord and delete it. Then click Finish Edit Mode  to complete the change. There will be warning message and you can look into detail later.





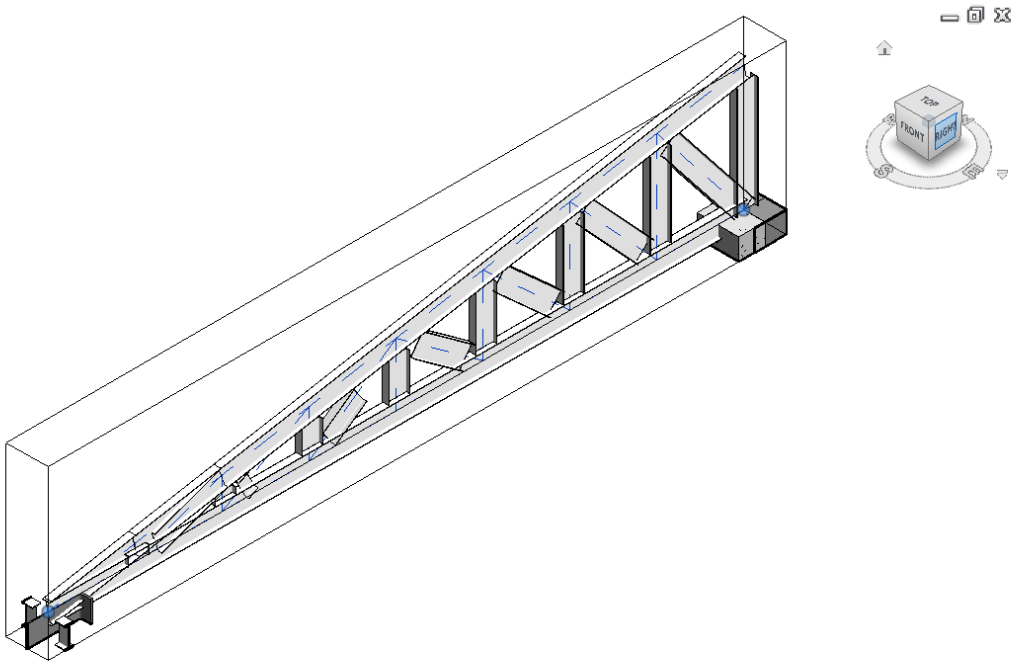
5 You can edit the other trusses similar to step 4 above but in 3D view.



7.3 Coping Diagonal Members of a Truss

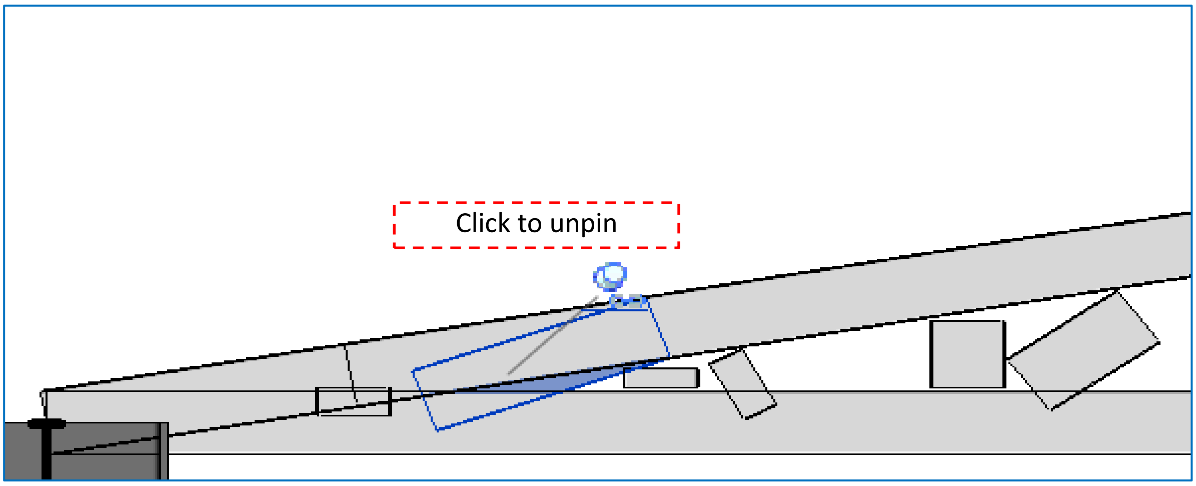
1 Go to 3D view and select the truss along Grid-2.

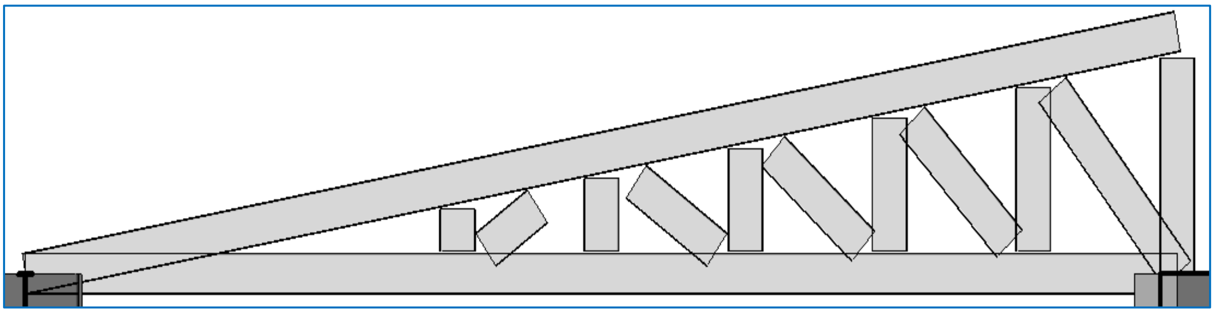
2 Click Selection Box  in the View panel.



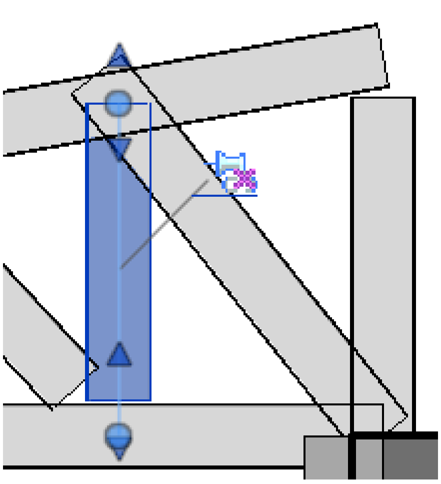
3 Click "Right" on the view cube to view the truss in elevation.

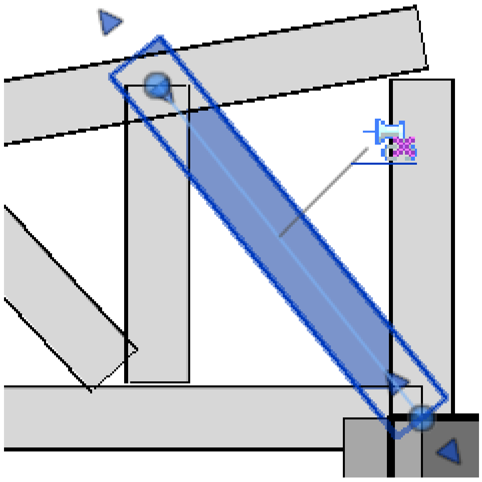
4 Remove overlapping / redundant member by selecting it, unpin  and delete  it. (Use the "tab" key to help your selection.)





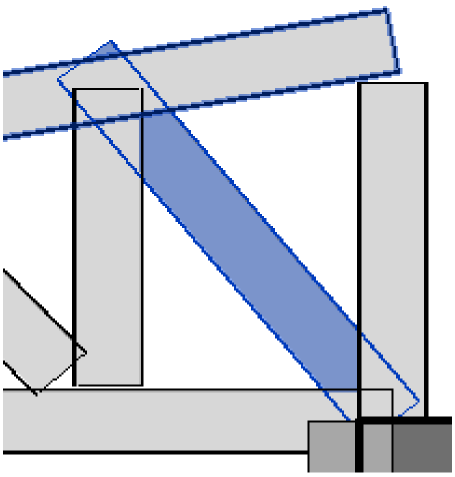
5 To cope the members, first unpin them and extend them into the chords.

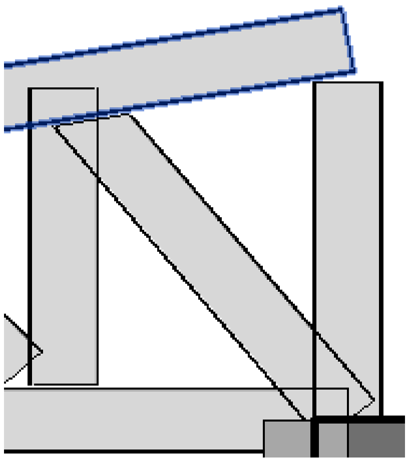




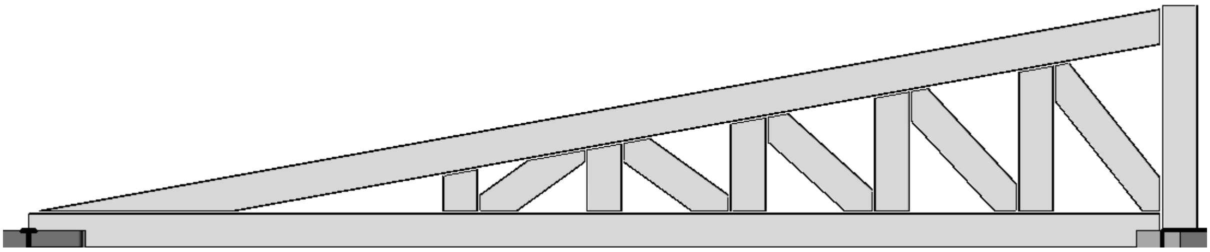
6 From the Modify tab ➢ Cope in Geometry panel. 

7 First select the element to be coped. Then select the element (top chord in this example) with which to cut.

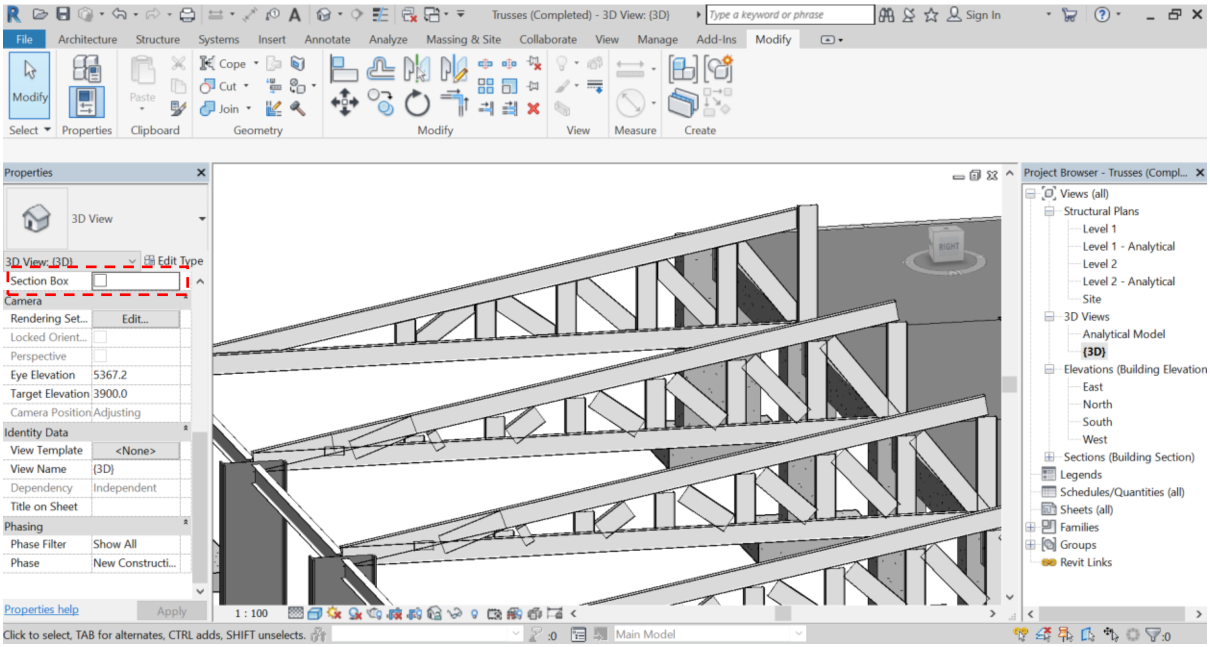




8 You can try to tidy up the truss using the cope tool. Remember to unpin and extend the members.



To view full 3D view, click on any empty space in the 3D view and uncheck the Section Box in the property palette.

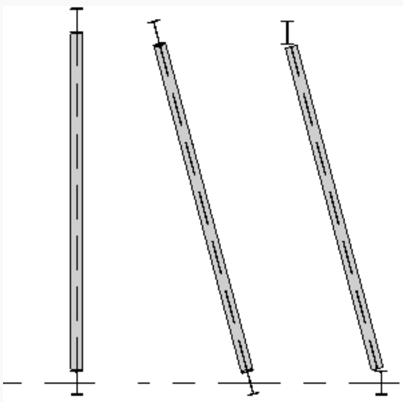


7.4 Detail Description of Truss Instance Properties

(source: Autodesk Knowledge Network)

https://knowledge.autodesk.com/support/revit-products/learn-explore/caas/CloudHelp/cloudhelp/2018/ENU/Revit-Model/files/GUID-3BB898A5-CD06-4091-AB37-908686B1A807-htm.html

Name	Description
Constraints	
Reference Level	The level from which the Start and End Level Offsets are measured. This value is dependent on the work plane of the truss. If the truss is detached from its work plane, you can set this parameter.

Name	Description
Start Level Offset	Specifies the vertical offset from the Reference Level for the start point of the location line.
End Level Offset	Specifies the vertical offset from the Reference Level for the endpoint of the location line.
Structural	
Create Top Chord	Creates the Top Chord. Clear the check box when you do not want the top chord to be created. This is useful for preventing overlaps when creating 3D space trusses.
Create Bottom Chord	Creates the Bottom Chord. Clear this check box when you do not want the bottom chord to be created. This is useful for preventing overlaps when creating 3D space trusses.
Bearing Chord	Specifies the chord bearing, determining the position of the truss with respect to the location line.
Rotation Angle	Sets truss axial rotation.
Rotate Chords With Truss	Aligns the chords with the truss plane on rotation. Deselect to align the chords with the placement plane of the truss. 
Bearing Vertical Justification	Sets the Vertical Justification parameter in bearing chord members.
Stick Symbol Location	Specifies the location of the coarse view plan representation of the truss: either Top Chord, Bottom Chord, or Bearing Chord.
Dimensions	
Truss Height	Specifies the distance between the top and bottom reference planes in the truss layout family.

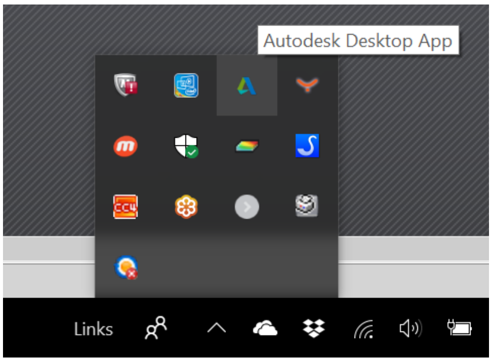
Name	Description
Non Bearing Offset	Specifies the horizontal offset of the non-bearing chord from the location line.
Span	Specifies the furthest extent of the truss along the location line. In many cases, this parameter does not correspond to the value of the Length parameter in the truss family.
Identity Data	
Engineering Type	The text field that can be used to tag trusses. The value is non-unique.
Comments	A field for entering comments about the truss.
Mark	The standard Object parameter. The value is unique per instance.
Tag new members in view	Specifies the view in which to display new beam elements added to the truss.
Phasing	
Phase Created	Indicates in which phase the brace component was created. For more information, see Project Phasing.
Phase Demolished	Indicates in which phase the brace component was demolished. See Project Phasing.
Other	
Number Panels	Displays the number of truss panels, based on the Actual Panel Width. This is a read-only value.
Max Panel Width	Specifies the width of a single truss panel.
Actual Panel Thickness	Displays the width of each truss panel, based on the overall width and number of panels for the selected truss. This is a read-only value.

7.5 Revit Built-in Steel Connections

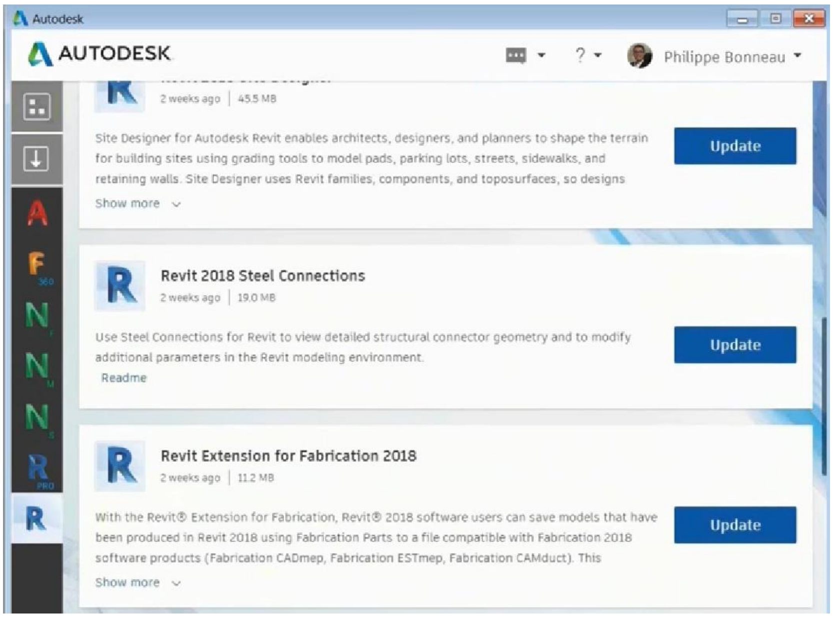
In Revit 2018, there is a library of Steel Connection components to help you model the connection between steel members. You will learn how to install and load them for use in a project. Two of the connections will be used as examples to demonstrate this powerful tool.

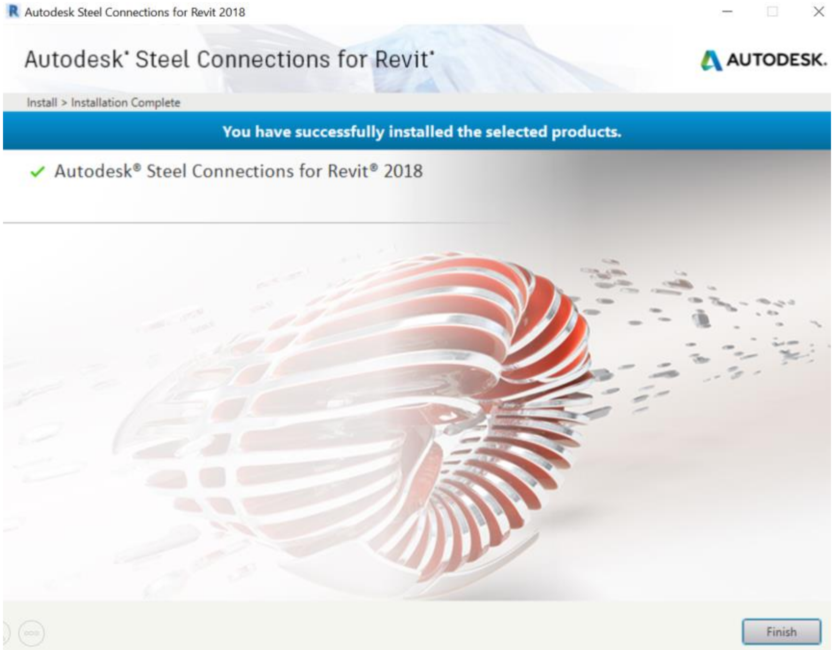
7.5.1 Install and Load Steel Connections

1 Open Autodesk Desktop App from your Window Toolbars.



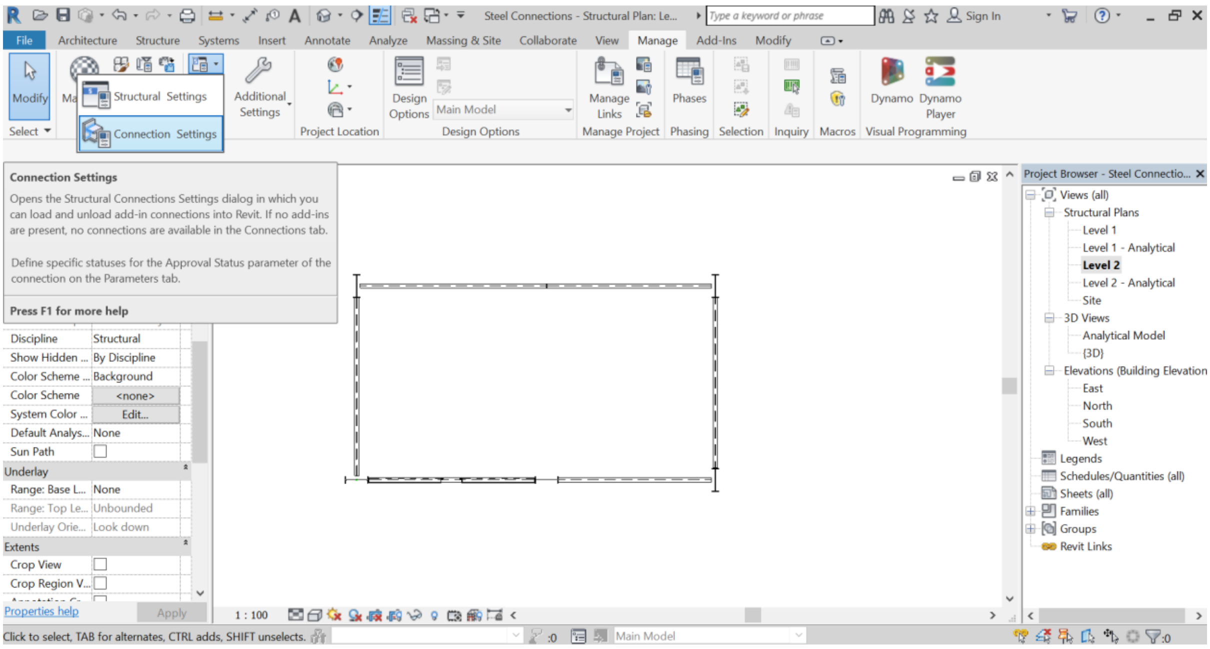
2 Scroll down until you find the Revit 2018 Steel Connections and click "Update" to install.



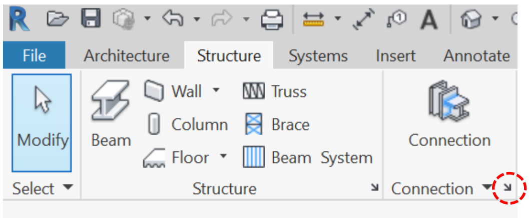


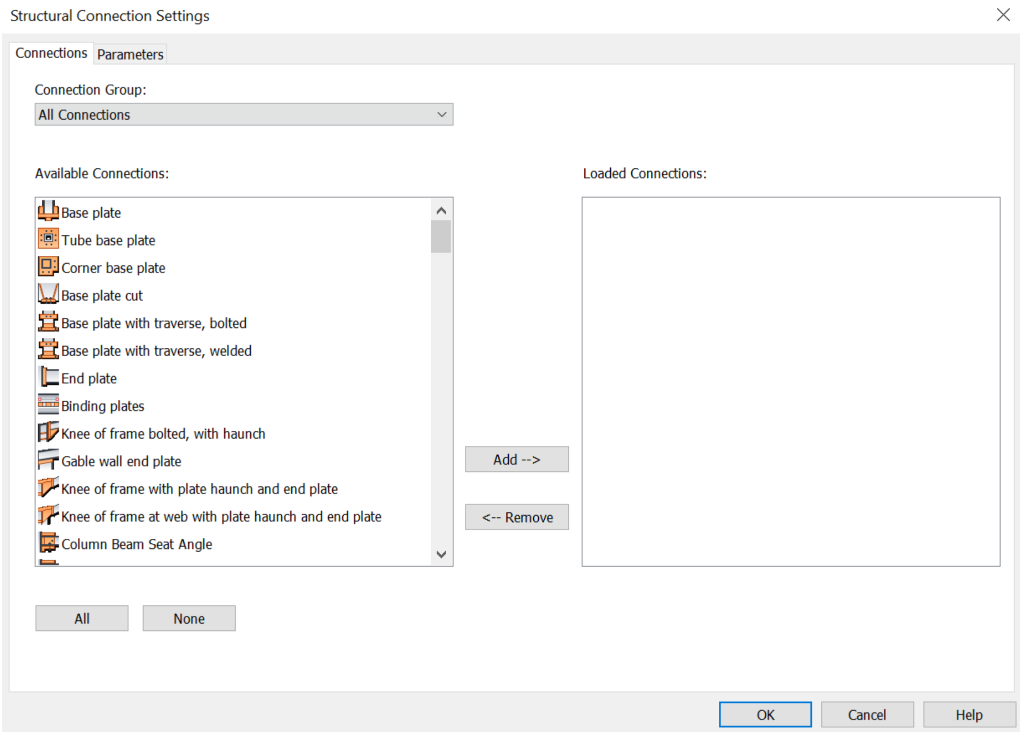
3 After installation, open from the course folder the "Steel Connections.rvt" file.

4 Click Manage Tab ➢ Settings panel ➢ Connection Settings to load steel connection components.



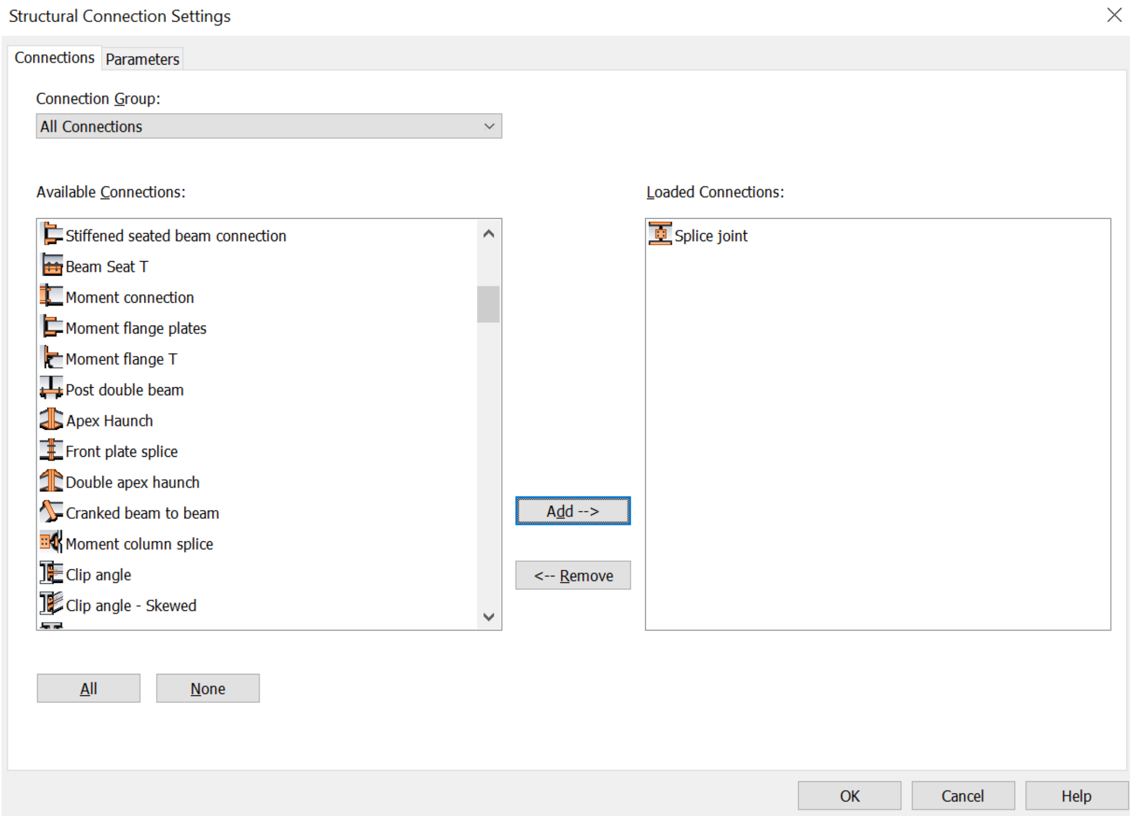
(*Alternately, you can open Connection Settings by clicking on the little arrow at the bottom right of the Connection panel in Structure Tab.)





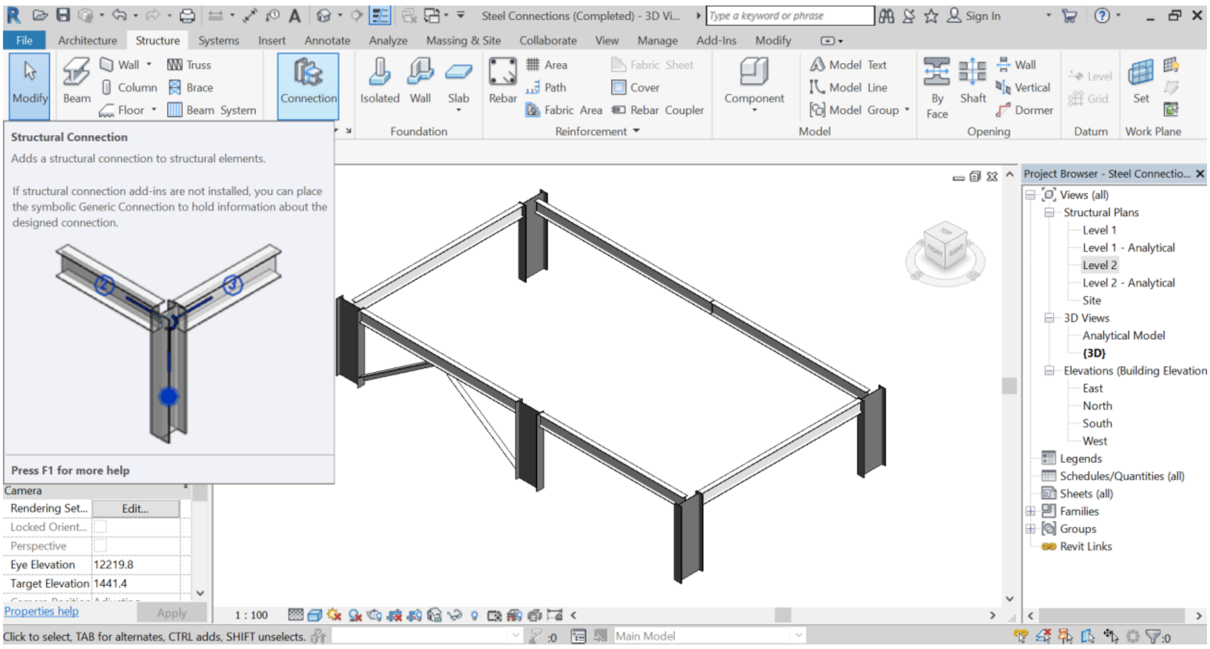
5 You can see there are many steel connection components such as “Base Plate” and “Column Beam Seat Angle”.

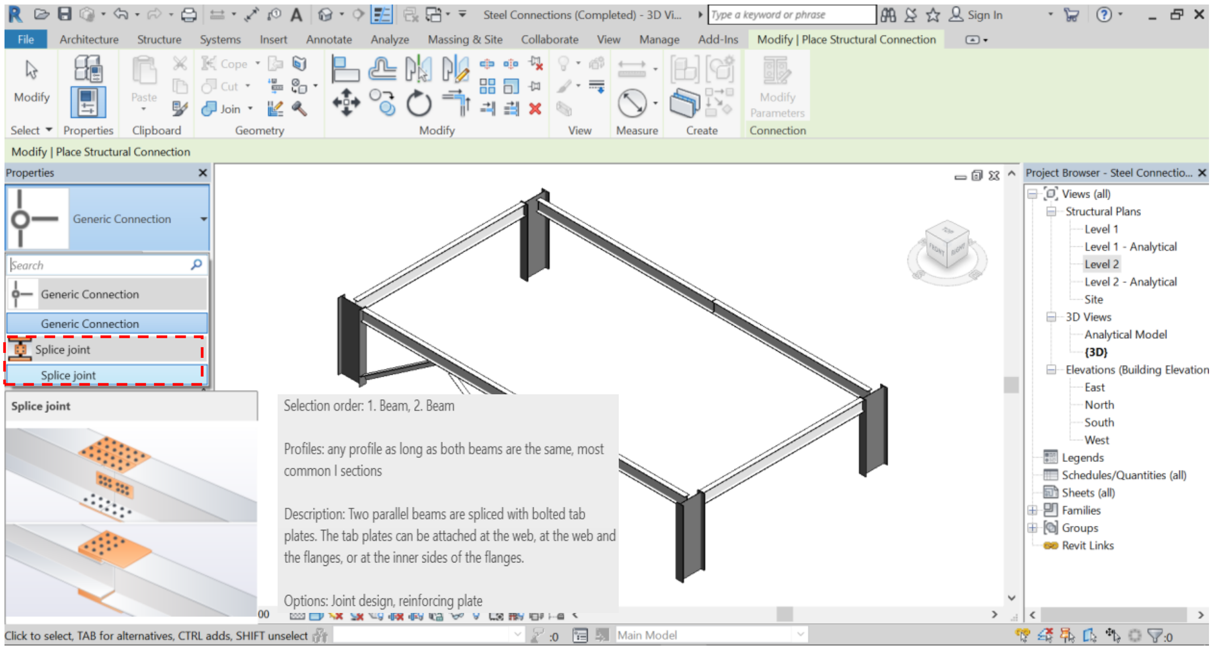
Scroll down the list of Available Connections and find “Splice joint”. Select it and click “Add”. Then click “OK” and the connection component “Splice joint” is loaded into the project.



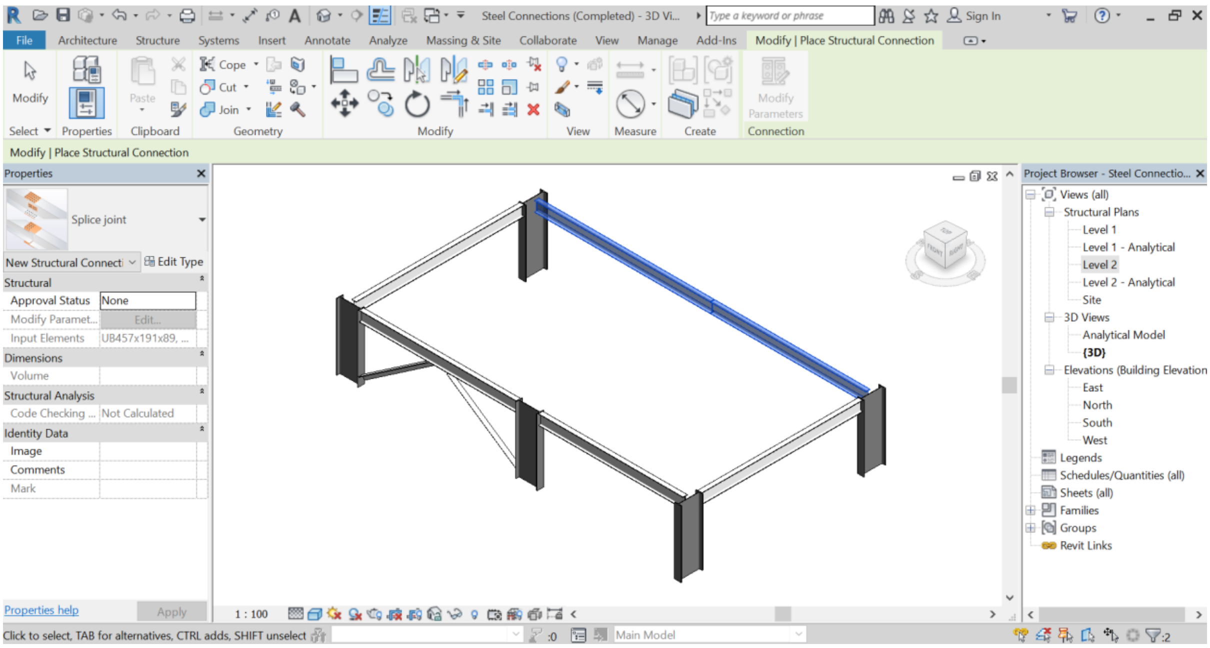
7.5.2 Create a Splice Joint (joining two steel beams)

1 Go to 3D view and click Structure Tab ➢ Connection panel ➢ Connection and select “Spice joint” from the Type Selector in Properties Palette.

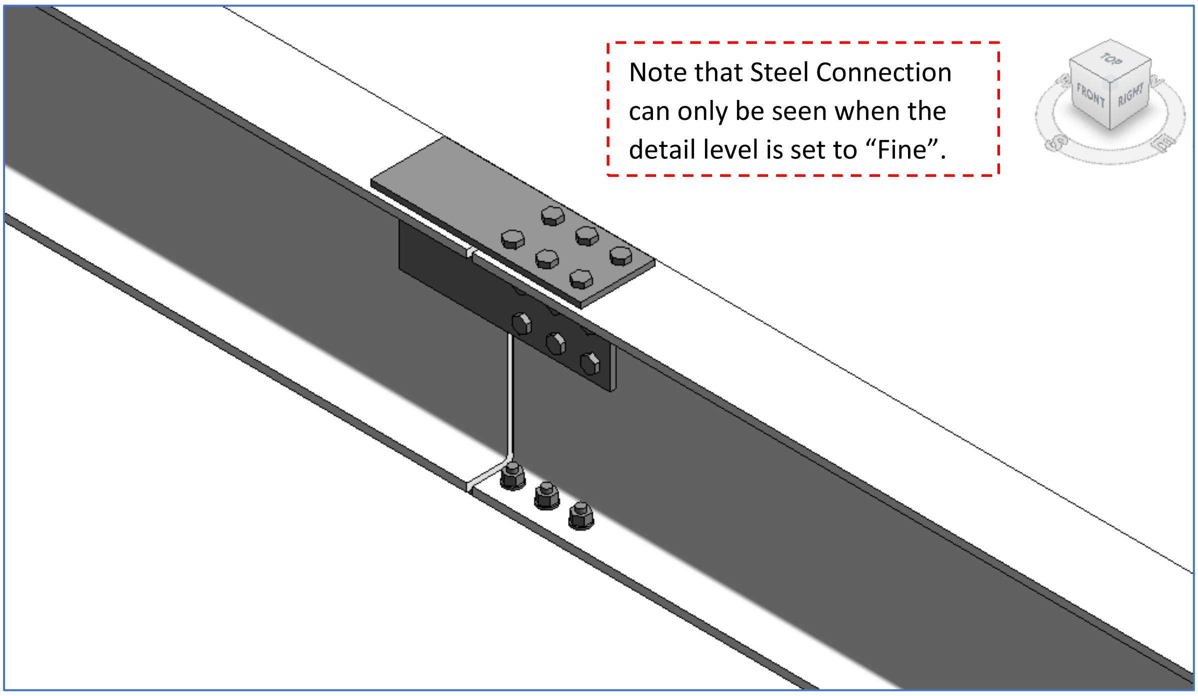




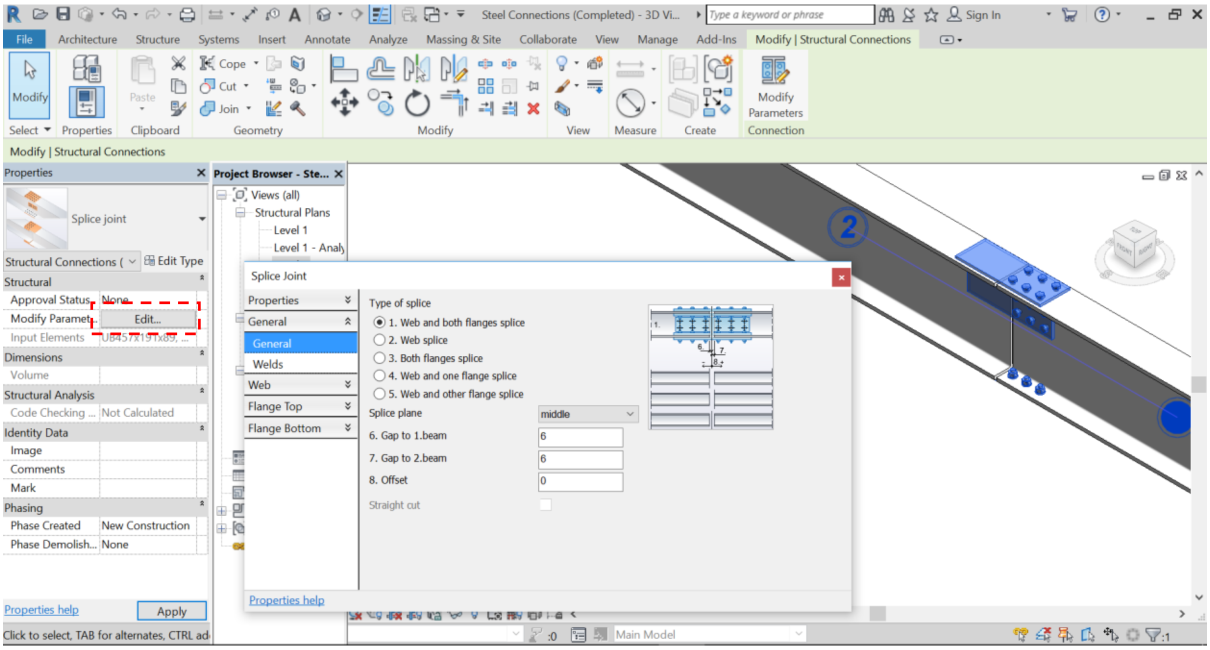
2 Select the two steel beams as shown below. (Hold down "ctrl" key while selecting the beams.) Then press "spacebar" and a splice joint is added to join the selected beams.



3 Zoom in to see the splice joint just added.



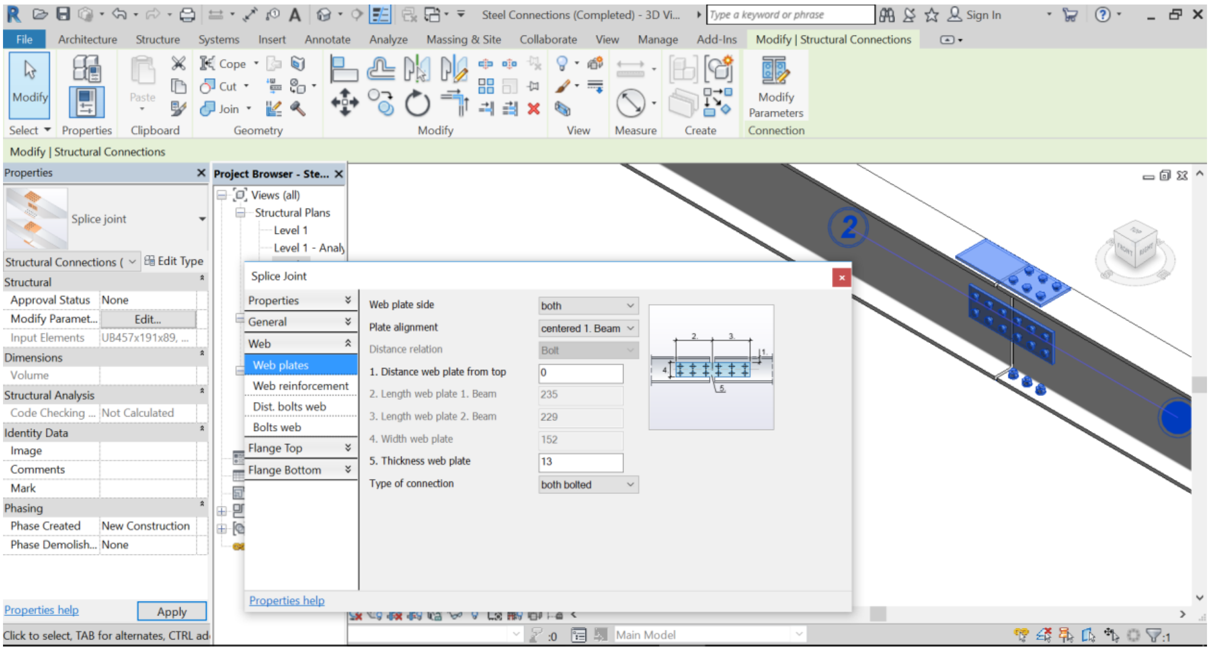
4 Select the splice joint and click "Edit" to modify the detail of this joint. Notice the number "2" on the left beam. This indicates that it is the "2.beam" in this joint configuration.



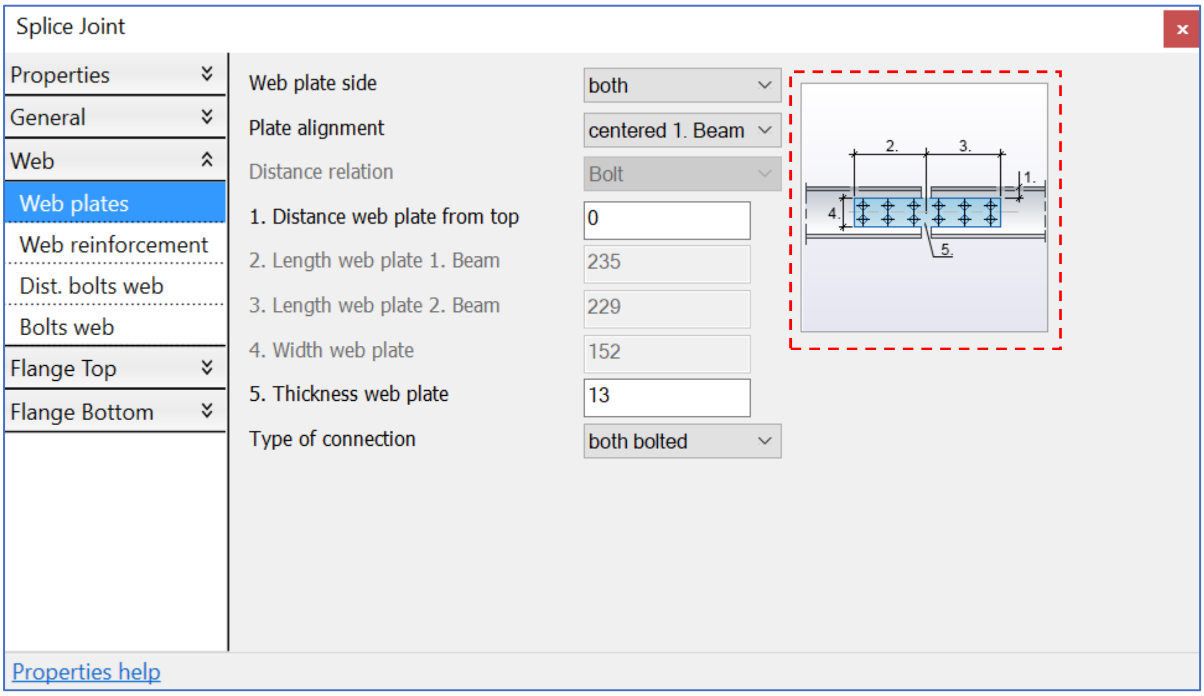
5 From the "Web" menu, select "Web plates" and set

- Plate alignment: "centered 1.Beam"
- Distance web plate from top: "0"
- Type of connection: "both bolted"

The splice joint will be updated automatically as you enter the changes.

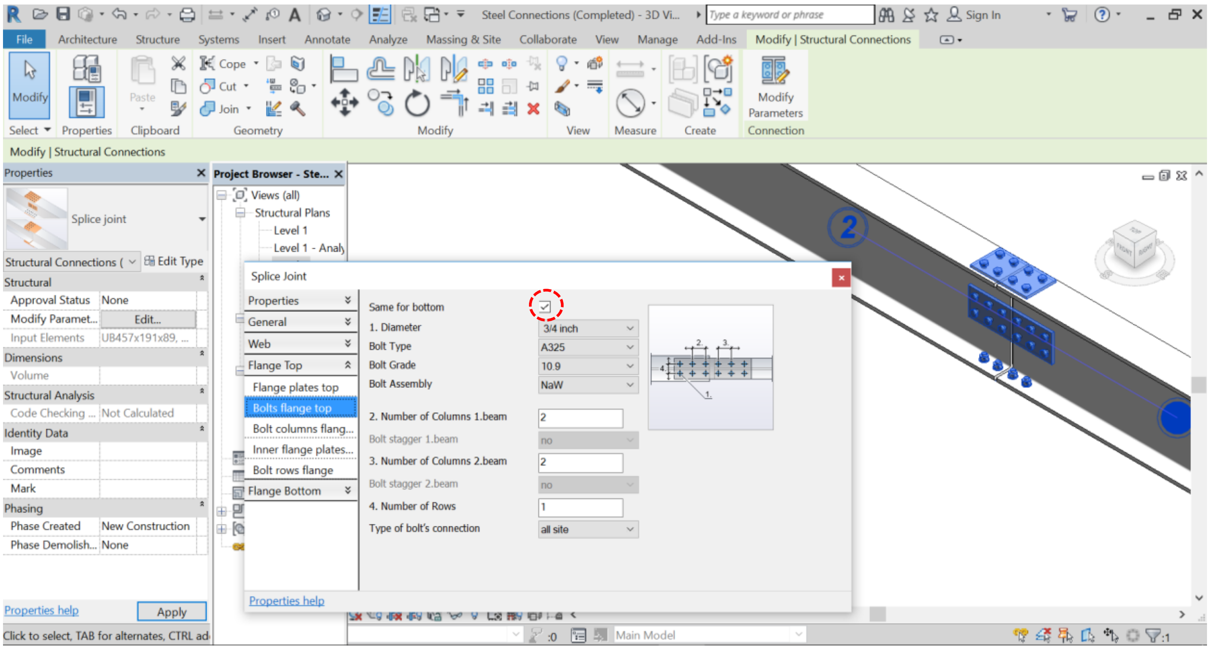


Note the little diagram with annotation to help you understand what parameter you are changing.

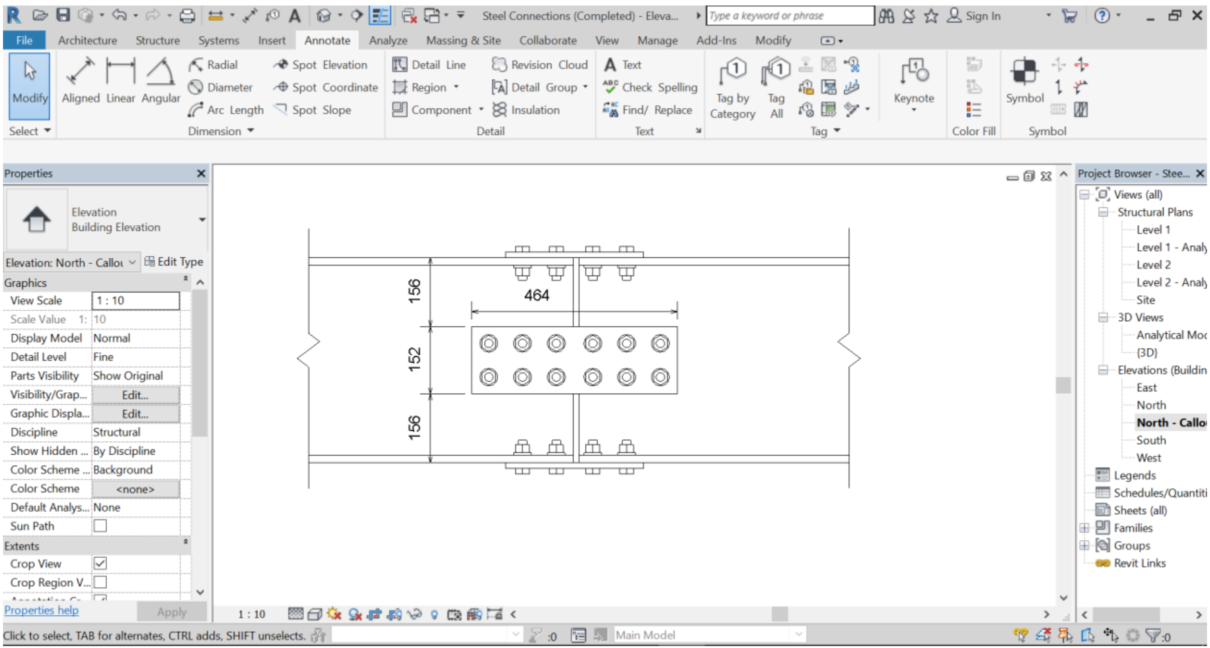


6 Next, from the “Flange Top” menu, select “Flange plates top” and change “Type of connection” to “both bolted”. Then select “Bolts flange top” and change both “Number of Columns 1.beam” and “Number of Columns 2.beam” to “2”.

Note the “Same for bottom” option. Uncheck it if you want the joint setting for top and bottom flanges to be independent of each other.

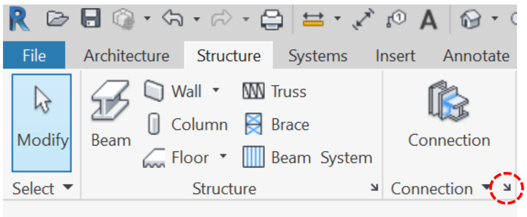


7 You can create a callout of the joint from the north elevation of the model. Then annotate the callout for drawing production.

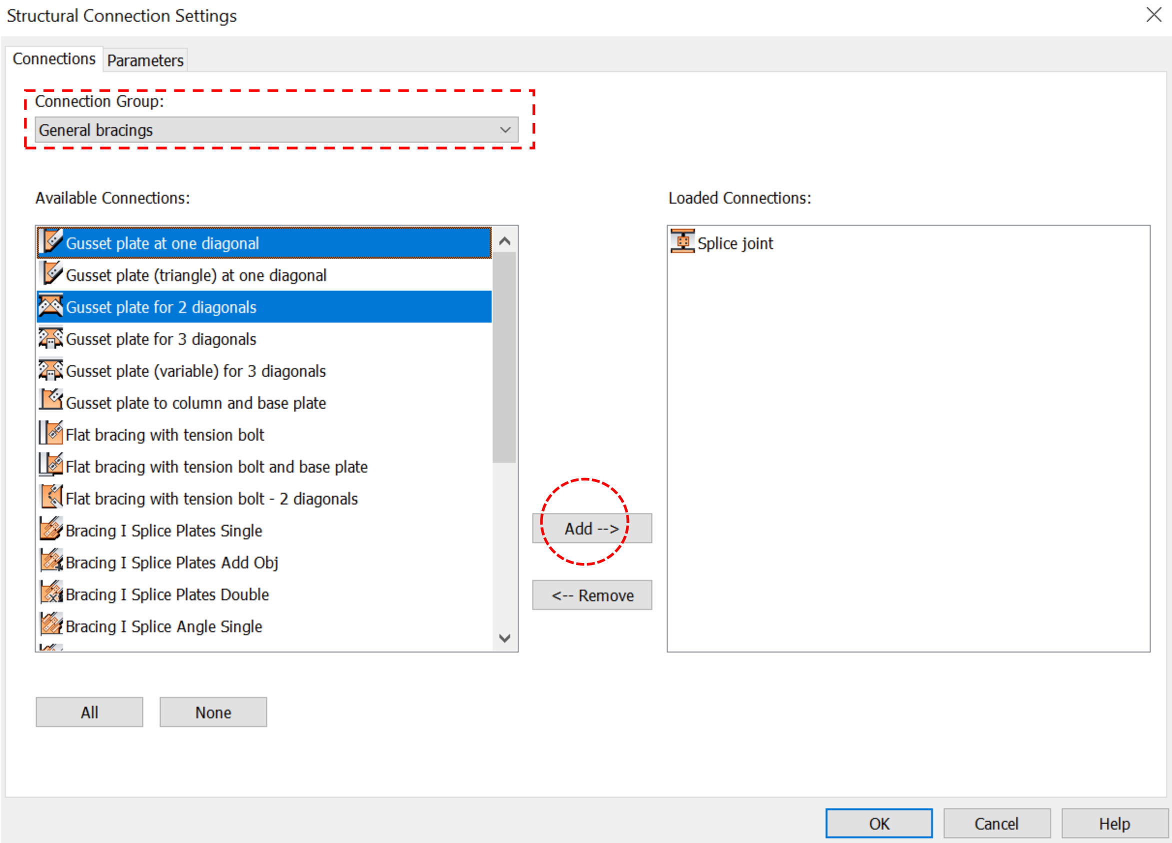


7.5.3 Create Gusset Plate joints for Bracings

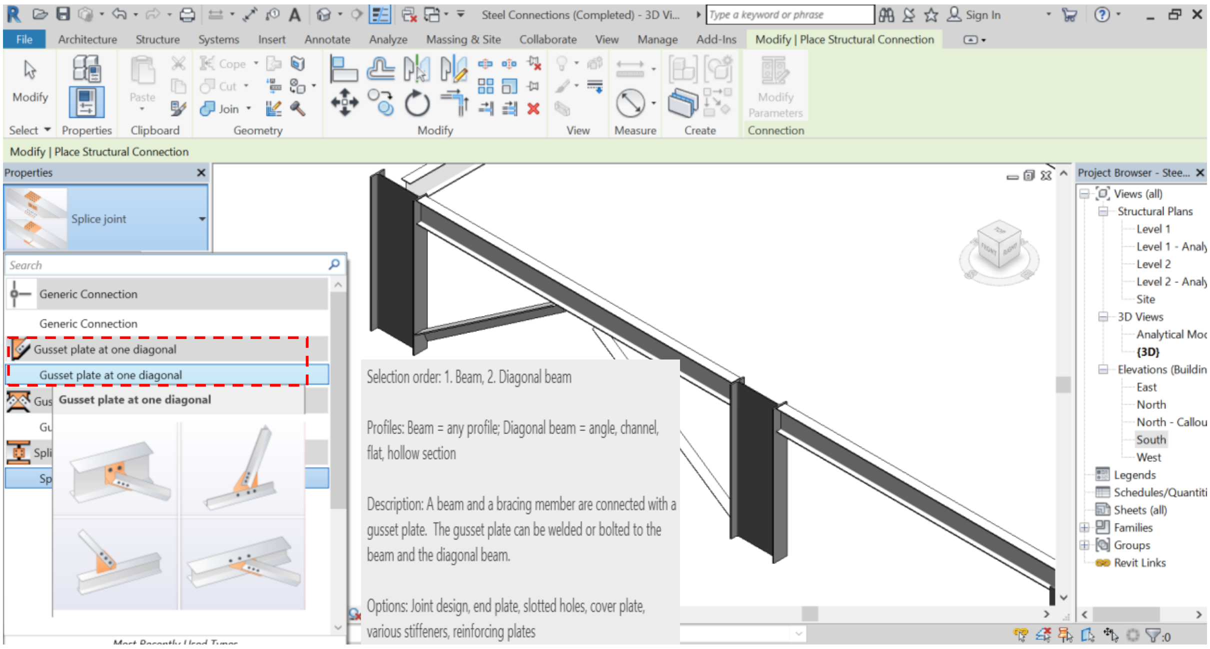
1 Go to 3D view and turn on the Connection Settings by clicking the little arrow in the Connection panel (Structure Tab).



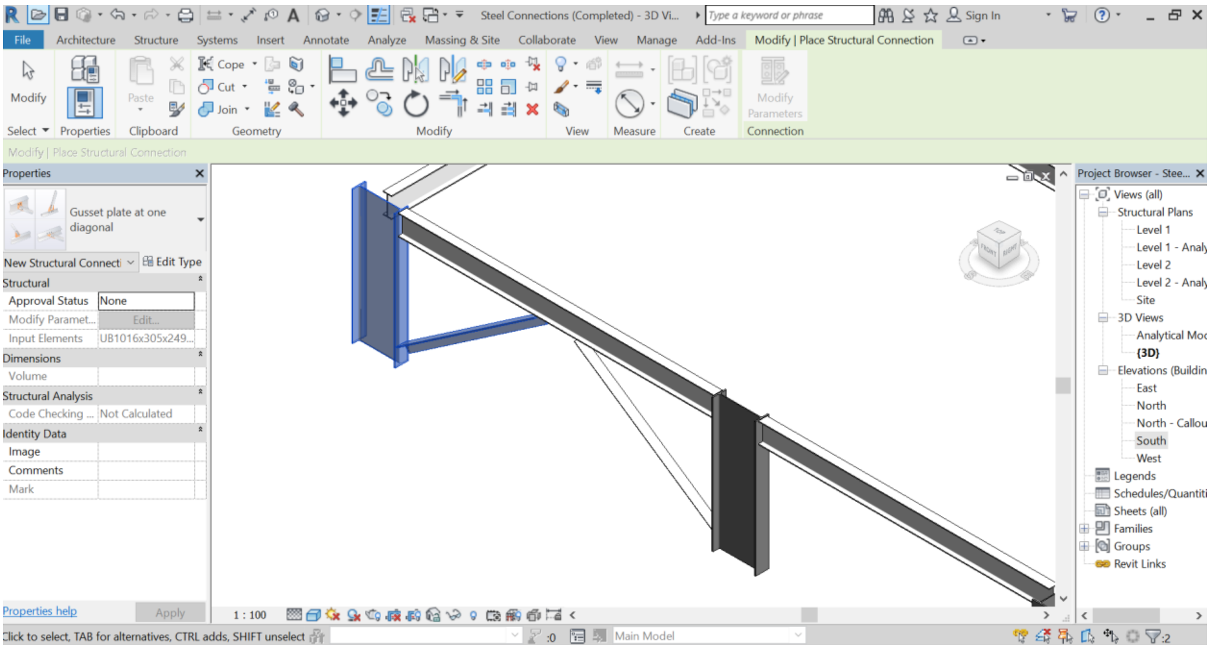
Choose “General bracings” in “Connection Group”. Then select “Gusset plate at one diagonal” and “Gusset plate for 2 diagonals” and click “Add”. Click “OK” to load these connections into the project.

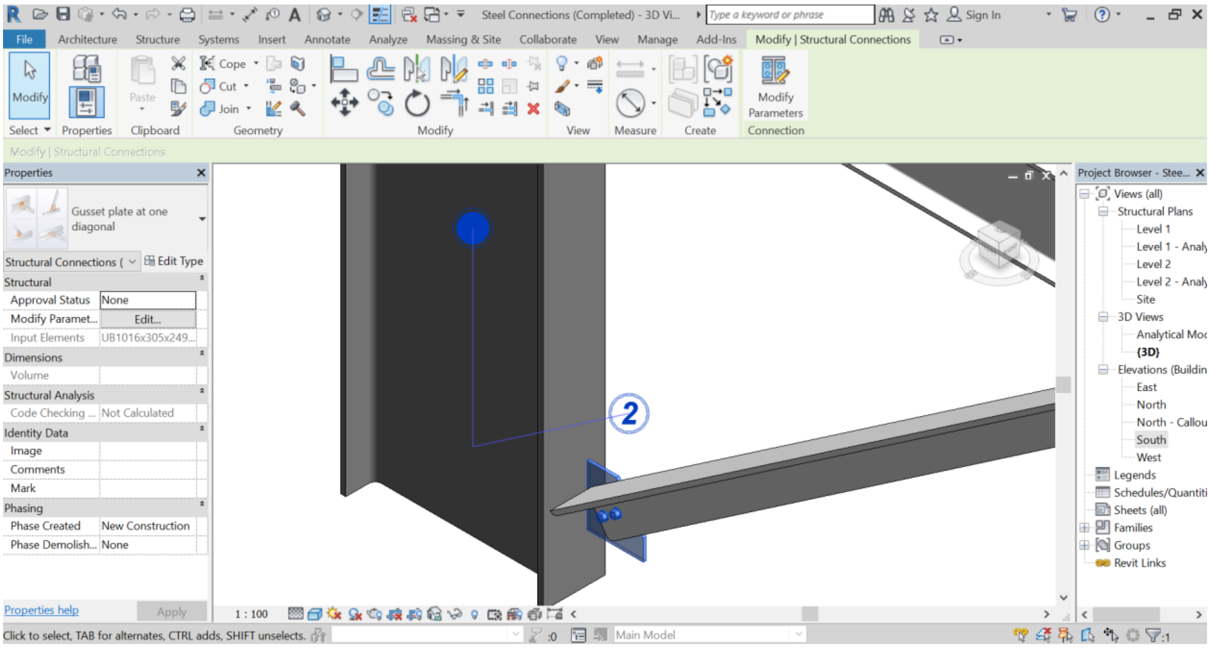


2 Click Structure Tab ➢ Connection panel ➢ Connection and select “Gusset plate at one diagonal” from the Type Selector in Properties Palette.

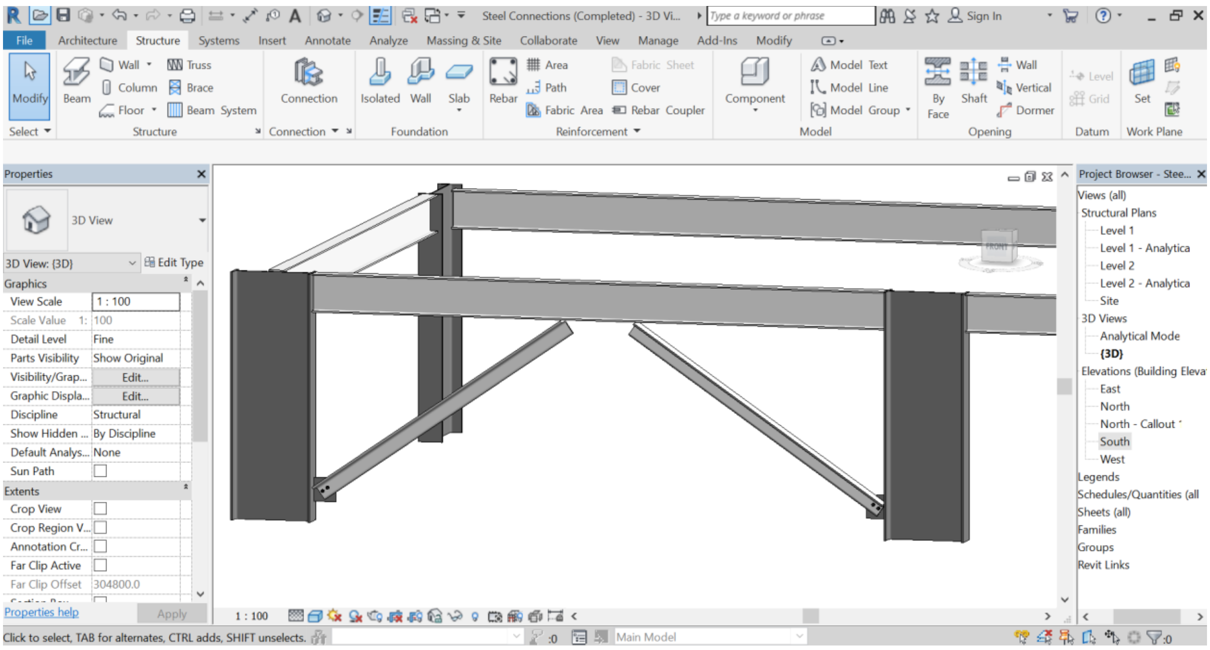


3 Select the left brace and the column as shown below. (Hold down “ctrl” key while selecting.) Then press “spacebar” and a gusset plate joint is added to join the selected brace to the column.



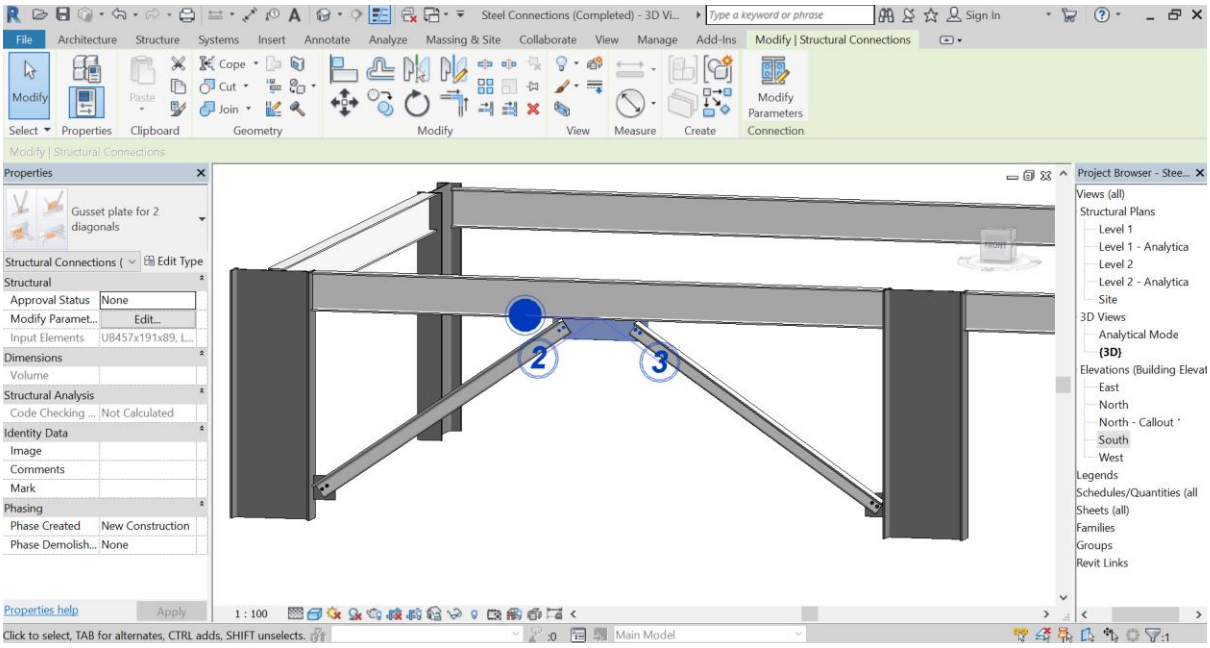


4 Create another gusset plate joint for the right brace.

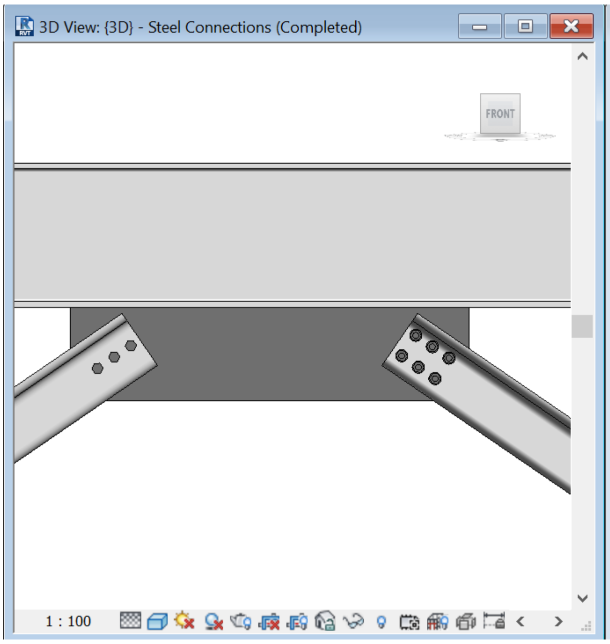


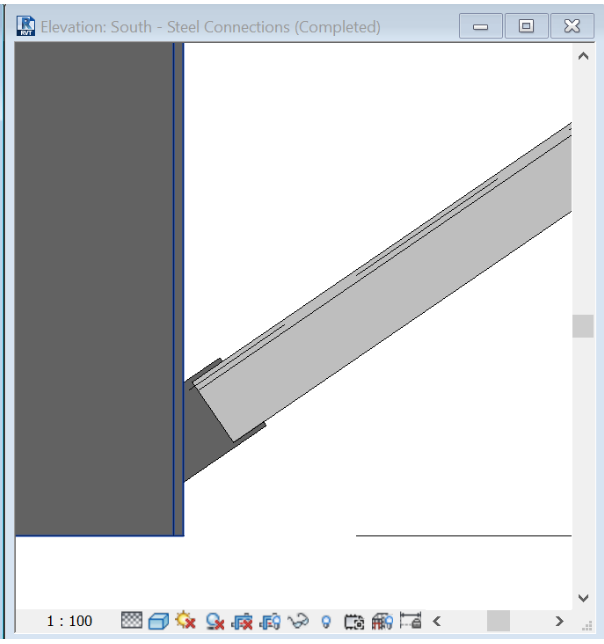
5 Now, click Structure Tab ➢ Connection panel ➢ Connection and select "Gusset plate for 2 diagonals" from the Type Selector in Properties Palette.

Select the two braces and the horizontal beam (Hold down "ctrl" key while selecting). Then press "spacebar" and a gusset plate joint is added to join the selected braces to the column.



You can select any steel connections created and click "Edit" to modify the details of the joints. Try changing the setting of the joints so that they look like the details shown below.





CHAPTER 8 - STEEL BRACINGS

Bracing is called Brace in Revit and is placed vertically by default. You can Create/Customize brace family for special use (not to be covered in this course). In this section, you will learn

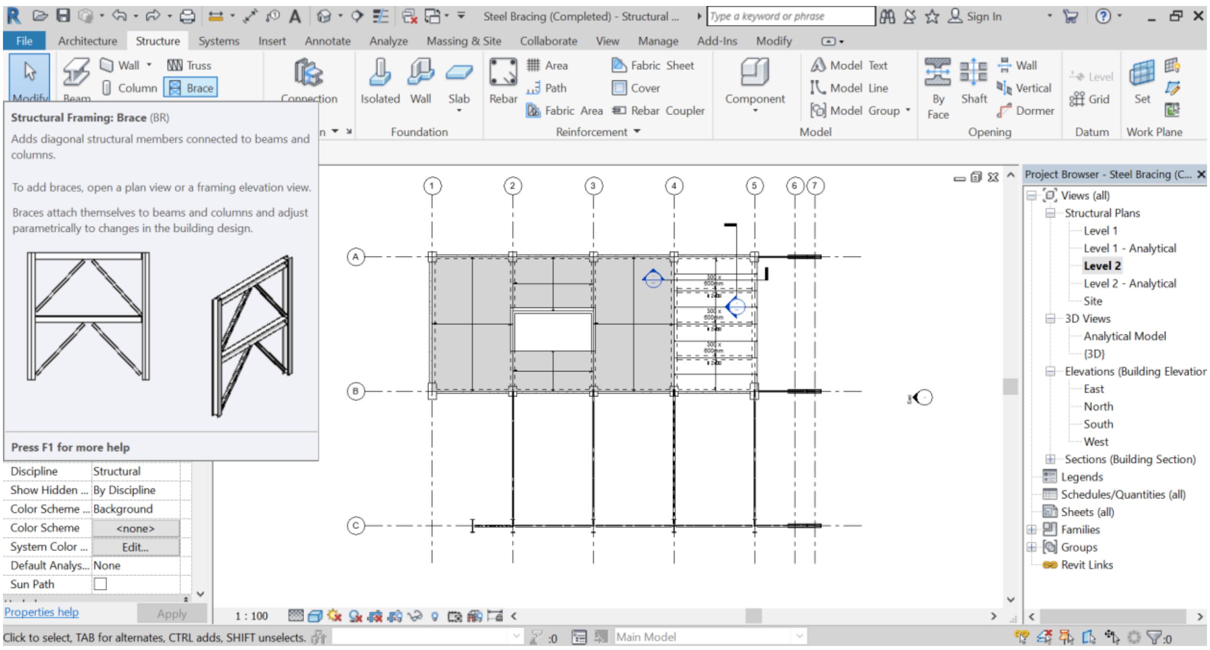
- How to add bracings.
- How to edit bracings and cope them.
- Options in bracing placement.
- Symbolic representations of Bracing.

8.1 Adding Bracings with default settings

To add bracing in Revit,

1 Open from the course folder the “Steel Bracing.rvt” file and go to Level 2 (Structural Plan).

2 Click Structural Tab ➢ Brace in Structure panel.

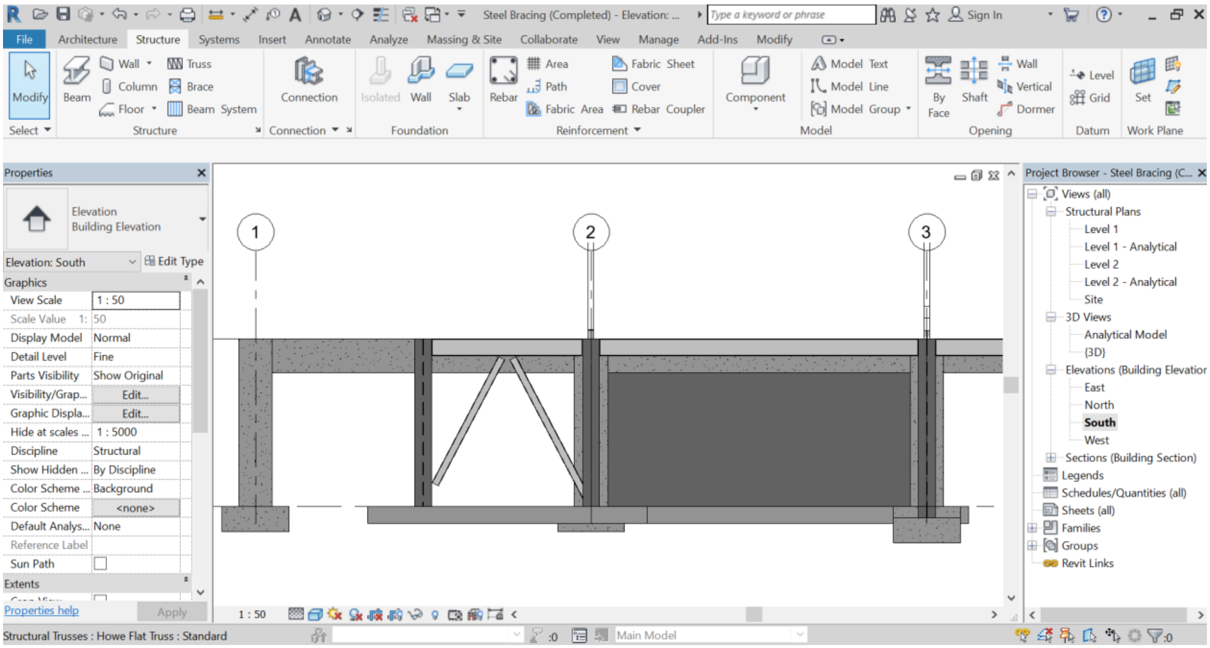


3 In the Properties Palette, select “UB127x76x13” type from UB-Universal Beams types.

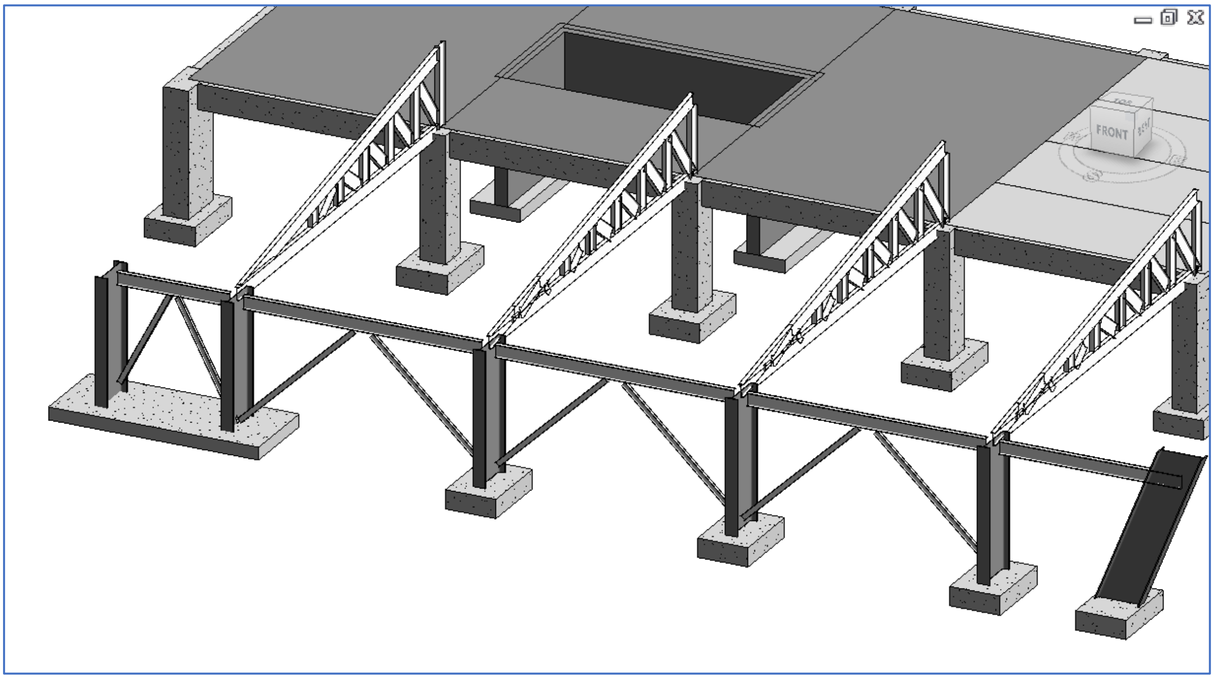
4 Keep all options unchanged. Pick the column between Grid-1 and Grid-2 as the bracing start point. Then pick the mid-point of the beam from that column (you can type “SM” to snap to mid-point). Press “esc” to start adding another bracing.

5 Pick the column at Grid-2 as the bracing start point. Then pick the mid-point of the beam towards Grid-1. Press “esc” a couple of times.

6 Go to South Elevation to view the bracings added.



6 Add similar bracings along Grid-C and from Grid-2 to Grid-5.



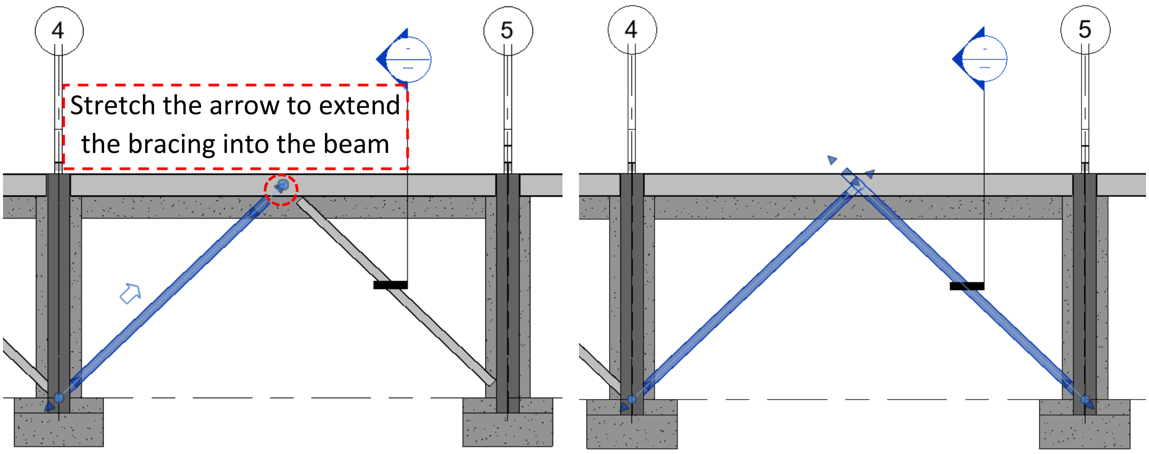
8.2Coping the Bracings

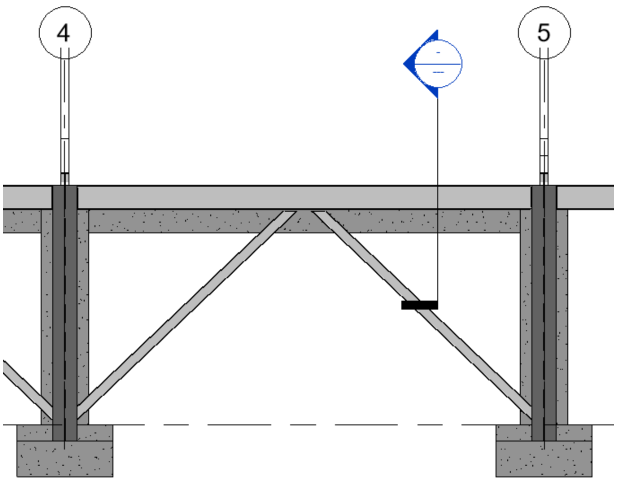
1 Go to South Elevation.

2 Select and extend the bracings between Grid-4 and Grid-5.

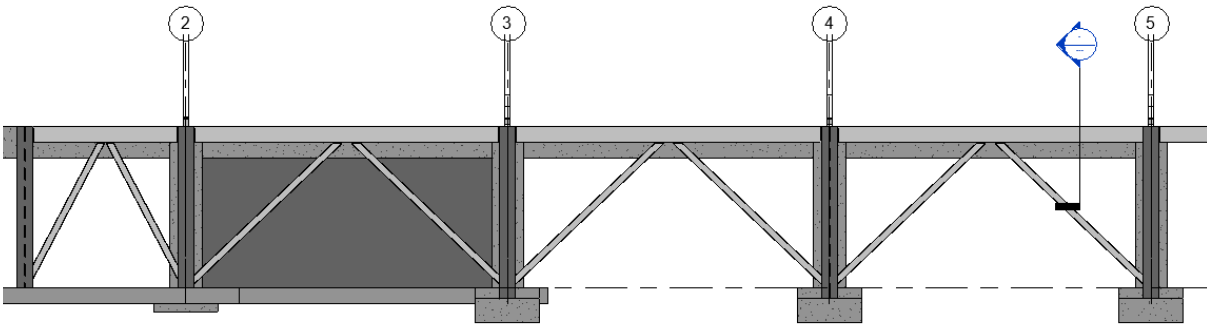
3 From the Modify tab ➢ Cope in Geometry panel. 

4 First select the element to be coped. Then select the element (top chord in this example) with which to cut.





5 Extend and cope all bracings as shown below.



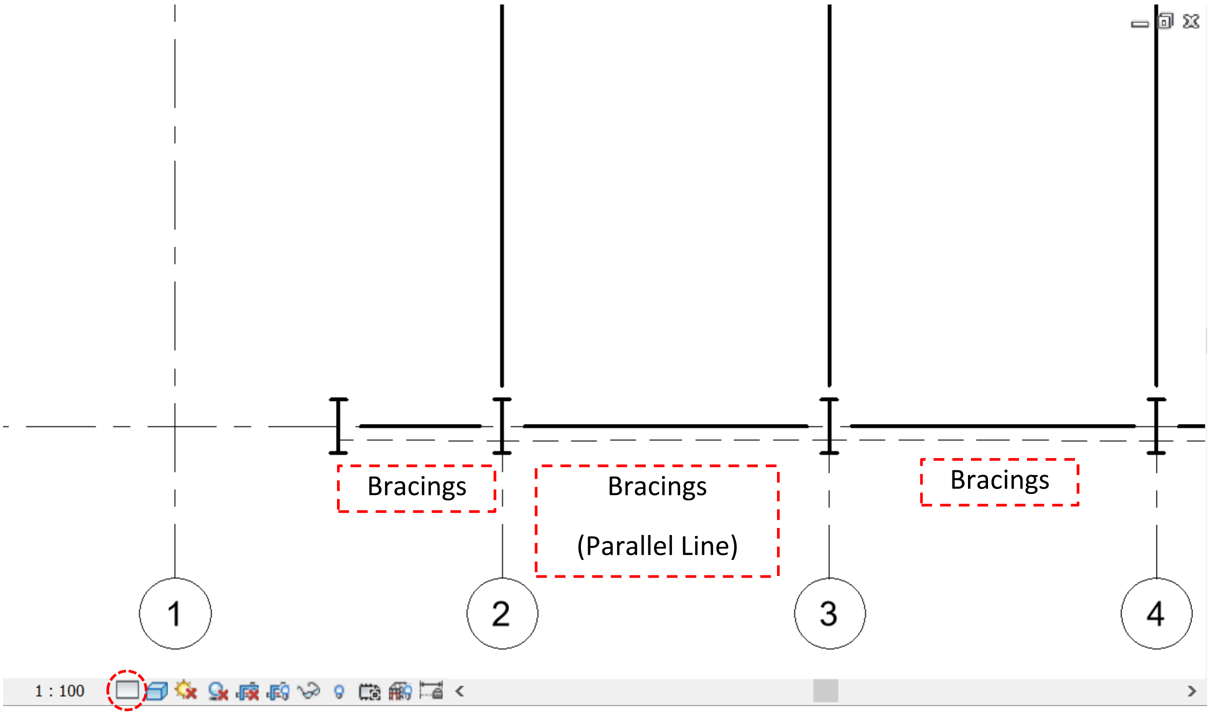
8.3Options in Bracing Placement

- Tag on Placement.

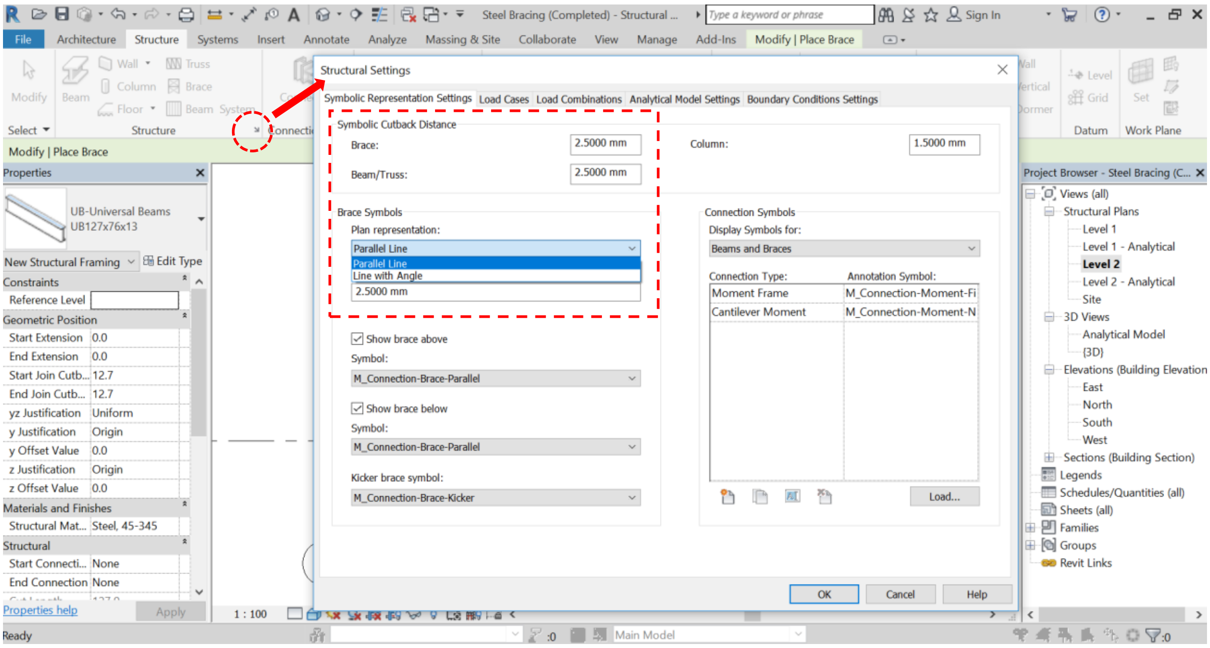
- Start Level (select from available structural levels) and Offset from it.
- End Level (select from available structural levels) and Offset from it.
- 3D Snapping

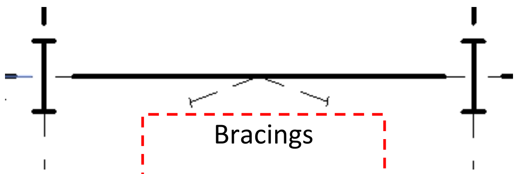
8.4Symbolic Representations of Bracing

To display bracings on plans instead of showing them as hidden lines, the display Detail Level has to be set to Coarse. This is a more practical representation of structural bracing on plans.



To set how bracings are represented on plan, go to Structural Setting in Structure Tag.





8.5 Detail Description of Truss Instance Properties

(source: Autodesk Knowledge Network)

https://knowledge.autodesk.com/support/revit-products/learn-explore/caas/CloudHelp/cloudhelp/2016/ENU/Revit-Model/files/GUID-035A3219-12A1-4978-9496-3B01728A2741-htm.html

Name	Description
Constraints	
Reference Level	The constraining level.
Cross-Section Rotation	Parameter that controls rotating beams and braces. The angle of rotation is measured from the beam's work plane and the direction of the center reference plane.
Construction (Pre 2014 projects)	
Start Extension	The dimension between the edge of the start end of the brace and the element to which it is connected. This parameter is only available in projects created in versions prior to Autodesk Revit 2014 containing brace families that have not been updated. It is recommended to use the Geometric Position Start Extension below. Using both is an additive extension.

Name	Description
End Extension	The dimension between the edge of the finish end of the brace and the element to which it is connected. This parameter is only available in projects created in versions prior to Autodesk Revit 2014 containing brace families that have not been updated. It is recommended to use the Geometric Position Start Extension below. Using both is an additive extension.
Materials and Finishes	
Structural Material	Controls the hidden view display of structural elements. Concrete or Precast will display as hidden. Steel or Wood will be visible when another element is in front of it. Unassigned will not display if hidden by another element. See Change the Physical Properties of a Material.
Structural	
Start Connection	The moment frame or cantilever symbol at the start end of brace, if applicable.
End Connection	The moment frame or cantilever symbol at the finish end of brace, if applicable.
Cut Length	The physical length (not the analytical length). This is a read-only value.
Structural Usage	Specifies Vertical Bracing, Kicker brace, Web, or Other.
Start Attachment Level Reference	The constraining level of the start end of the brace.
Start Attachment Elevation	The elevation of the start end referenced from the Start Attachment Level Reference.
End Attachment Level Reference	The constraining level of the finish end of the brace.
End Attachment Elevation	The elevation of the finish end referenced from the End Attachment Level Reference.
Start Attachment Type	Distance or Ratio. Type of measure of distance between brace start end and a designated beam end. Applies to the brace end attached to a beam.
Start Attachment Distance	Available when Start Attachment Type is specified as Distance. Distance of separation of the brace start end from the designated beam end. Applies to the brace end attached to a beam.
Start Attachment Ratio	Available when Start Attachment Type is specified as Ratio. Ratio (percentage) of separation of the brace start end from the designated beam end. Applies to the brace end attached to the beam.

Name	Description
End of Referenced Element	Available when attached to a beam. Designated end (Start or End) of the referenced beam to which the start end of brace is attached. Applies to the brace end attached to the beam.
End Attachment Type	Distance or Ratio. Type of measure of distance between brace finish end and a designated beam end. Applies to the brace end attached to a beam.
End Attachment Distance	Available when End Attachment Type is specified as Distance. Distance of separation of the brace finish end from the designated beam end. Applies to the brace end attached to a beam.
End Attachment Ratio	Available when End Attachment Type is specified as Ratio. Ratio (percentage) of separation of the brace finish end from the designated beam end. Applies to the brace end attached to the beam.
End of Referenced Element	Available when attached to a beam. Designated end (Start or End) of the referenced beam to which the finish end of brace is attached. Applies to the brace end attached to the beam.
Rebar Cover - Top Face	Applies to concrete beams only. The rebar cover distance from the column top face.
Rebar Cover - Bottom Face	Applies to concrete beams only. The rebar cover distance from the column bottom face.
Rebar Cover - Other Faces	Applies to concrete beams only. The rebar cover distance from the column to adjacent element faces.
Estimated Reinforcement Volume	Specifies the estimated reinforcement volume of the selected element. This is a read-only parameter that only displays when rebar has been placed.
Enable Analytical Model	Displays the analytical model and includes it in analytical calculations. Selected by default. See Disable an Analytical Model.
Dimensions	
Length	The length of the brace location line. This is a read-only value.
Volume	The volume of the selected brace element. This is a read-only value.
Elevation at Top	Indicates the elevation used for tagging the top of the brace. This is a read-only parameter that reports Varies for sloped planes.
Elevation at Bottom	Indicates the elevation used for tagging the bottom of the brace. This is a read-only parameter that reports Varies for sloped planes.
Identity Data	

Name	Description
Comments	User comments.
Mark	A label created for the brace. Possible use: shop mark. This value must be unique for each brace in a project. Revit warns you if the number is already used but allows you to continue using it. You can see the warning using the Review Warnings tool. See Review Warning Messages.
Phasing	
Phase Created	Indicates in which phase the brace component was created. See Project Phasing.
Phase Demolished	Indicates in which phase the brace component was demolished. See Project Phasing.
Geometric Position	
Start Extension	Applies to steel braces only. A dimension that adds brace geometry beyond the start end of the brace.
End Extension	Applies to steel braces only. A dimension that adds brace geometry beyond the finish end of the brace.
Start Join Cutback	Applies to steel braces only. The dimension between the edge of the start end of the brace and the element to which it is connected. Only for the joined start of an element.
End Join Cutback	Applies to steel braces only. The dimension between the edge of the finish end of the brace and the element to which it is connected. Only for the joined end of the element.
yz Justification	Applies to steel braces only. Either Uniform or Independent. Uniform allows setting the same parameters to both start and end of a brace. Independent allows setting different parameters to start and end of a brace.
y Justification	Applies to Uniform justified steel braces only. Specifies the location of the physical geometry with respect to the location line: either Origin, Left, Center, Right.
y Offset Value	Applies to Uniform justified steel braces only. The numeric value that offsets the geometry. The distance between the location line and the characteristic point set in y Justification parameter.
z Justification	Applies to Uniform justified steel braces only. Specifies the location of the physical geometry with respect to the location line: either Origin, Top, Center, Bottom.
z Offset Value	Applies to Uniform justified steel braces only. The distance between the location line and the characteristic point set in z Justification parameter.
Start y Justification	Applies to Independent justified steel braces only. Specifies the location of the physical geometry of the start end of the brace with respect to the location line: either Origin, Left, Center, Right.
Start y Offset Value	Applies to Independent justified steel braces only. The numeric value that offsets the geometry at the start end of the brace. The distance between the location line and the characteristic point set in the Start y Justification parameter.

Name	Description
Start z Justification	Applies to Independent justified steel braces only. Specifies the location of the physical geometry with respect to the location line at the start end of the brace: either Origin, Top, Center, Bottom.
Start z Offset Value	Applies to Independent justified steel braces only. The numeric value that offsets the geometry at the finish end of the brace. The distance between the location line and the characteristic point set in the Start z Justification parameter.
End y Justification	Applies to Independent justified steel braces only. Specifies the location of the physical geometry of the finish end of the brace with respect to the location line: either Origin, Left, Center, Right.
End y Offset Value	Applies to Independent justified steel braces only. The numeric value that offsets the geometry at the start end of the brace. The distance between the location line and the characteristic point set in the End y Justification parameter.
End z Justification	Applies to Independent justified steel braces only. Specifies the location of the physical geometry of the finish end of the brace with respect to the location line: either Origin, Left, Center, Right.
End z Offset Value	Applies to Independent justified steel braces only. The numeric value that offsets the geometry at the start end of the brace. The distance between the location line and the characteristic point set in the End z Justification parameter.
Other	
Start Extension Calculation	Applies to steel braces only. Specifies family parameters; defines maximum distance of start extension parameter. This parameter is only available in projects created in versions prior to Autodesk Revit 2014 containing brace families that have not been updated. This is a read-only value.
End Extension Calculation	Applies to steel braces only. Specifies family parameters; defines maximum distance of end extension parameter. This parameter is only available in projects created in versions prior to Autodesk Revit 2014 containing brace families that have not been updated. This is a read-only value.

CHAPTER 9 - STAIRCASES AND RAMPS

In Revit, Staircases and Ramps are non-structural elements. These elements are introduced in this course to complete a building model. In this section, you will learn

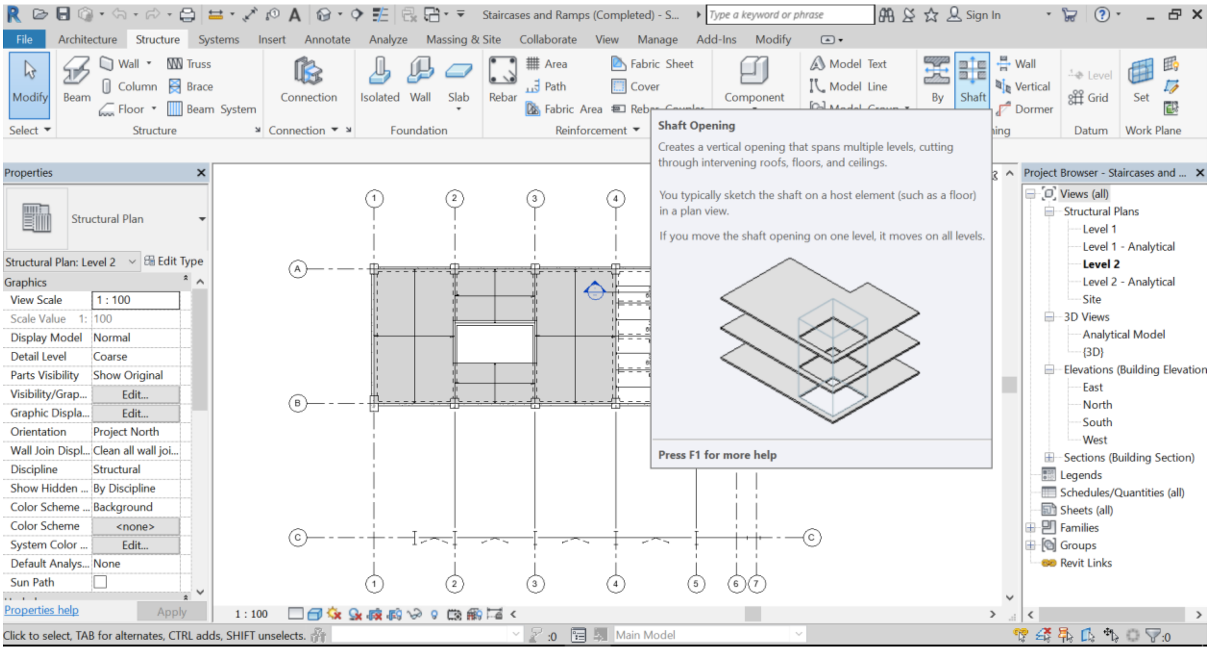
- How to add shaft opening (for lift / staircases) that run through the entire building height.
- How to create staircases.
- Options in staircase placement.
- How to create Ramps.

9.1 Adding a shaft opening with default settings

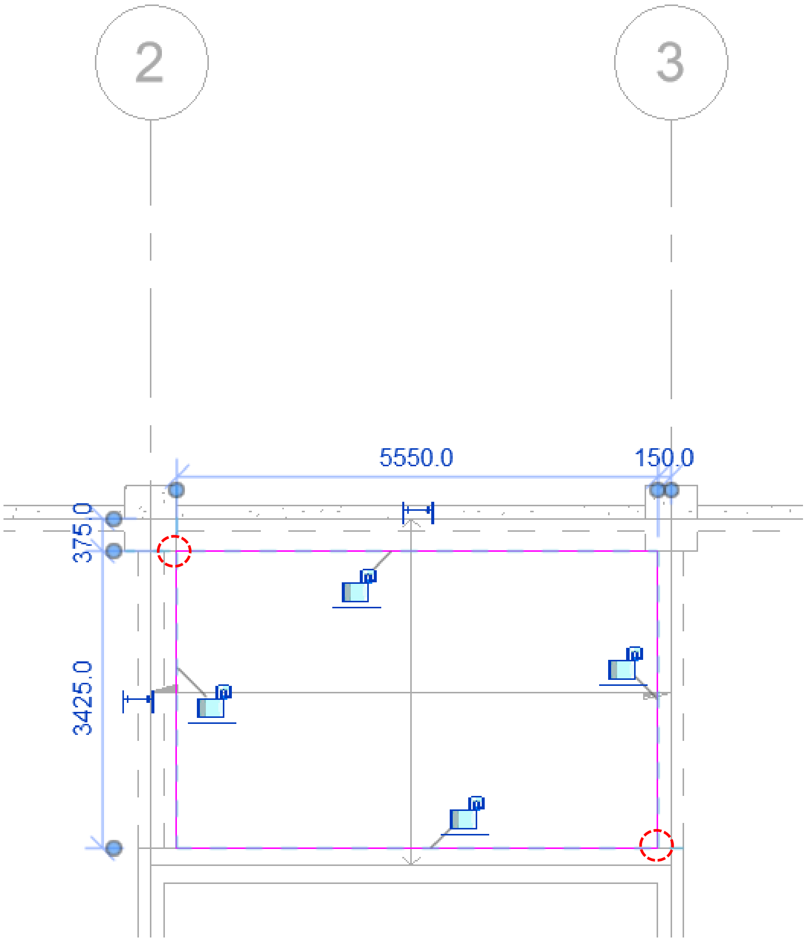
To add a shaft opening in Revit,

1 Open from the course folder the "Staircases and Ramps.rvt" file and go to Level 2 (Structural Plan).

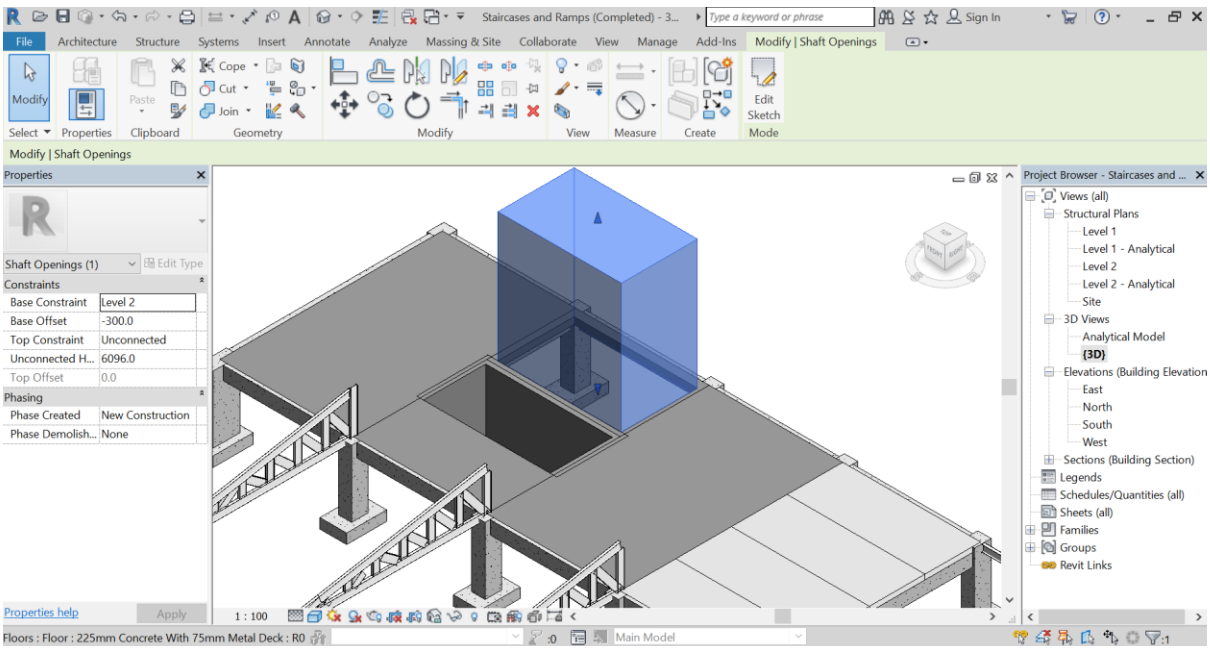
2 Click Structure Tab ➢ Opening panel ➢ Shaft.



3 Sketch the opening by Rectangle. Pick the first point at the lower right corner of the column at Grid-A-2 and the second point at the intersection of wall edge and beam along Grid-3 as shown below.



4 Go to 3D view to view the opening.

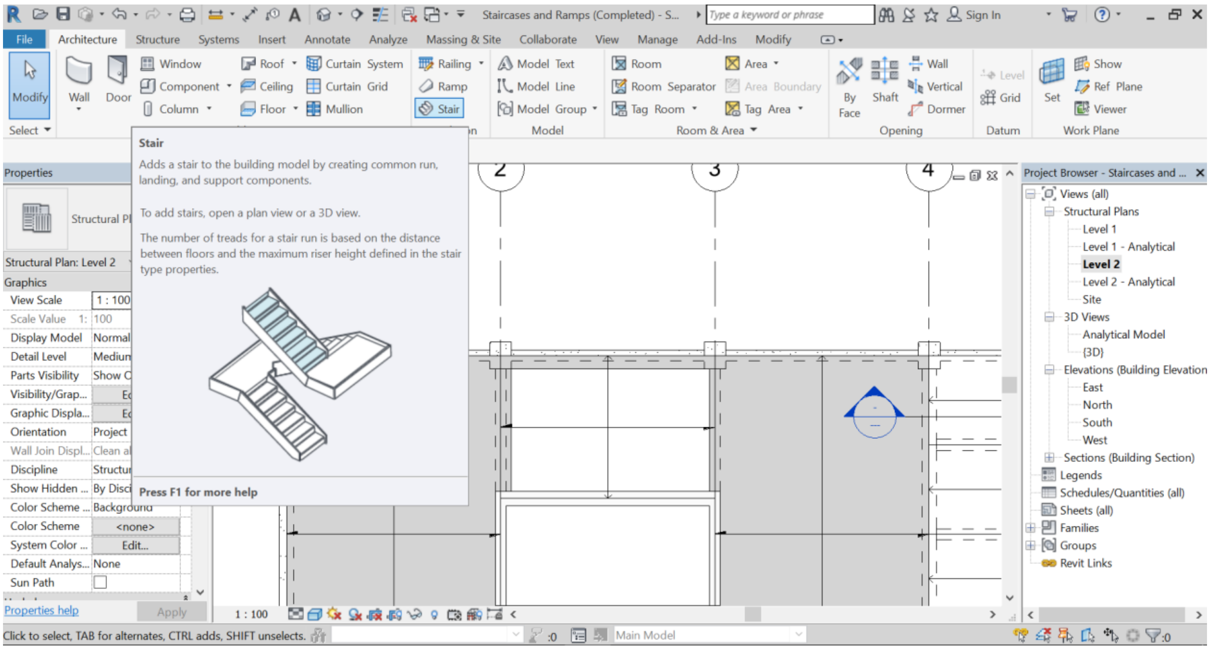


9.2Create Staircases

To create staircases in Revit,

1 Go to Level 2 (Structural Plan).

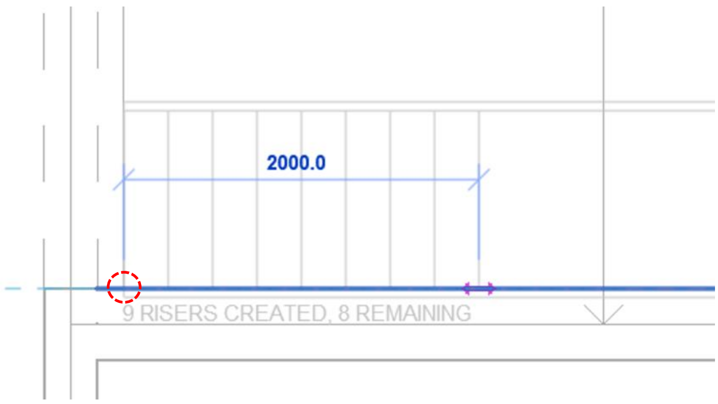
2 Click Architect Tab ➢ Circulation panel ➢ Stair.



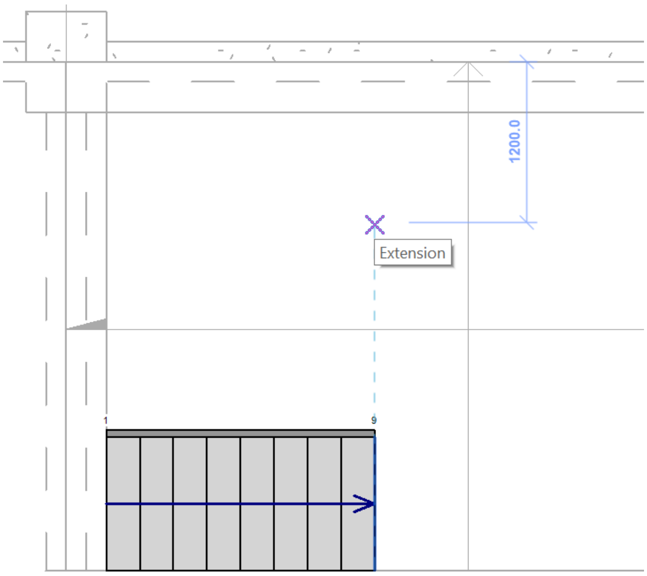
3 From the Properties Palette, select “190mm max riser 250mm going” from Assembled Stair.

4 Keep all default settings. Start sketching the staircase at the circled corner shown below. (Notice that as you sketch the staircase, Revit will show you how many risers created and how many remaining.)

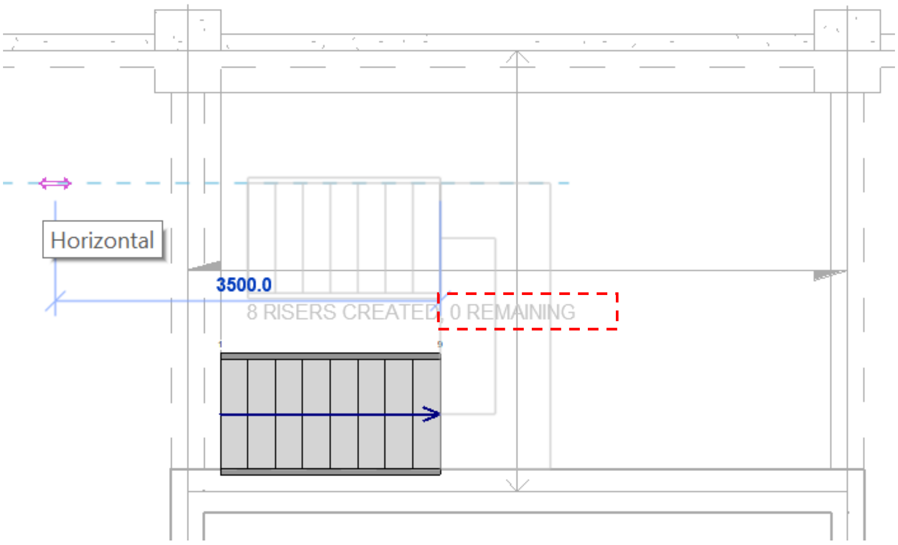
5 Pick another point after 2000mm to the right.



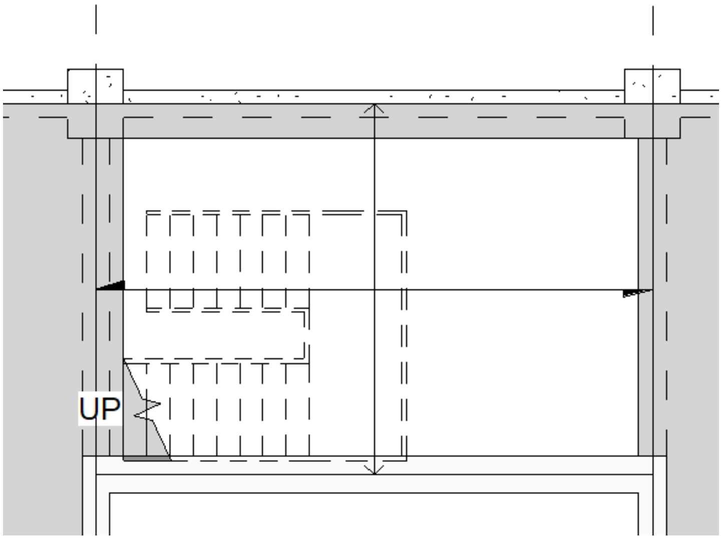
6 Pick the third point as shown below.



7 Then extend the staircase to the left until 0 (zero) riser remaining.

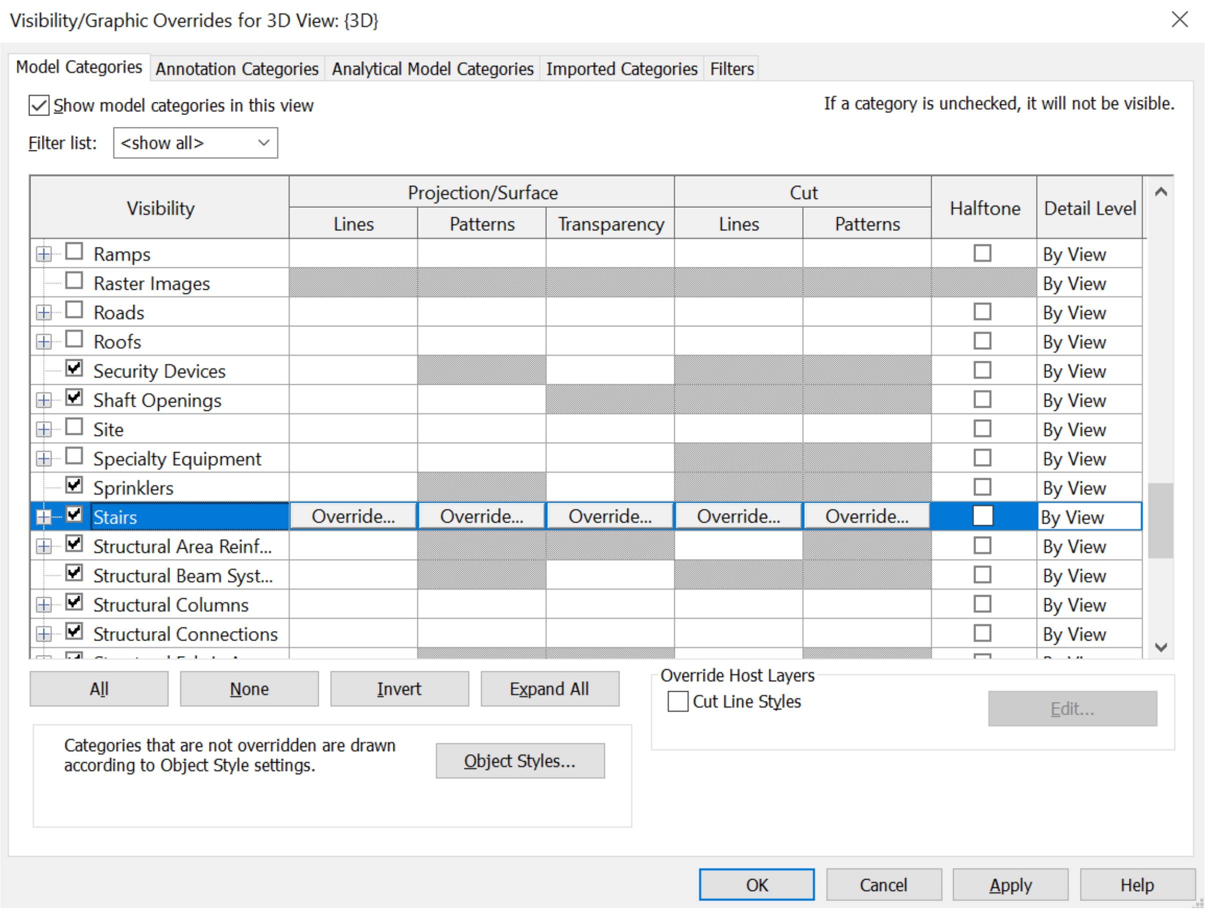


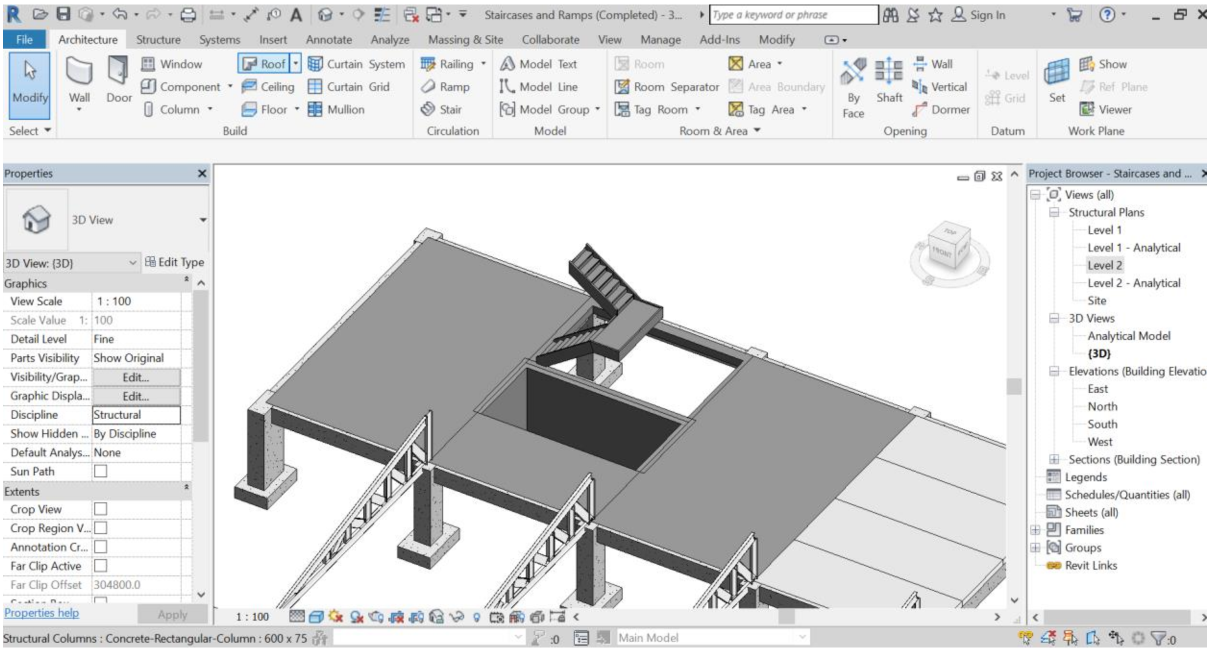
8 Click Finish Edit Mode  to add the staircase.



9 If you go to the 3D view, you will notice that you cannot see the staircase just created. This is because Stair is turned off in Visibility Control.

10 In the 3D view, type “vv” (or “vg”) for Visibility/Graphic Overrides. In Model Categories scroll down to find Stair and enable its visibility. Click “OK”.

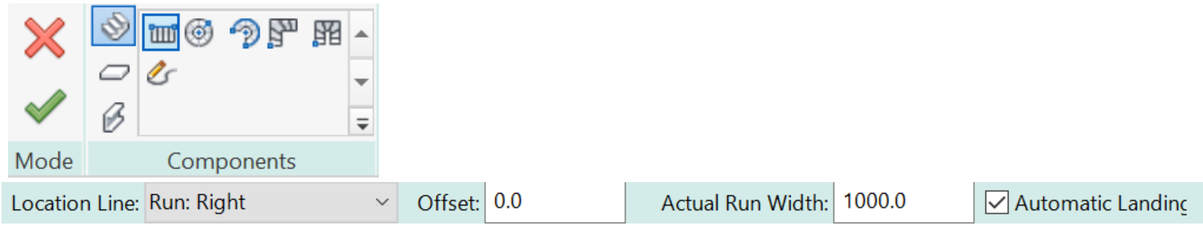




9.3 Options in Creating Staircase

There are many options available to help you to define a staircase in Revit. Many of the options are self-explanatory. You can hover over the option and see its description and press F1 for more help. Under the contextual ribbon tab, the options for creating staircase are:

- Draw (Components): Run / Landing / Support
- Draw (Components) by: Straight / Full-Step Spiral / Center-Ends Spiral / L-Shape Winder / U-Shape Winder / Create Sketch
- Location Line: Exterior Support: Left / Run: Left / Run: Center / Run: Right / Exterior Support: Right
- Options: Offset / Actual Run Width / Automatic Landing

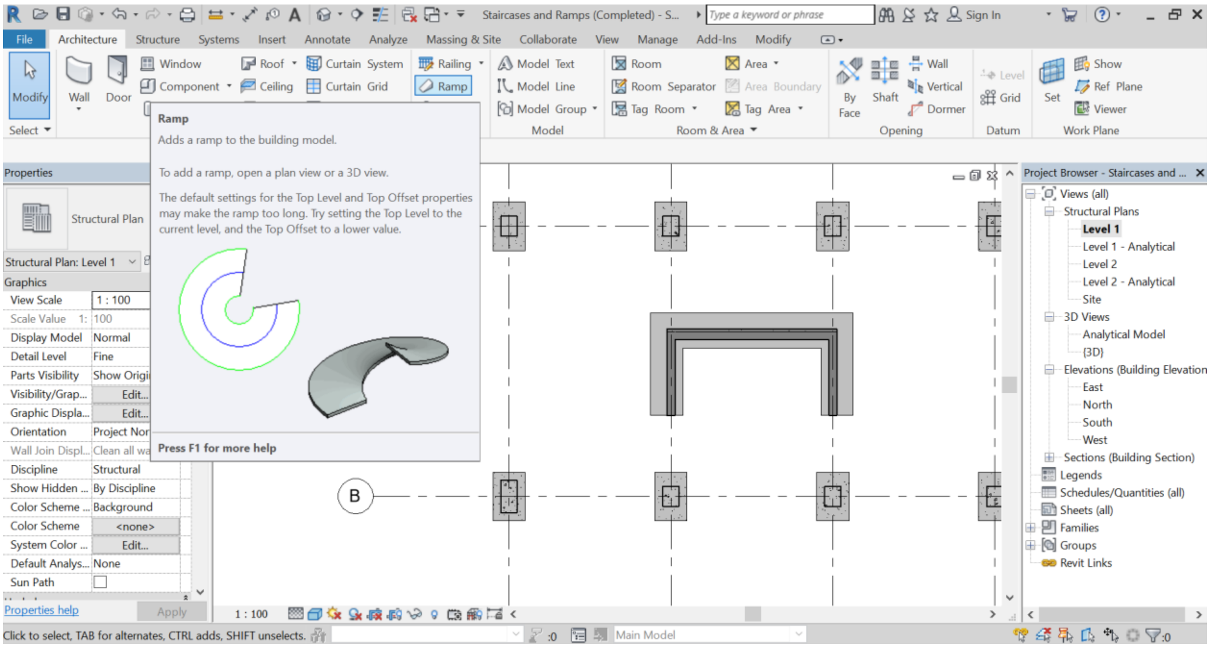


9.4Creating Ramps

To add a ramp in Revit,

1 Go to Level 1 (Structural Plan).

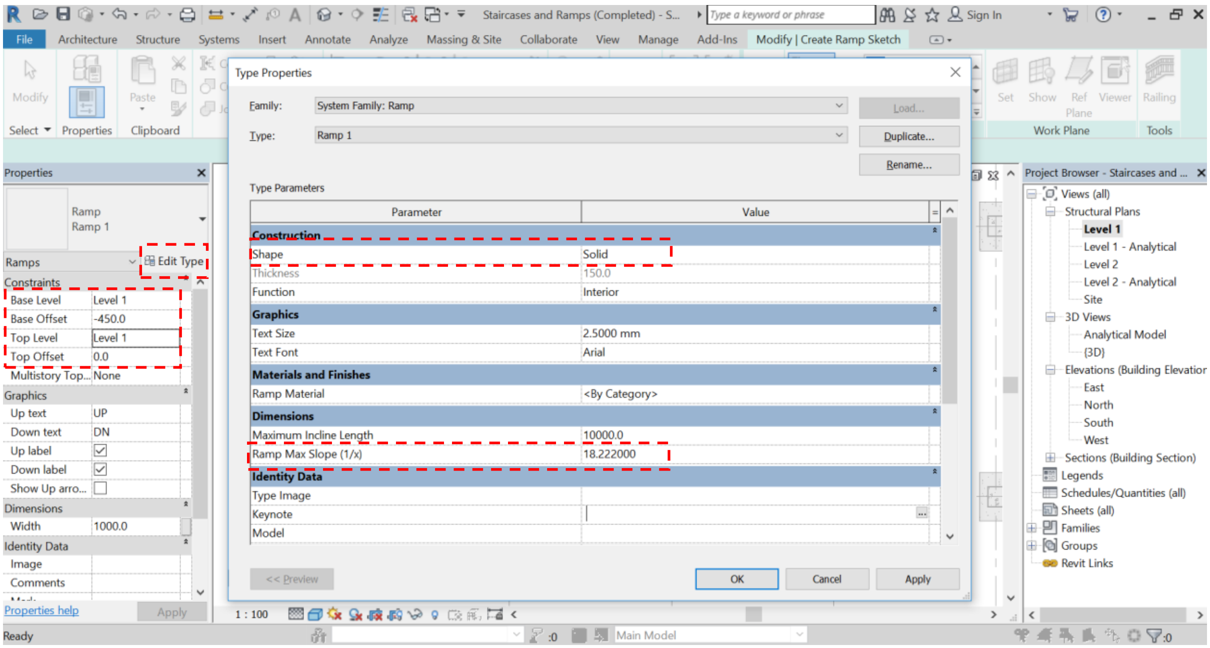
2 Click Architecture Tab ➢ Circulation panel ➢ Ramp.



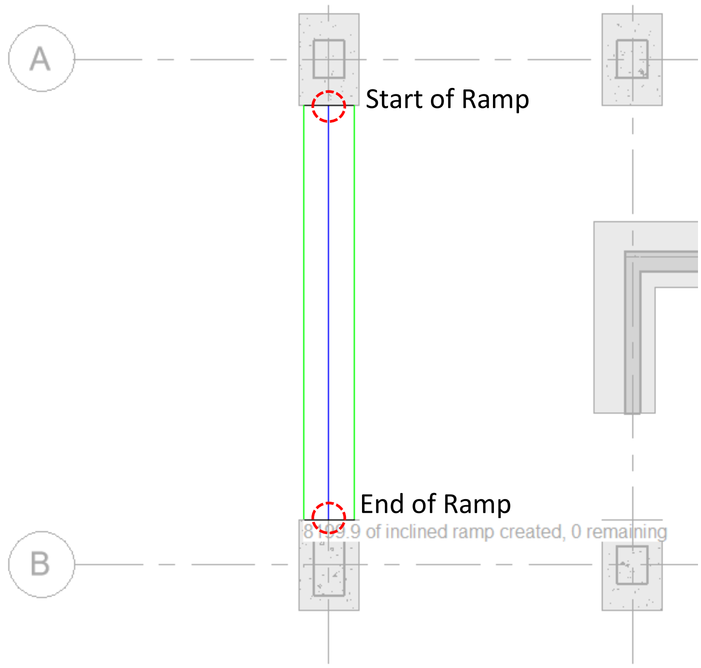
3 Set Base Level and Base Offset to Level 1 and -450 respectively.

4 Set Top Level and Top Offset to Level 1 and 0 respectively.

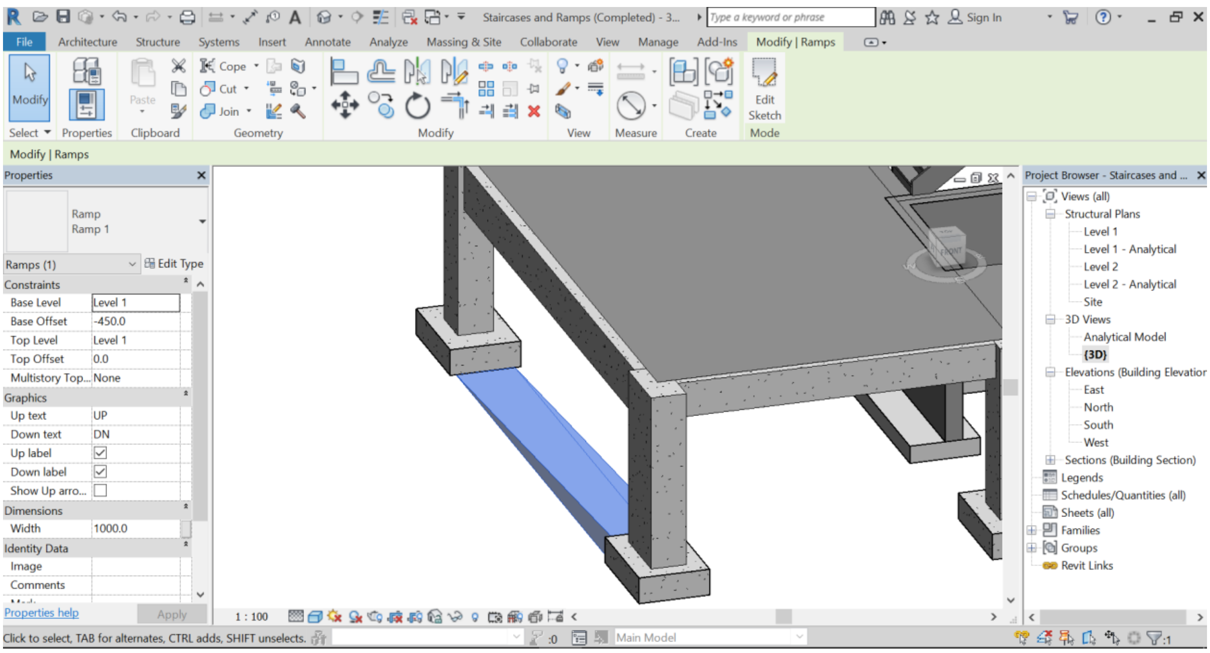
5 Click "Edit Type". Then change Shape to "Solid" and Ramp Max Slope(1/x) to 18.222.



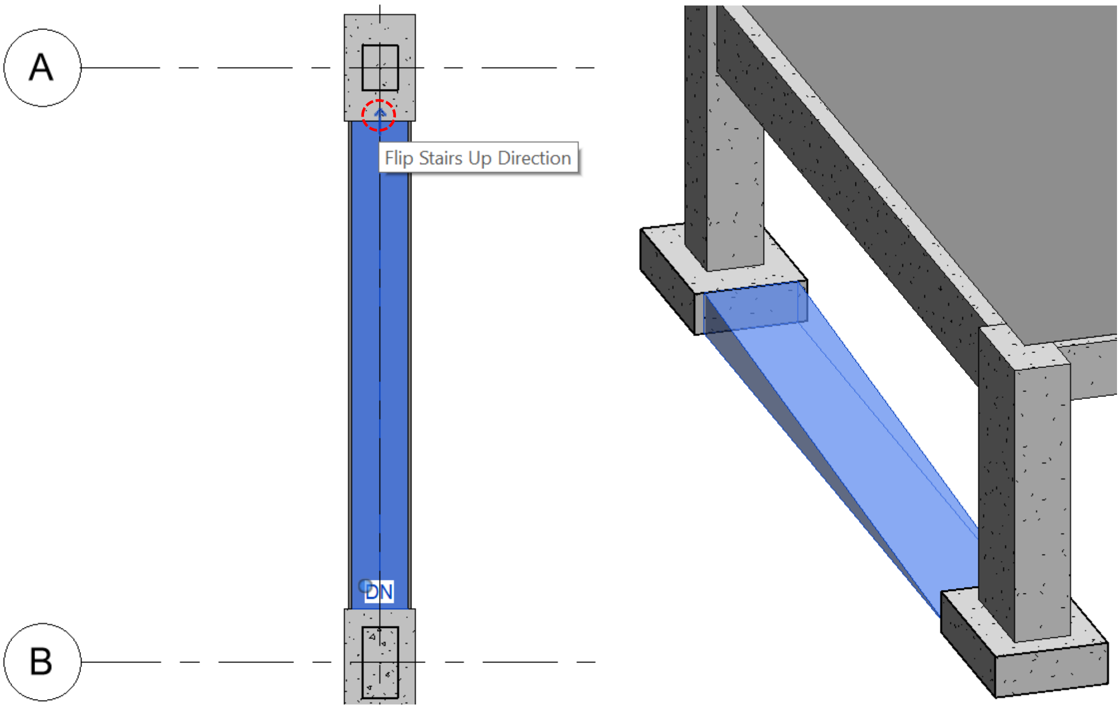
6 Pick the start and end points of the ramp as shown below.



7 Click Finish Edit Mode to add the ramp.



8 When you select the ramp, you can change the ramp up direction by clicking the direction arrow (highlighted below).



9.5 Detail Description of Ramp Instance Properties

(source: Autodesk Knowledge Network)

https://knowledge.autodesk.com/support/revit-products/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/Revit-Model/files/GUID-14D7B7D6-F29A-4AD2-8A14-B91AD4A99ECA-htm.html

Name	Description
Constraints	
Base Level	Sets the base of the ramp.
Base Offset	Sets the ramp's height from its base level.
Top Level	Sets the top of the ramp.
Top Offset	Sets the ramp's offset from the top level.
Multistory Top Level	Sets the top of the ramp in a multi-story building.
Graphics	
Up text	Specifies the up text.
Down text	Specifies the down text.

Name	Description
Up label	Indicates whether the up text appears.
Down label	Indicates whether the down text appears.
Show Up arrow in all views	Indicates whether the up arrow appears in all views.
Dimensions	
Width	Width of the ramp.
Identity Data	
Comments	Specific comments about the ramp.
Mark	A unique identifier for the ramp.
Phasing	
Phase Created	The phase when the ramp was created. See Create a phase.
Phase Demolished	The phase when the ramp was demolished. See About Demolishing Elements.

CHAPTER 10 - PILES and PILE CAPS

Piles and Pile Caps are loadable families In Revit. You can Create/Customize these families for special use. In this section, you will learn

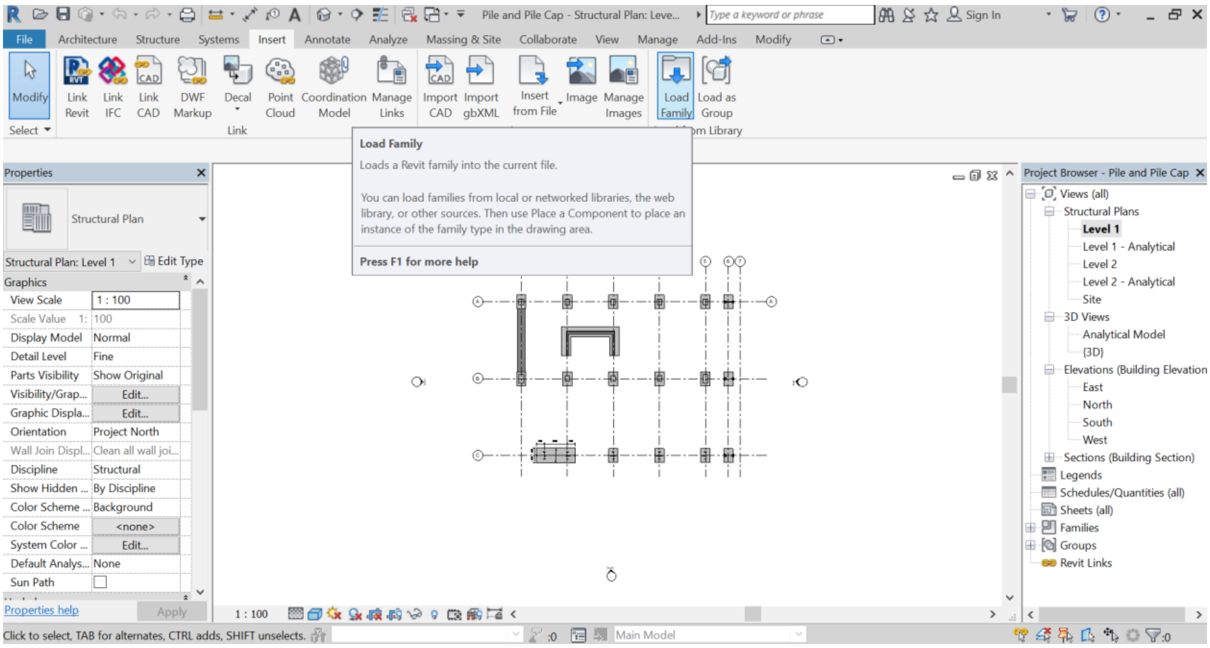
- How to load Pile and Pile Caps families.
- How to place Pile and Pile Cap in Revit.
- How to create new Pile/Pile Caps type from existing types.

10.1 Loading Pile and Pile Caps Families in Revit

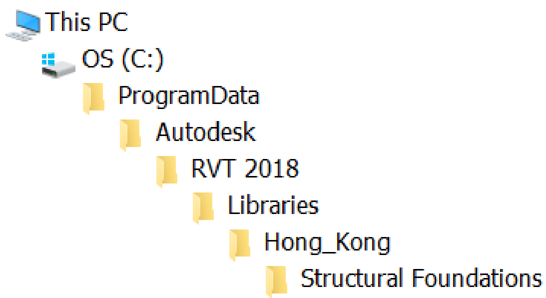
Pile/Pile Cap families are not automatically loaded in Revit. To load a Pile/Pile Cap family,

1 Open from the course folder the “Piles and Pile Caps.rvt” file and go to Level 1 (Structural Plan).

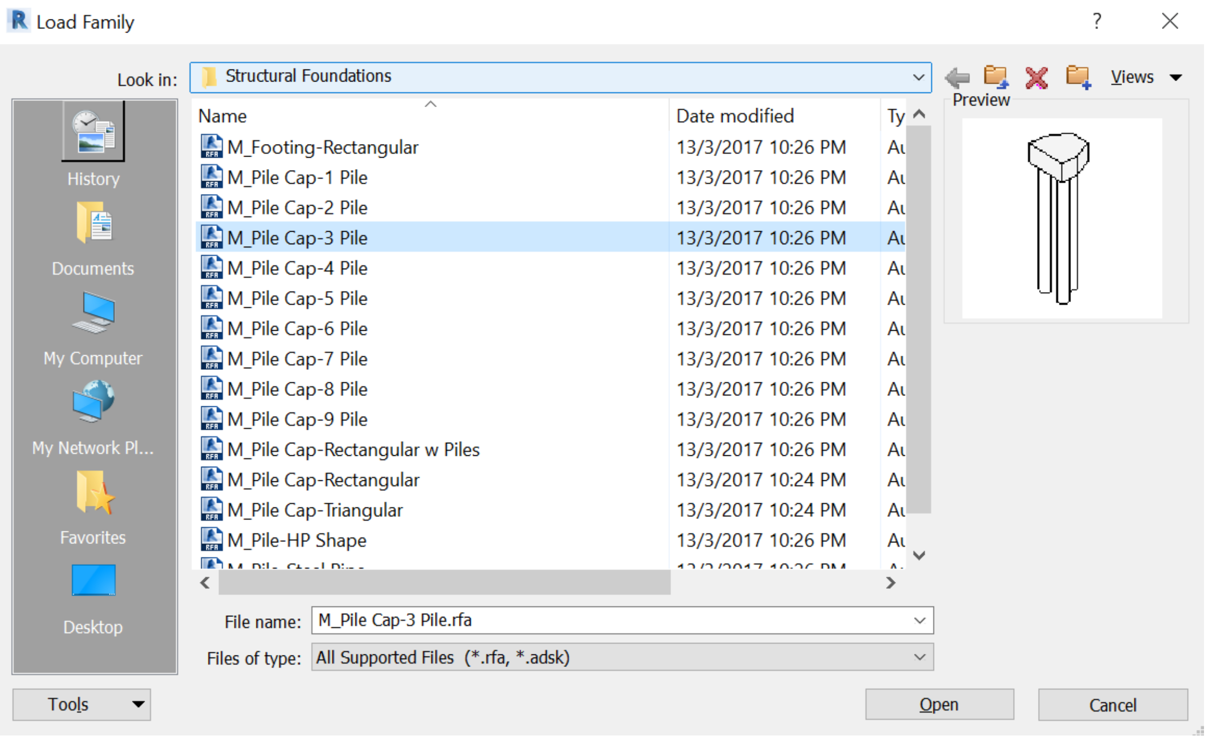
2 Click Insert Tab ➢ Load from Library panel ➢ Load Family.



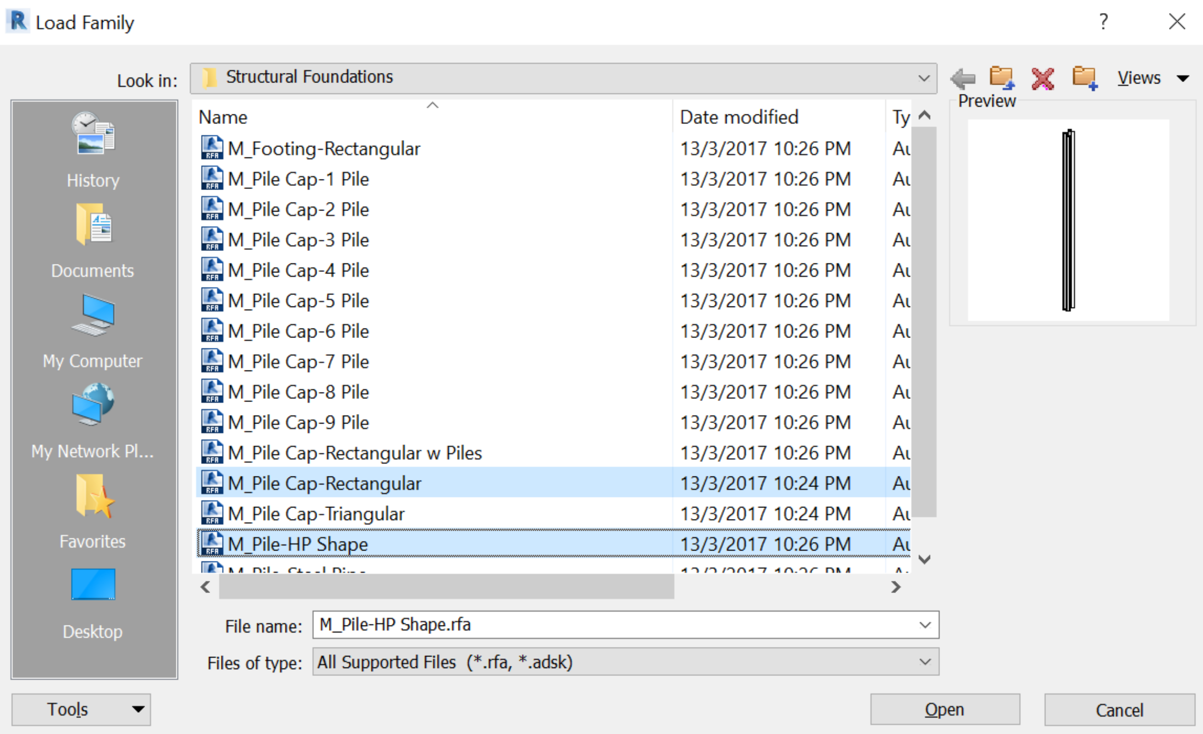
3 Select the path for the families to be loaded.



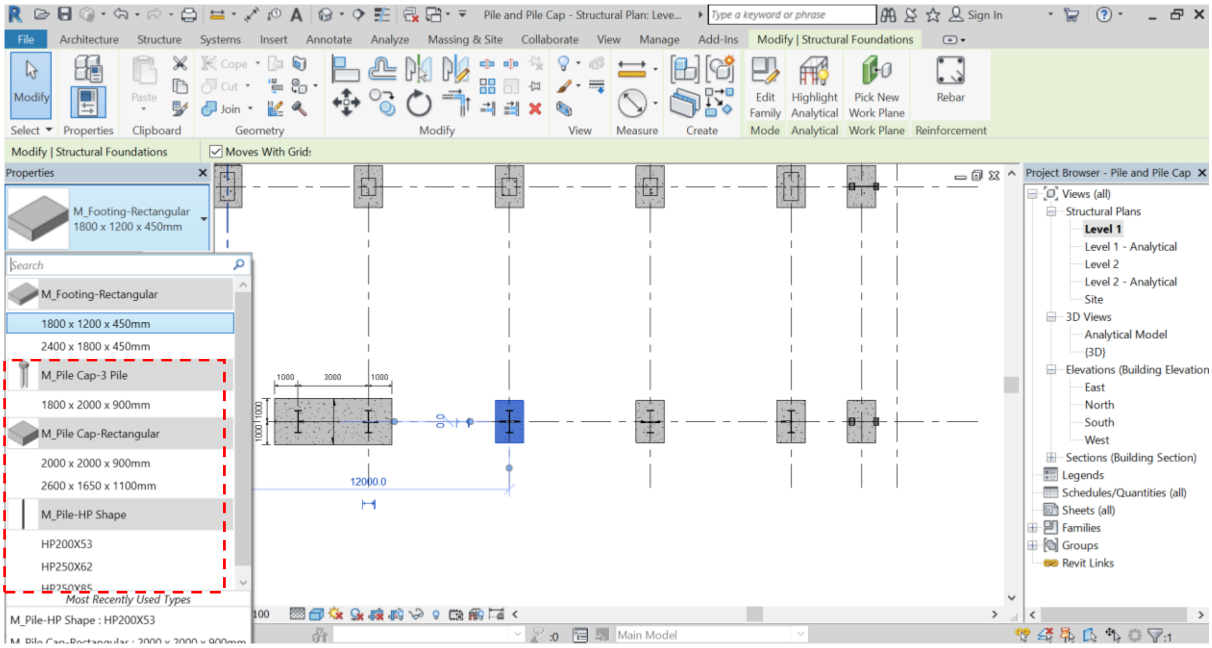
4 Select M_Pile Cap-3 Pile and click OK.



5 Also load “M_Pile Cap-Rectangular” and “M_Pile-HP Shape” families into the project.



Now when you select/create a foundation element, you can find the loaded Pile/Pile Cap in the Type Selector from the Properties Palette.

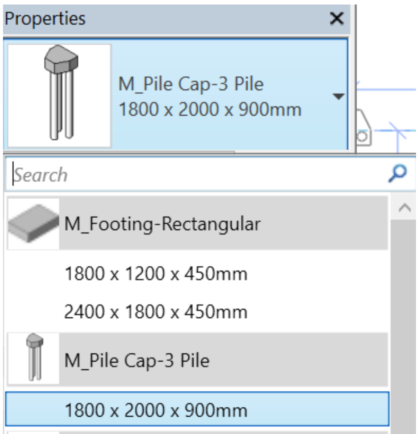


10.2 Adding Pile and Pile Caps in Revit

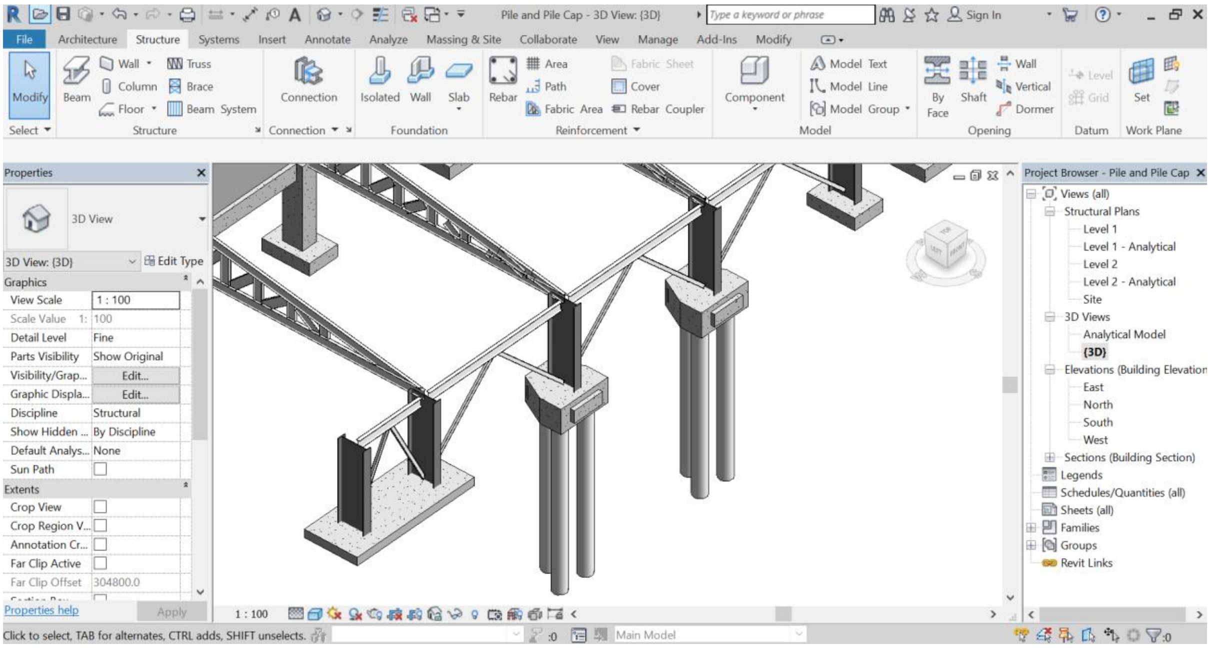
To place Piles/Pile Caps in Revit,

1 Click Structure Tab ➢ Foundation panel ➢ Isolated.

2 Select "1800 x 2000 x 900mm" from M_Pile Cap-3 Pile family.

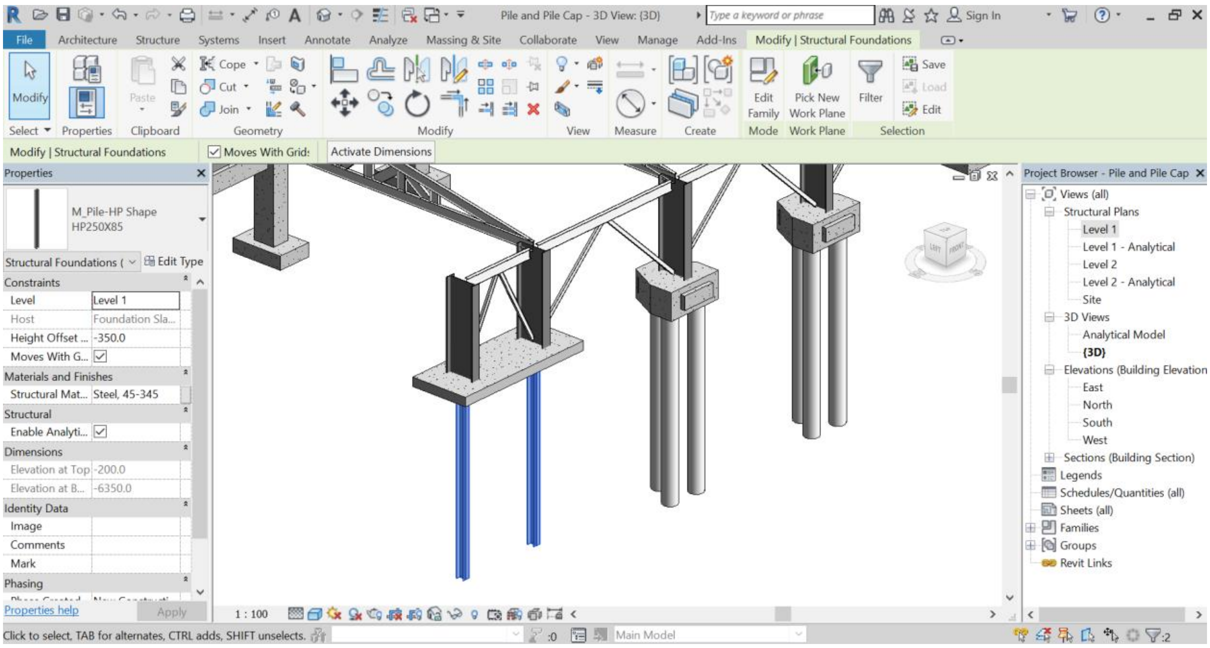


3 Place the Pile Cap at Grid-C-3, Grid-C-4 and Grid-C-5. Press "esc" a couple of times.



4 Now, place 2 piles to support the footing at Grid-C-2. Go to Level 1 (Structural Plan) and select Isolated from foundation panel in Structure Tab.

5 Select "HP205x80" pile and place at the column locations on the footing.

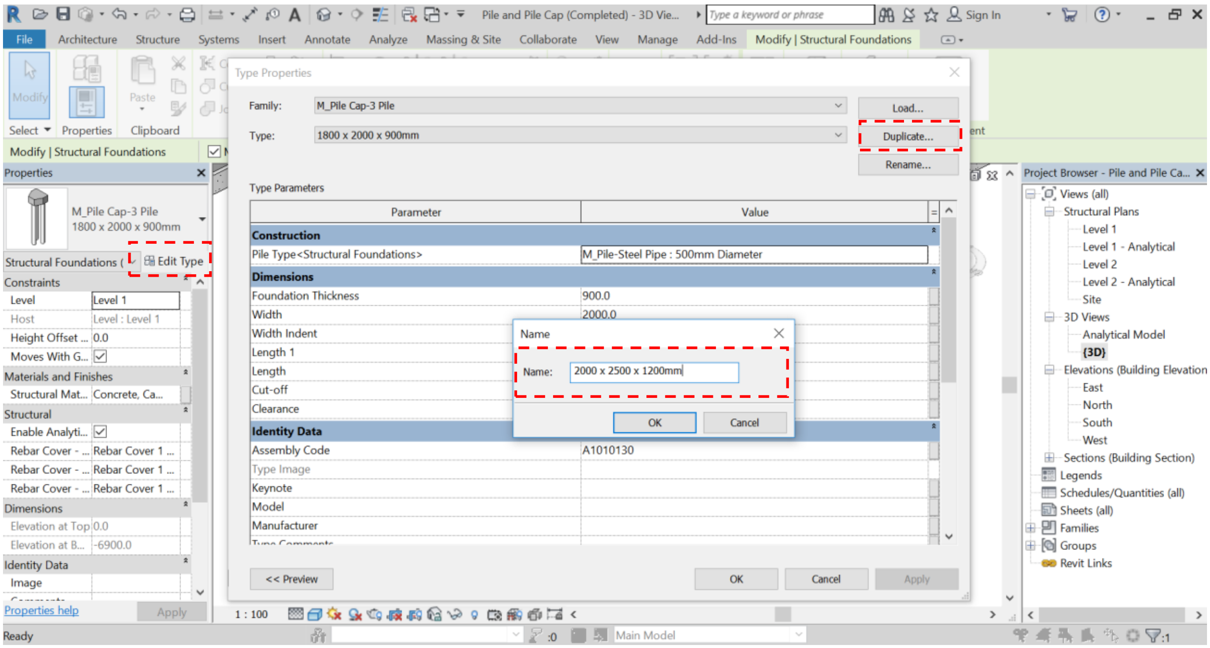


10.3 Create New Pile and Pile Cap (from loaded type)

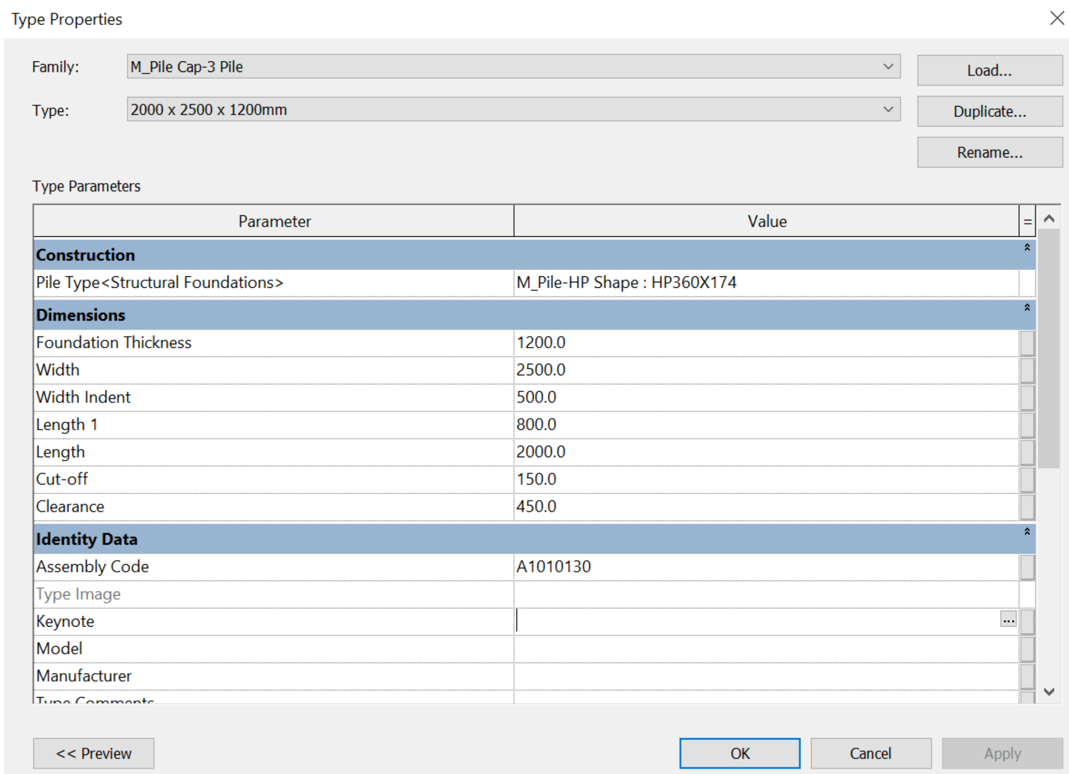
You can create a new pile and pile cap by duplicating a loaded type.

1 Select an existing pile cap (at Grid-C-3) and click "Edit Type".

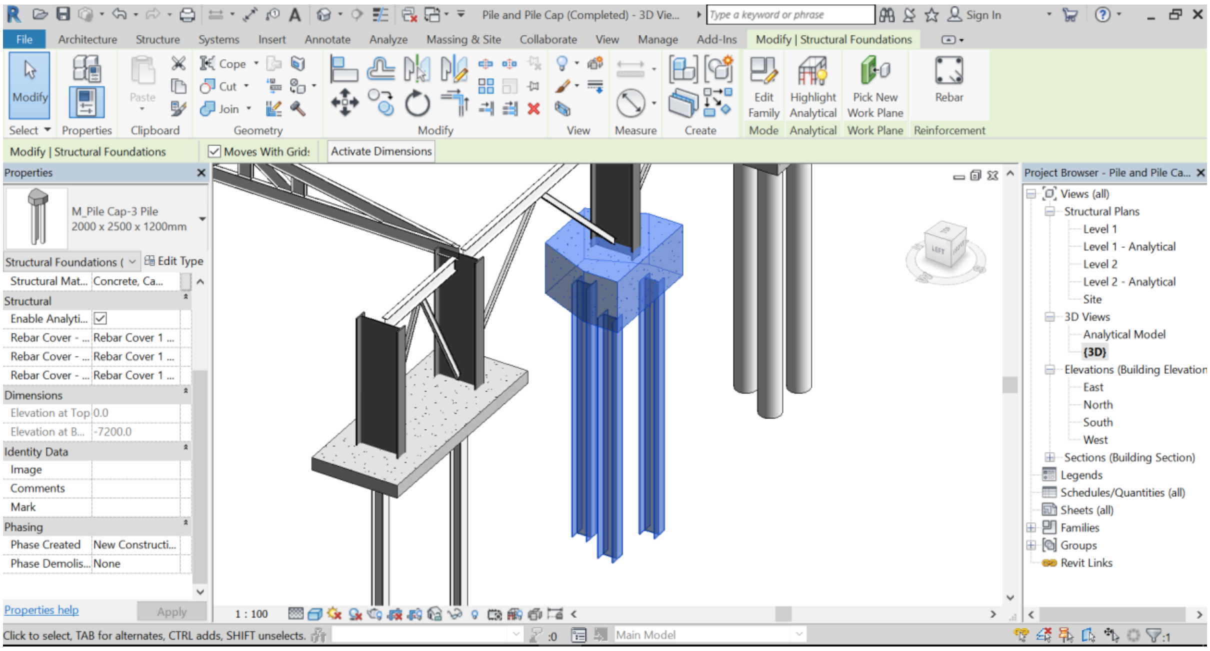
2 Click “Duplicate” and change the name to “2000 x 2500 x 1200mm” then click OK.



3 Change the Pile Type to HP360X174 instead of 500mm Diameter. Also change the Foundation Thickness, Width and Length to 1200, 2500 and 2000 respectively.



4 Click OK and a new Pile Cap type is created.


Pile and Pile Cap (Completed) - 3D Vie...
File
Architecture
Structure
Systems
Insert
Annotate
Analyze
Massing & Site
Collaborate
View
Manage
Add-Ins
Modify | Structural Foundations
Modify | Structural Foundations
Moves With Grids
Activate Dimensions
Properties
M_Pile Cap-3 Pile
2000 x 2500 x 1200mm
Structural Foundations
Edit Type
Project Browser - Pile and Pile Ca...
Views (all)
Structural Plans
Level 1
Level 1 - Analytical
Level 2
Level 2 - Analytical
Site
3D Views
Analytical Model
{3D}
East
North
South
West
Legends
Sheets (all)
Families
Groups
Revit Links
Click to select, TAB for alternates, CTRL adds, SHIFT unselects.
Main Model

Chapter 11 - 3D REINFORCEMENT

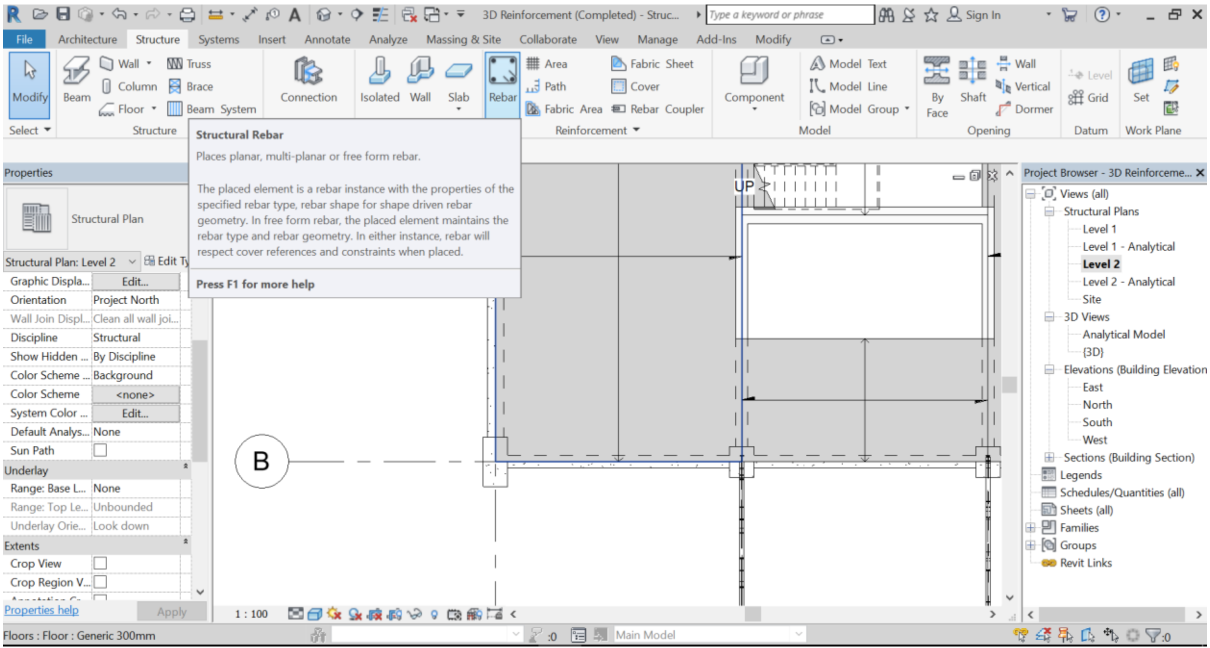
3D reinforcement (rebar) can be modelled in Revit. Reinforcement must be hosted by a valid host such as structural wall, column and floor. In this section, you will learn

- How to place reinforcement for Columns.
- How to place area reinforcement for Floors.
- How to control visibility of rebars.
- Various reinforcement settings in Revit.

11.1 Modelling Reinforcement for Columns

1 Open from the course folder the “3D Reinforcement.rvt” file and go to Level 2 (Structural Plan).

2 Click Structure Tab ➢ Reinforcement panel ➢ Rebar.

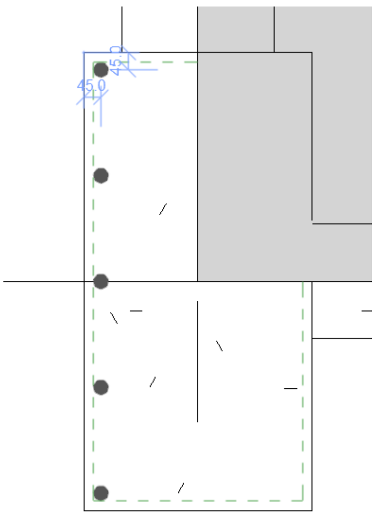


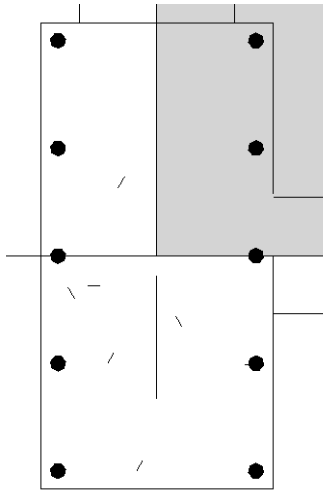
3 First, place the main (longitudinal) rebars with the following settings:

- Use default 40R rebar
- Placement Plane: Near Cover Reference
- Placement Orientation: Perpendicular to Cover
- Rebar Set: (Layout: Fixed Number and Quantity: 5)

Hover over to the left side of the column at Grid-B-1 and place the rebars. Revit will show you boundary of the cover in green dotted lines.

4 Select the rebars just created and mirror (type “MM”) them to the other side of the column.

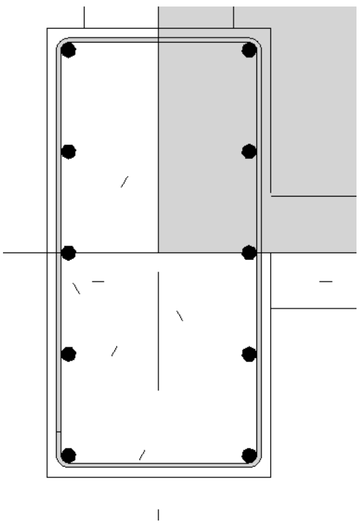




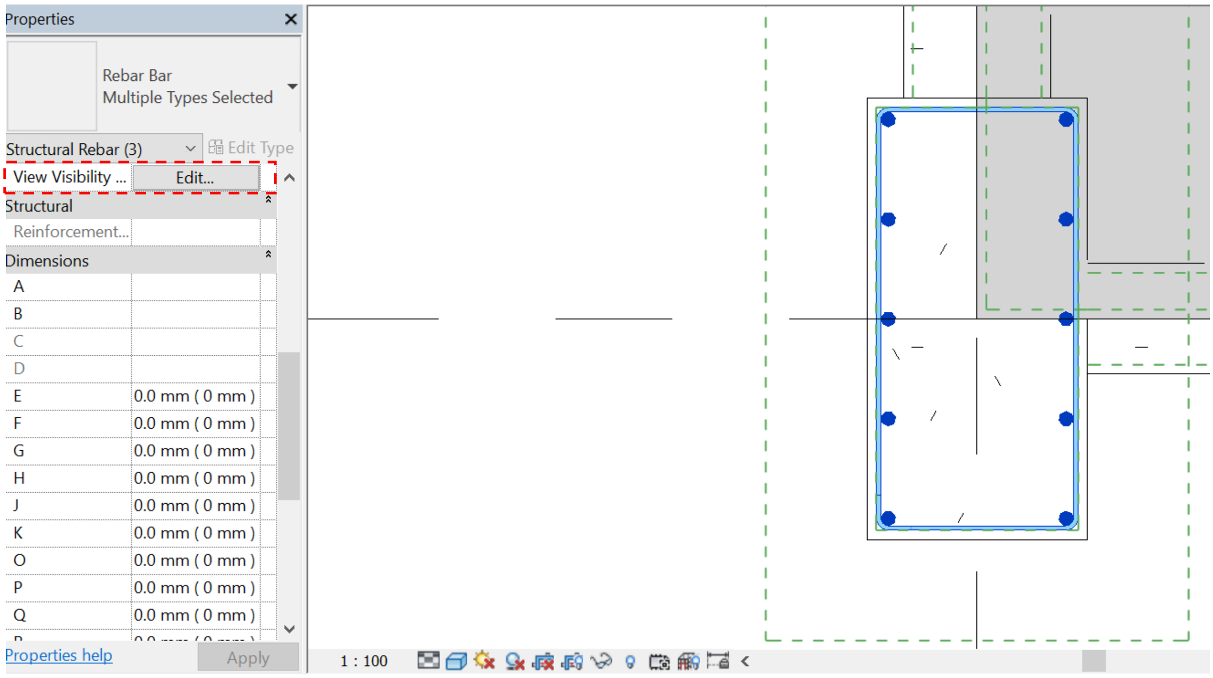
5 Next, place the links to the column. Click Structure Tab ➢ Reinforcement panel ➢ Rebar.

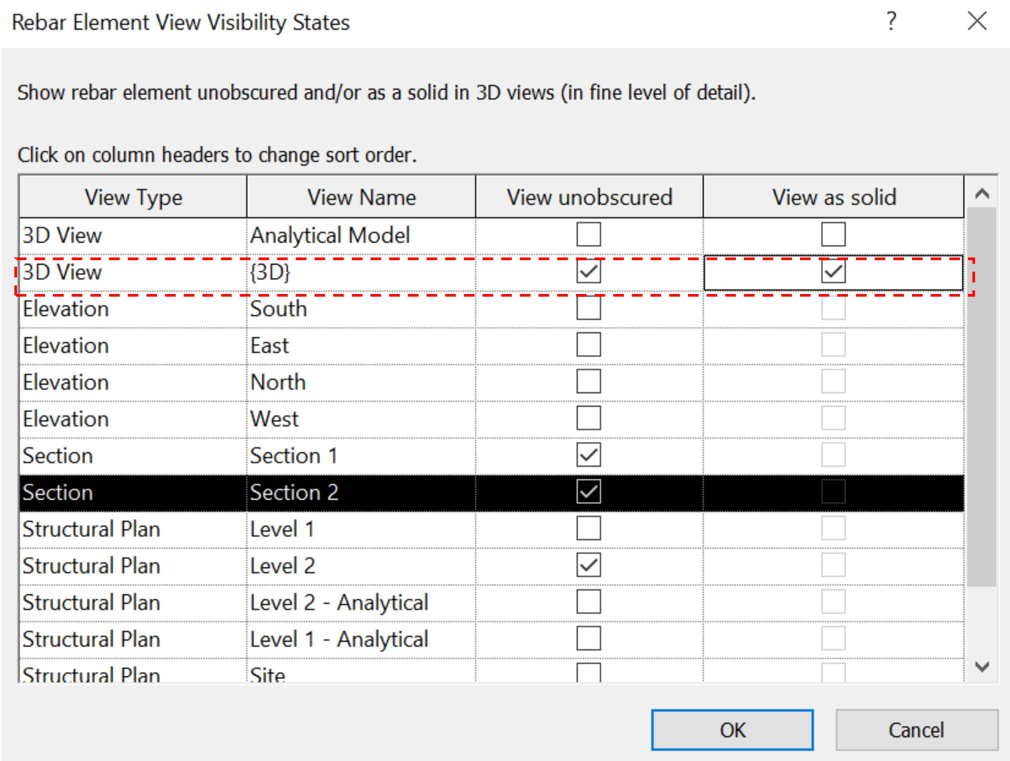
6 Select

- Rebar Shape: 51
- 12T rebar
- Placement Plane: Near Cover Reference
- Placement Orientation: Parallel to Cover
- Rebar Set: (Layout: Maximum Spacing and Spacing: 300mm)

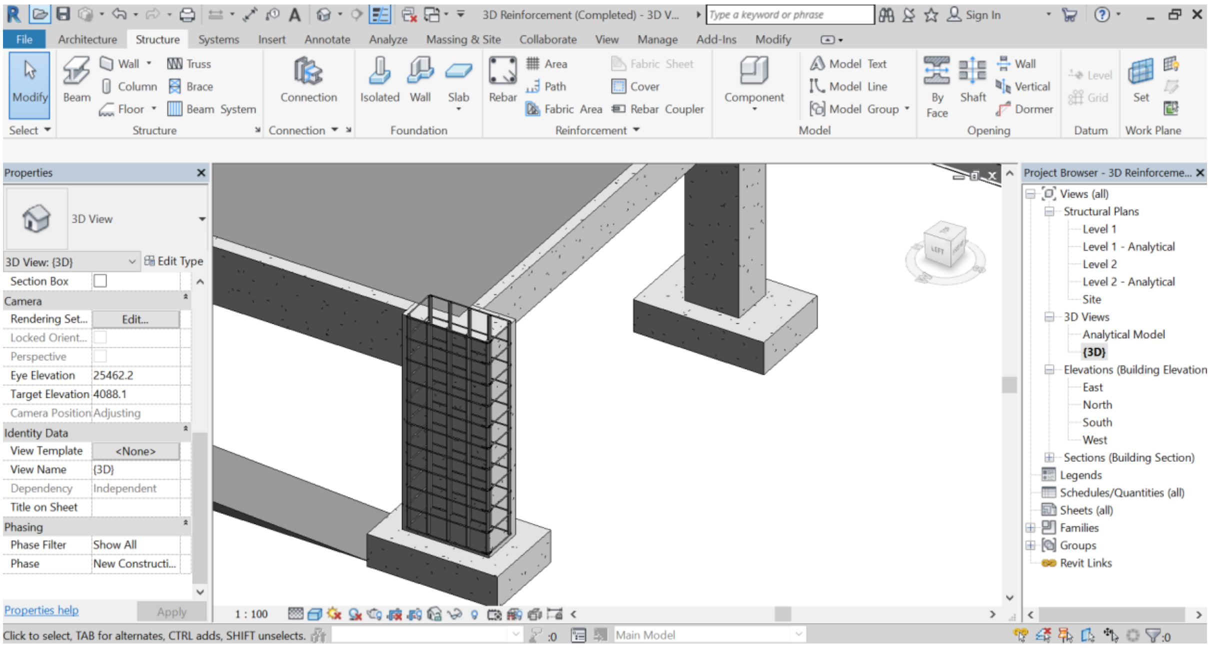


7 Now change the view visibility of these rebars. Select all the rebars and click “Edit” in View Visibility from the Properties Palette and make the rebars “View unobscured” and “View as solid” in 3D View {3D}. Then click OK.

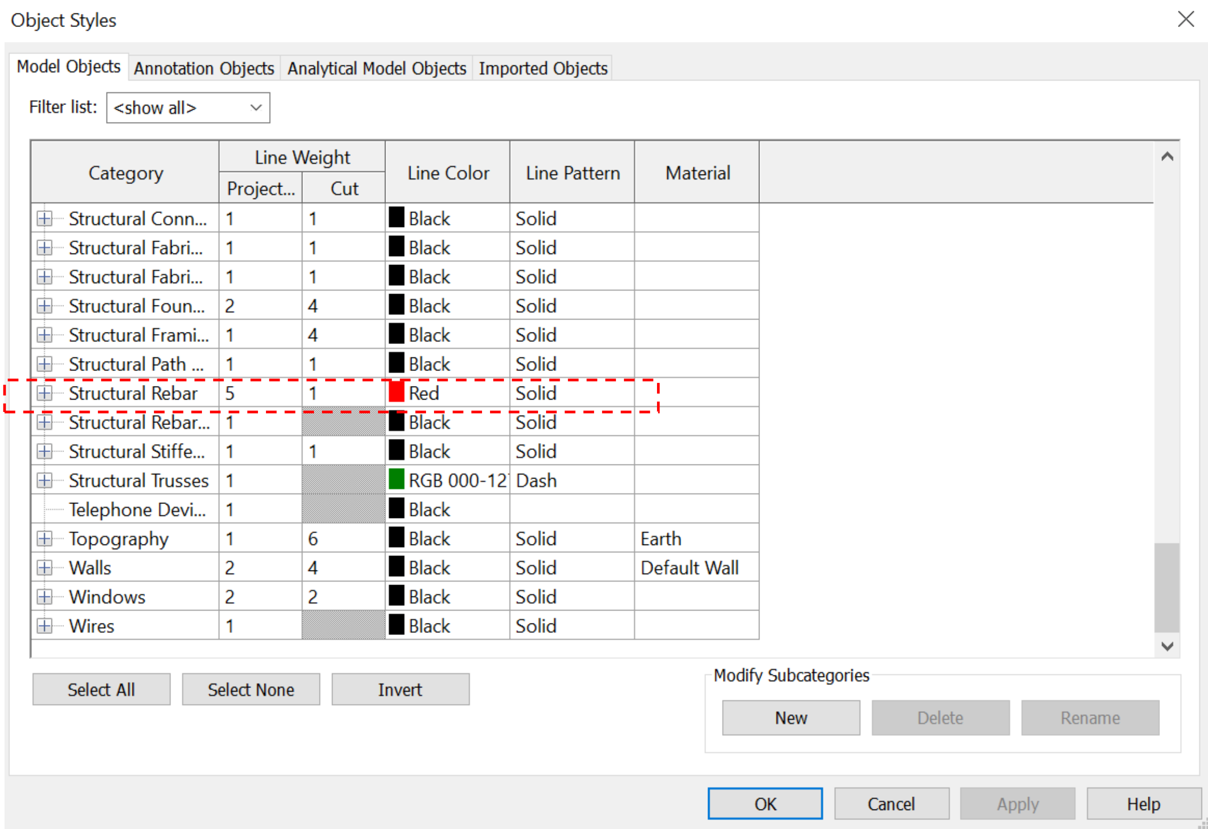




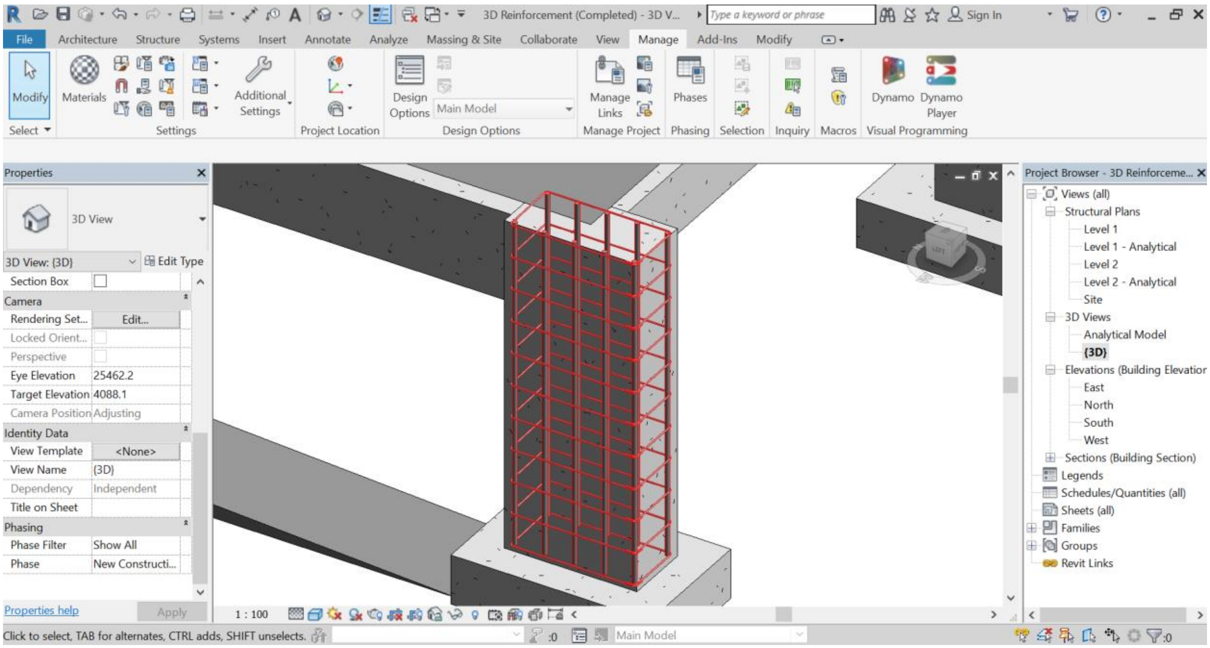
8 Go to 3D view and the rebars can be seen unobscured now.



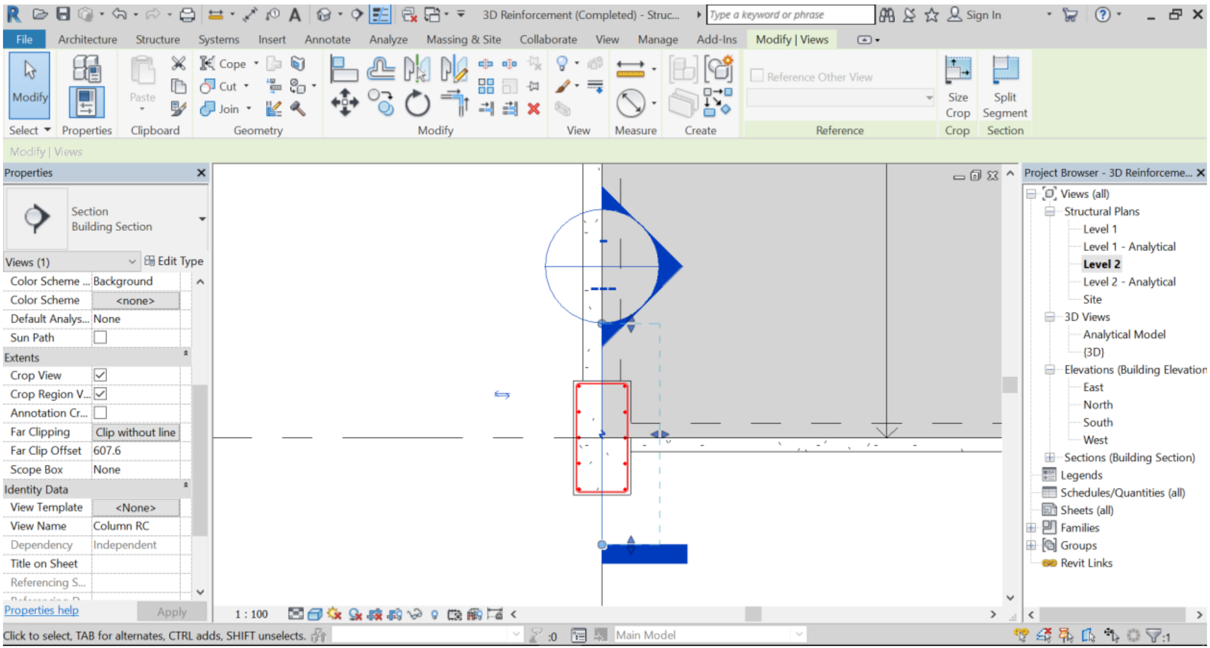
9 You can set the Structural Rebar Object Style and make it easier to view. (Click Manage Tab ➢ Settings panel ➢ Object Styles)



Category	Line Weight		Line Color	Line Pattern	Material
	Project...	Cut			
Structural Conn...	1	1	Black	Solid	
Structural Fabri...	1	1	Black	Solid	
Structural Fabri...	1	1	Black	Solid	
Structural Foun...	2	4	Black	Solid	
Structural Frami...	1	4	Black	Solid	
Structural Path ...	1	1	Black	Solid	
Structural Rebar	5	1	Red	Solid	
Structural Rebar...	1		Black	Solid	
Structural Stiffe...	1	1	Black	Solid	
Structural Trusses	1		RGB 000-12	Dash	
Telephone Devi...	1		Black		
Topography	1	6	Black	Solid	Earth
Walls	2	4	Black	Solid	Default Wall
Windows	2	2	Black	Solid	
Wires	1		Black	Solid	



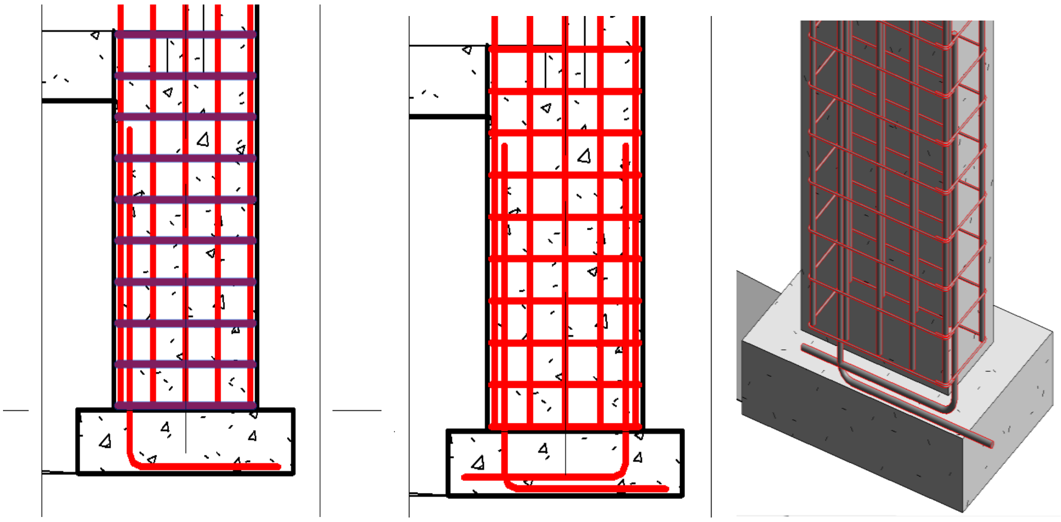
10 Next, place starter rebars. First cut a (building) Section of the column and then go to this section view.



11 Click Structure Tab ➢ Reinforcement panel ➢ Rebar. Select

- Rebar Shape: 00
- Placement Methods: Sketch Rebar and select the footing
- 40R rebar
- Placement Plane: Near Cover Reference
- Placement Orientation: Parallel to Cover
- Rebar Set: (Layout: Maximum Spacing and Spacing: 300mm)

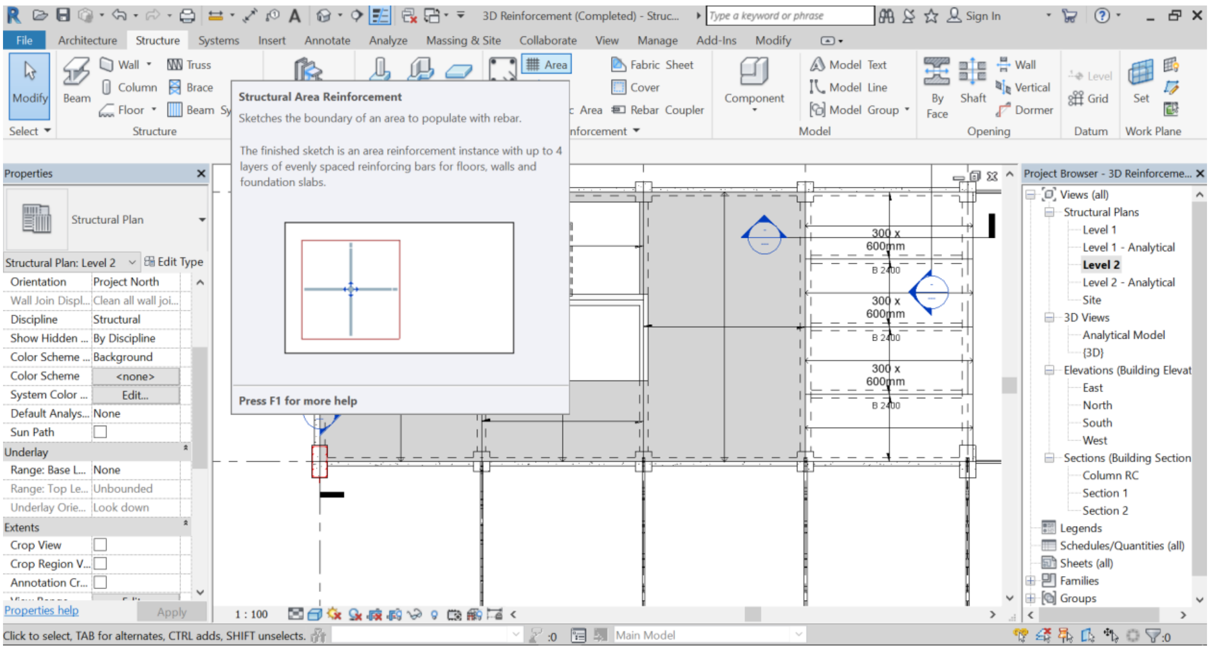
Sketch the starter rebar as shown below. Then mirror this starter rebar to the opposite face and move it slightly upward.



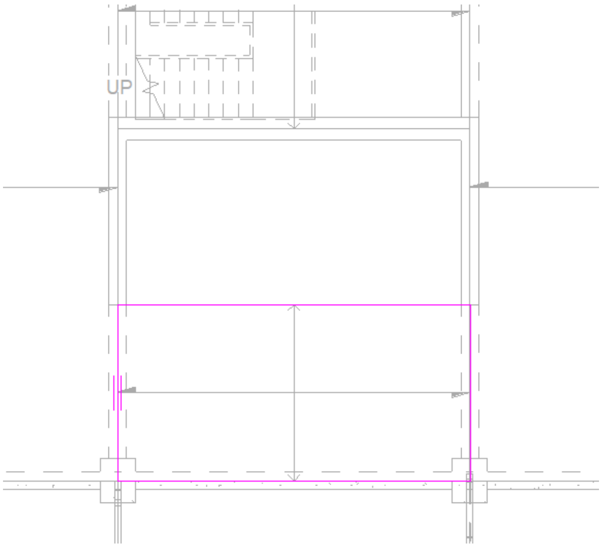
11.2 Placing Area Reinforcement for Floors (Slabs)

1 Go to Level 2 (Structural Plan).

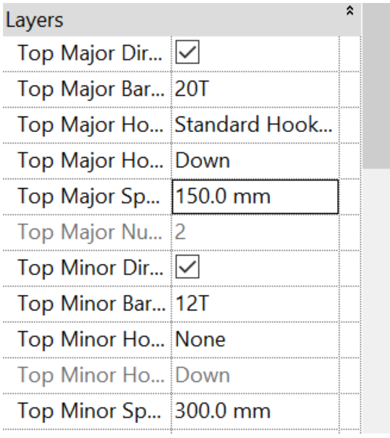
2 Click Structure Tab ➢ Reinforcement panel ➢ Area.



3 Select the slab below the corewalls on plan. Then sketch the area of the reinforcement on plan using Rectangle as shown below.

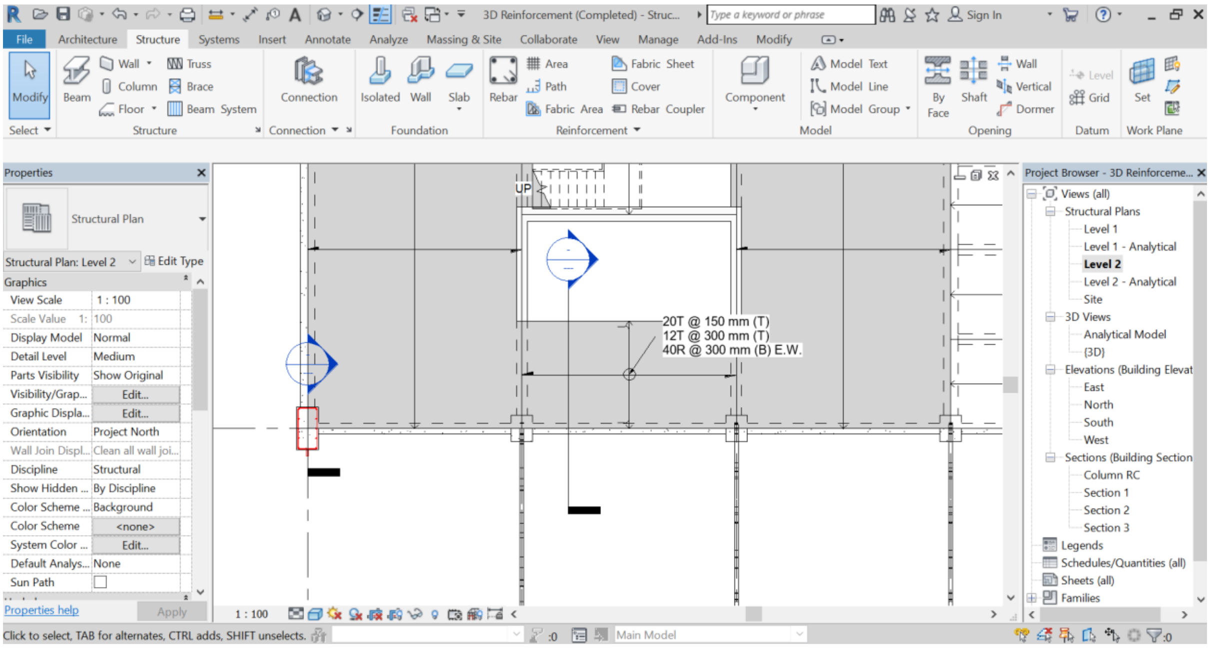


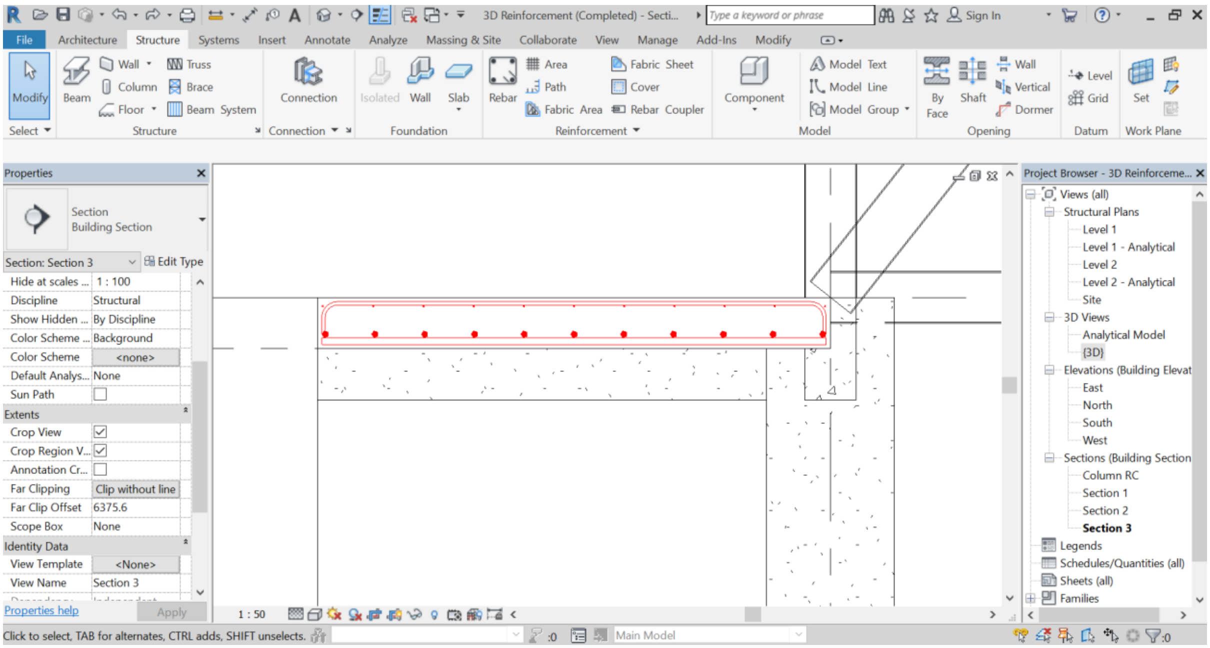
4 Change the area reinforcement properties as follow.



5 Click Finish Edit Mode ✔ to add the area reinforcement.

6 Create a (building) Section across the slab.

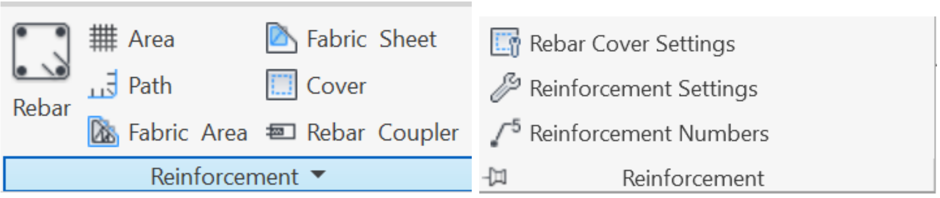




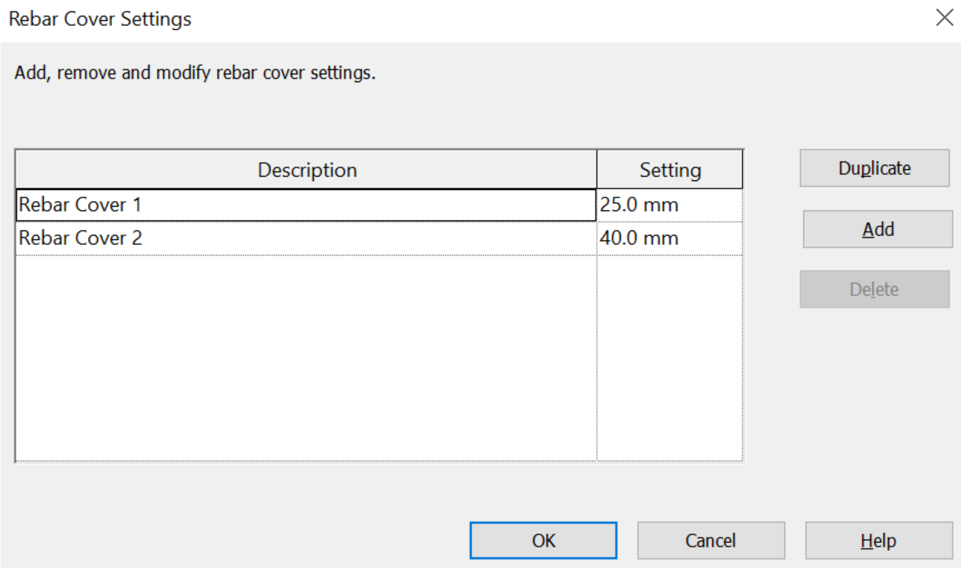
11.3 Reinforcement Settings

There are many parameters to control the reinforcement in Revit. To check/revise the setting,

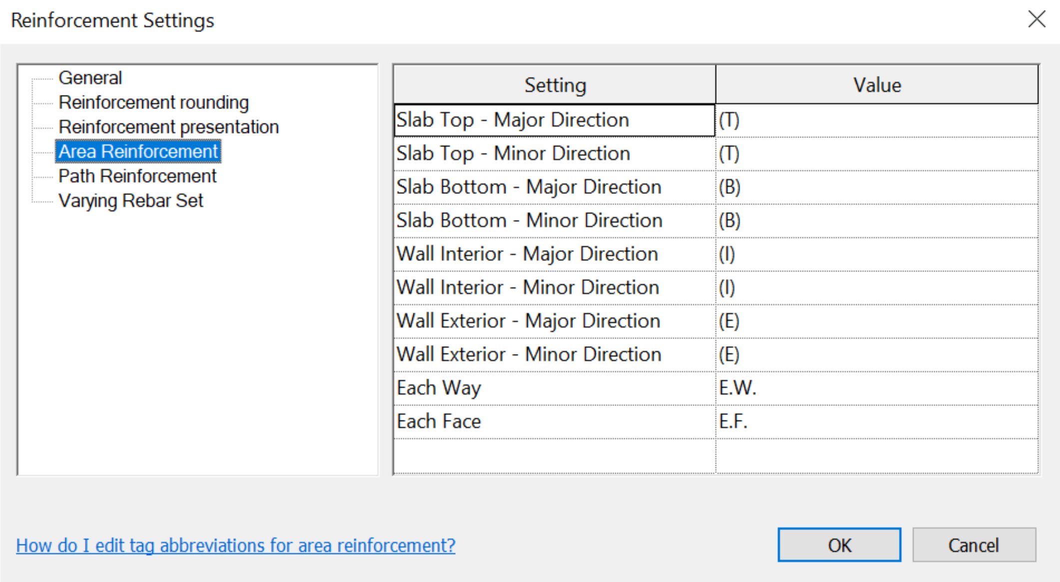
Click Structure Tab ➢ Reinforcement panel



- Rebar Cover Settings



- Reinforcement Settings (e.g.



- Reinforcement Numbering

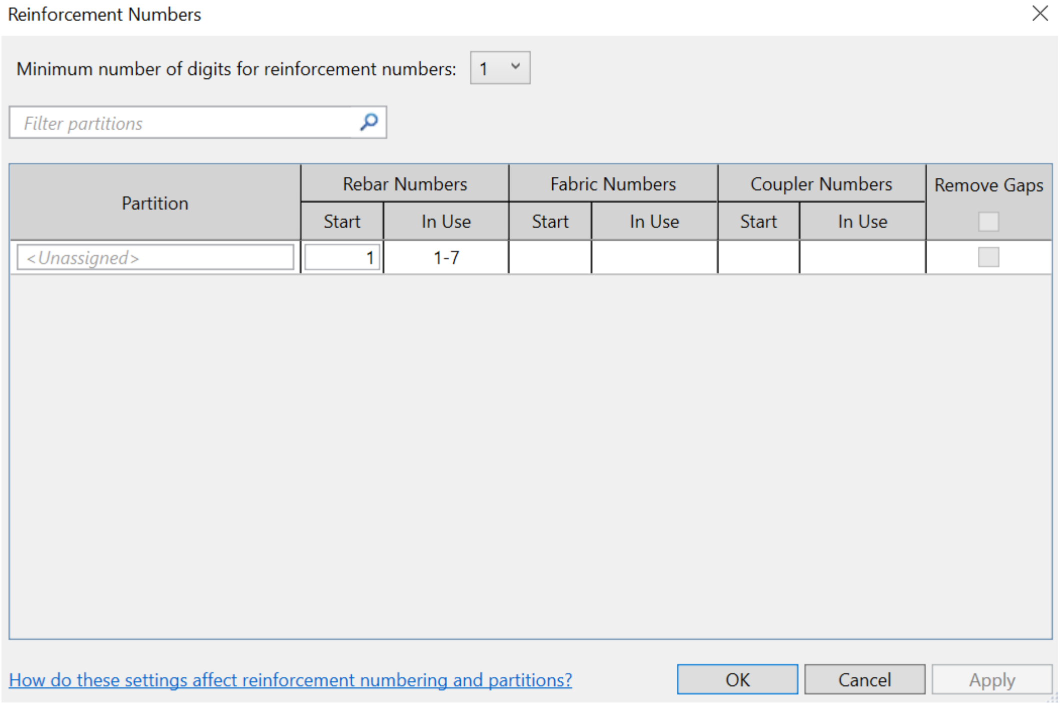


Table of Contents

CHAPTER 12 - WORKING WITH FAMILIES

One of the advantages in using Revit is its ability to create and customize families for users to build their own parametric objects to be used in a project. There are many aspects in families and once you master them, you can model almost anything parametrically.

In this section, you will learn

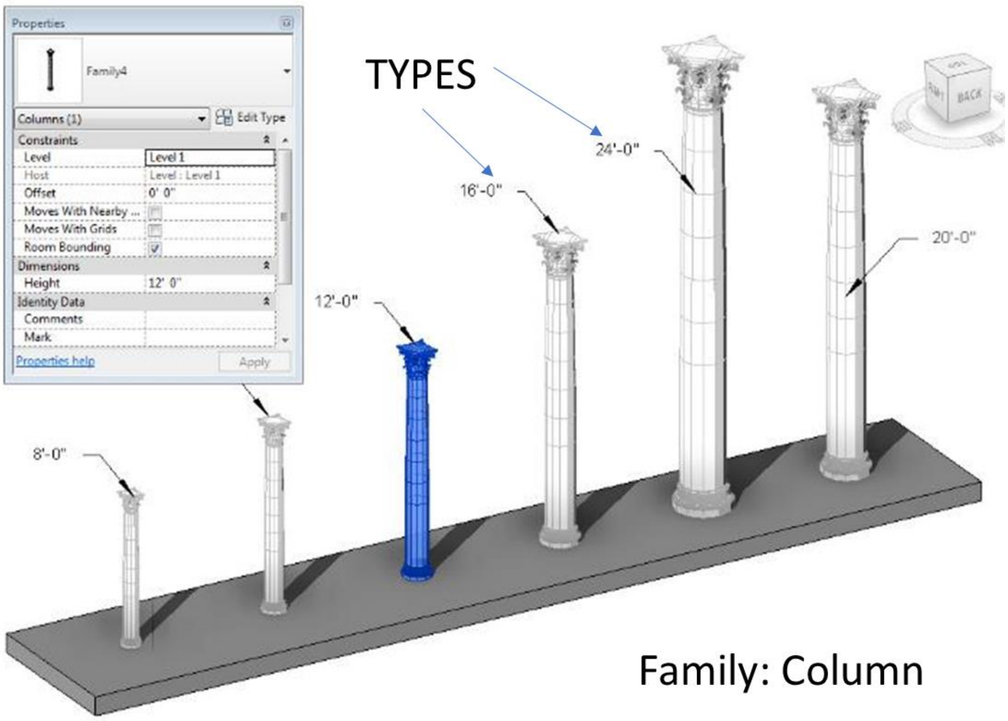
- The concept of "family" and "parameter" in Revit.
- How to create family elements with parametric dimensions.
- how to create parametric frameworks with constraints and parametric dimensions

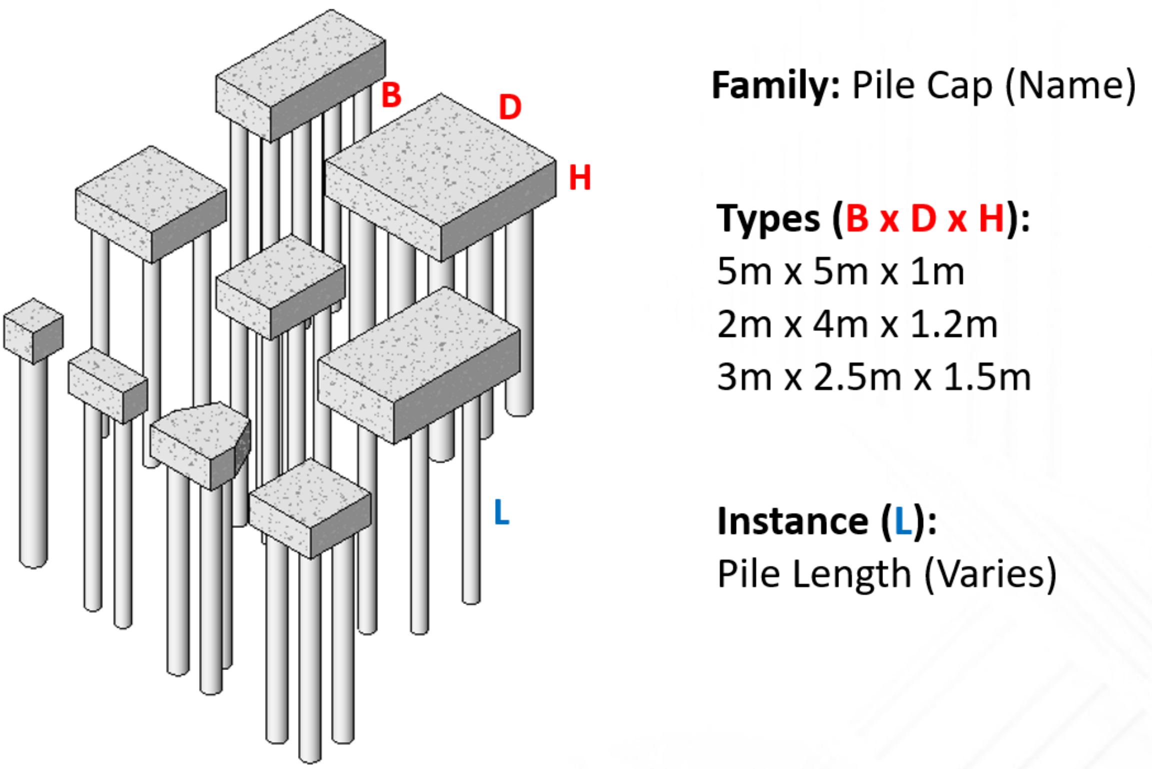
12.1 Introduction to Families

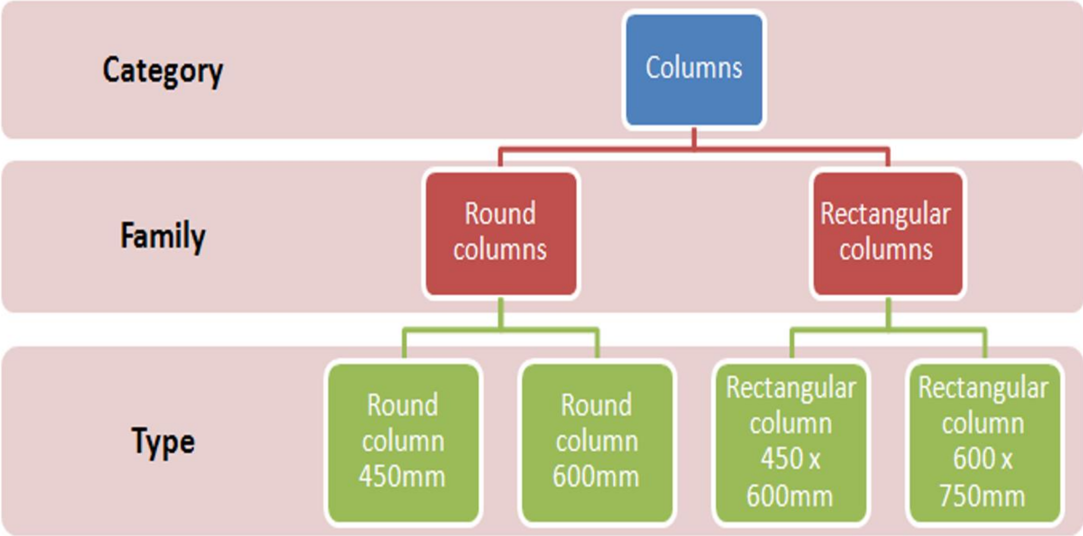
Family in Revit is a group of components that shares a set of common properties, parameters, and graphical representation. Components in a family can have different values for each property and parameter. Various components in architecture, structure, and MEP can be made as families. It is like "block" in AutoCAD that these two are preset symbol for certain components. The differences are that families are objects with properties and generally drawn in 3-dimensional way with parameters to allow user to adjust detail features of each component.

Image from http://www.revit-content.com/content/window/

Generally, Revit elements have a "Family Category > Family > Type > Instance" hierarchy. For a window family depicted above, the structure is "Windows > Casement Windows > 0610 x 1220mm". Categories control the overall organization, visibility, graphical representations, and scheduling options of Families within a project. Families serve to represent a discrete building or documentation elements in a project. It defines parametric, graphical, and documentation requirements. Types are defined by distinct parametric, graphical, and documentation characteristics which makes it unique from other types in a family. An Instance is an individual representation of a type. There are 3 kinds of families in Revit.







12.1.1 System Families

System families are very basic components in architectural, structural, and MEP representation. They can be created in a project with basic functions. In Revit MEP, pipes, ducts, cable trays, and conduits are system families. The shape (more specifically, the section profile) of these families cannot be changed, but the dimensions can be controlled. They cannot be loaded, but they can host some of "loadable families". They are stored within a project and pre-set and can be transferred to another project using Transfer Project Standards.

12.1.2 Standard Component (Loadable) Families

Standard component families can be created and edited in the family editor. Loadable families include diffusers, dampers, valves, tags, and so on. They are saved in separate files in ".rfa" files (not in project file with ".rvt" extension) and they can be loaded to projects for the use. Some of the parameter values are referred from lookup tables, type catalogues.

12.1.3 In-Place Families

In-place families are project specific families with custom shapes and parameters. In one project, multiple in-place families can be used and each of them can be copied. Unlike system family or loadable family, multiple types cannot be created. They are also transferable to other projects.

12.1.4 Family Types

Family types are included in system families and loadable families. Objects under the same type share type parameters and properties. Family instances are individual objects under a certain type in a family. Instances have instance parameters and properties. The image below shows various types of "Casement Windows" and objects under "0915 x 1200mm" type with different "Sill Height" instance parameter value.

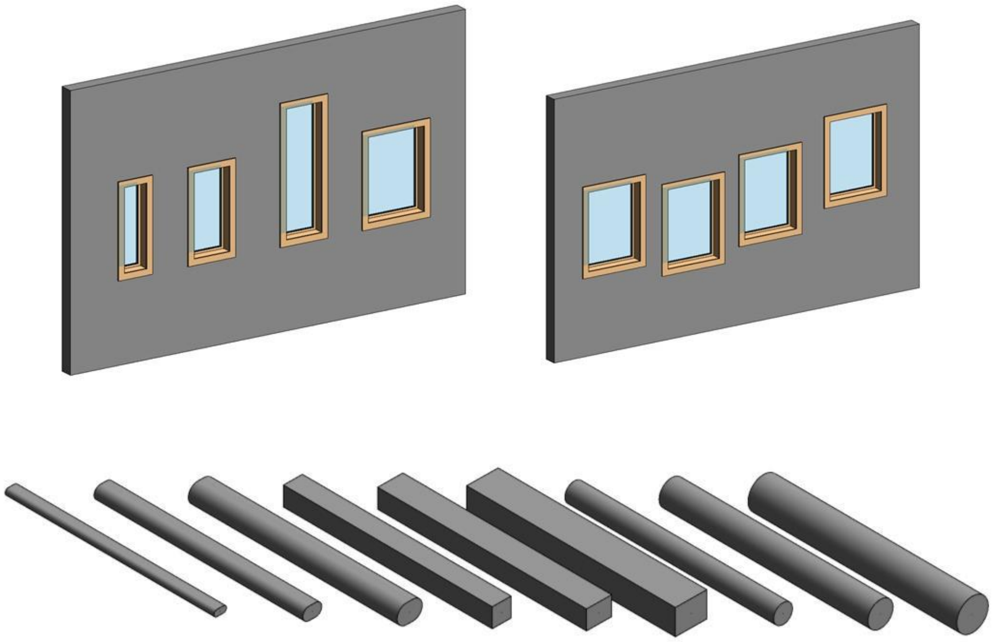


Figure 1 Various Revit components in families

From "Type Properties", you can find Type parameters, and from "Properties" palette, you can control the parameter of each family instance. The below is "0915 x 1220mm" type properties an instance Properties of a window object.

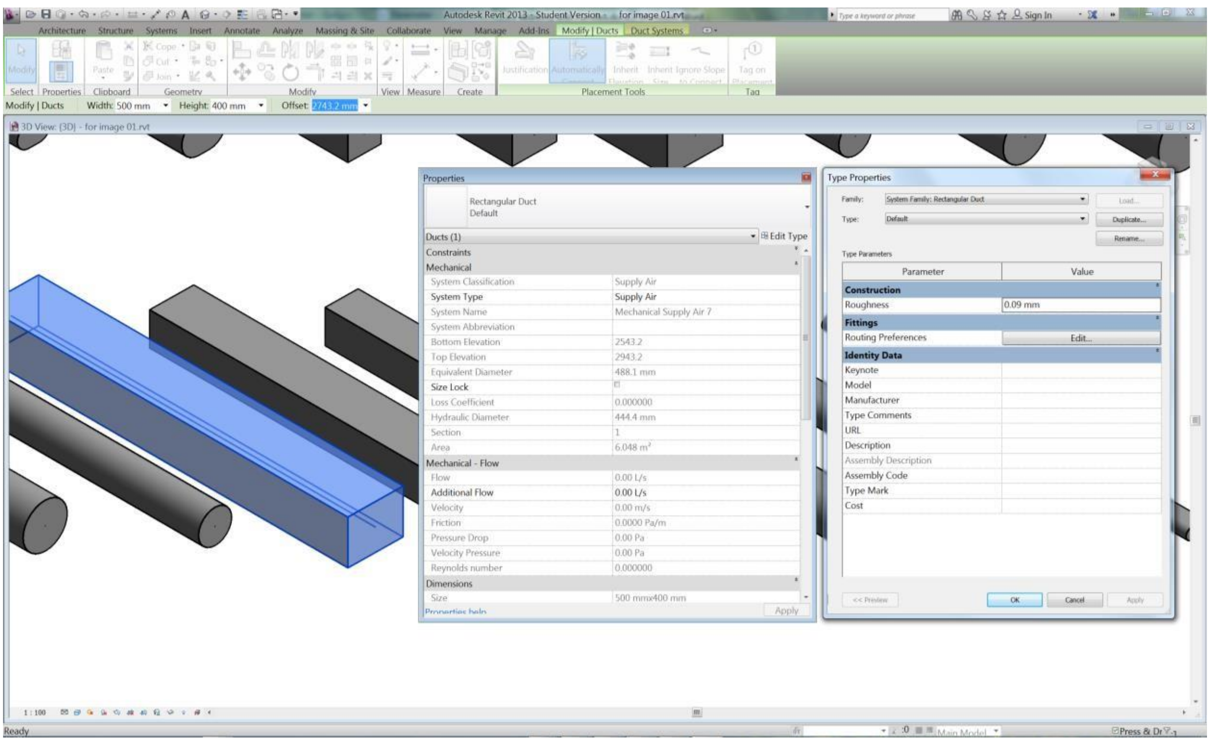


Figure 2 Family Parameters in an instance's properties and type properties

12.1.5 Parameters

In Revit, parameters are values that control the geometry or properties of model components. In a model component, relationships between parameters are defined that users can easily have desired results by changing parameters. For successful implementation of Revit in a project, it is very important to understand how parameters behave. In Revit, there are 4 kinds of parameters.

Family Parameters

Family parameters control properties and graphical representations of families. They can be added and edited only in the family editor. In the project editor, parameter values can be changed.

Project Parameters

This type of parameters is used in a project file. It is mainly used for schedules, i.e., to add new field to schedules or calculations in schedule.

Shared Parameters

Shared parameters can be used in families, projects, tags, filters, and schedules. They are applicable to all kinds of categories. You can add your own parameters that are not provided by family templates or project templates. They are stored in separate text files and loadable from other projects or families.

System Parameters

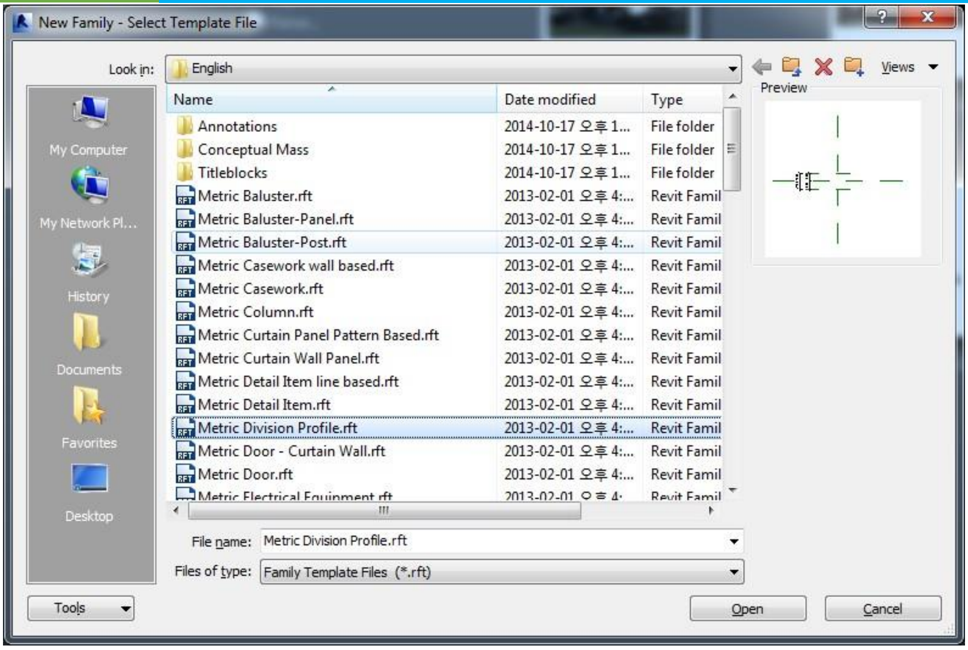
System parameters are defined inside of every family. Most of them can be used for schedules and Tags.

12.1.6 How to add loadable family libraries to Revit

Family libraries are not installed with Revit program by default. To have Autodesk's family libraries, additional features need to be installed from control panel. (Control Panel > Control Panel Items > programs and Features > Autodesk Revit 2017). Select "Add or Remove Features", from "Select Content", check "English – US Metric". With default setting, libraries will be installed at "C:\ProgramData\Autodesk\RVT 2017\Libraries\US Metric

12.1.7 Family templates

Family templates are preset canvas for family creation. They define how the family objects behave. To create a loadable family, template is essentially required.



Default path for Family Templates is C:\ProgramData\Autodesk\RVT 2017\Family Templates \English.

Major family templates	
Wall-based Template	Template for wall mounted family objects. With "opening" component, doors and windows can be created. In Revit MEP, lightings and fire alarm families use this kind of template.
Ceiling-based Template	Template for ceiling mounted family objects. This can be used to create ceiling mounted lightings, sprinklers, and diffusers.
Floor-based template	Template for floor mounted family object. In Revit MEP, heating register families can be created with "opening" component.
Roof-based Template	Template for roof mounted family objects.
Standalone Template	Template for standalone family objects. The family objects do not require host, they can be placed anywhere in a project model. (e. g., furniture, duct fittings)
Line-based Template	Template for family objects along lines
Face-based Template	For reference plane mounted family objects. The host of the family objects is changeable and could be placed in any kind of faces.

12.1.8 Reference plane

Understanding the concept of reference plane is fundamental in creation of Revit family and parametric frameworks. It is very important for users to understand how it works to implement Revit it a successful way. It is basically guide plane that your model object can refer to in order to establish parametric frames works for family objects. There are 3 main use of reference plane.

1. Users can draw base sketches on it to create extrusions and other "formworks" in Revit.

2. Faces of models can be aligned to reference planes to create constraints.
3. In between reference planes, parametric dimensions can be added to create parametric framework of a model.

While creating parameters and constrains, test the model as many as you can. This will make sure you are creating parameters and constraints as you intend, it is also helpful to use least number of views for constrains for the future edit and use of the model.

12.2 How to create a family?

9 Steps to create a family

This is supposed to give you a clearer picture of what we are about to cover. The order can be changed, and some steps can be skipped when you become familiar with family creating & editing process.

1. Design what you want kind family to you want to create. More specifically, you need to know what are required specifications, parameters, category, and shape of your family. You can do some sketch if you need.
2. Select template file.

   Template defines types and hosts for your family. More specifically, it determines your family is going to be either wall-mounted or ceiling-mounted. If you want to create annotation, or title instead of models, you have to choose right template. Once template is selected for family creation, this cannot be changed in later stages. In other words, a wall-mounted lighting family cannot be a spot lighting family.

3. Set category of your family.

   Base on the category you set for your family, the function will be determined. Some parameters will be added as well.

4. Create parameters of your family

   Create “type parameters” and “instance parameters” for your family. This can be edited in the process.

5. Arrange reference planes/Reference planes works as guide for the shape of your family. They are also used to set constrains.

6. Create components and set parameters to family

   Components can be solids for shapes, or texts for annotations. If you set dimension, this can be turned into parameters. Based on the function of each component, you should make it refer to certain database (catalogue, lookup tables) or constraints.

7. Test your family

   In the process, you should do as many tests as possible to make sure your family works as you designed.

8. Set graphical representation

   Set graphical representation of your family in 2D and 3D views.

9. Save and load into a project file

   Save your family in “rfa” format and load it into a project file. Here you need to test your family again to check if it works well with other families in the project.

You do not have to fully understand what we have covered so far. They will be picked up in the process of modeling exercises.

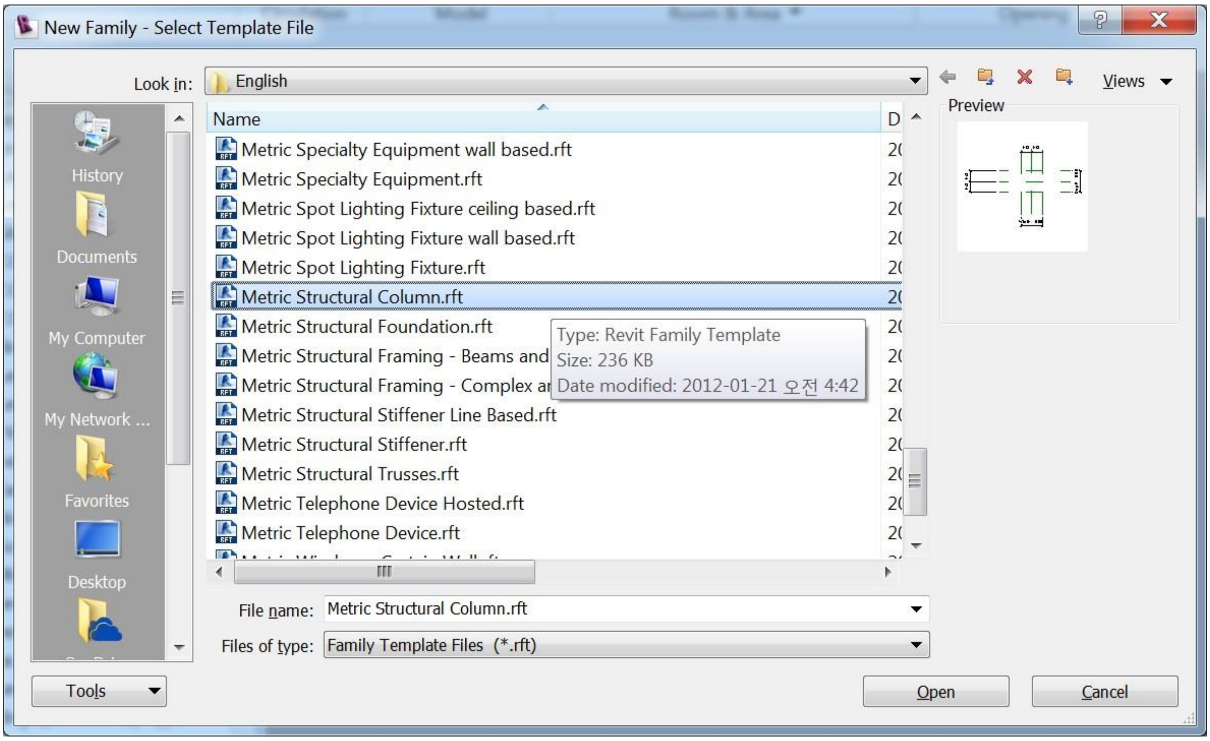
12.3 Overview of Family Creation with Structural Column Family

The best way to understand the concept and functionality of family editor is to create a family. In this tutorial, we will explore the functions in the family editor with a simple structural column example. Through the exercise, we will look ways to create family elements with parametric dimension, and how these things work together to create a parametric framework.

Family Editor is a tool to edit and create families. It has a very similar look to project editor. It provides functions to create forms and set properties for families. There are six taps; “Create”, “Insert”, “Detail”, “Modify”, “View” and “Manage”. The detail functions will be covered in modeling exercises.

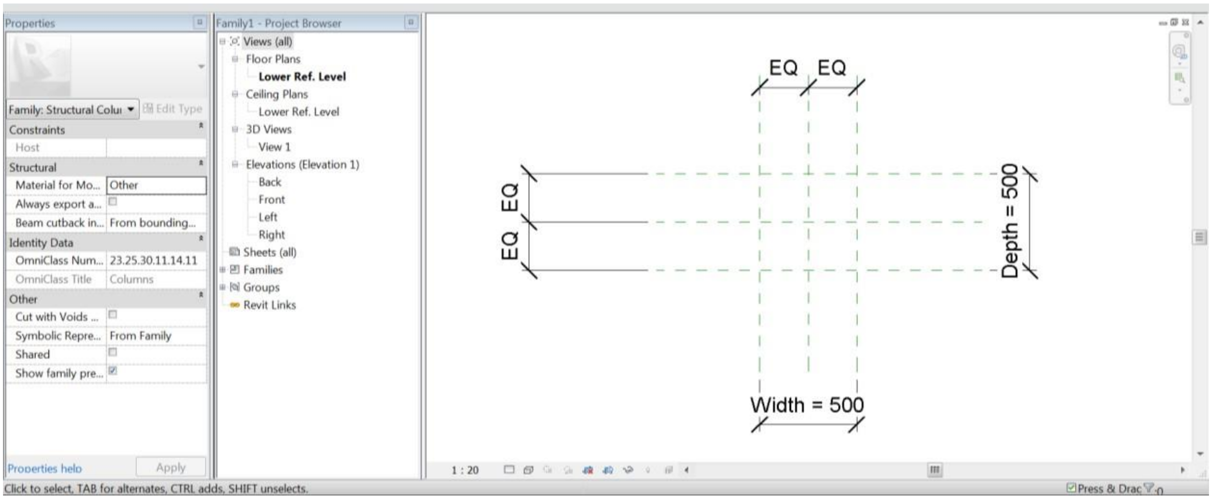
12.3.1 Template

Start your Revit software, and go to “File > New > Family”. Then you will see a window pops up with a list of templates. Family templates set basic features of families. In order to create a structural column family, select “Metric Structural Column.rft” file and click “Open” button. If your window does not show the folder, go to the path “C:\ProgramData\Autodesk\RVT 2017\Family Templates\English”.

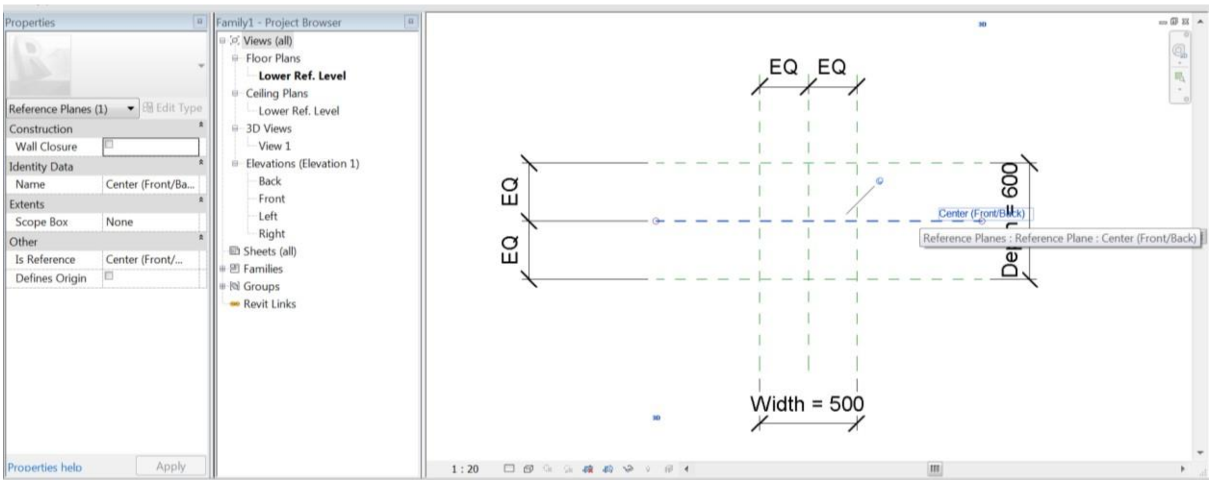


12.3.2 Parameter Set up

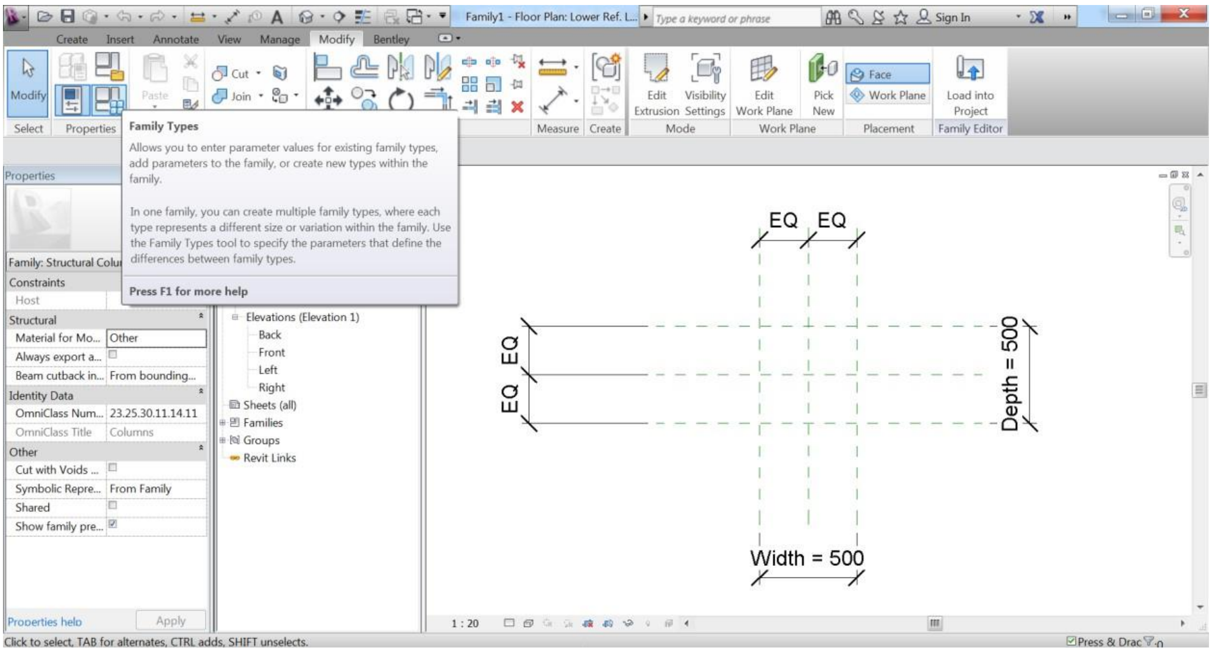
Once the family editor begins, you will see some dimensions and green dotted lines. The green dotted lines represent reference planes. The dimensions are associated with the reference planes to move the planes parametrically. This means that the three horizontal lines always have an equal distance in-between, and the total distance between the top one and the bottom one will be 500mm. The distance is controlled by a parameter called "Depth", which is currently 500mm. Three vertical reference planes are the same. These are preset features for your structural family, you will see different elements from different templates.



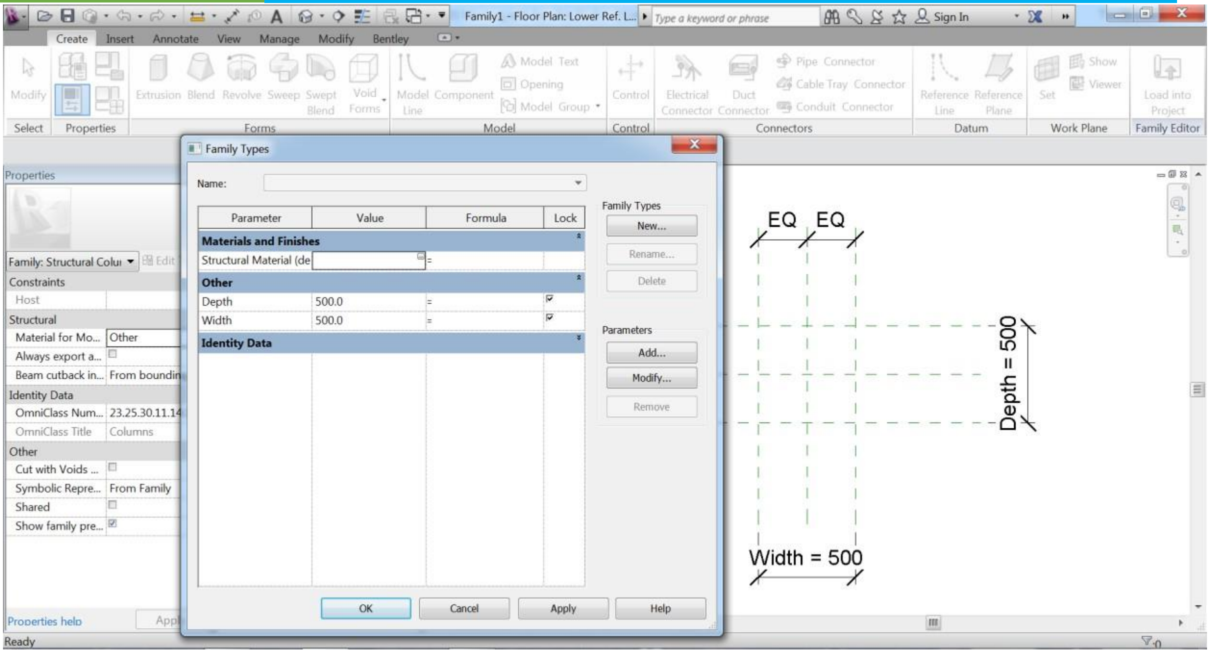
Select the horizontal reference plane in the middle. You can see that the reference plane is pinned. This means the reference plane will not move but not the two reference planes related by the dimensions.



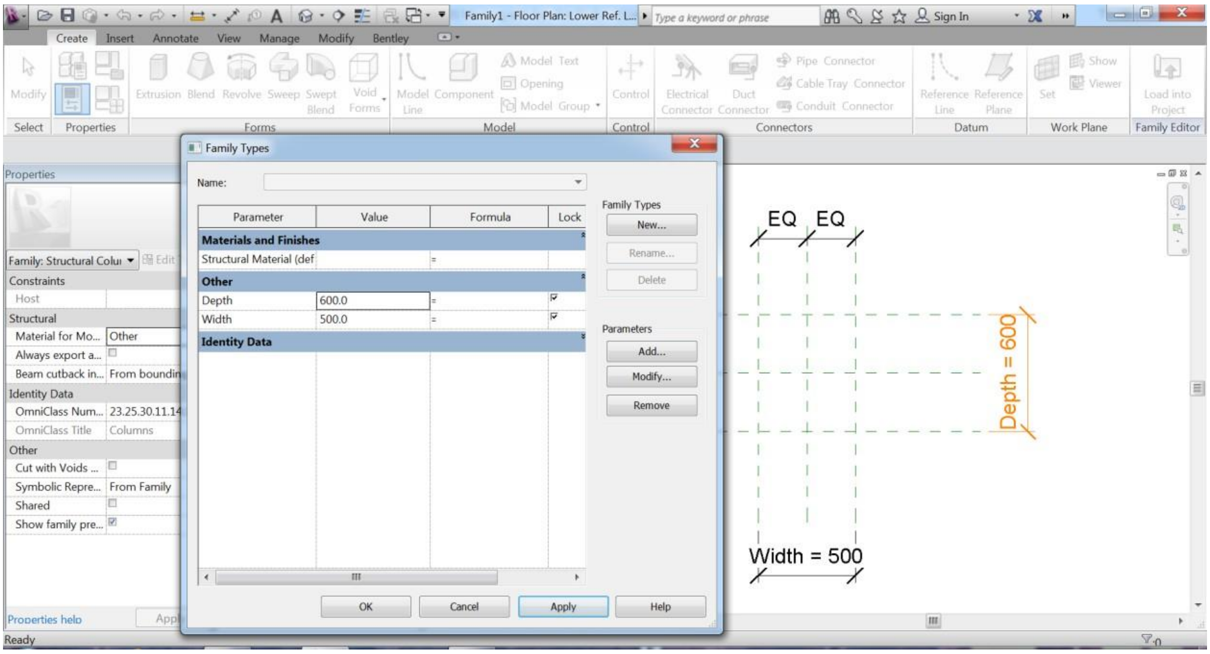
From the ribbon tab, go to “Properties” group and select “Family Types”



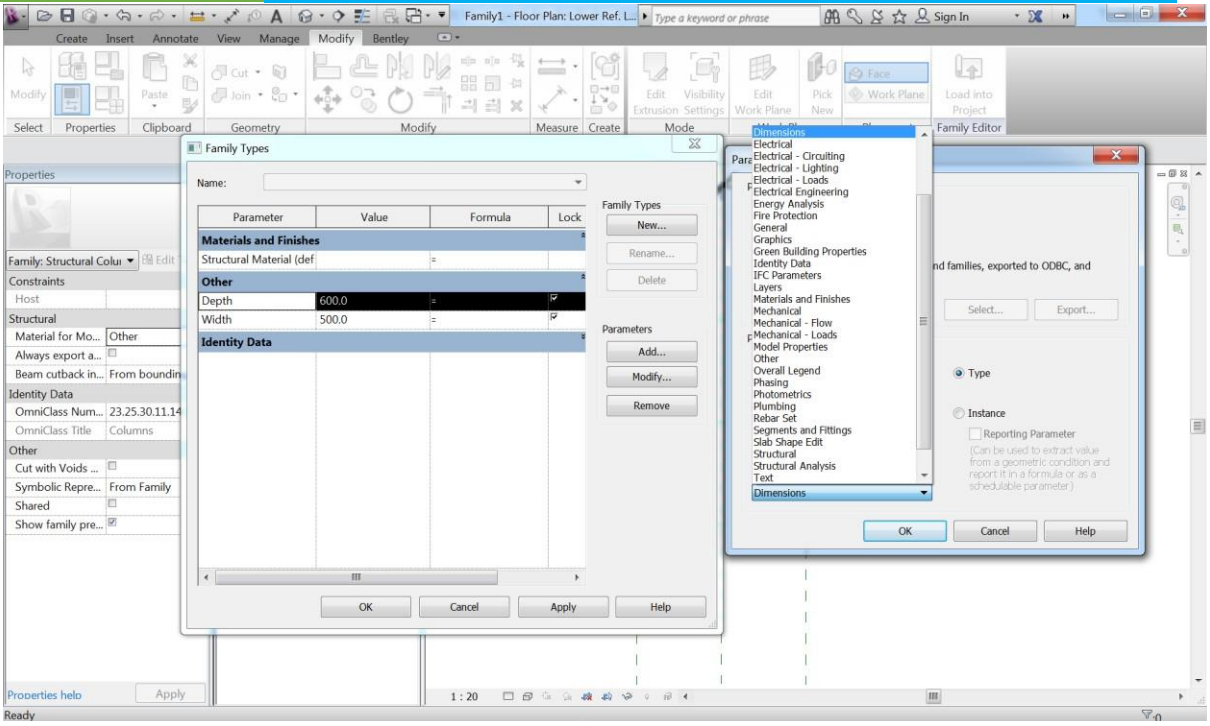
You can see that “Depth” and “Width” parameters are listed under “Other” Group.



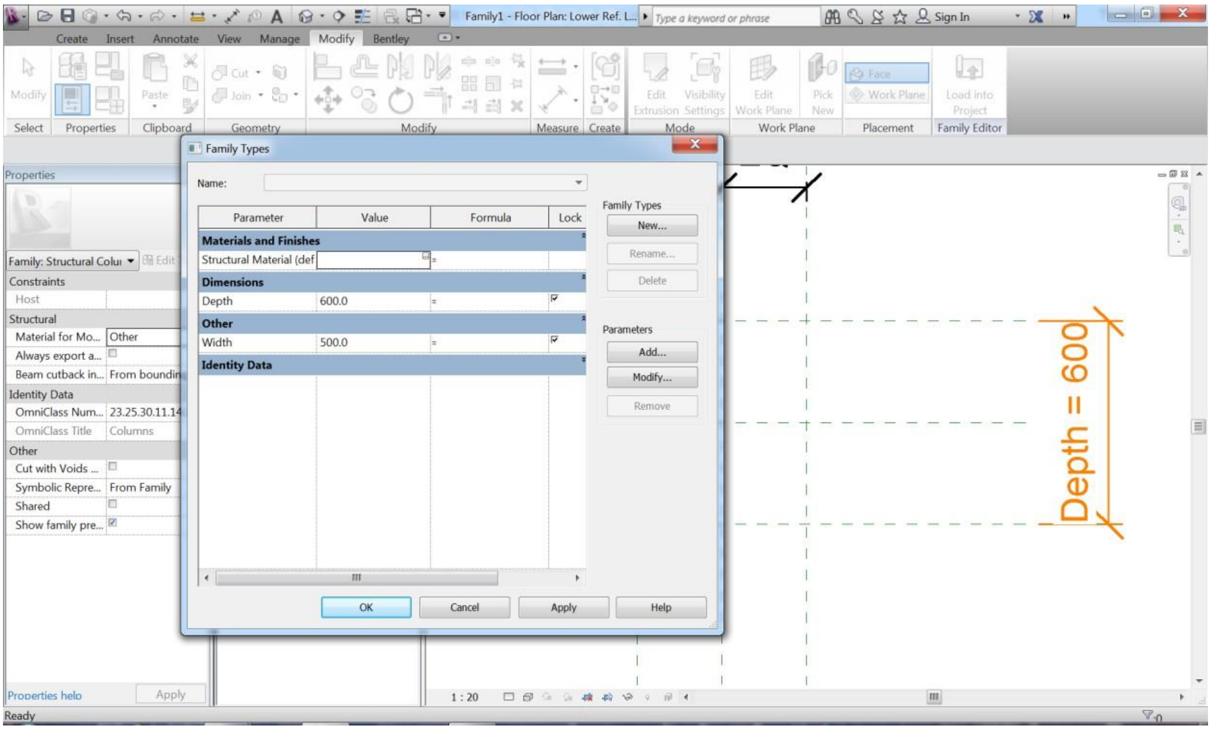
change the value of “Depth” parameter to “600”, or any other number. Then you can see that the dimension changes accordingly and so do the reference lines.



Now select the “Depth” parameter and click “Edit Parameter” button. “Parameter Properties” window will pop up. Change the “Other” under “Group parameter under:” to “Dimensions”.



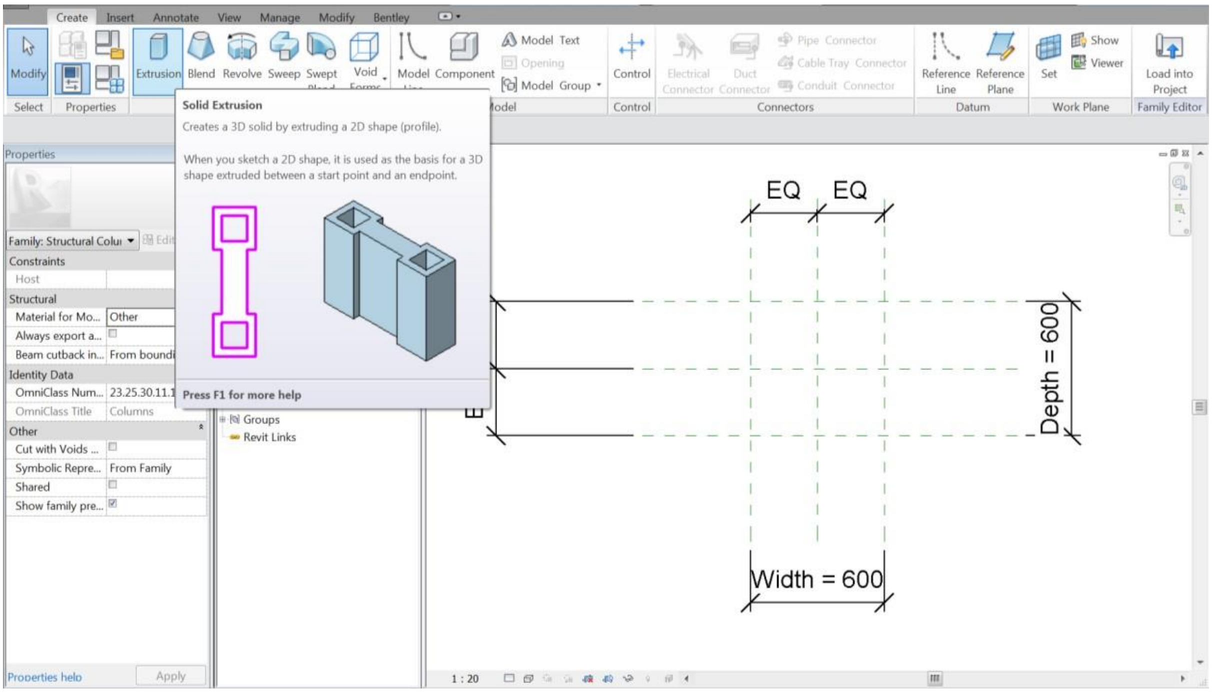
click “OK” button to close the “Parameter Properties” window. Now “Dimensions” group is added to the parameter list and “Depth” is under “Dimensions” now.



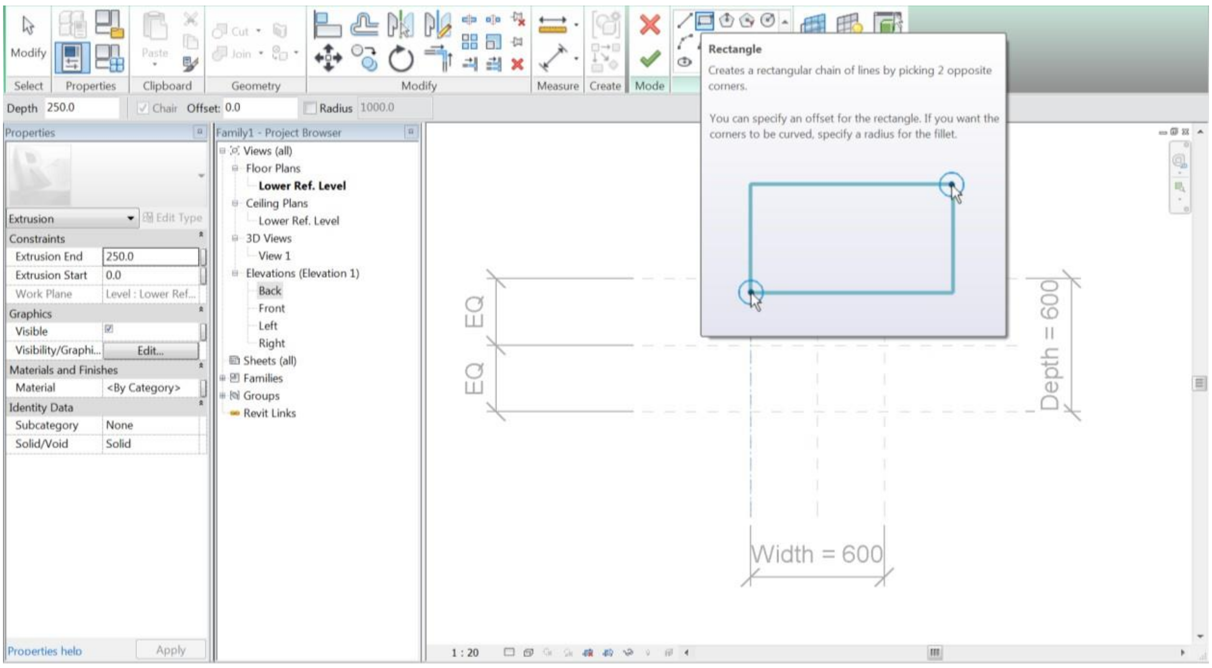
We are going to use a square section for the column family. So we do not need to control the “Depth” and “Width” parameter separately. Type in “Depth” to “Formula” of “Width” parameter. use the exact letters. This way you can associate parameters to each other. By changing one of the values, you can change the both parameters now.

12.3.3 Base Geometry

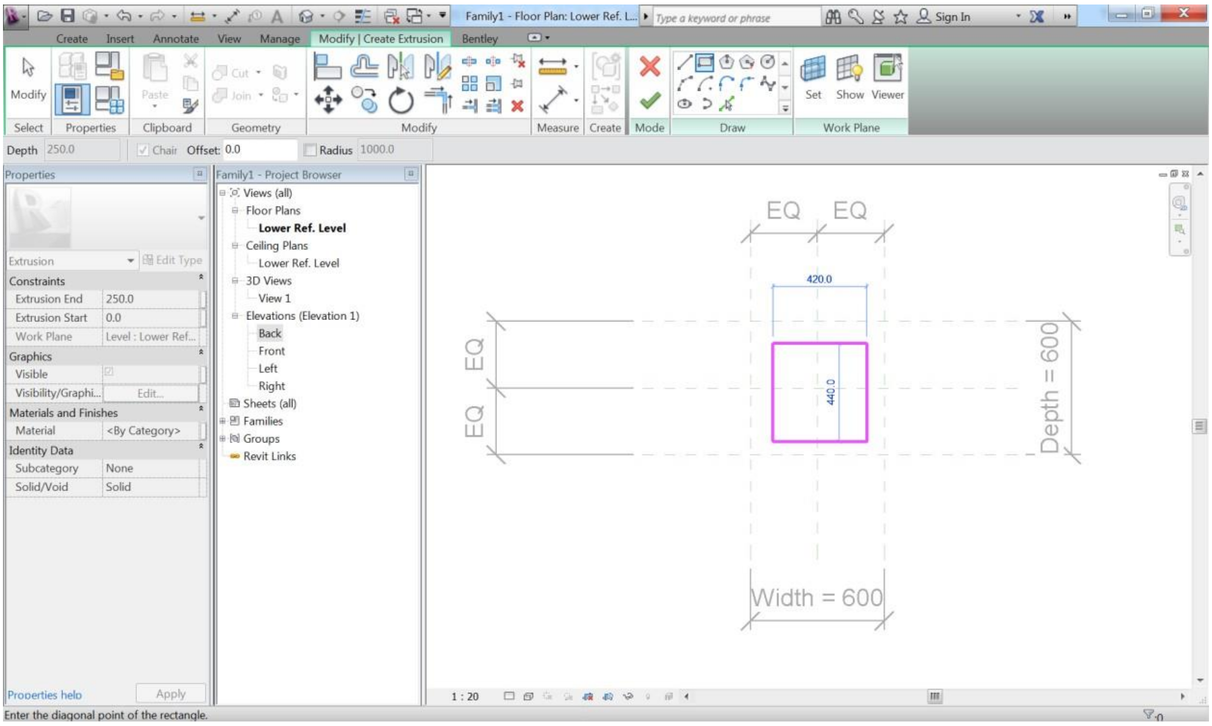
Now we are going to create a geometric object that is associated with the parameters we set. "Extrusion" is the most commonly used function to create a solid object. Go to "Create" tab and select "Extrusion" button from the ribbon tap.



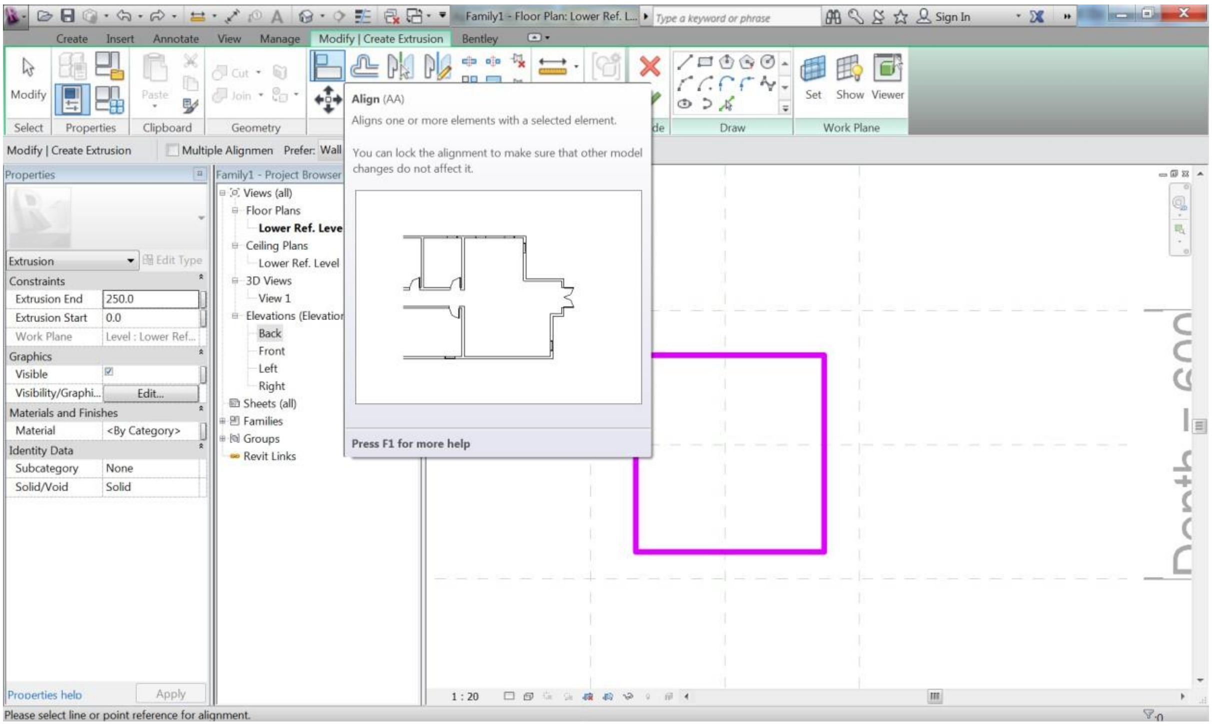
To create an extrusion, you need to create a two-dimensional sketch first. Once you click started the function, the ribbon tab will be automatically refreshed to show the sketch functions. Select "Rectangle" from the "Draw" menu.



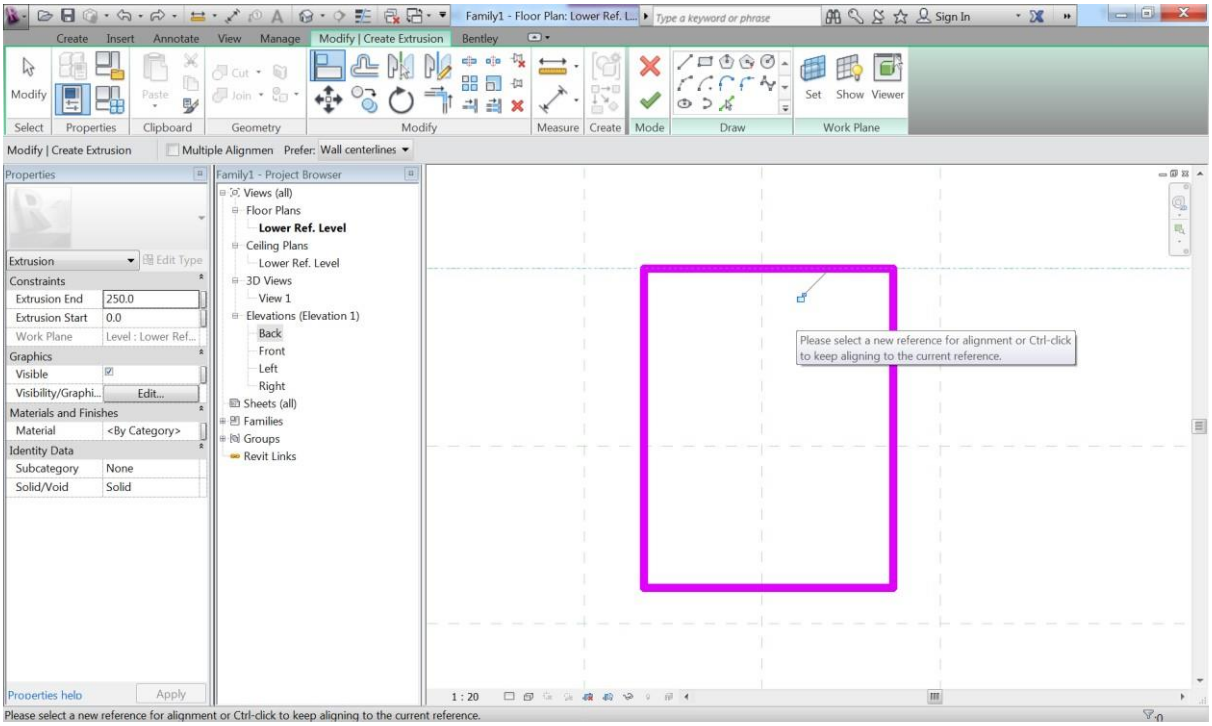
As the instruction suggests, designate the two corners of the rectangle and make sure that the two corners do not coincide with any of the reference planes. The size does not matter at the moment.



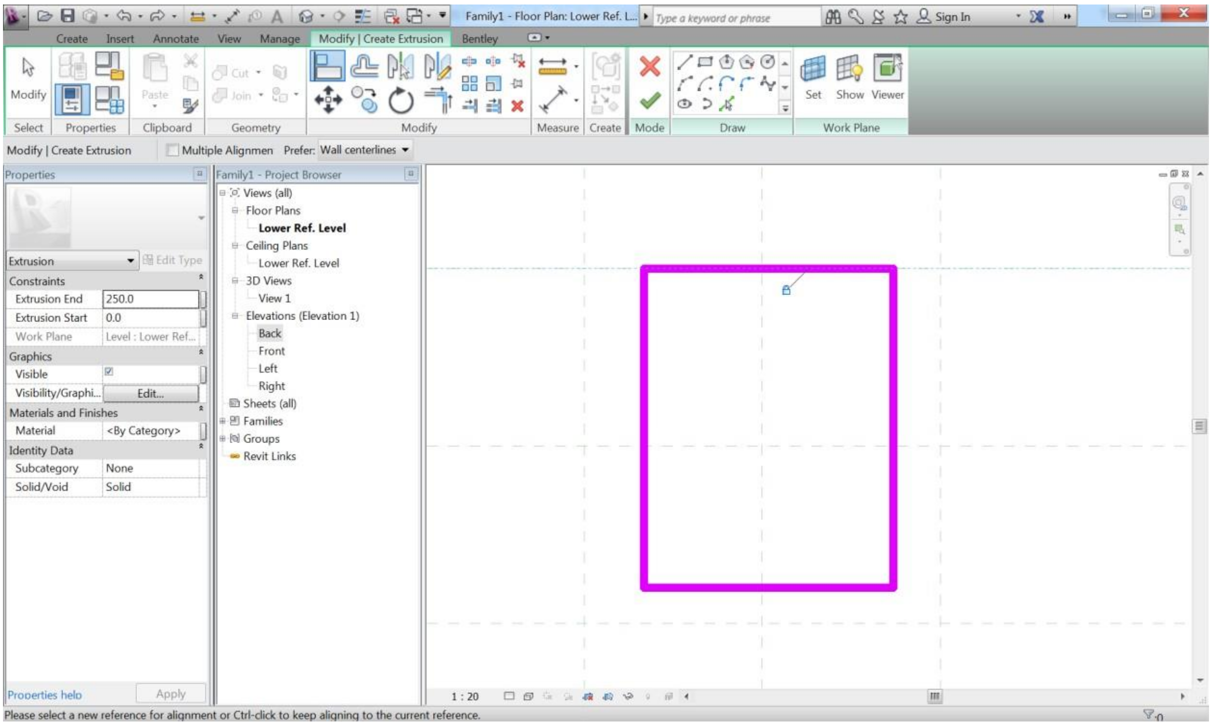
From the "Modify" tab, select "Align" function. As the name suggests, you can align two different object to each other.



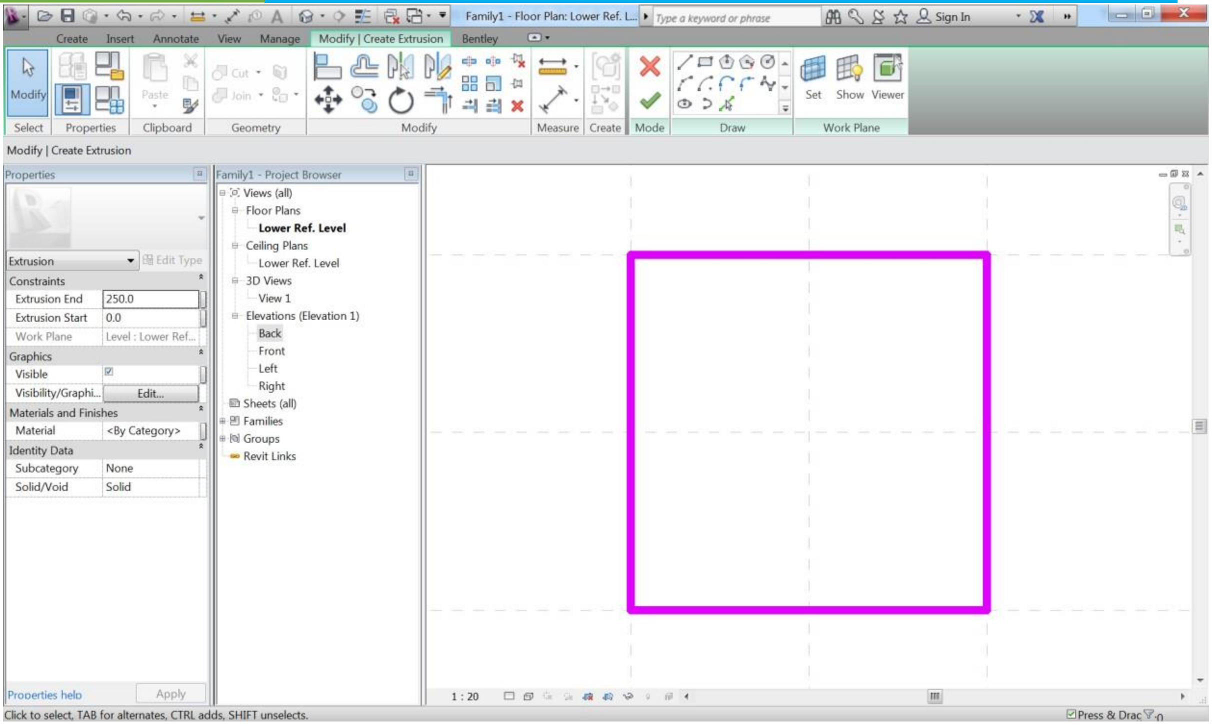
Select the top reference plane as the target. Then click the top edge of the rectangle. The edge will move and be aligned with the reference plane. A small lock will appear in the screen.



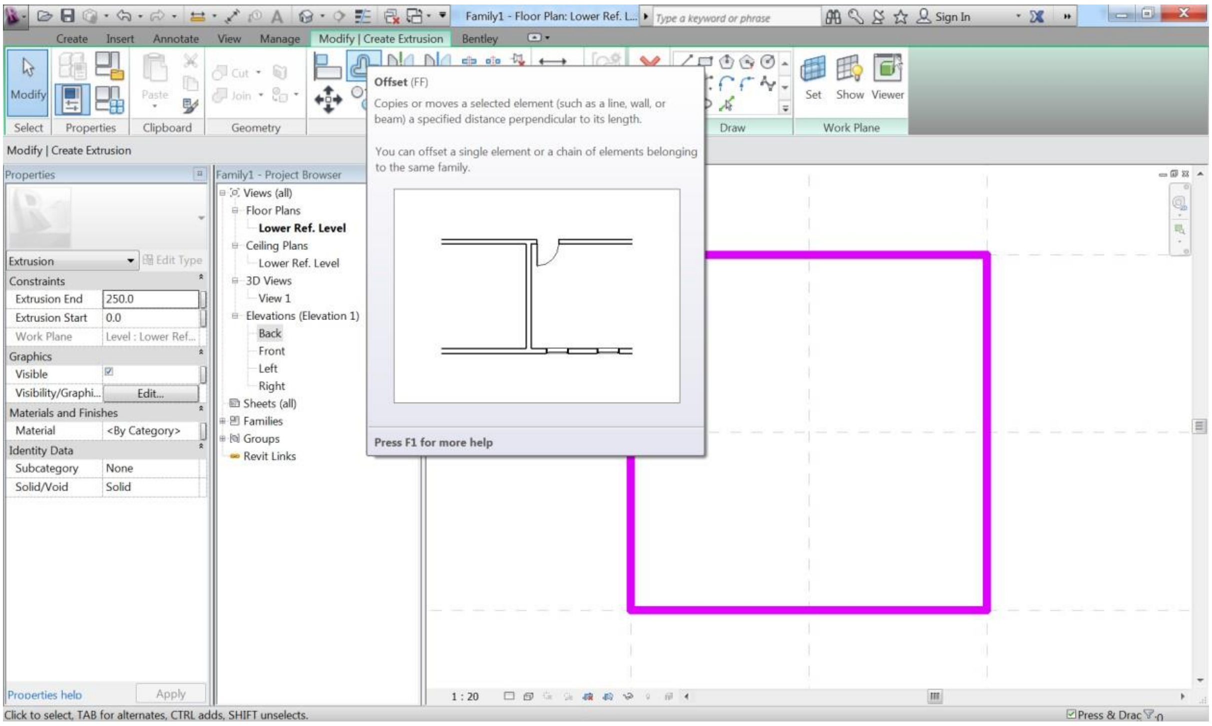
click the lock to close the lock. This means you just constrained your edge to the reference line and the line will move along with the reference line.



Repeat the same steps to other edges and lock up the lines so that all edges are constrained to the corresponding reference planes.

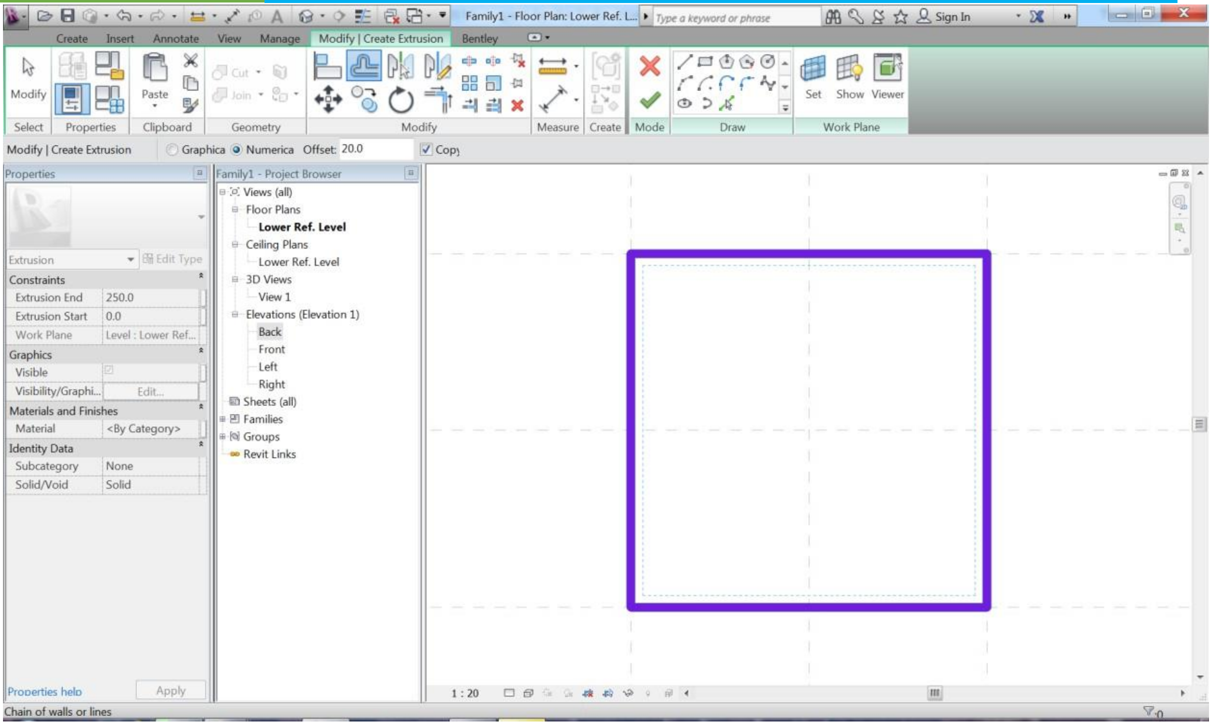


Now go back to "Modify" tab and select "Offset" function.

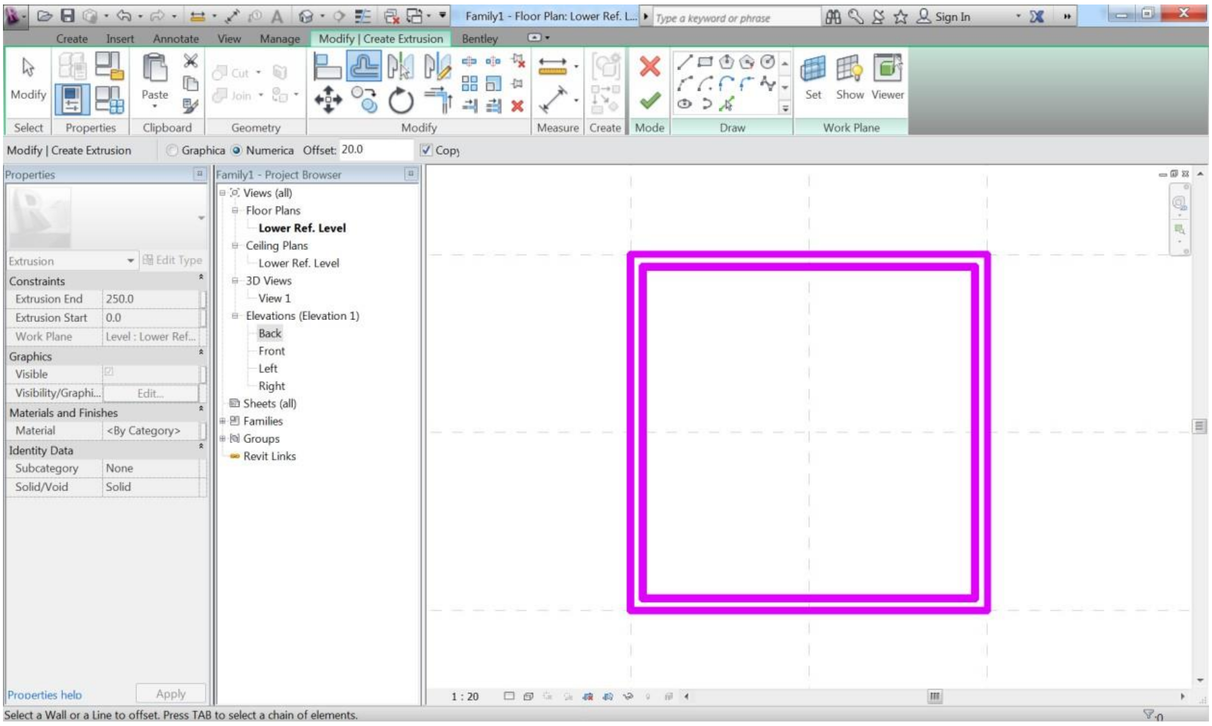


Set the "Offset" value to "20" from the option bar. Then click any empty spot in the canvas. Then bring your cursor over one of the edges and make sure a dotted line appears inside of the rectangle.

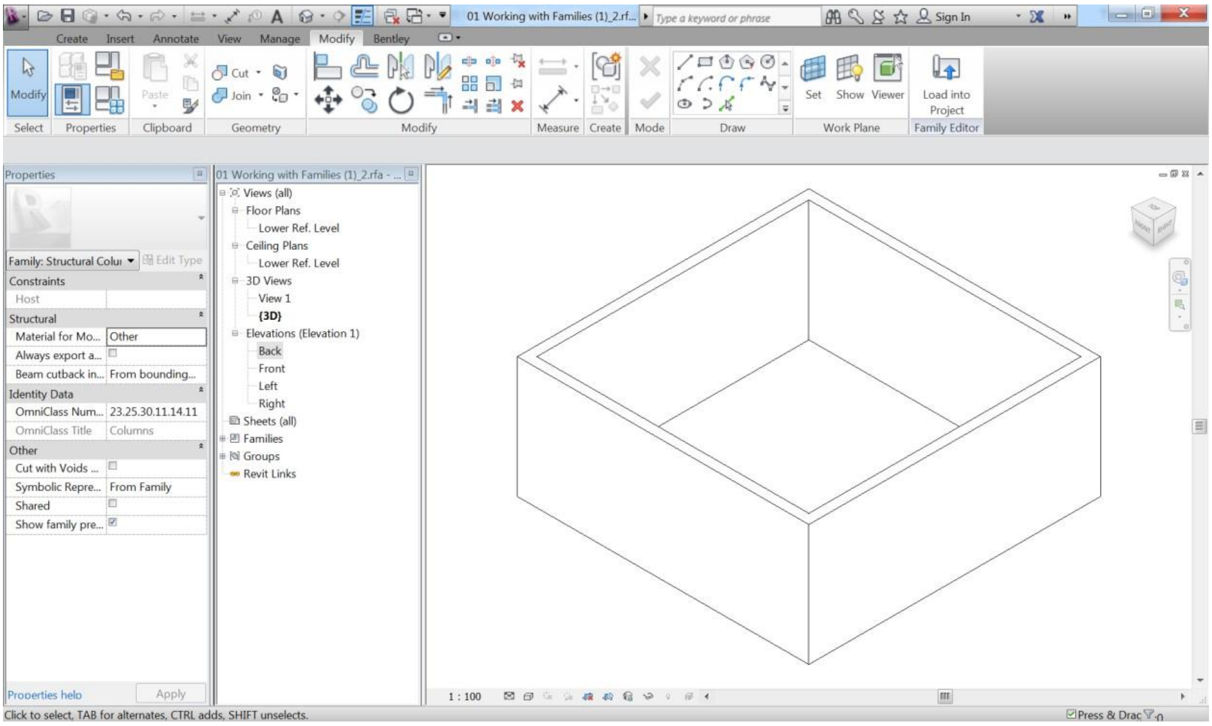
Hit "tab" button on your keyboard to offset all the edges together.



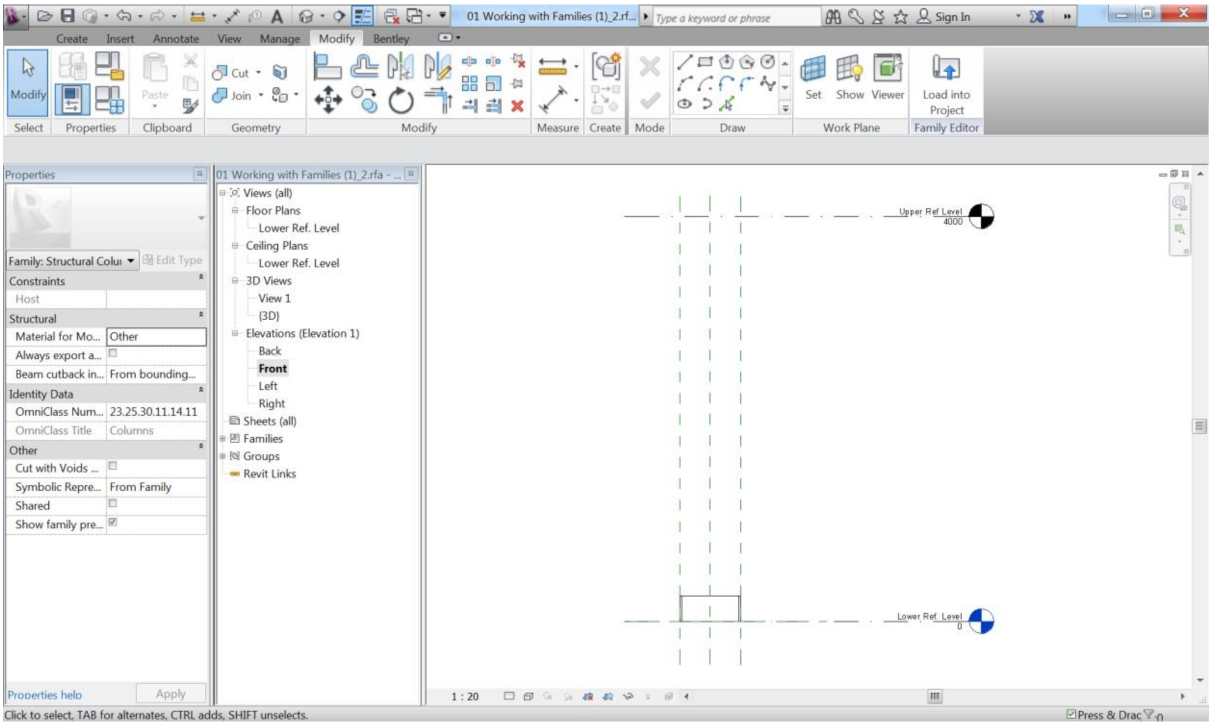
Once a dotted square appears inside the sketch, click the edge. Then the all lines will be offset together.



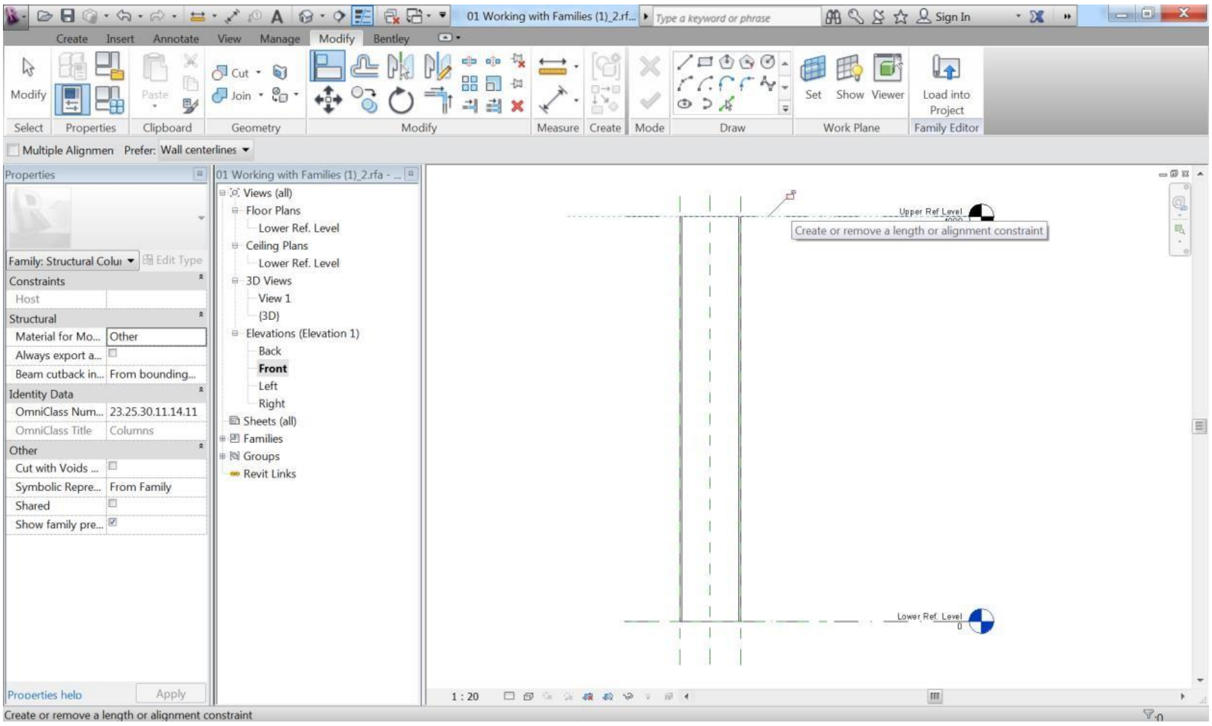
Finish the sketch mode and go to 3D view. You should see a solid geometry like the below image.



Go to "Front" view under "Elevation".

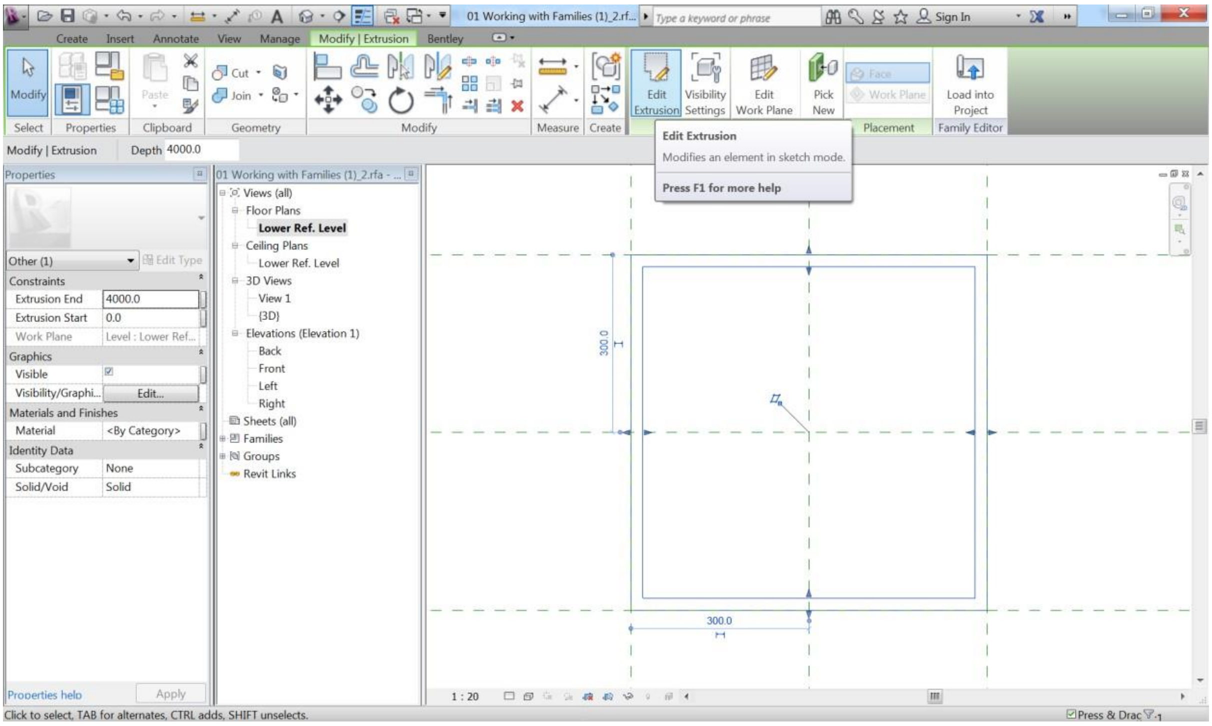


Use the align function again to coincide the top face of the solid and the "Upper Ref Level" reference plane.

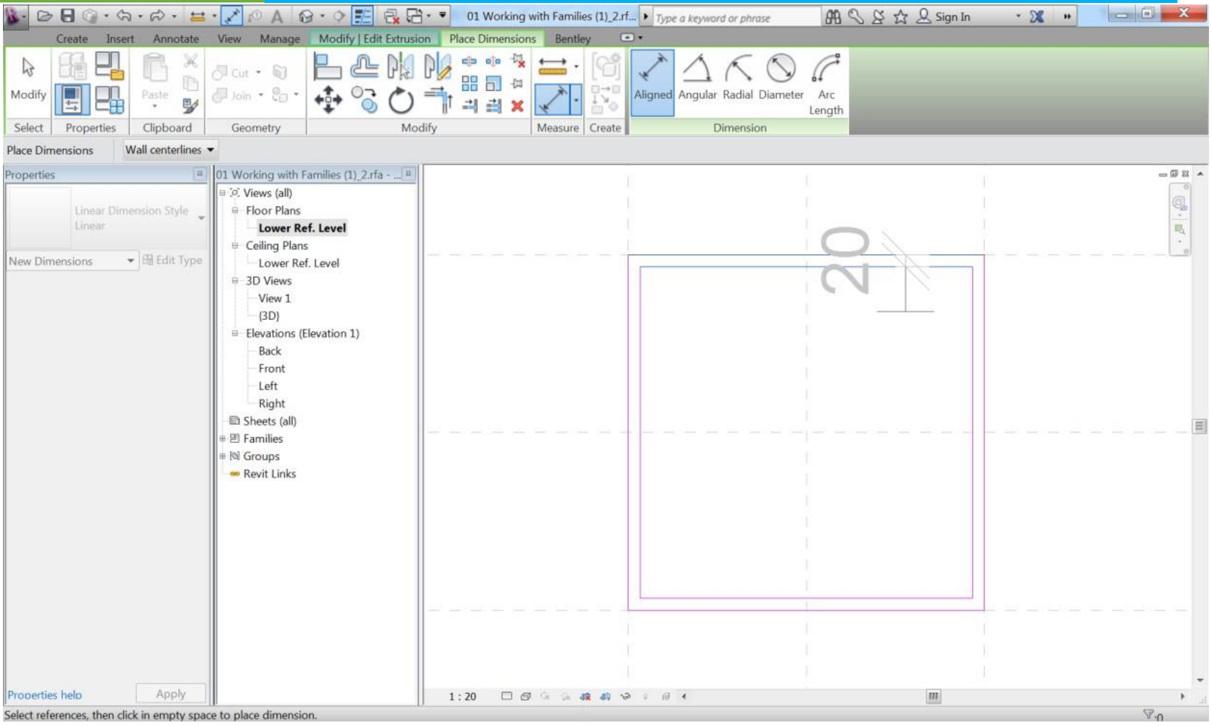


12.3.4 Parametric Dimension

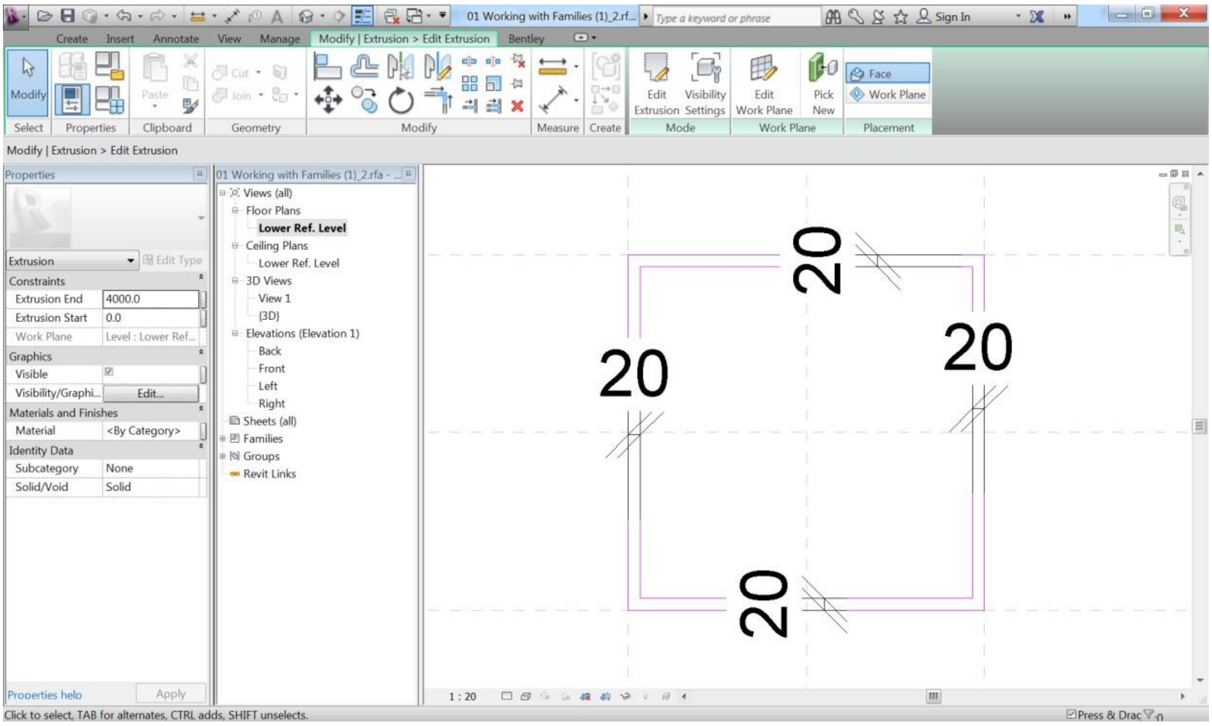
Go back to "Lower Ref. Level" view under "Floor Plans". Select the extrusion that you just created. Then "Edit Extrusion" button will appear on the ribbon tab. click the button.



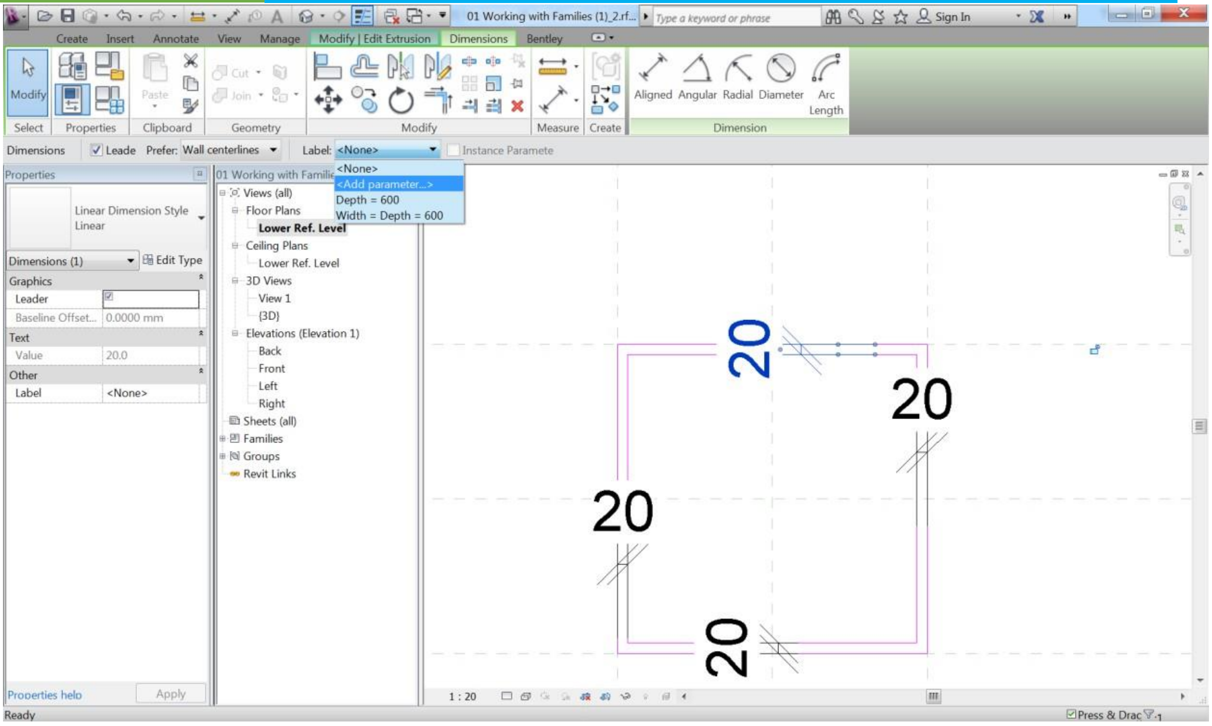
Then you can edit the sketch of the extrusion. Click the "Aligned Dimension" button under "Measure" group. Add a dimension in-between the top two edges.



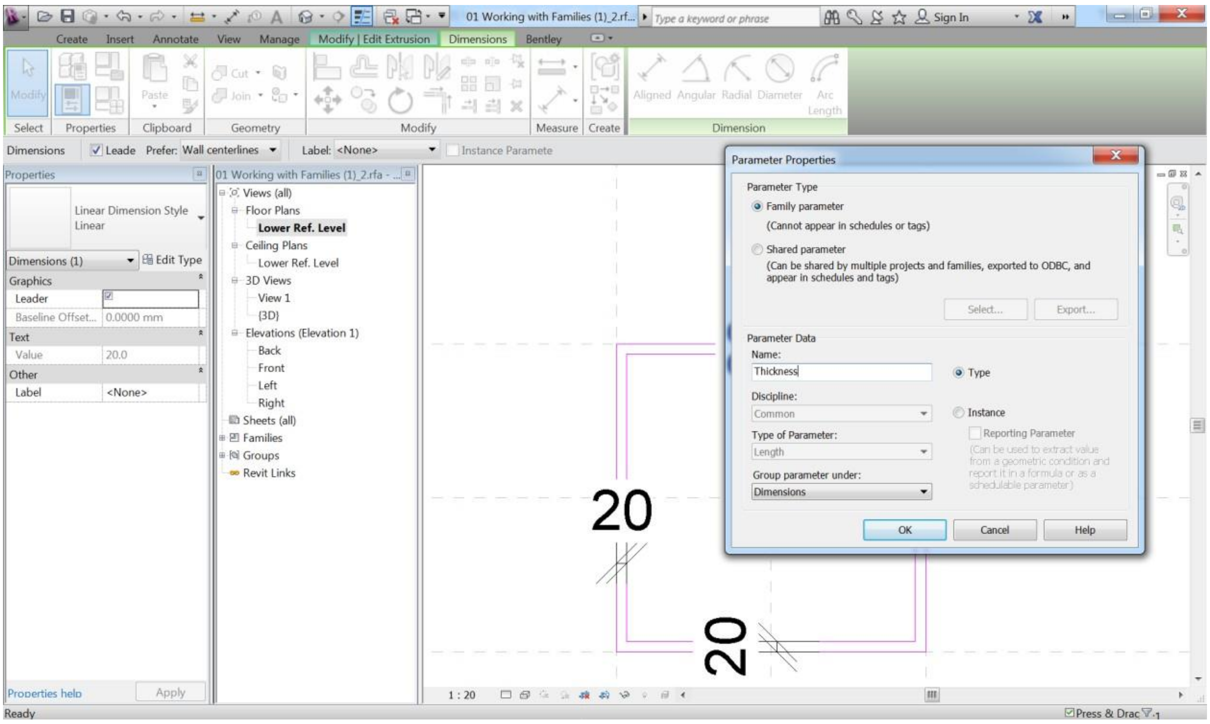
Repeat the same steps to the other sides.



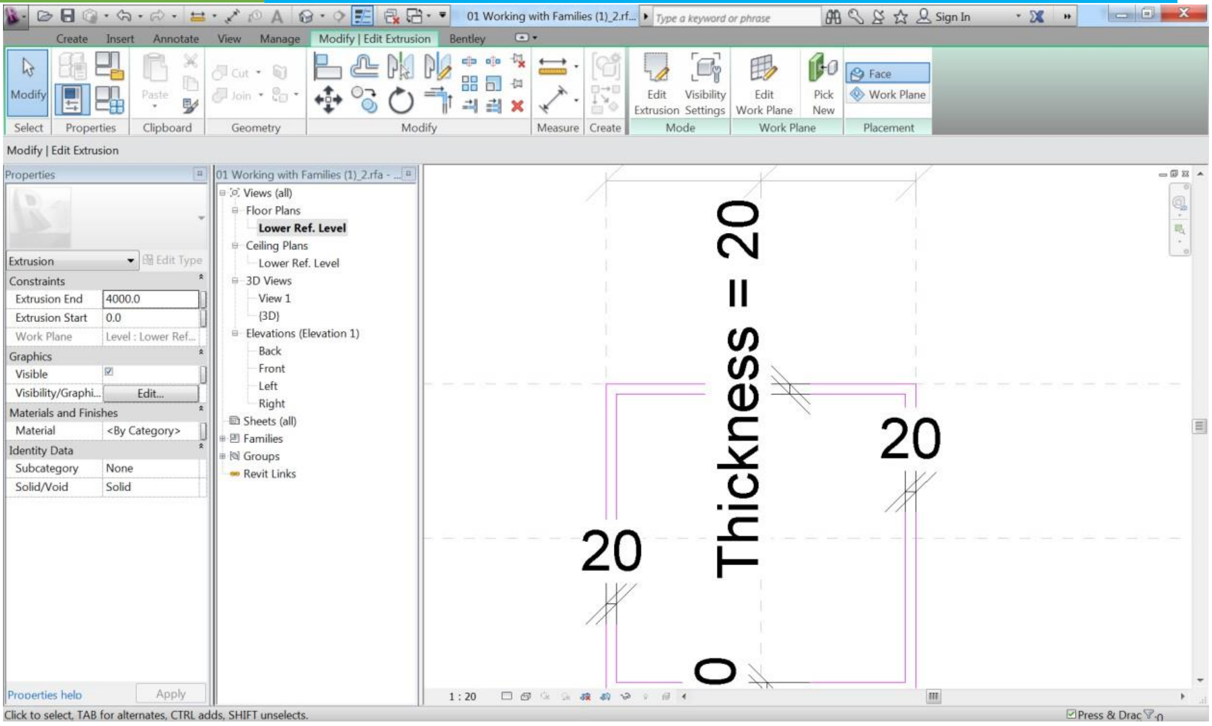
Now we are going to parameterize the dimensions. First, select one of the dimensions. Then from option bar, select "<Add parameter...>" from "Label".



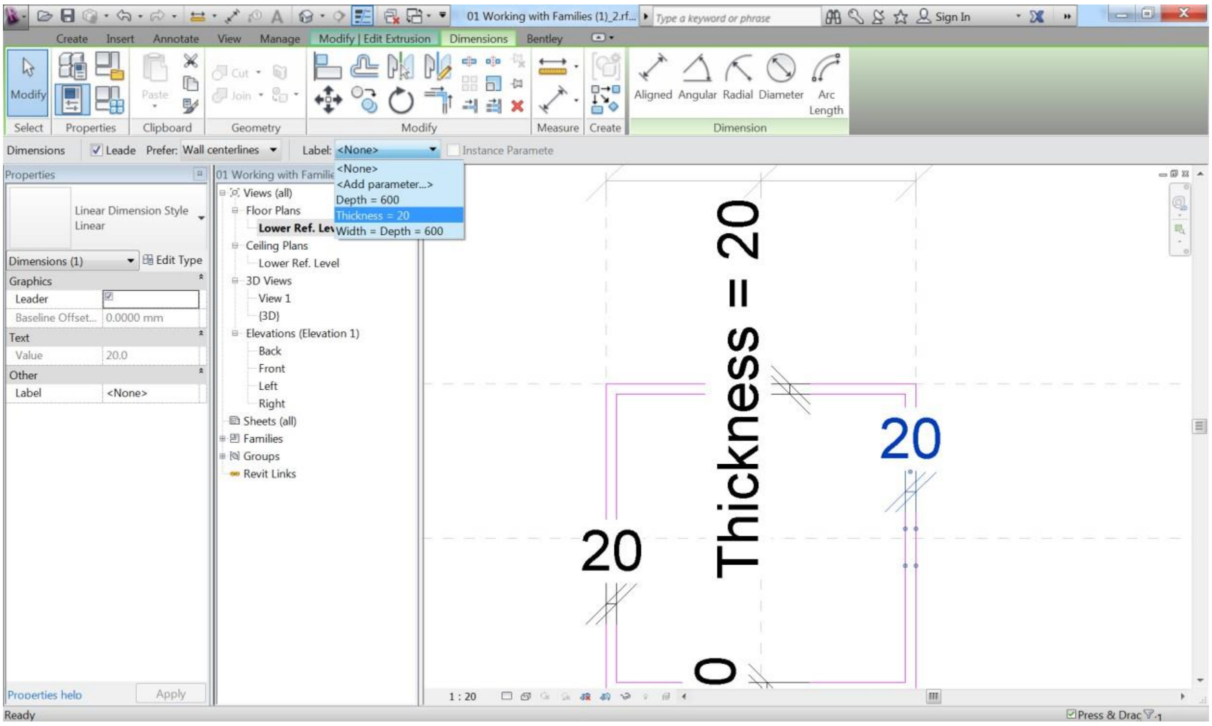
Then “Parameter Properties” window will appear. type in “Thickness” for the name and set “Dimension” for “Group parameter under:”, click “OK” button.



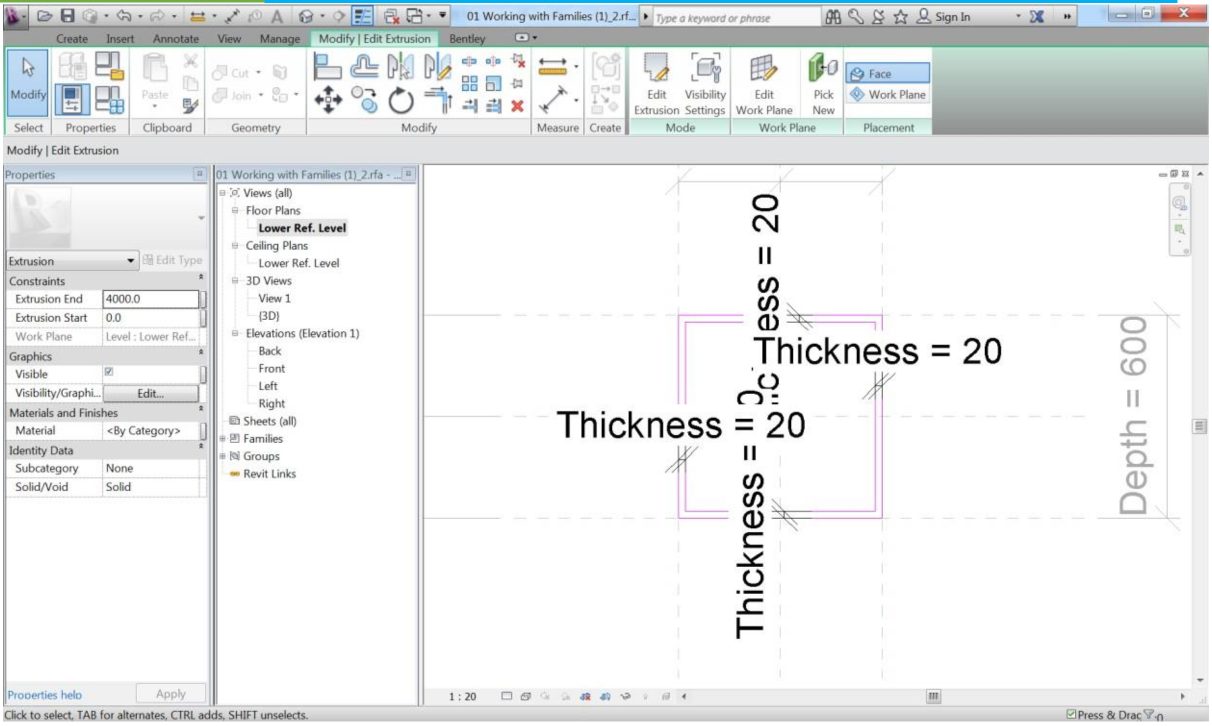
You can see that see that “Thickness” parameter is associated with the dimension.



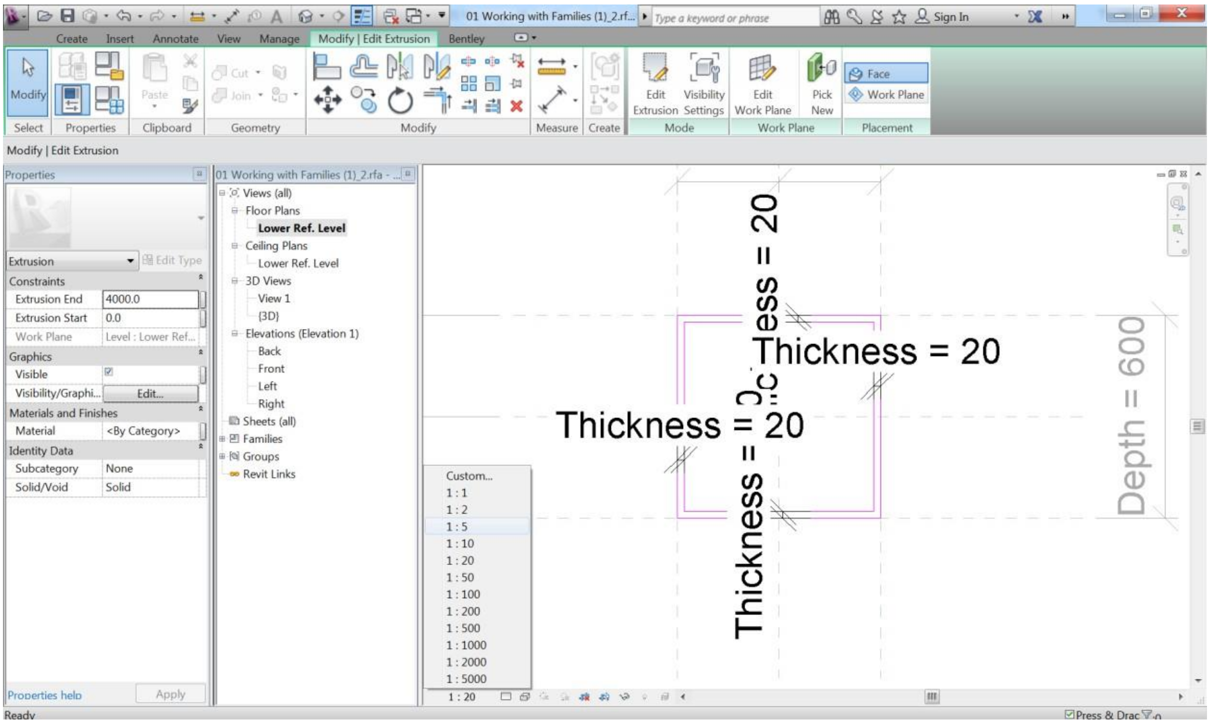
For the other dimensions, select the “Thickness = 20” for the “Label”.



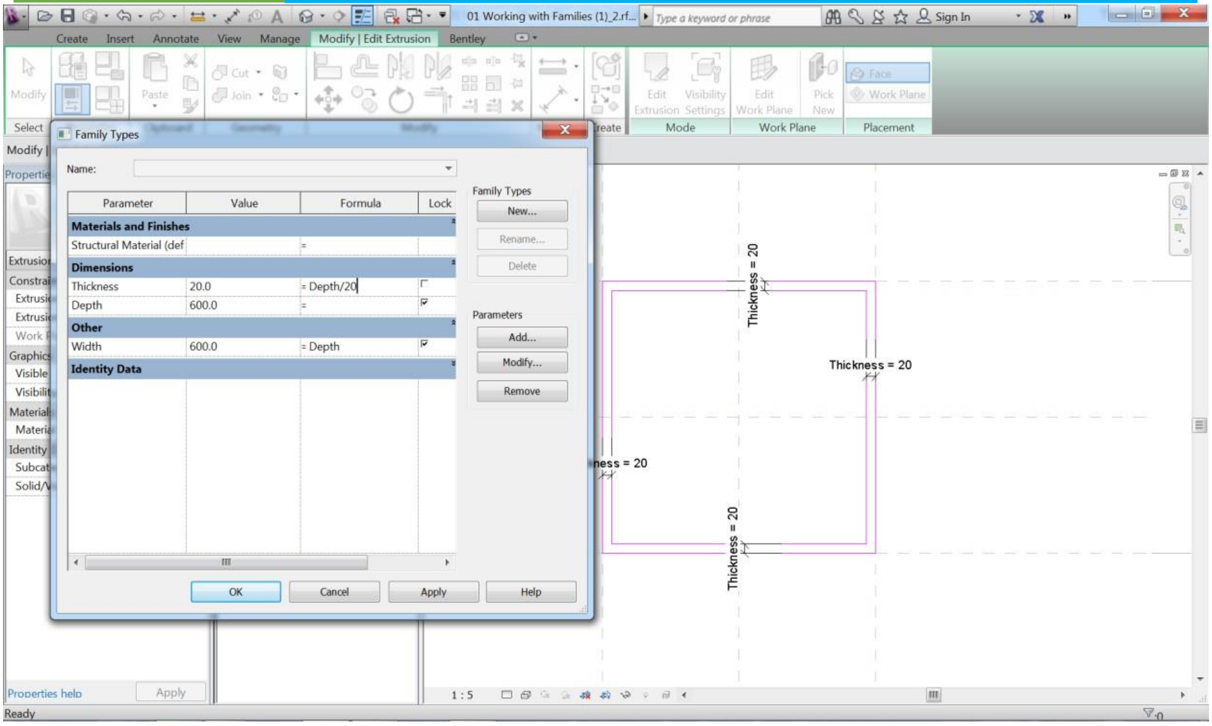
Repeat the same steps to the rest of the dimensions



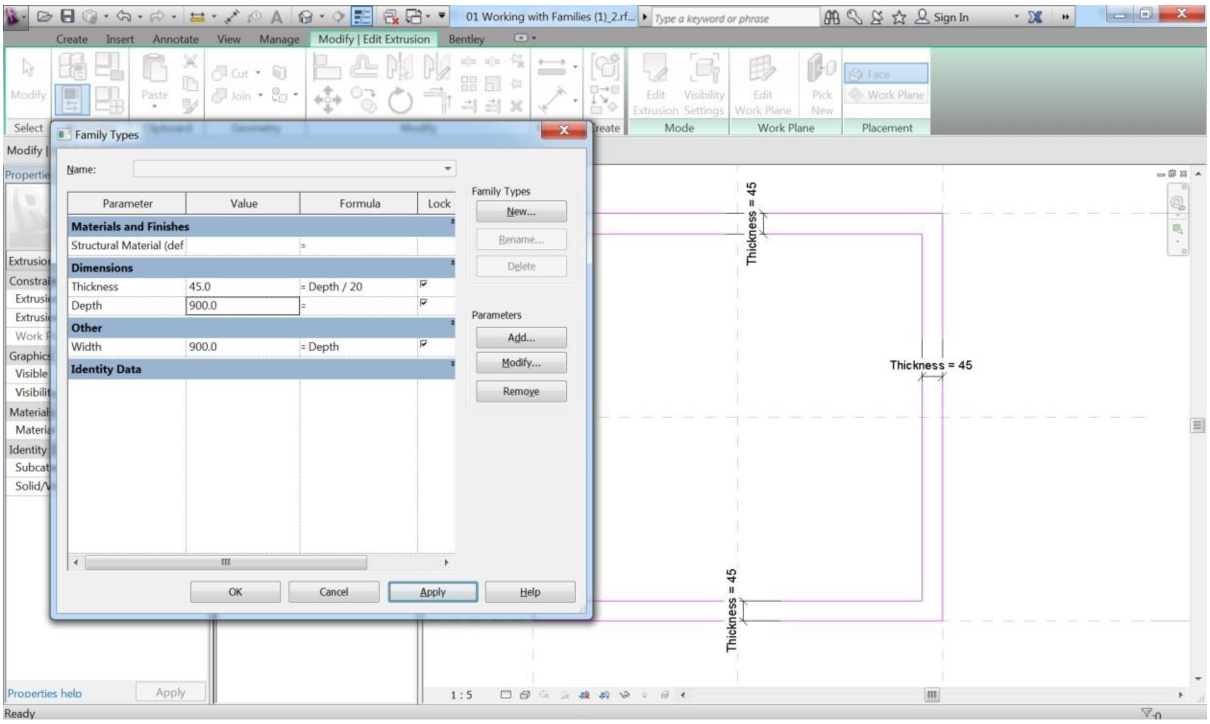
The dimensions are all jumbled together. To solve this, change the scale of the view to “1:5”.



Once the scale is set, click “Family Types” button. Set “Depth/20” for “Formula” of “Thickness” parameter.



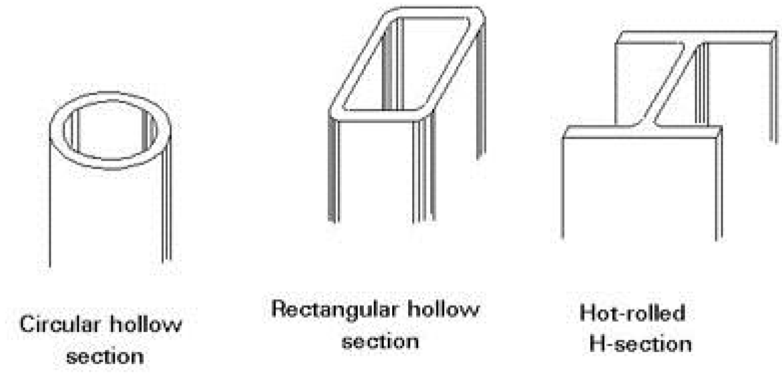
try other values for “Depth” parameter to check if the formula works well and the sketch changes along.



set “Depth” to “600” again and finish the sketch mode.

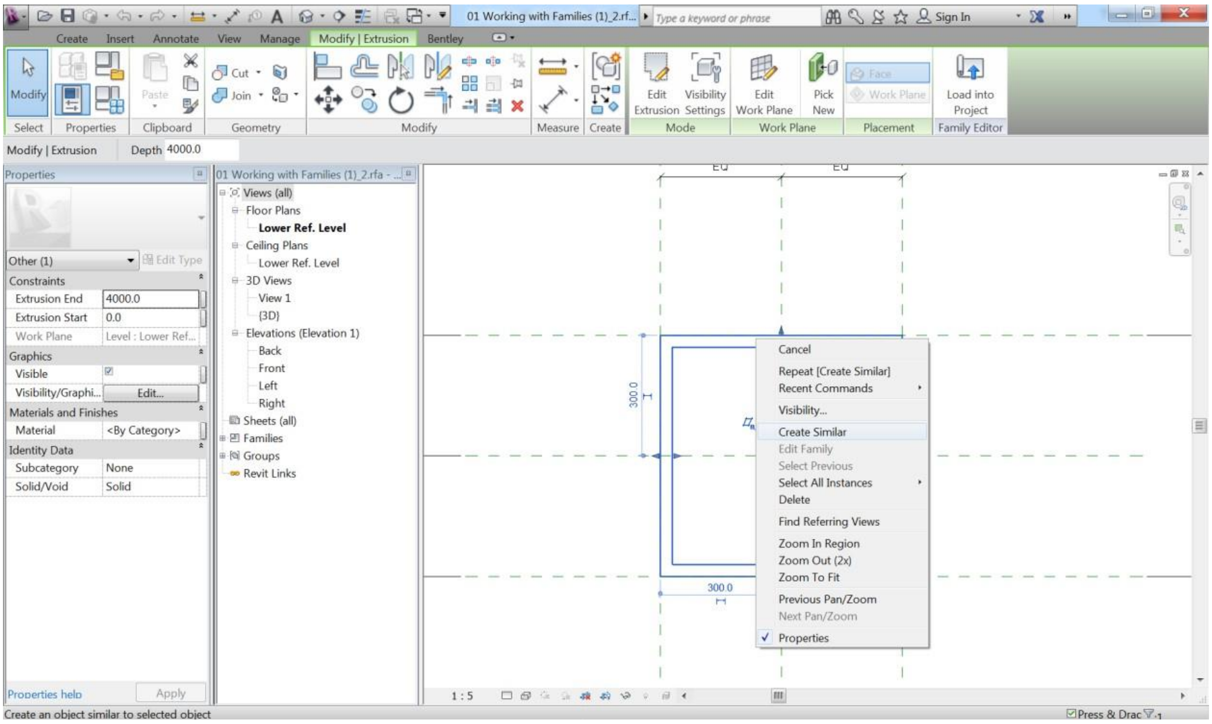
12.3.5 Base Geometry 2

Now we have the basic geometry of the column. We will add more elaborated geometry to the family to have more accurate representation of hollow steel columns. In reality, hollow steel columns have filleted edges. In computational representation, such fillet is additional features than requires more graphical computation. In Revit, there is a way to control the detail level of your family's representation.

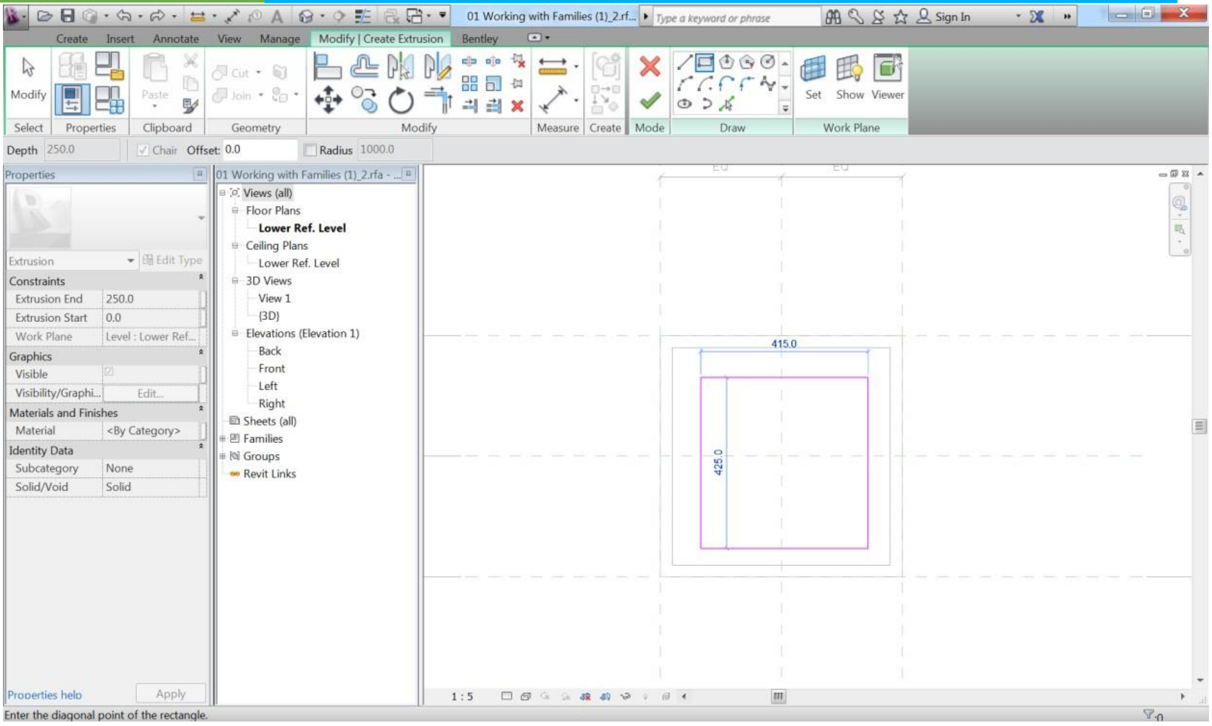


The above image is from http://www.tatasteelconstruction.com/

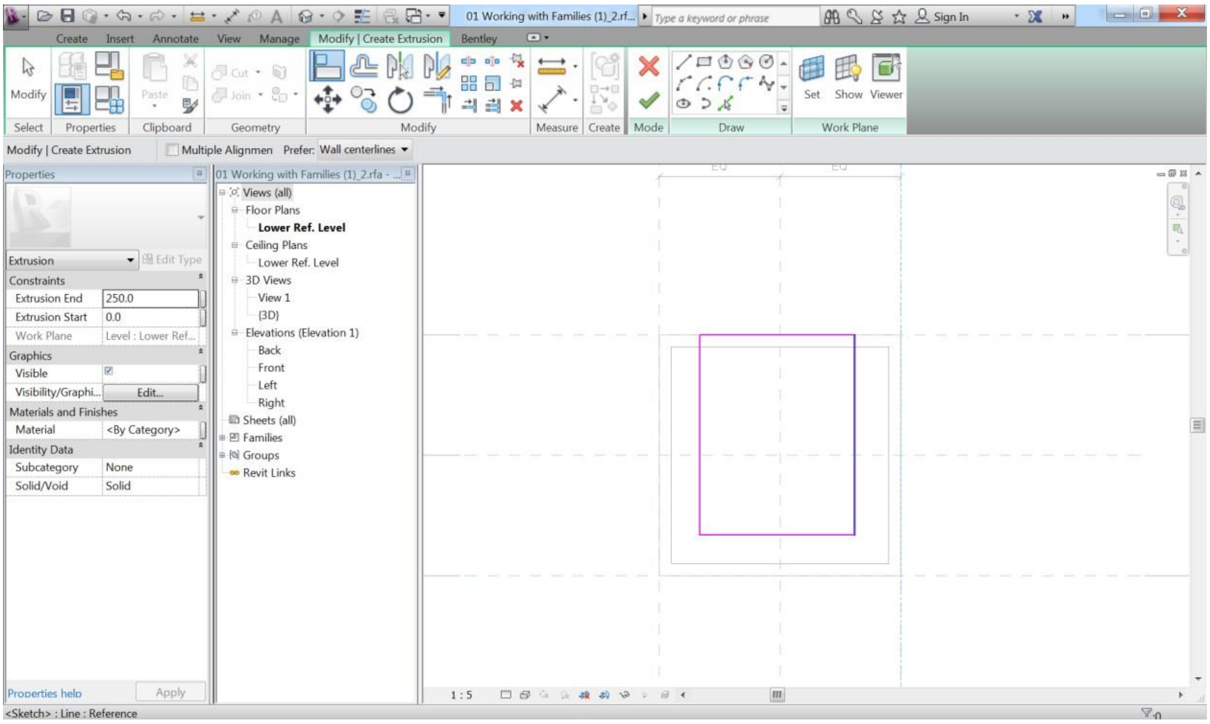
We need another extrusion with filleted edges. We can use the "Extrusion" function under "Create" tab. Here is another way, when we create something similar to what is already existing, we can use "Create Similar" function. From the "Lower Ref. Level" view, select the geometry and right click to select "Create Similar" from the pop-up menu.



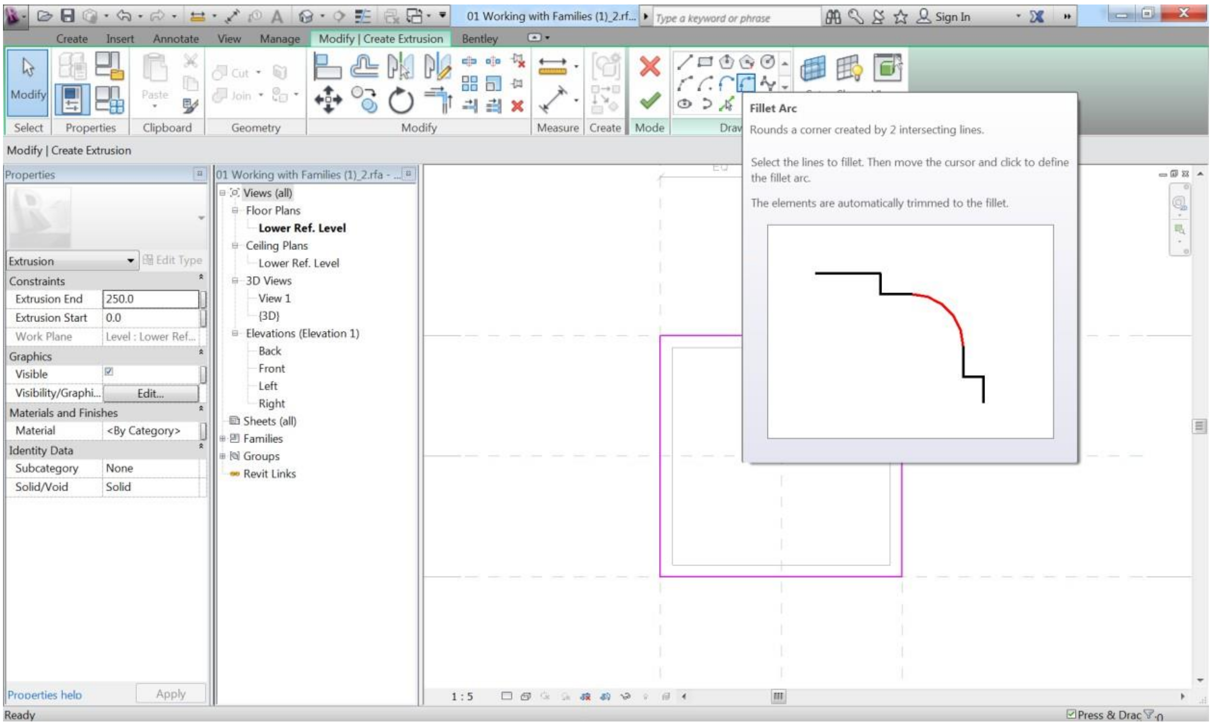
Once you select the function, you will be in sketch mode. Create a rectangle as we did for the existing extrusion.



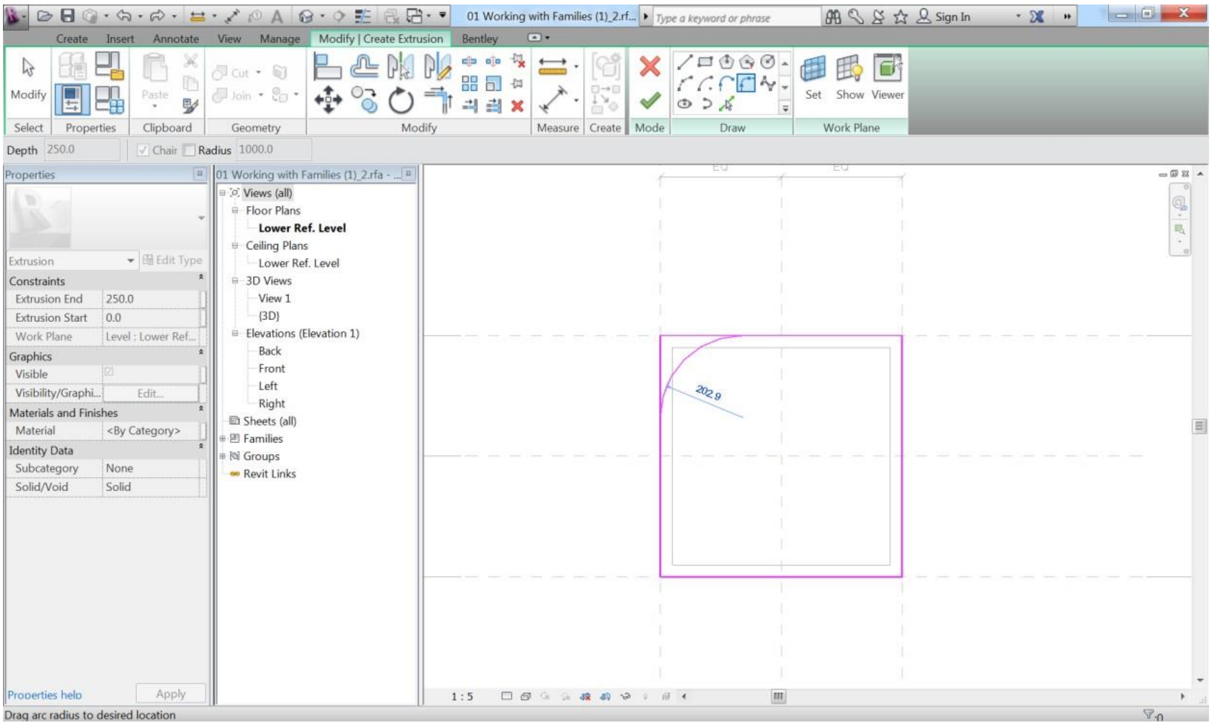
Align and constrain the edges to reference plane. make sure you select the reference planes as targets, not the side faces of the existing extrusion.



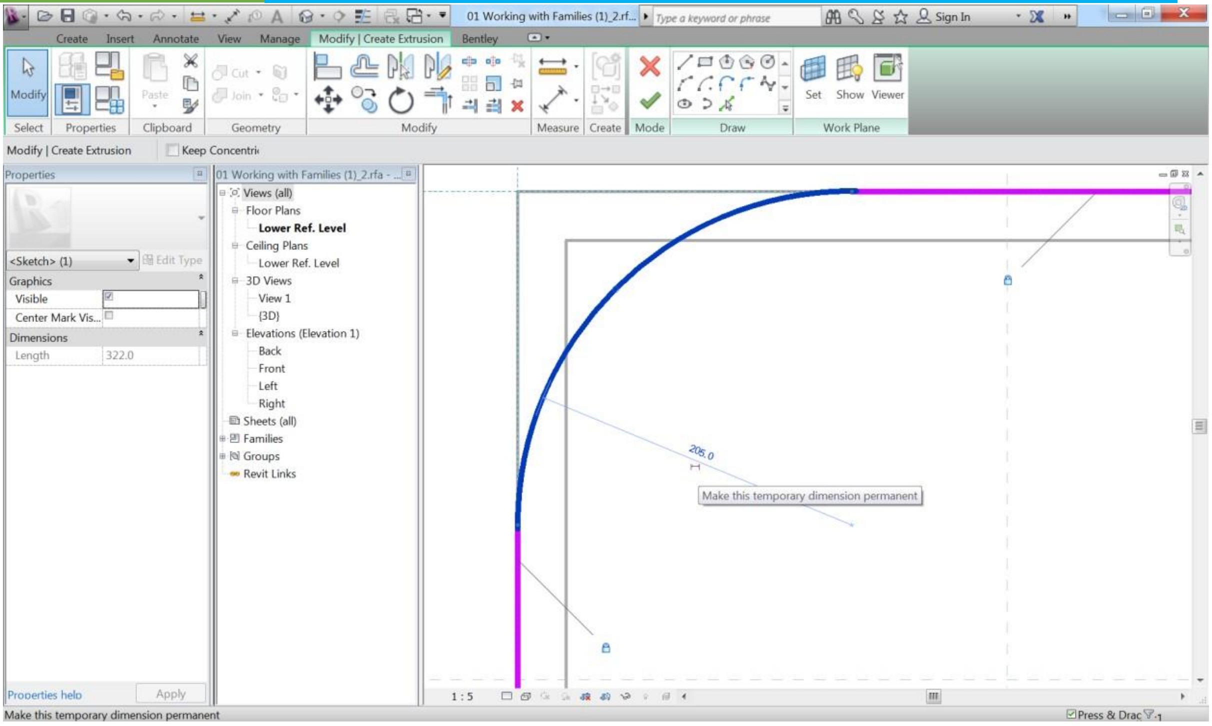
Now we need to fillet the corners. Select “Fillet Arc” function from the “Draw” option.



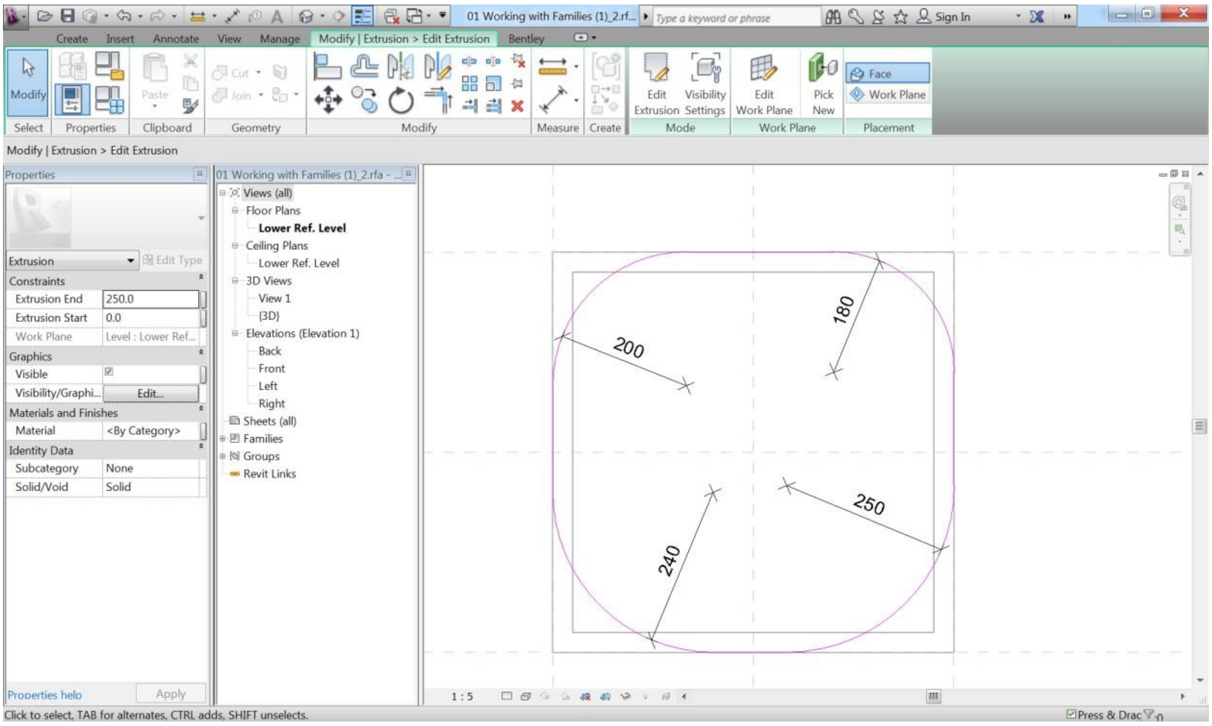
Then click two adjacent edges.



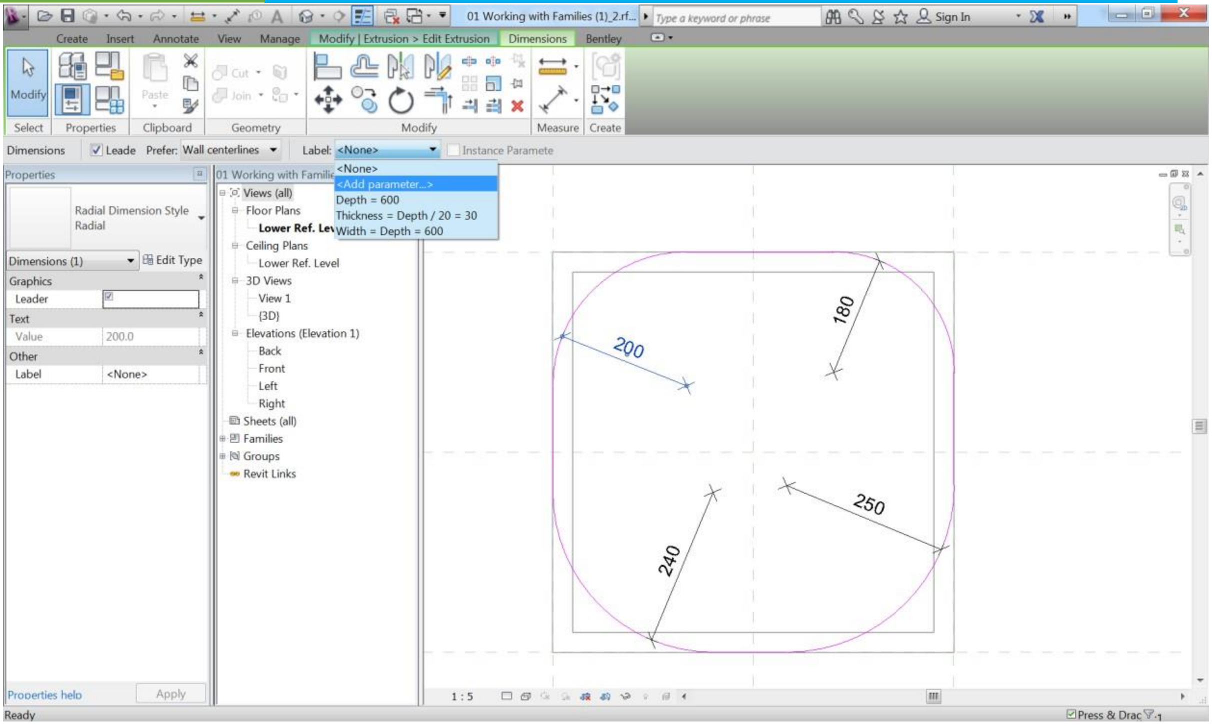
When fillet is done, select the created arc and click the small dimension icon appears”.



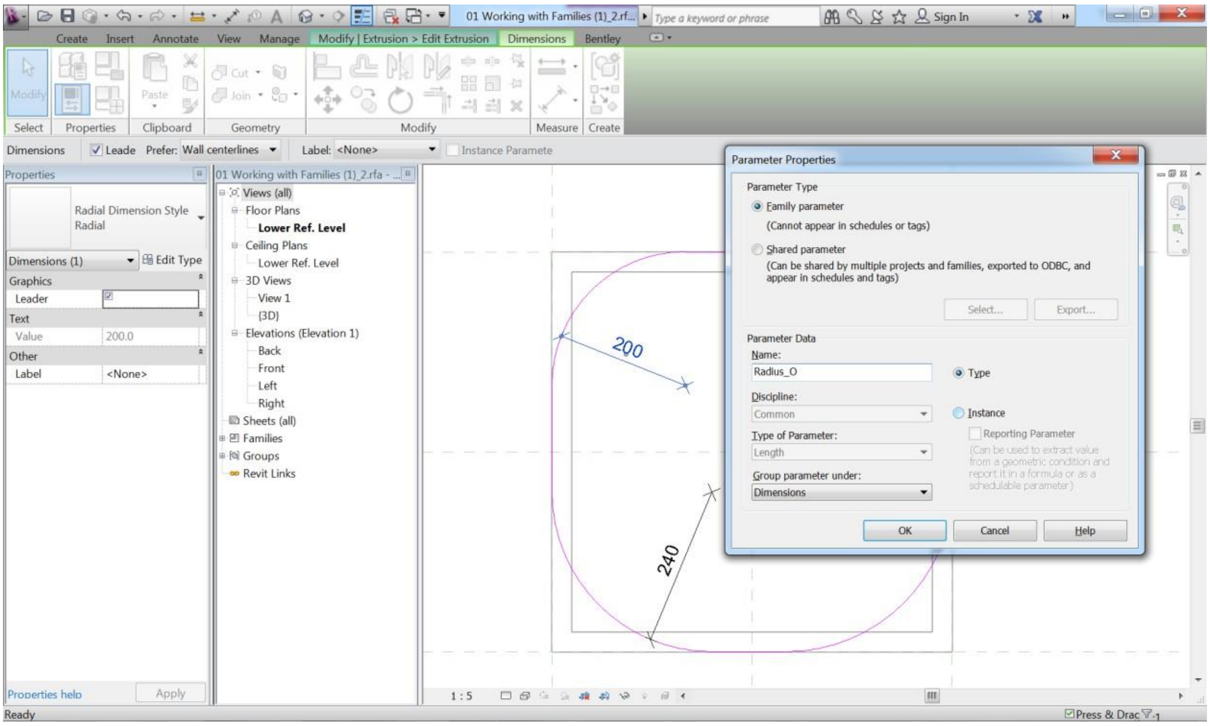
Repeat the same step to other corners.



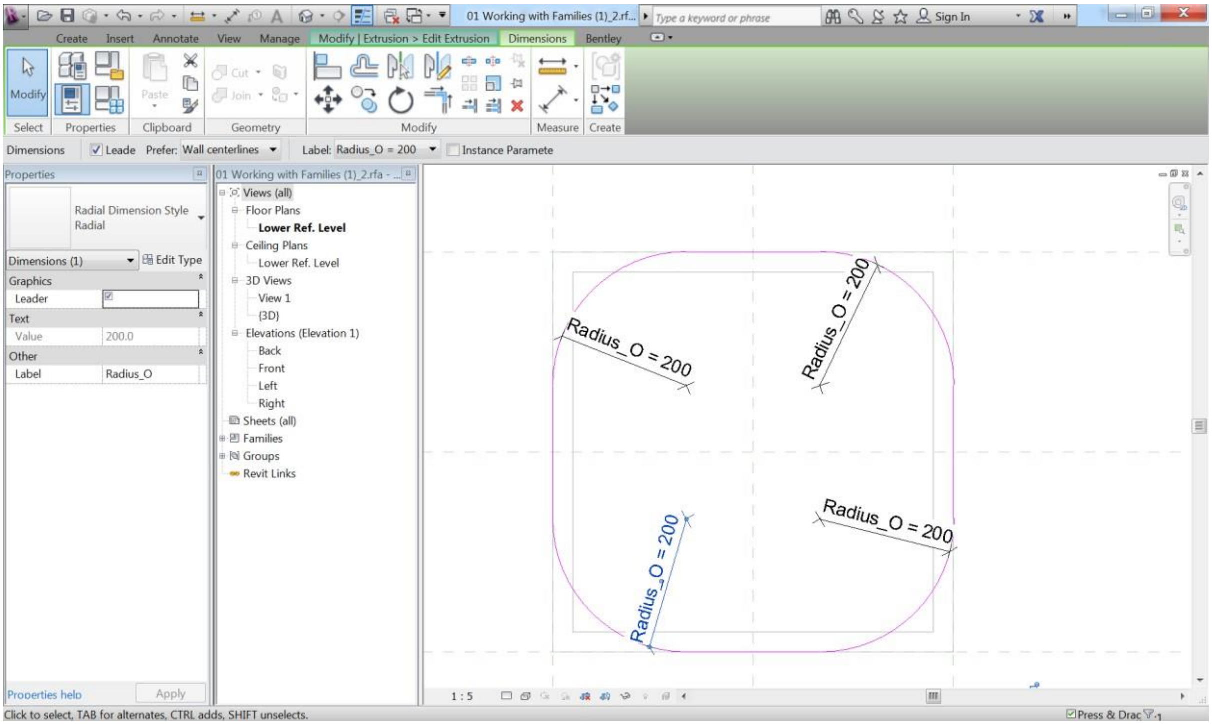
Now select one of the radius dimensions and add a new parameter.



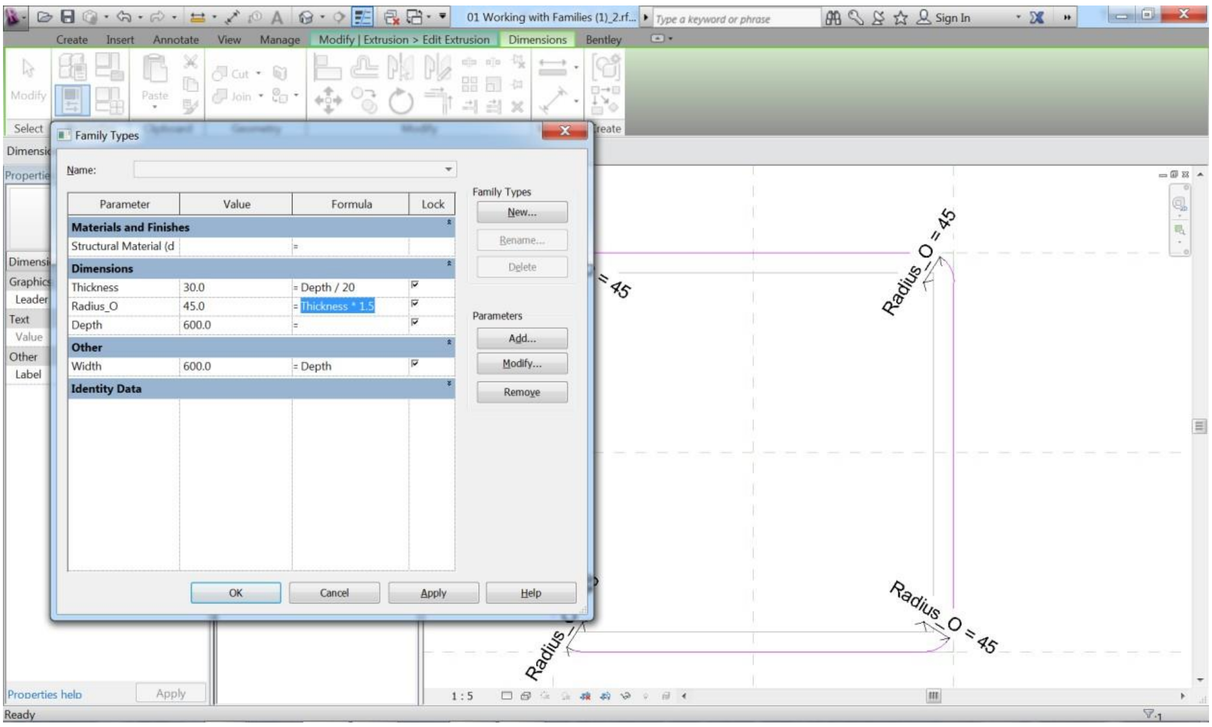
Name the new parameter “Radius_O” and click “OK” button.



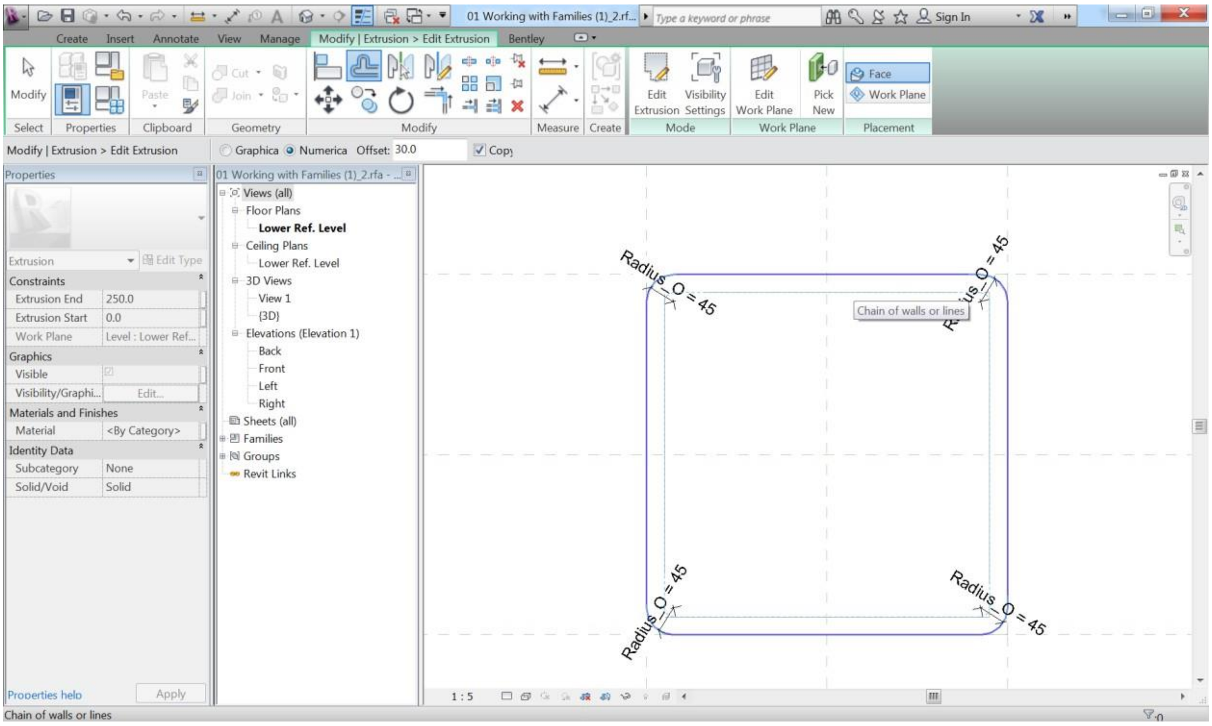
Apply the "Radius_O" parameter to all the corners.



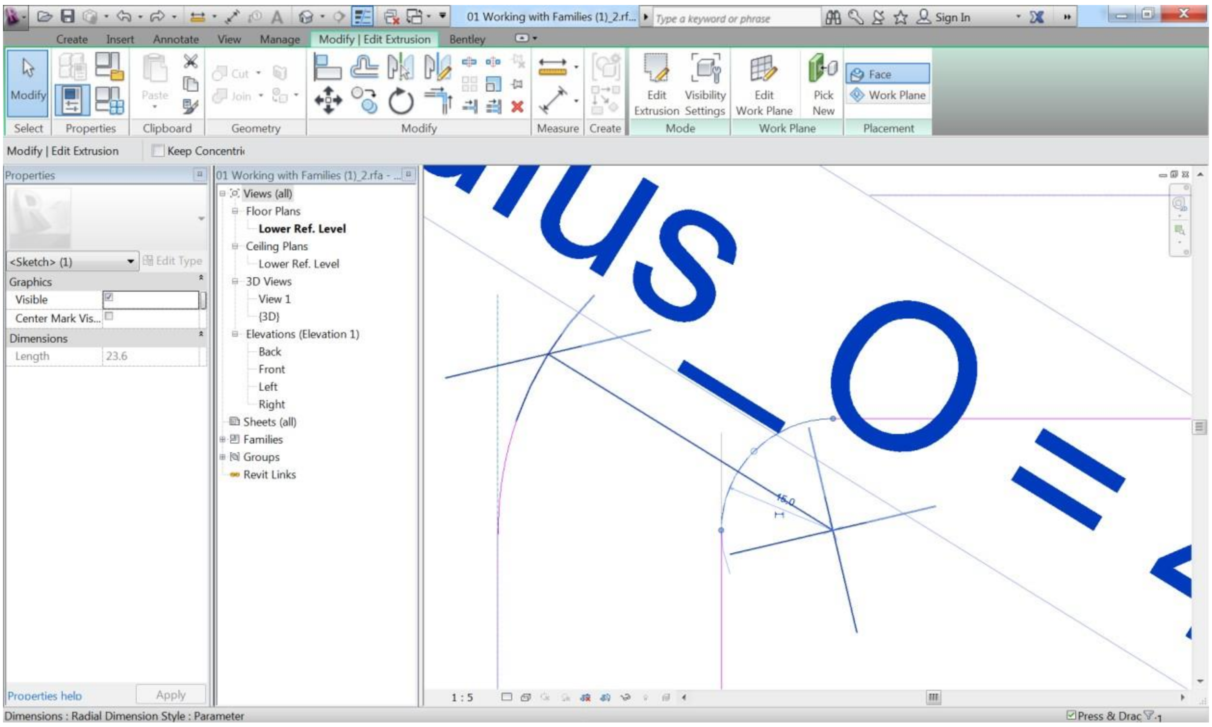
Go to "Properties" and click "Family Types" button. From the appeared window, set the formula of "Radius_O" to "Thickness * 1.5". Then click "Apply" button. This means that the radius for the outer arcs will always be 1.5 times of the thickness.



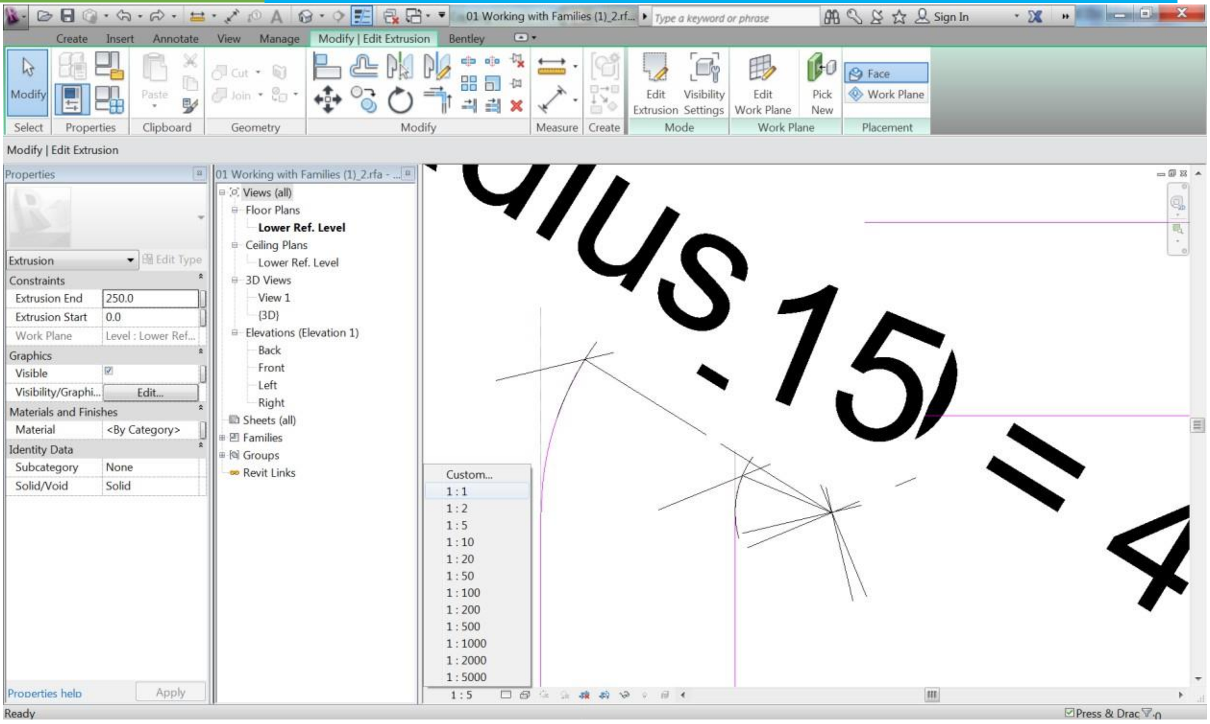
Click “OK” button to go back to the view. Then with “Offset” function under “Modify” tab, set the distance to 30, click empty spot in the canvas, and then put your mouse over to an edge. With “tab” button on your keyboard, select the all edges around to offset.



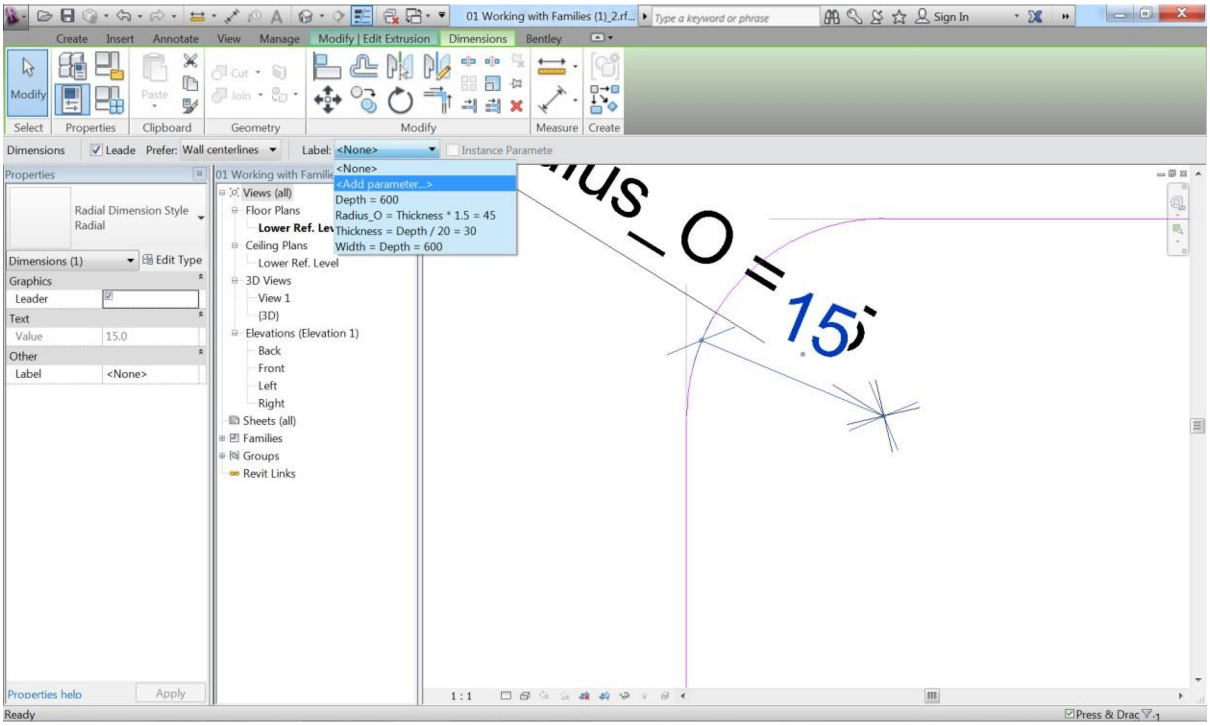
Once offset is done, select one of the inner arcs. Click the appeared small “dimension” icon.



You may want to set the scale to “1:1” to make the dimensions more eligible.

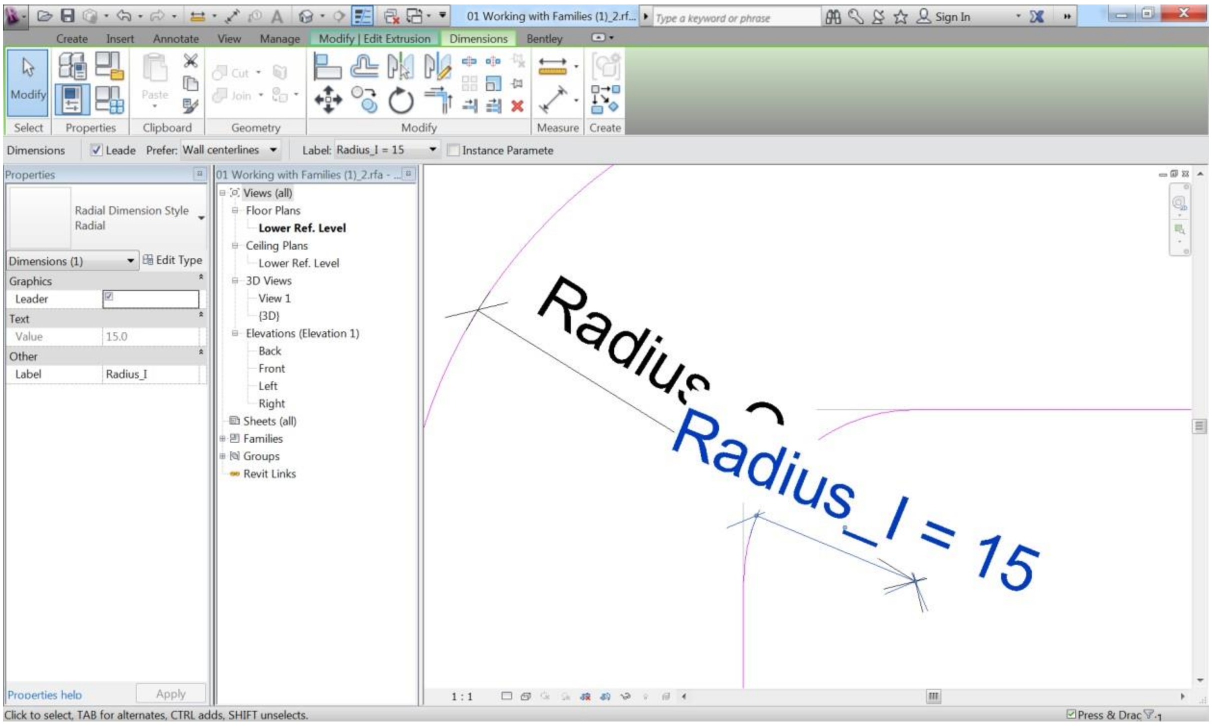


Then select the dimension again to add a new parameter.

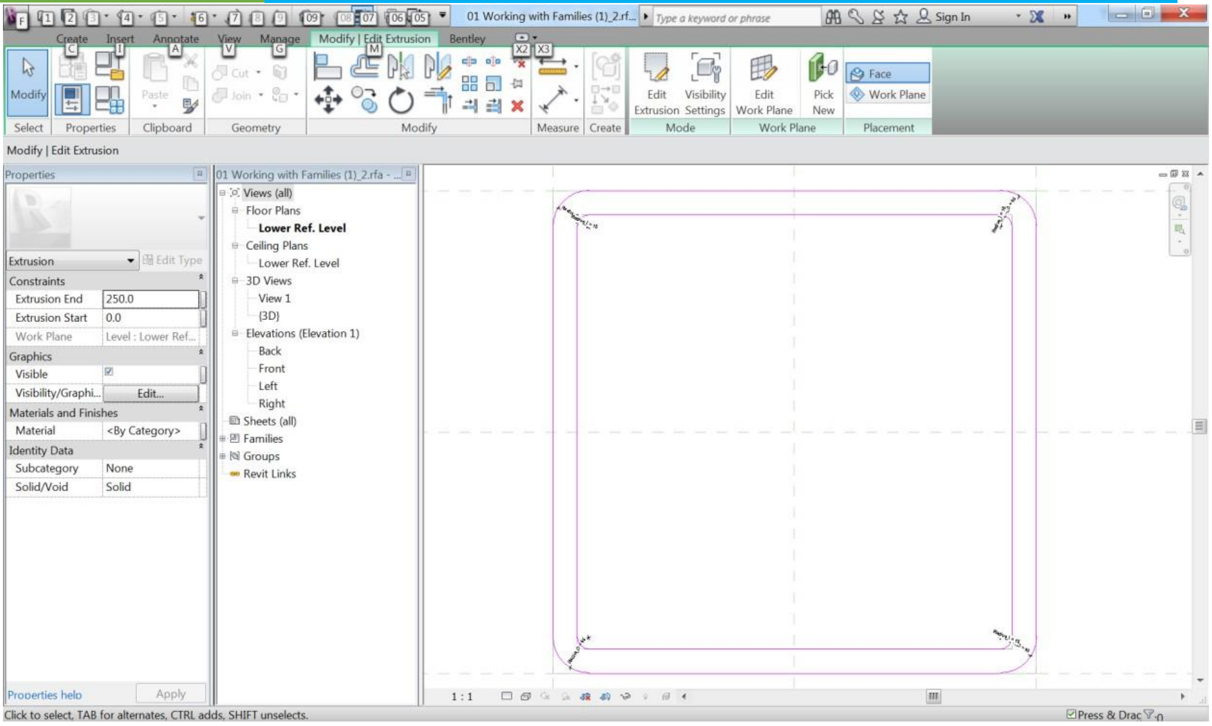


Type in “Radius_I” for the name of the new parameter. Then click “OK” button.

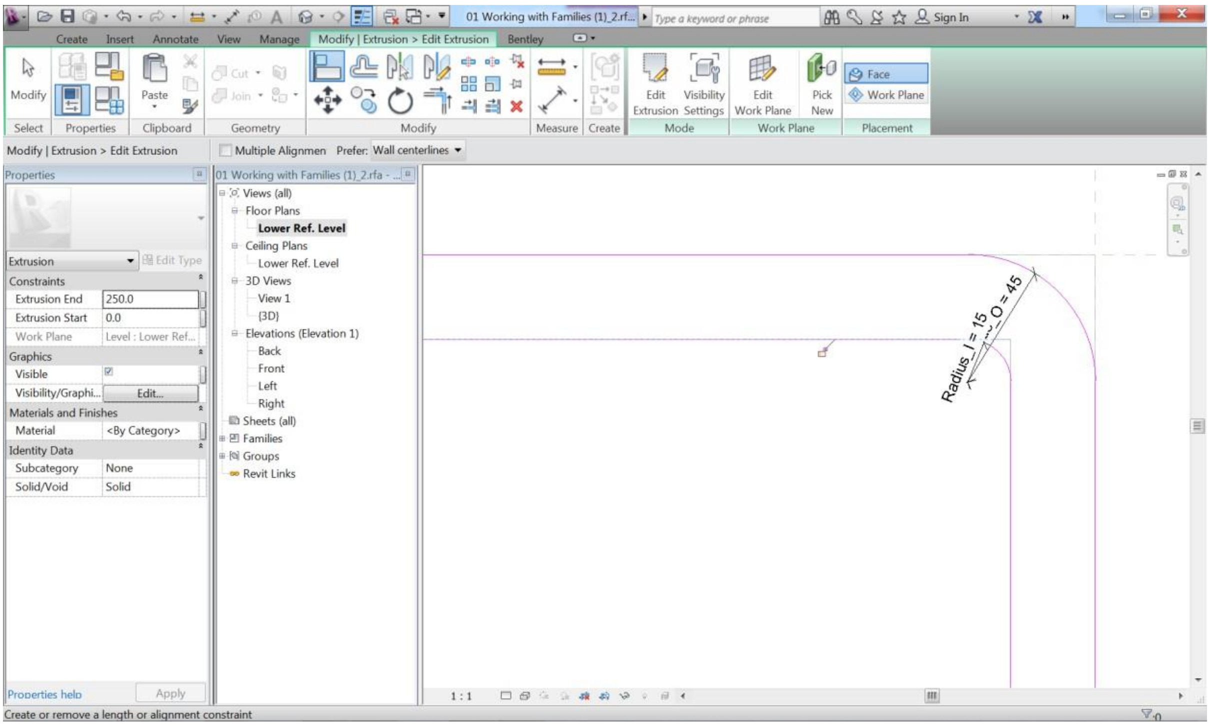
Now the dimension is labeled with "Radius_I" parameter.



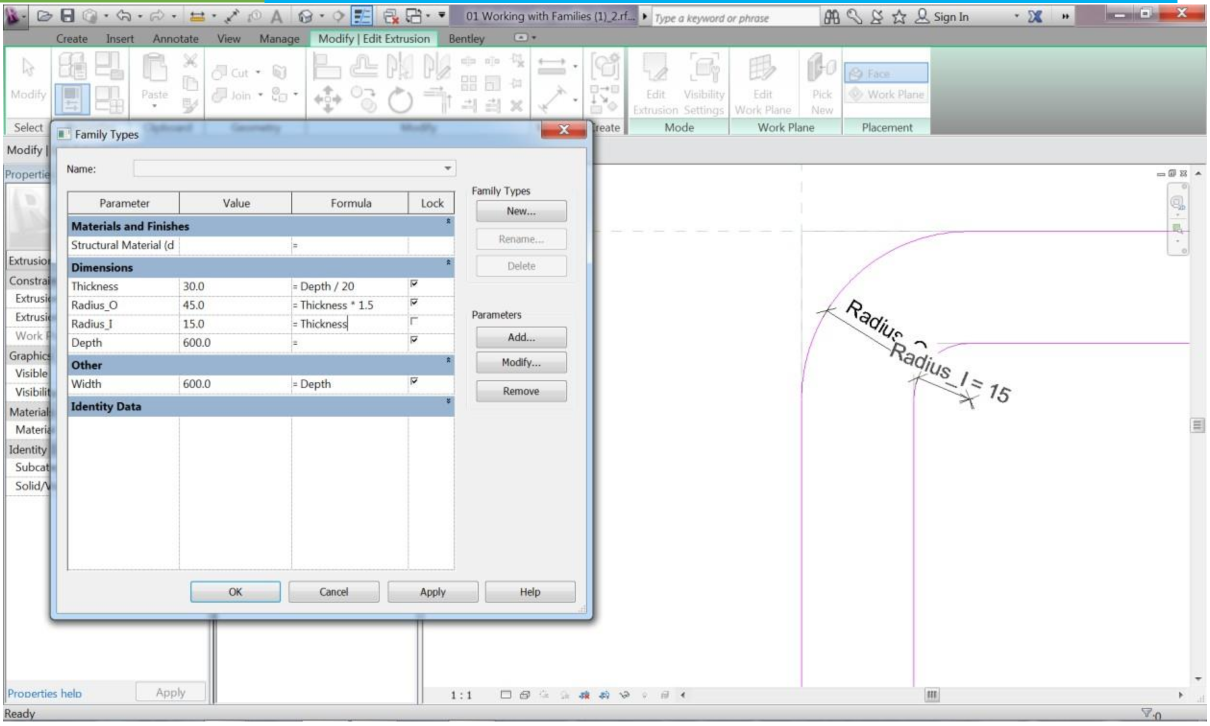
Create dimension for the other inner arcs and label them with "Radius_I".



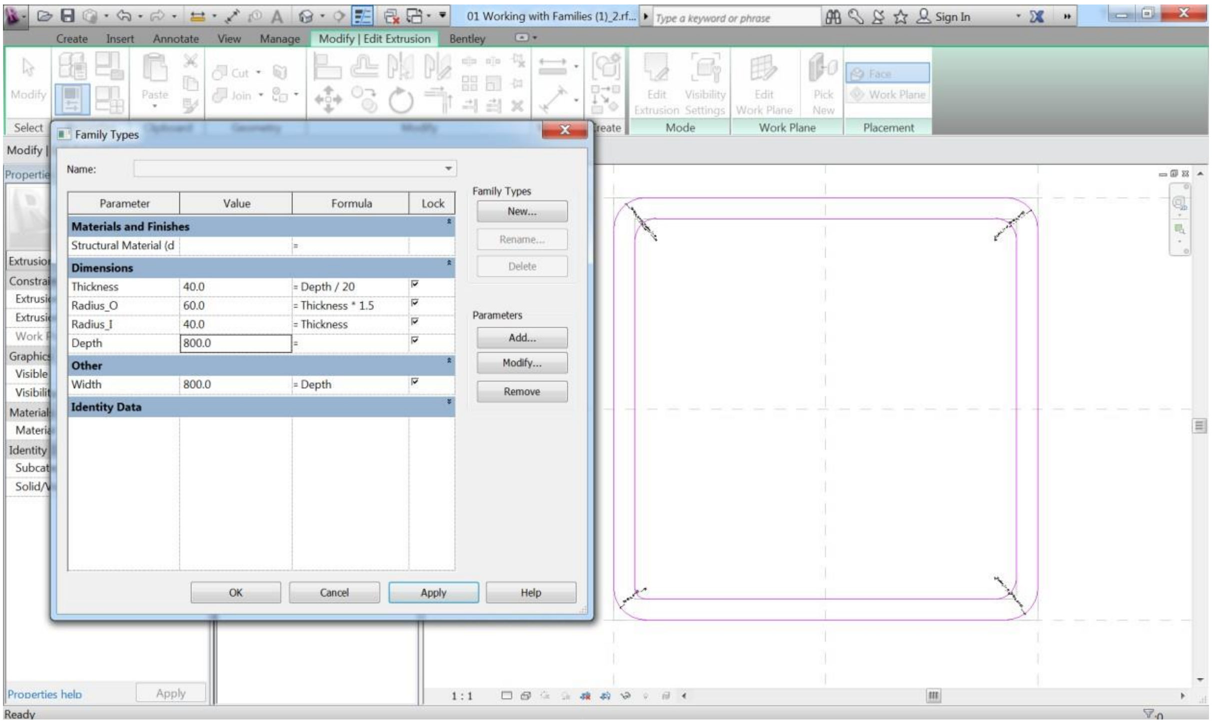
For the inner edges, align them to each corresponding inner face of the existing extrusion.



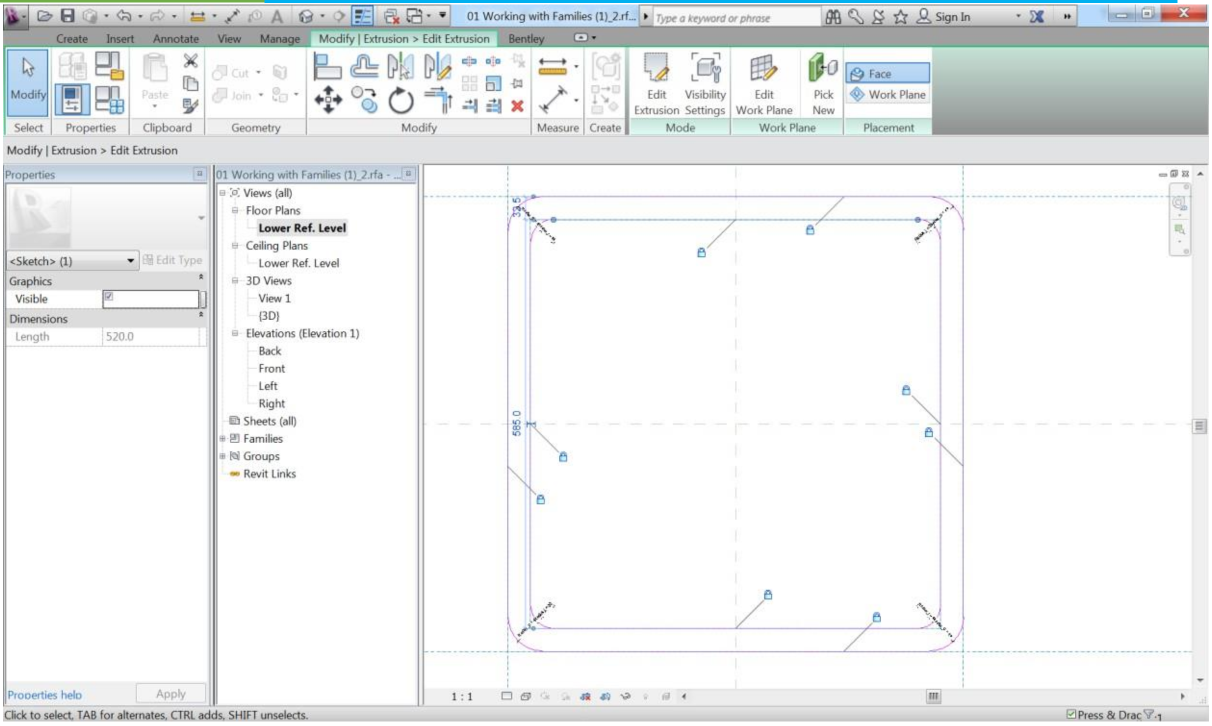
G to "Family Types", set the formula of "Radius_I" to "Thickness".



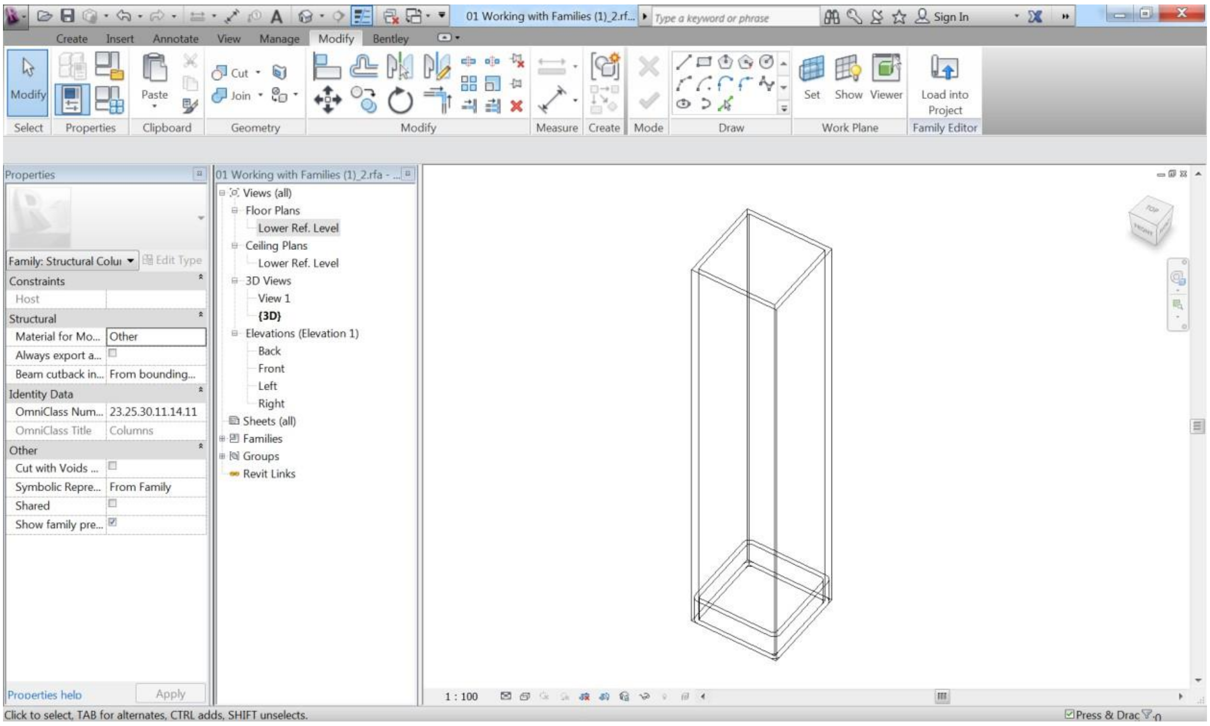
After applying the formula, change the value of "Depth" parameter to test your parametric constraints.



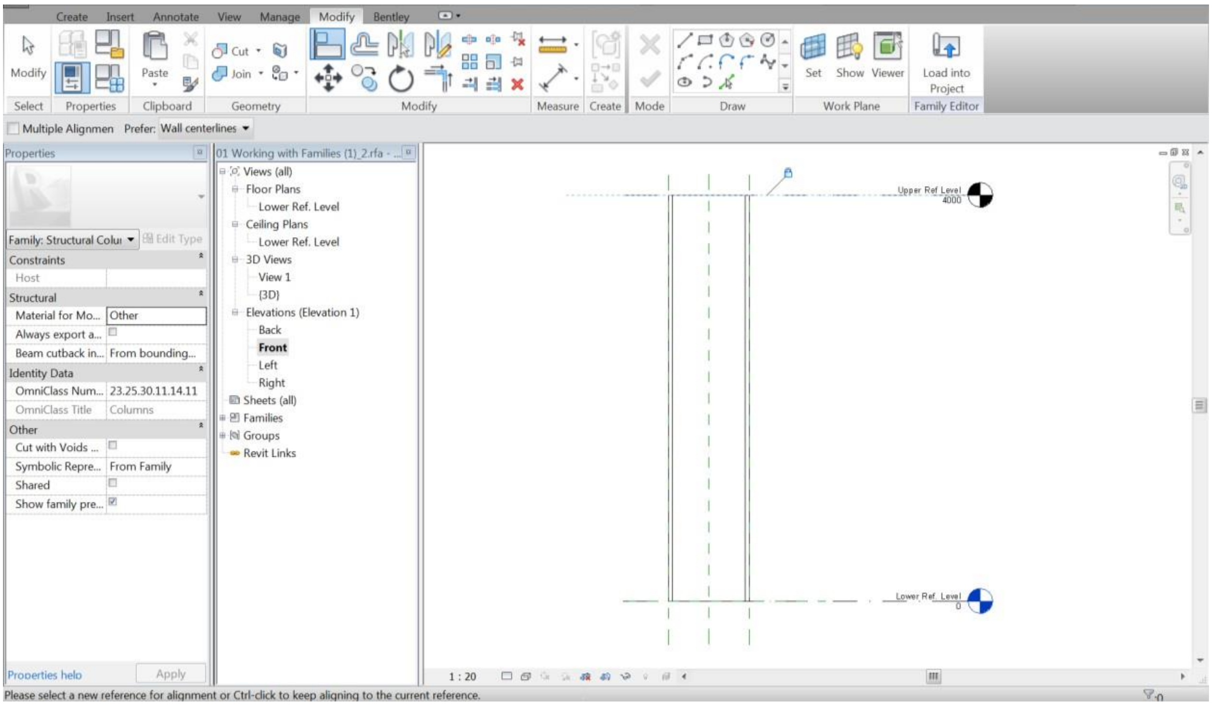
When select any sketch segment, it should look like below image.



Finish the sketch mode and go to 3D view. test our geometry again.

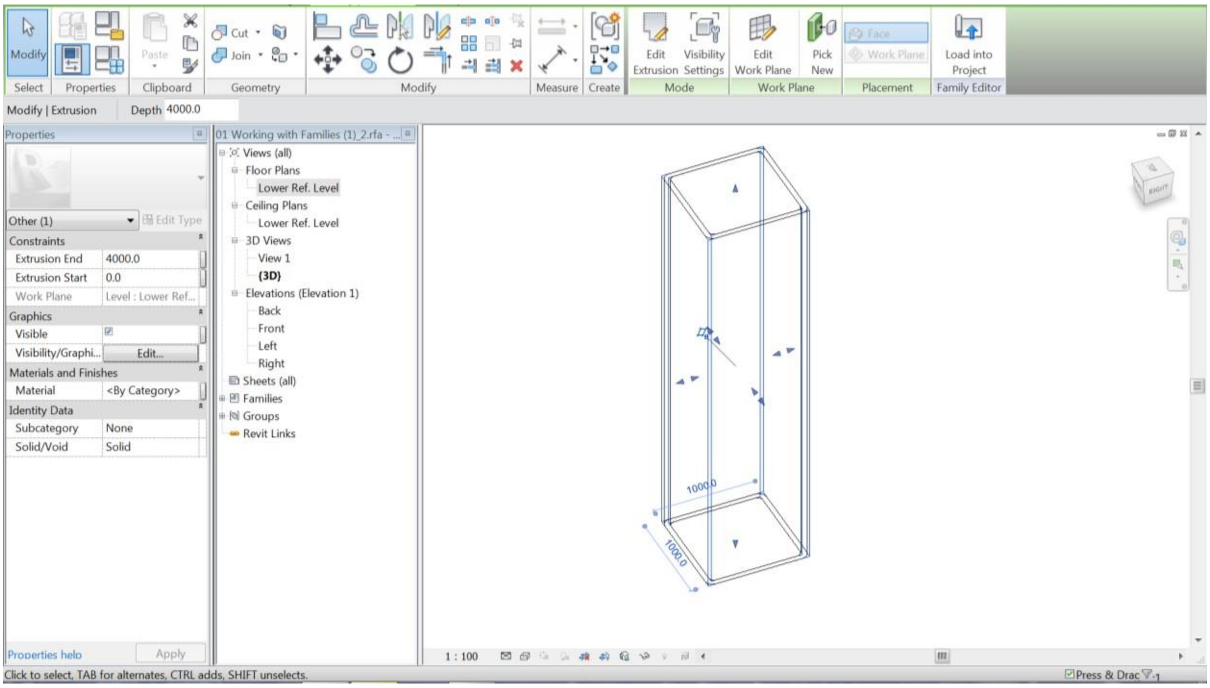


Go to "Front" elevation view. align and constrain the top face to the solid to "Upper Ref. Level" view.

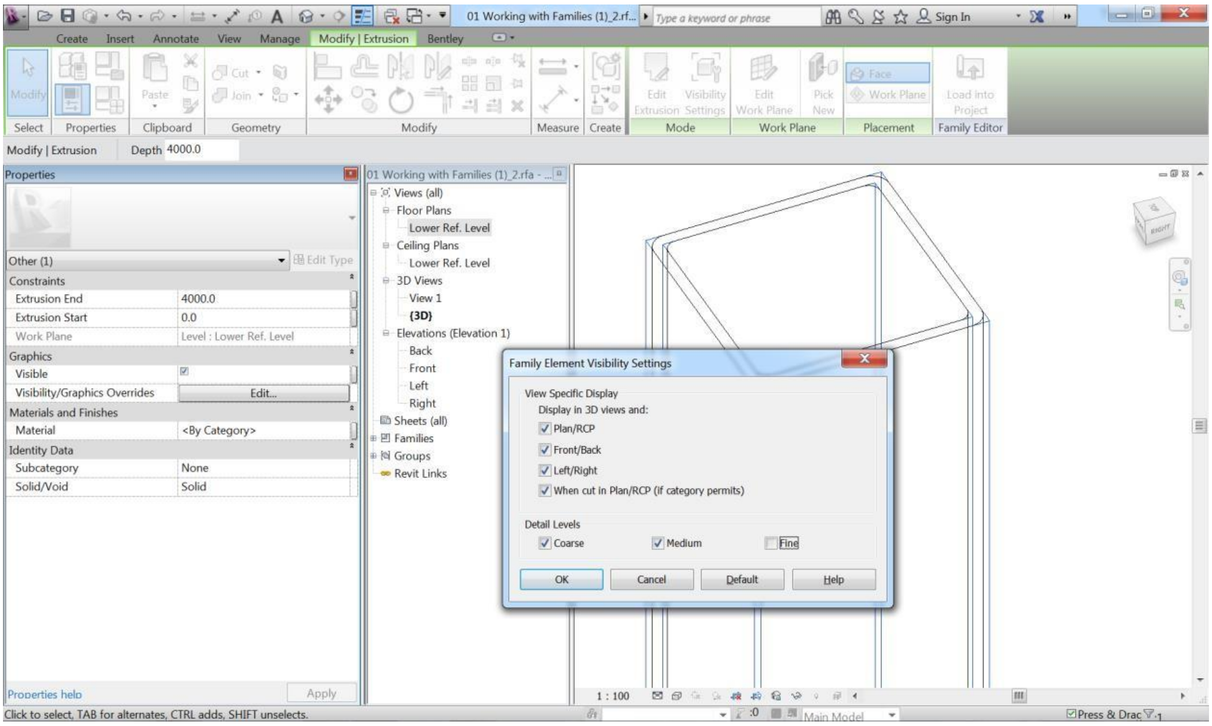


12.3.6 Visibility Setting

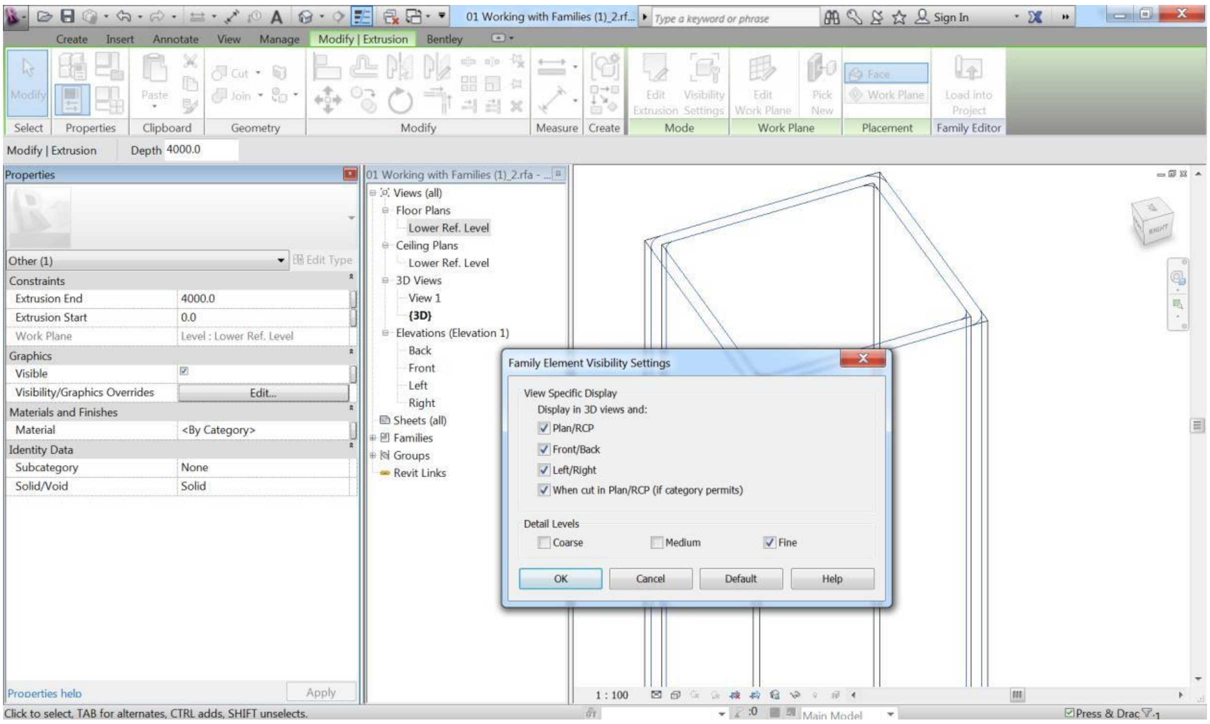
For the two geometries, you can set each to different detail level. Go to 3D view, and select the first extrusion with the right-angled corners. From the "Properties" select "Edit" button next to "Visibility/Graphics Overrides" field.



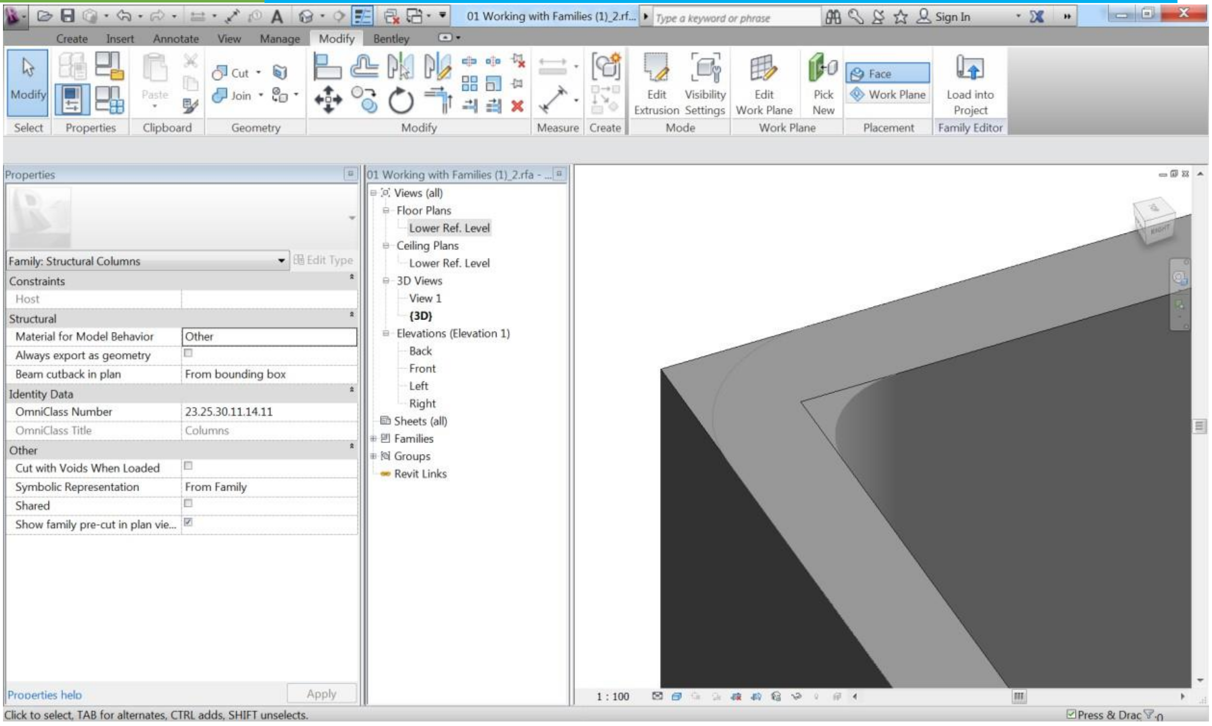
From the appeared window, deselect the "Fine" option from the list. Then click "OK" button.



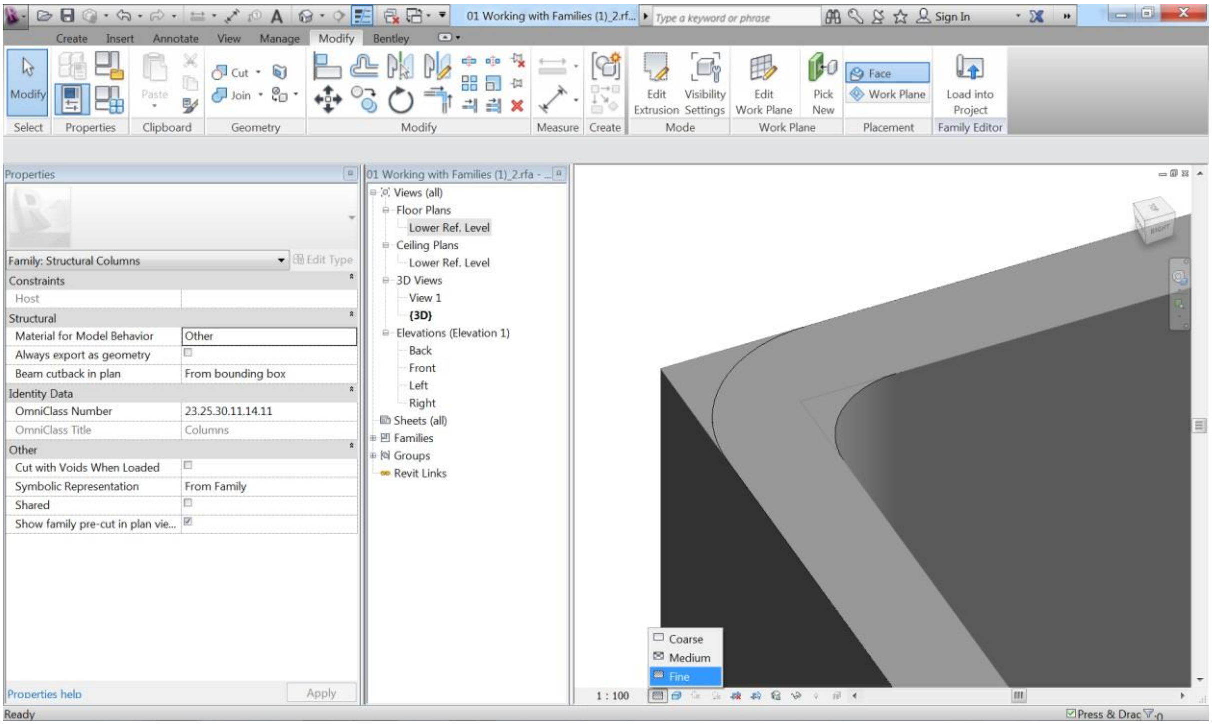
Then go back to the view and select the other extrusion. From the “Family Element Visibility Settings” window, deselect “Coarse” and “Medium” from the options.



Go back to the 3D view. Now the detail level of your model view might be “Medium”. As you just set, the filleted extrusion is in halftone.



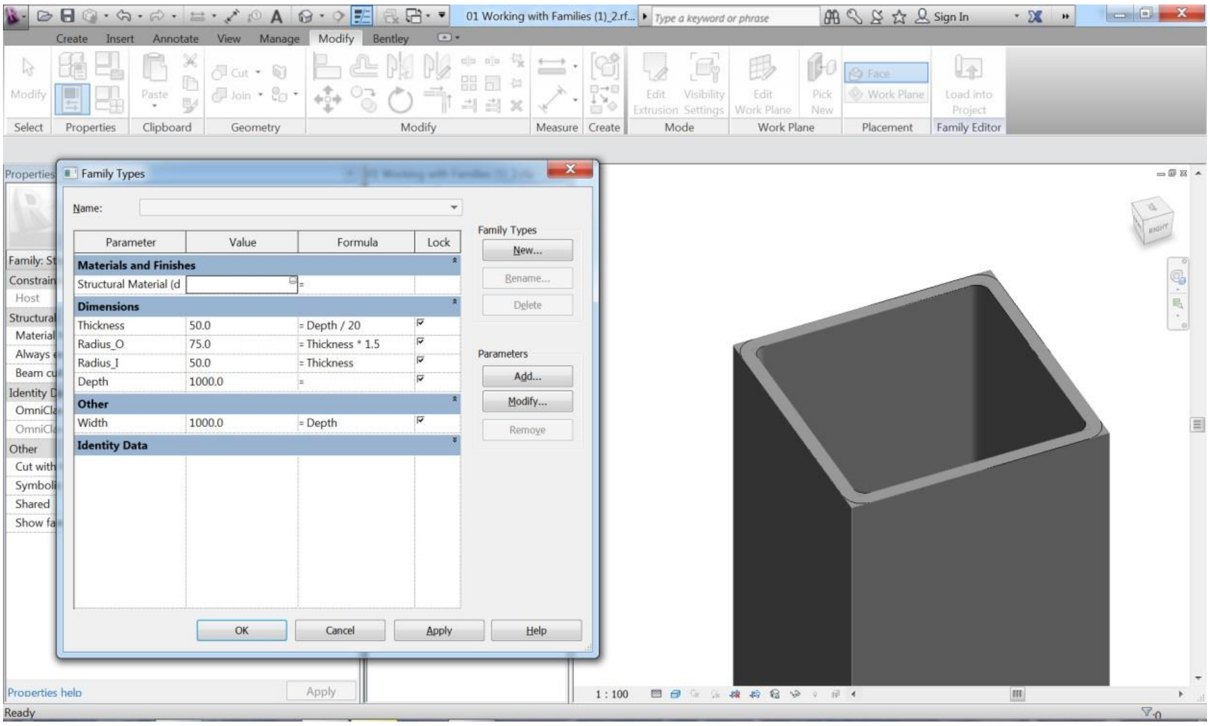
set the detail level to “Fine”. Then the right-angled extrusion will be in halftone.



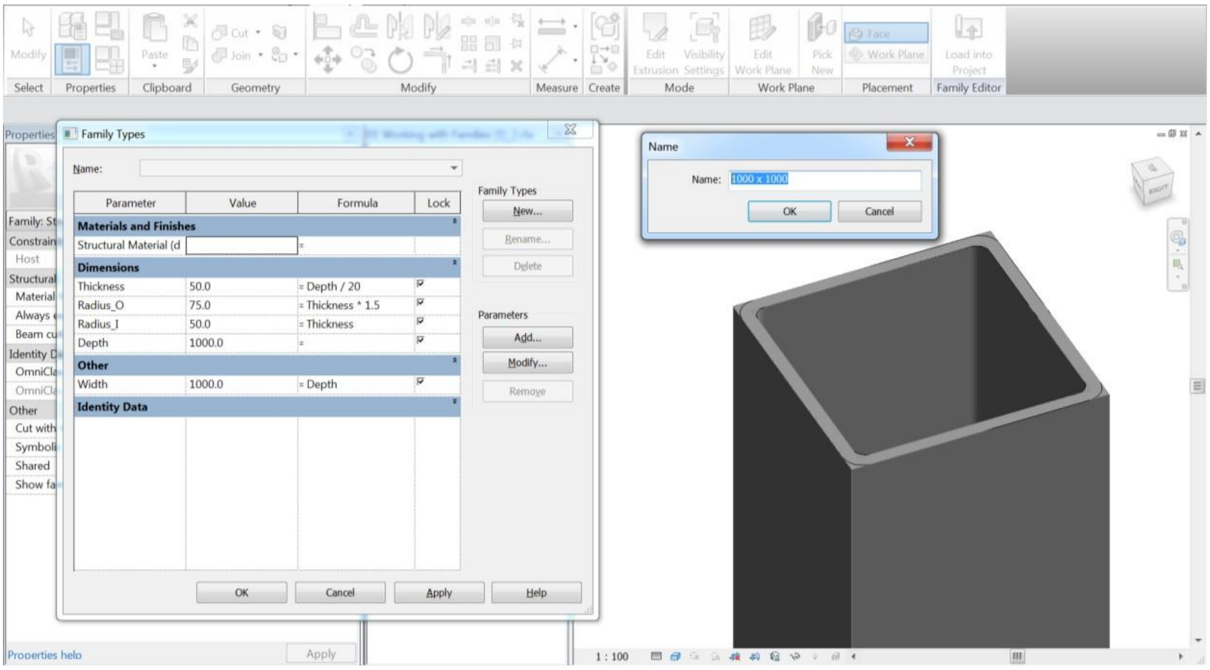
We will check how it works in a project in the later stage.

12.3.7 Type

From the “Family Types”, you can add types to your family. One of advantages of using a parametric modeling tool is that, you can use parametric frameworks to create multiple objects with different geometry.

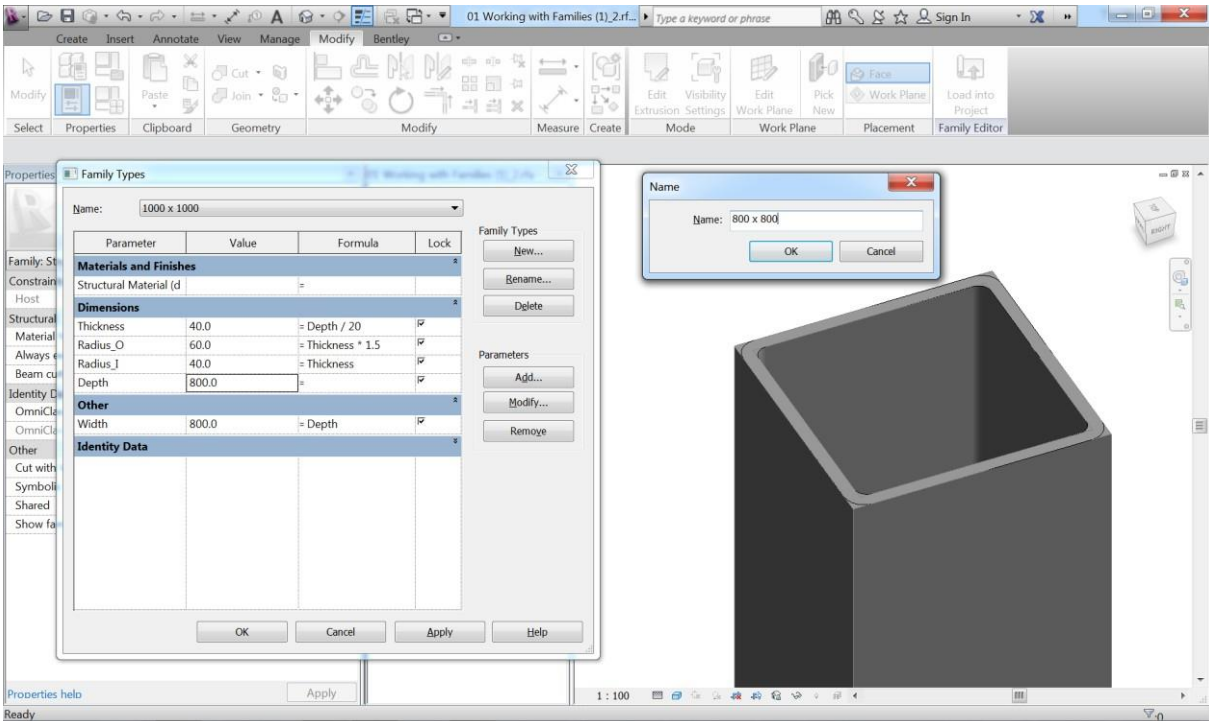


From “Family Types” window, set “Depth” parameter’s value to “1000”. Then click “New” button to add a type. Type in “1000 x 1000” for the name.

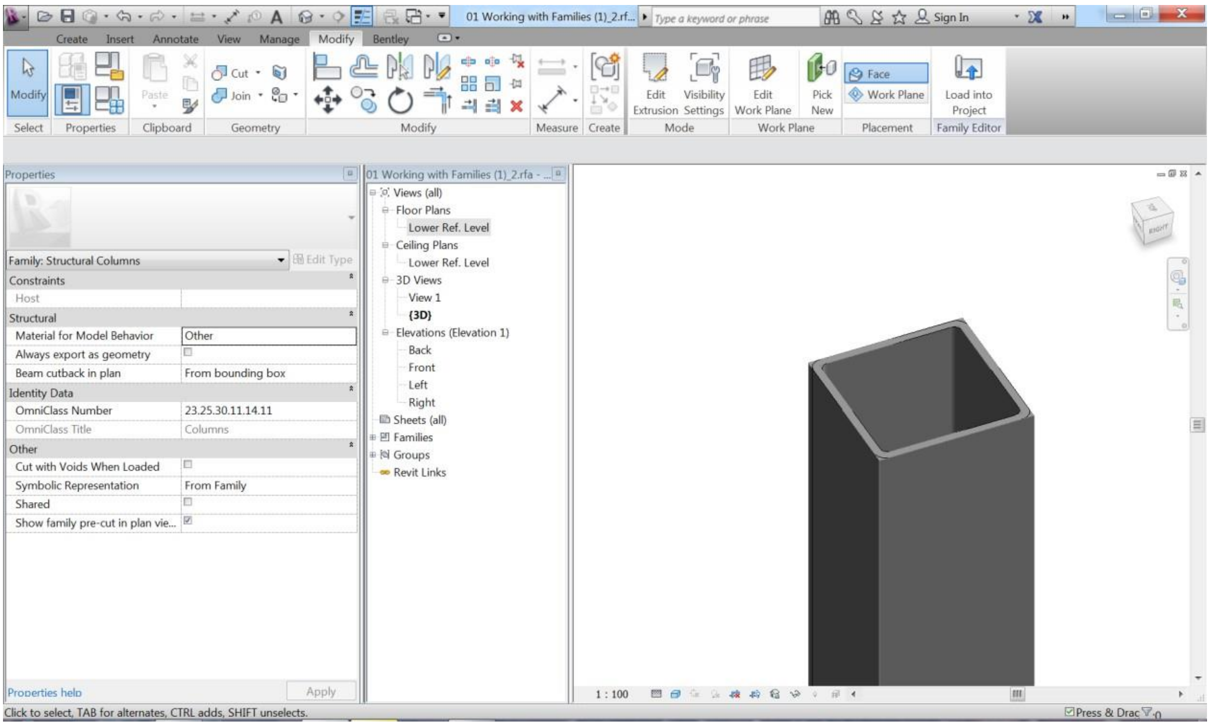


click “New” button and type in “800 x 800” for the name. Click “OK” button and set the

"Depth" parameter's value to "800". (note that you have to add a type first and then change the values.) Create another type "600 x 600" with the depth value of "600".

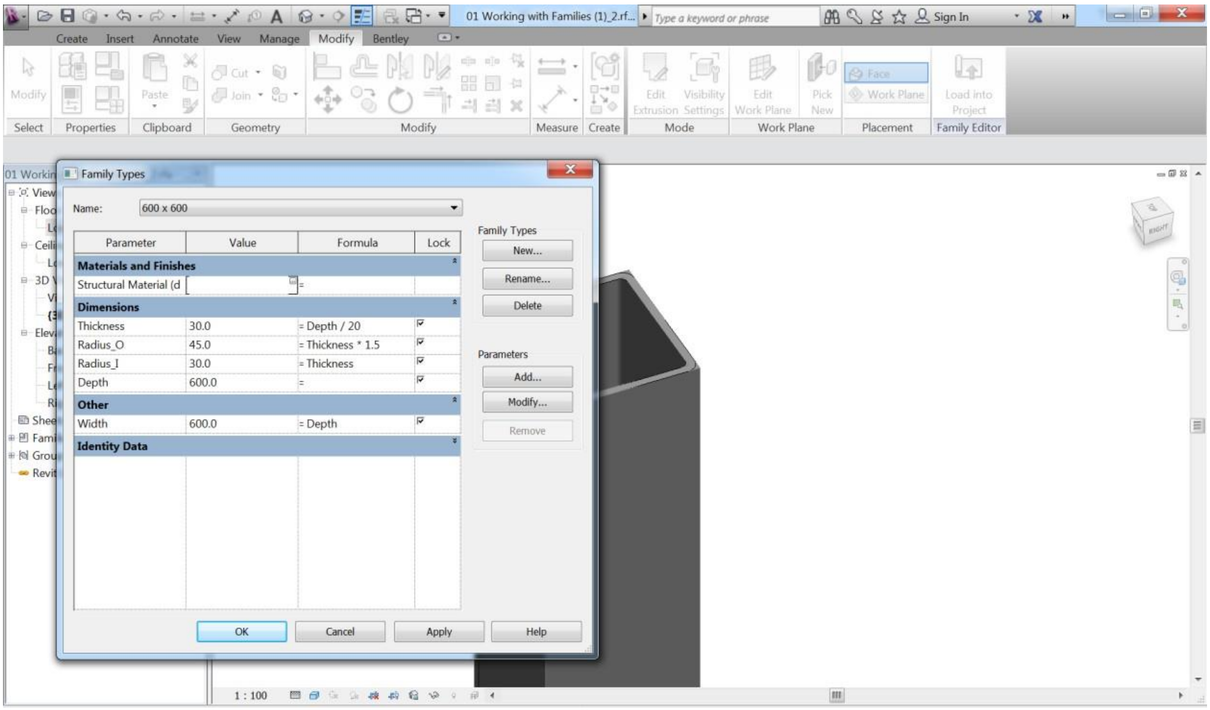


test your family by changing the type to different ones. You can also change the formulas to have different parametric rules and geometric results.

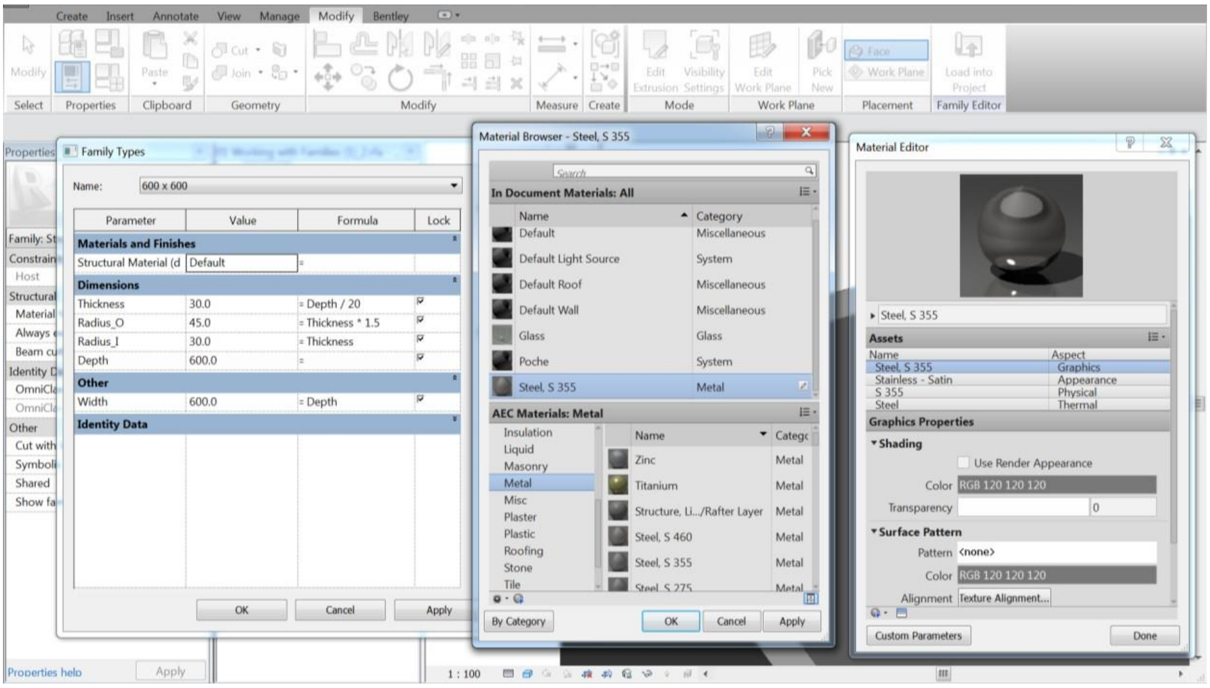


12.3.8 Material

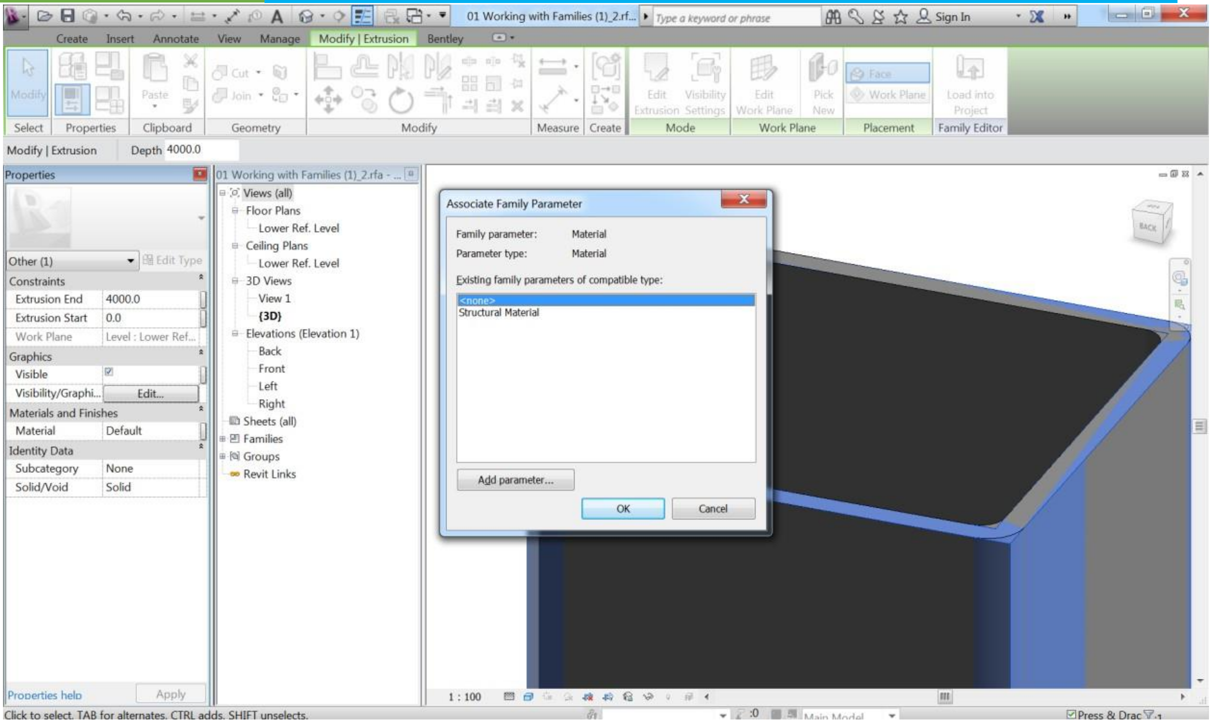
In the structural column family template, "Structural Material" parameter is set by default. From the "Family Types", click the small button on the right-top corner of "Value".



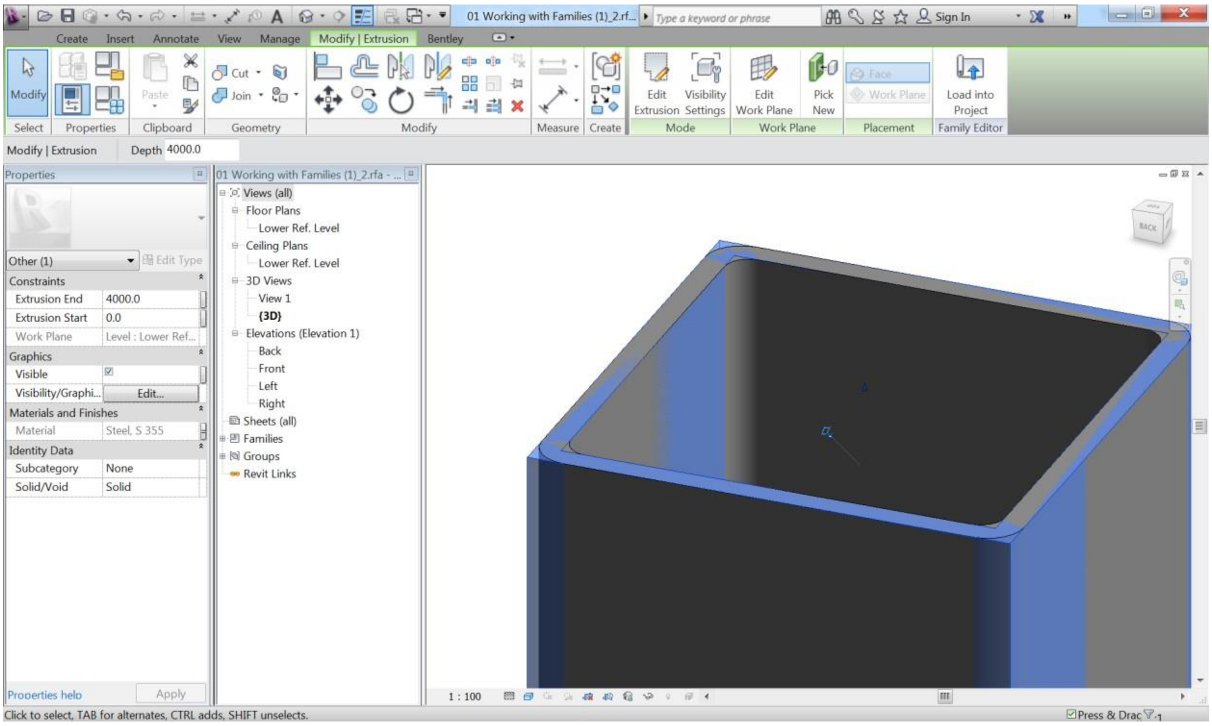
Then "Material Browser" window will appear. select "Metal" under "AEC Materials" then double click "Steel, S 355" to add it to the list above. Select "Steel, S 355" from the list above and click "OK" button. apply the same material to the other types.



go back to the 3D view, and select one of the extrusions. From the properties window, click the button next to "Material" under "Material and Finishes". The gray button in the 3rd column. "Associate Family Parameters" window will appear. Select "Structural Material" from the list.

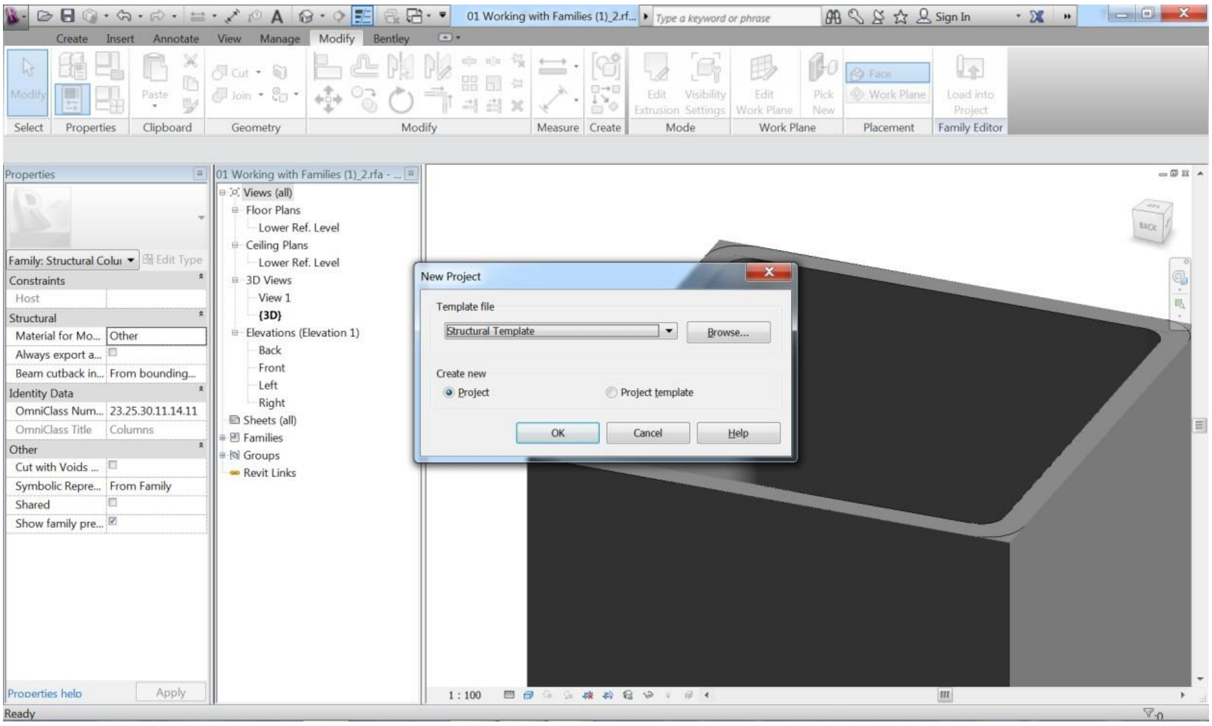


Then the structural material, "Steel S, 355" will be applied to the extrusion. repeat the same step to the other extrusion. You may want to save your steel column family.

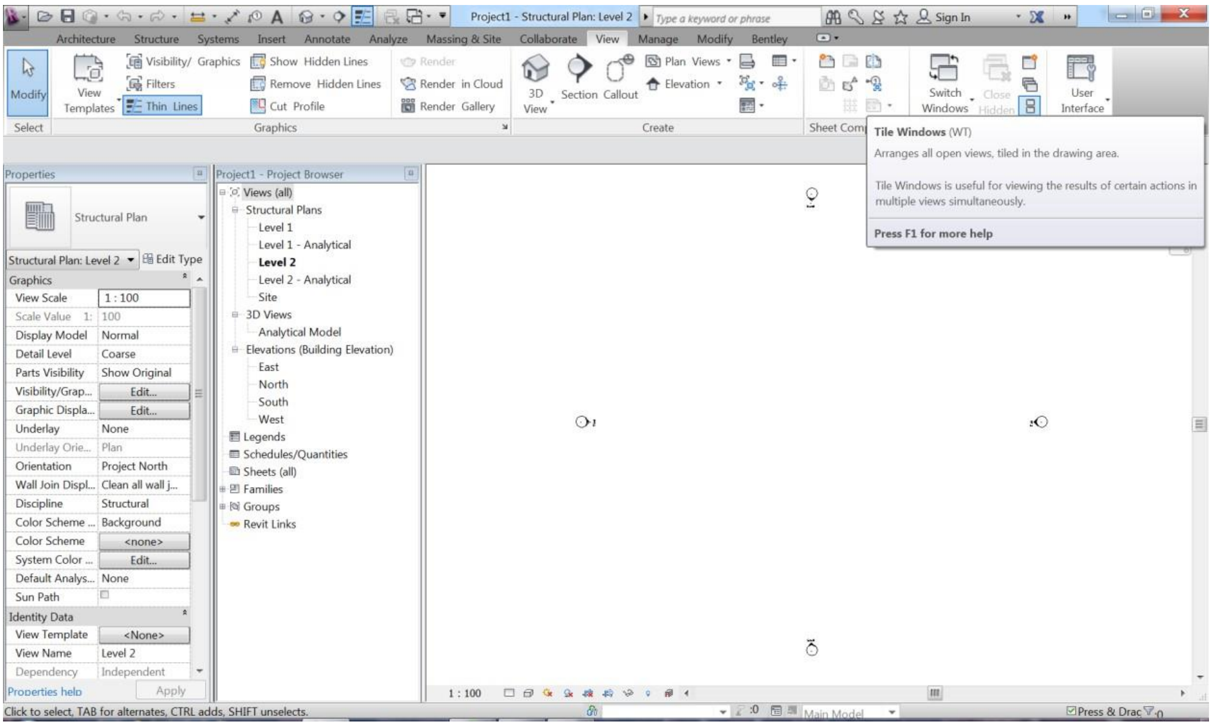


12.3.9. Load into the project

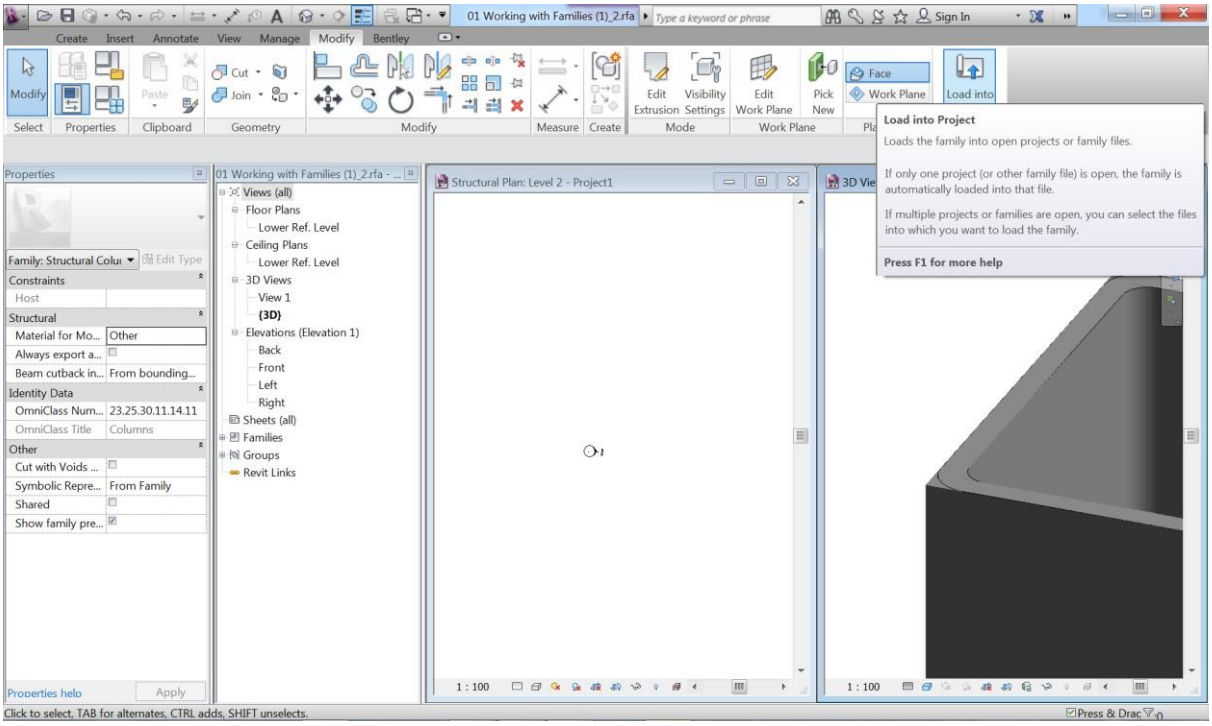
Lastly, you need to check if the family works well in a project file. start a new project with "Structural Template".



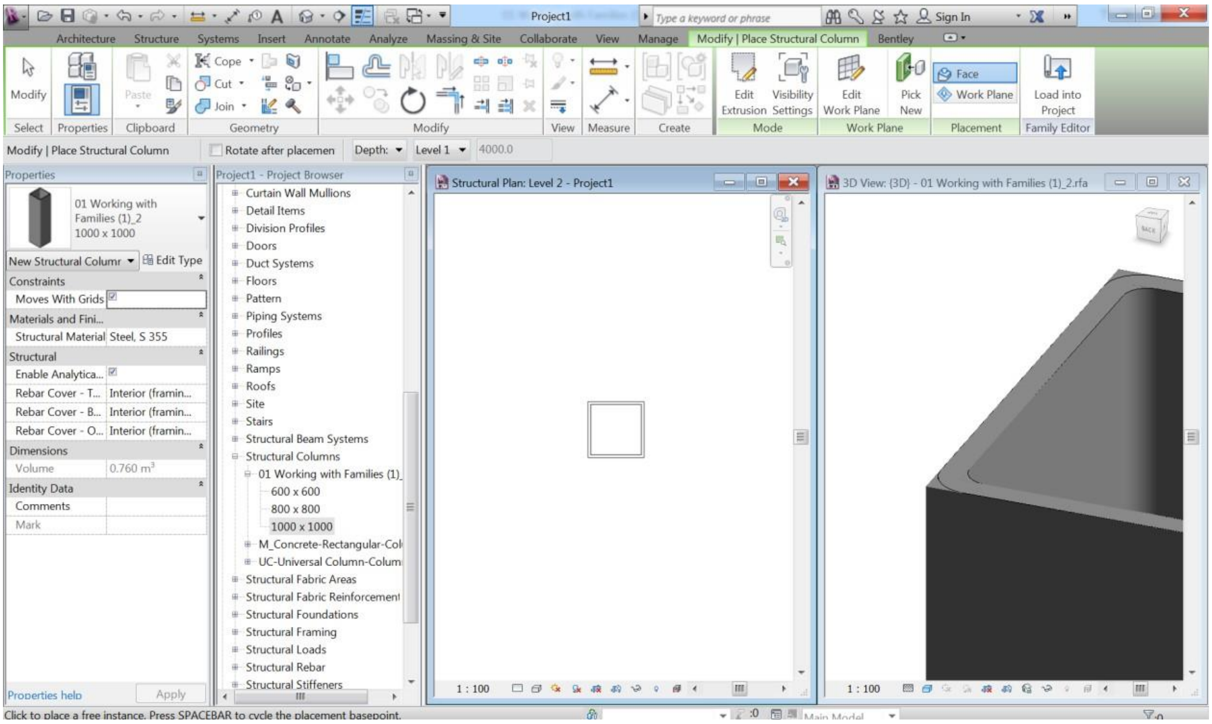
Once the new project begins, go to “View” tab and click “Tile Windows” button.



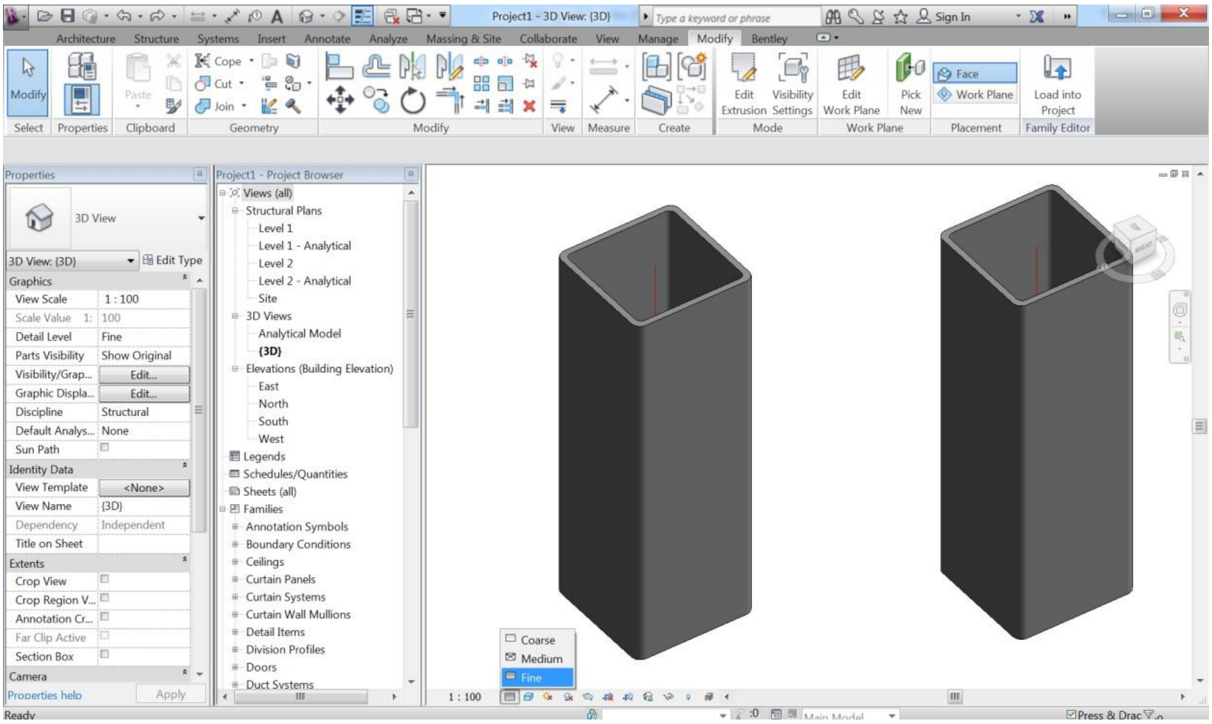
Then the model view sill be equally divided for the family editor’s view. Click any spot in the 3D view of the family editor and click “Load into Project” button.



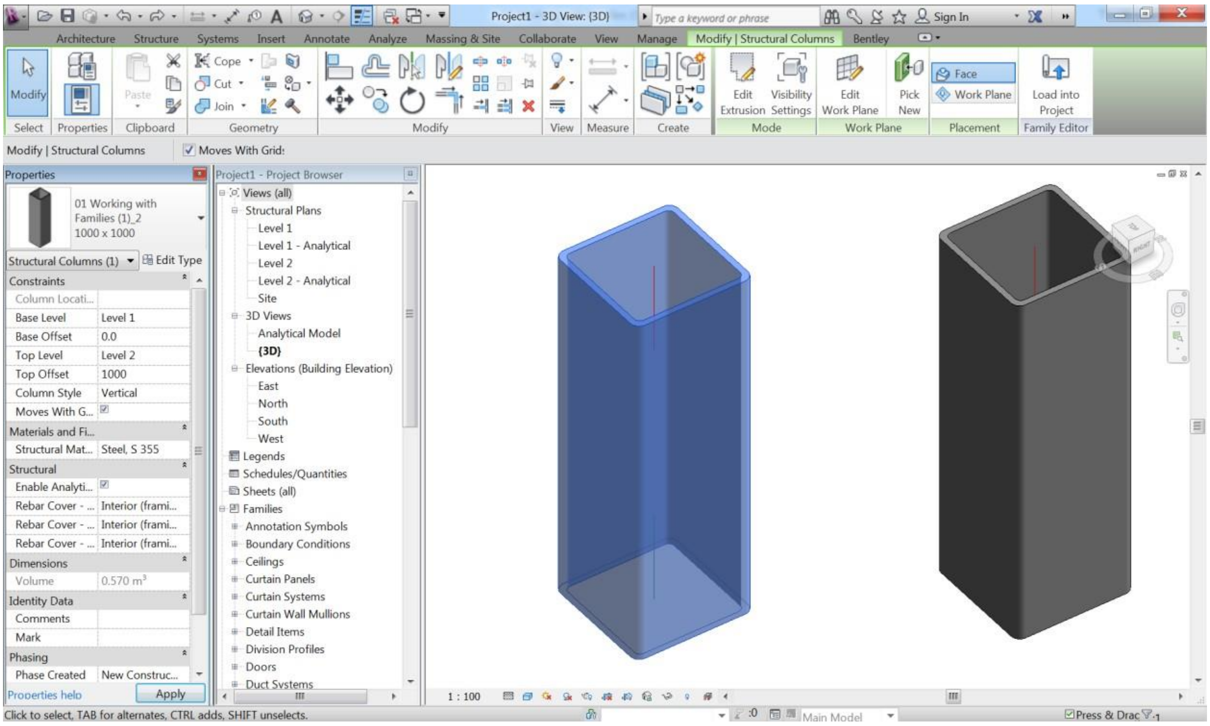
You will be automatically taken to the project view to place a column instance. place few columns in the project.



Go to 3D view and change the detail level of the view to check if the column geometry changes accordingly.

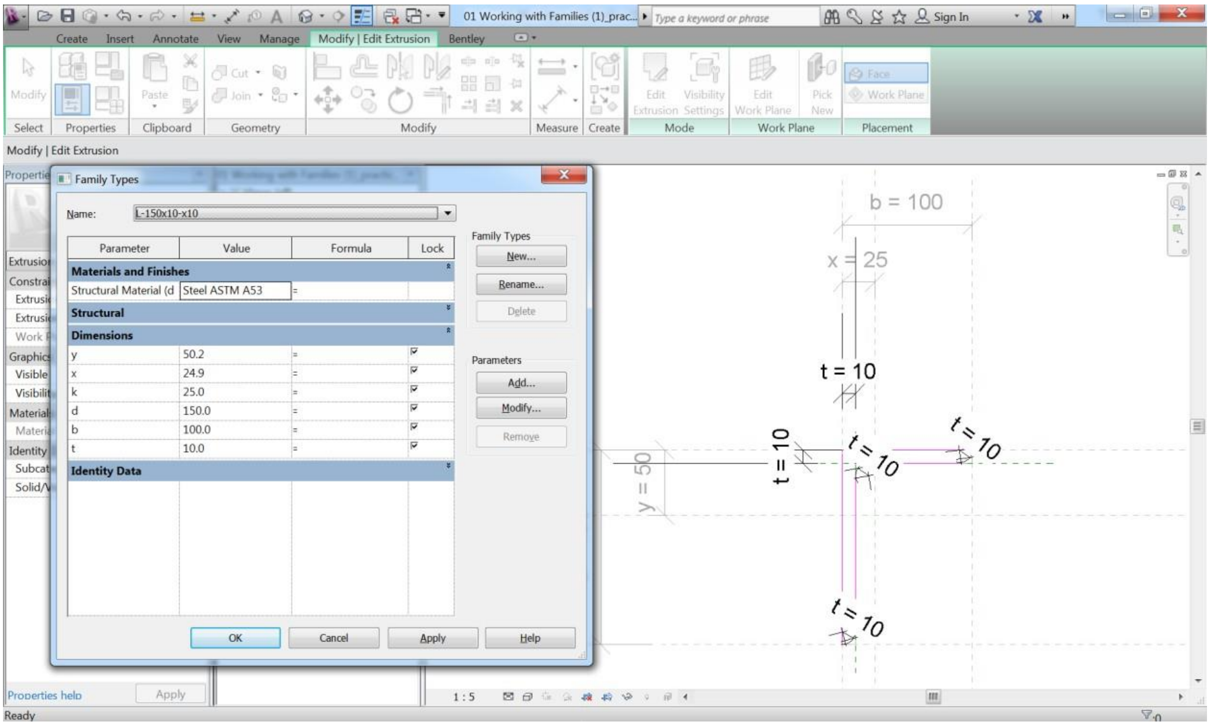


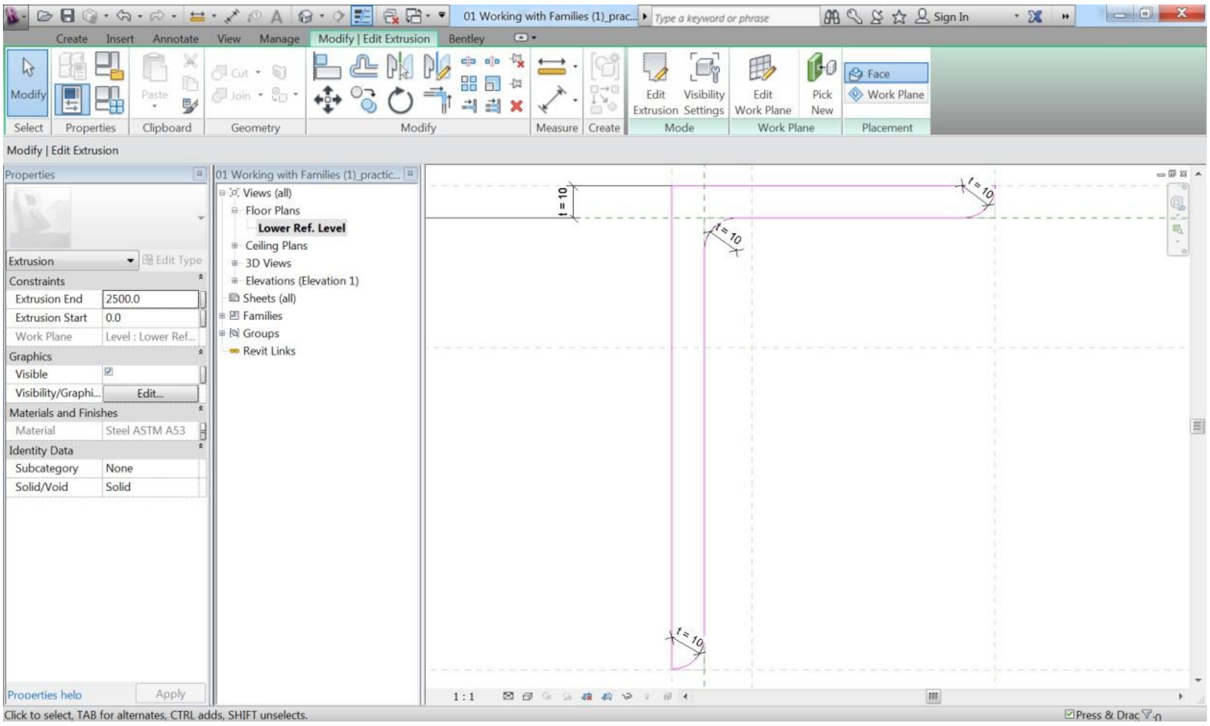
You can also change the instance parameters from the properties to see if it works well. You may want to click "Edit Type" button and try different values for the type parameters.



12.4 Modeling Exercise

refer to the previous chapter and the images below to create L-Angled Column





Disclaimer: The process described in this tutorial is only to help the readers understand the concept of the topic and how to use related functions. This tutorial does not offer the optimized way to implement a project in Revit.

CHAPTER 13 - Customize Column Family

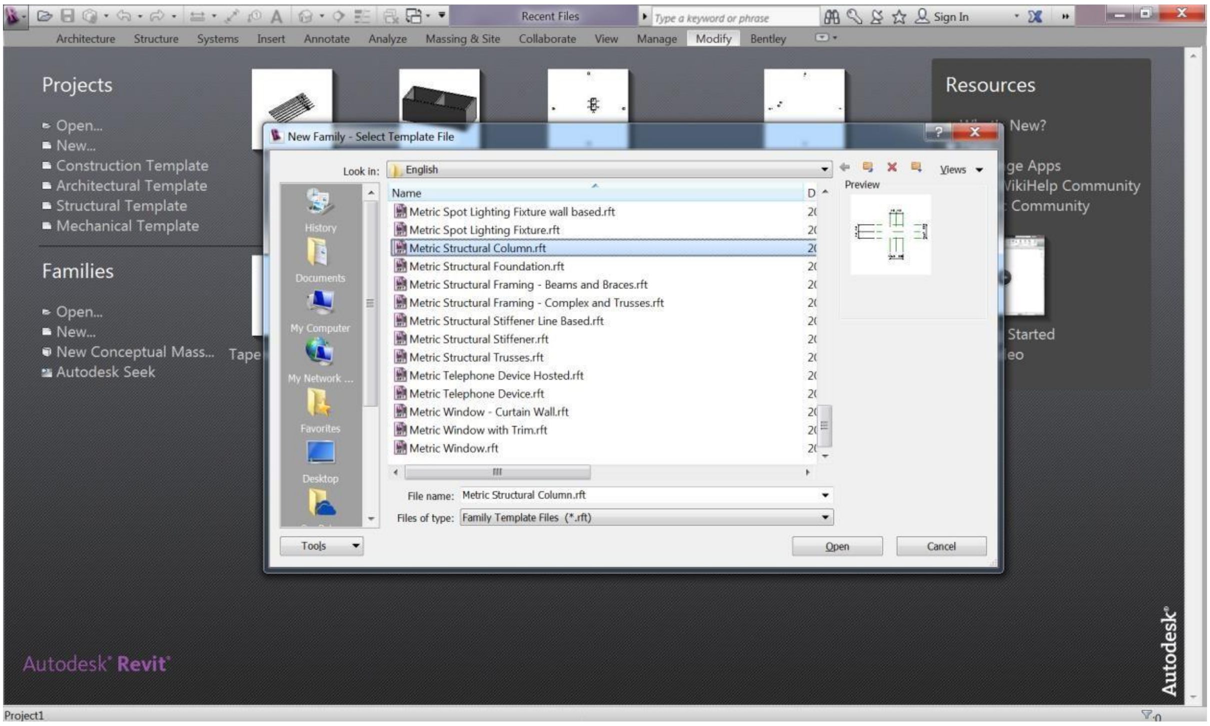
In this section, you will learn

- How to create a Tapered Concrete Column Family

<Image from “http://www.architectmagazine.com/photos/brockman-hall-for-physics>

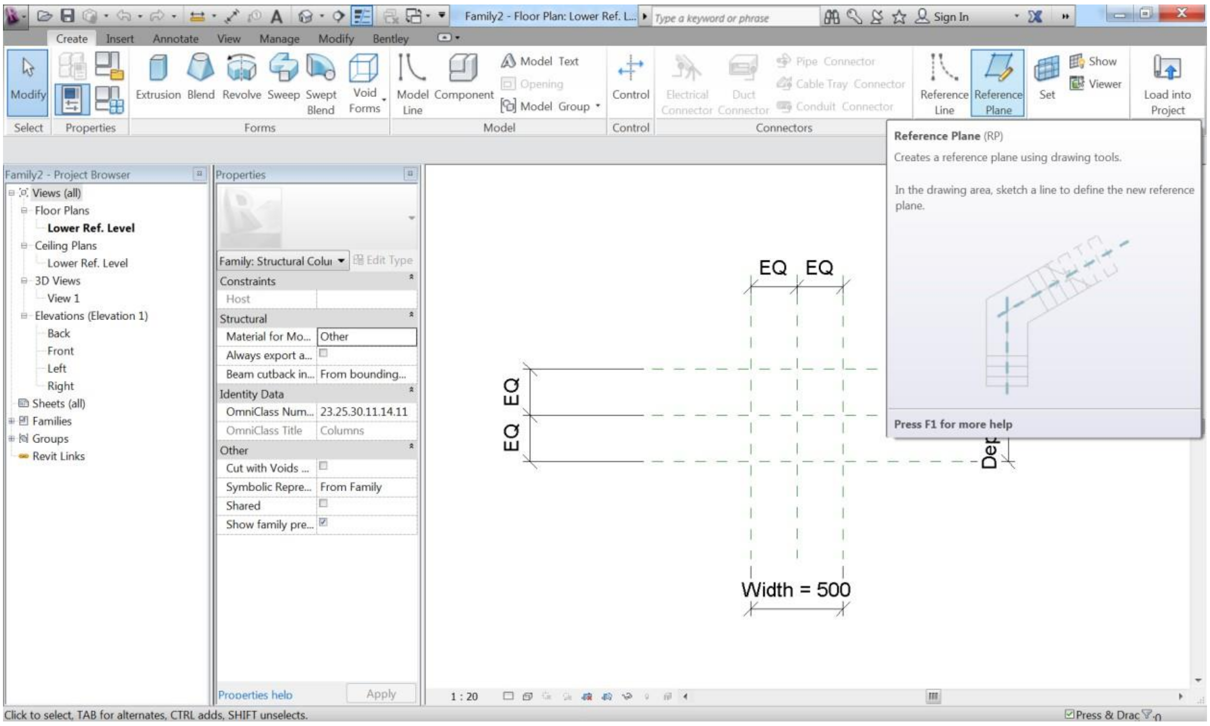
13.1 Template

Go to “File>New>Family” and select “Metric Structural Columns.rft” file for the template.

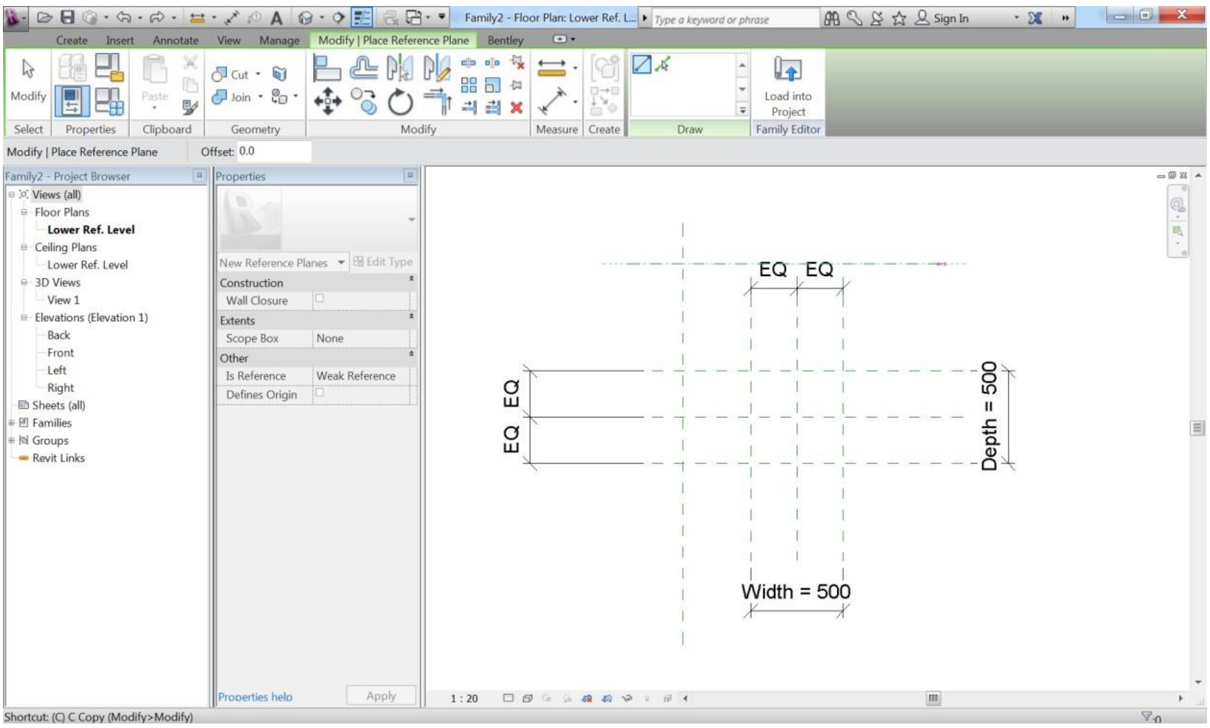


13.2 Reference Plane

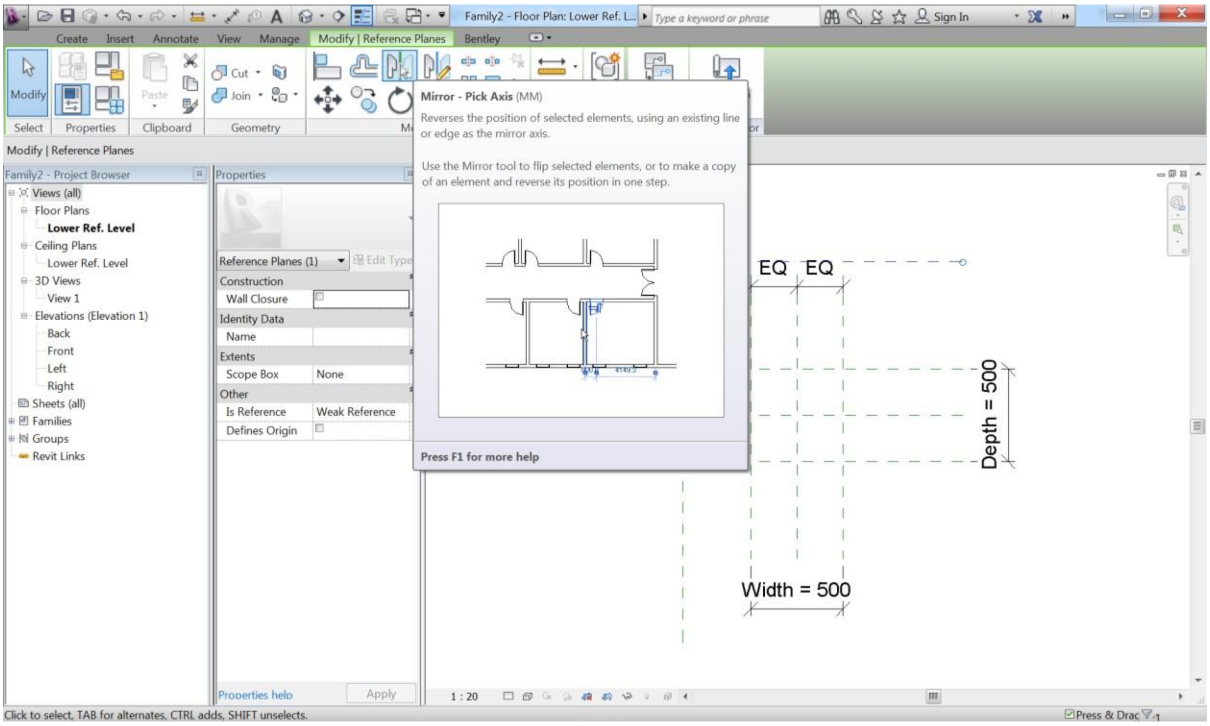
To create tapered column, you need to have two different profiles for the top and the bottom. In order to do so, we need a separate set of reference planes. Once family editor begins, go to "Create" tab and select "Reference Plane".



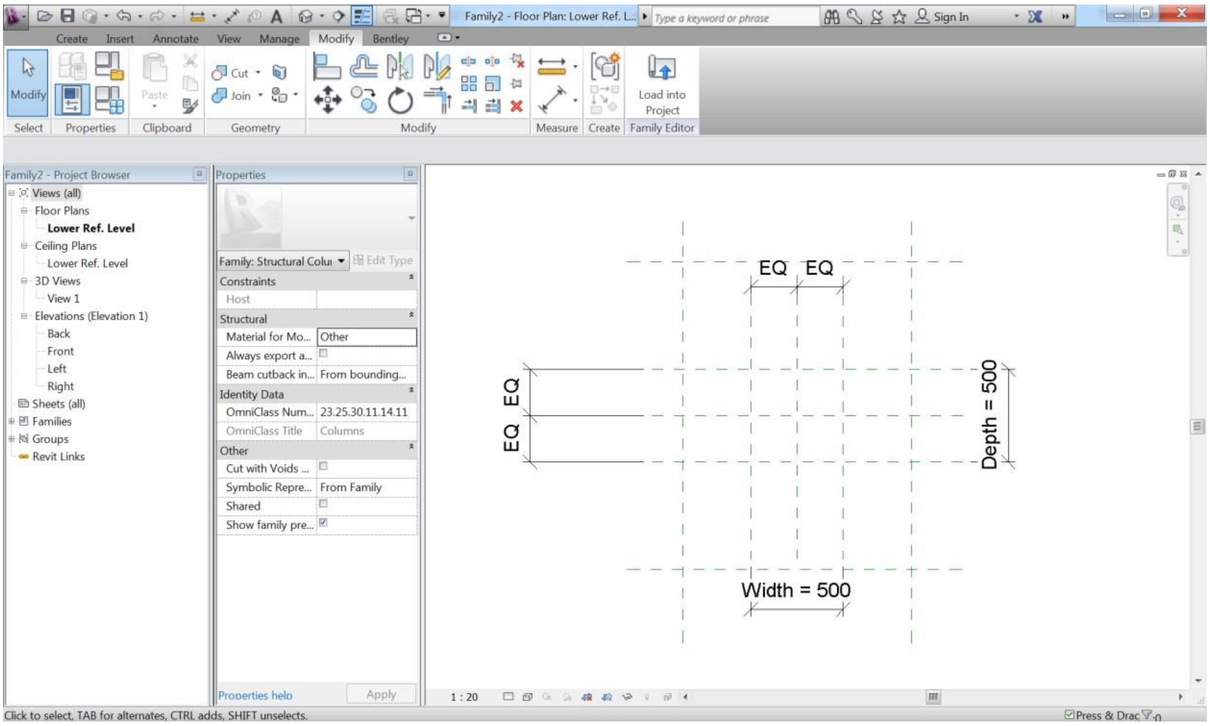
Draw two reference planes one on the left and one on the top of existing reference planes.



Select one of the new reference plane and go to "Modify" tab and select "Mirror – Pick Axis" button. Pick the center reference plane in parallel.

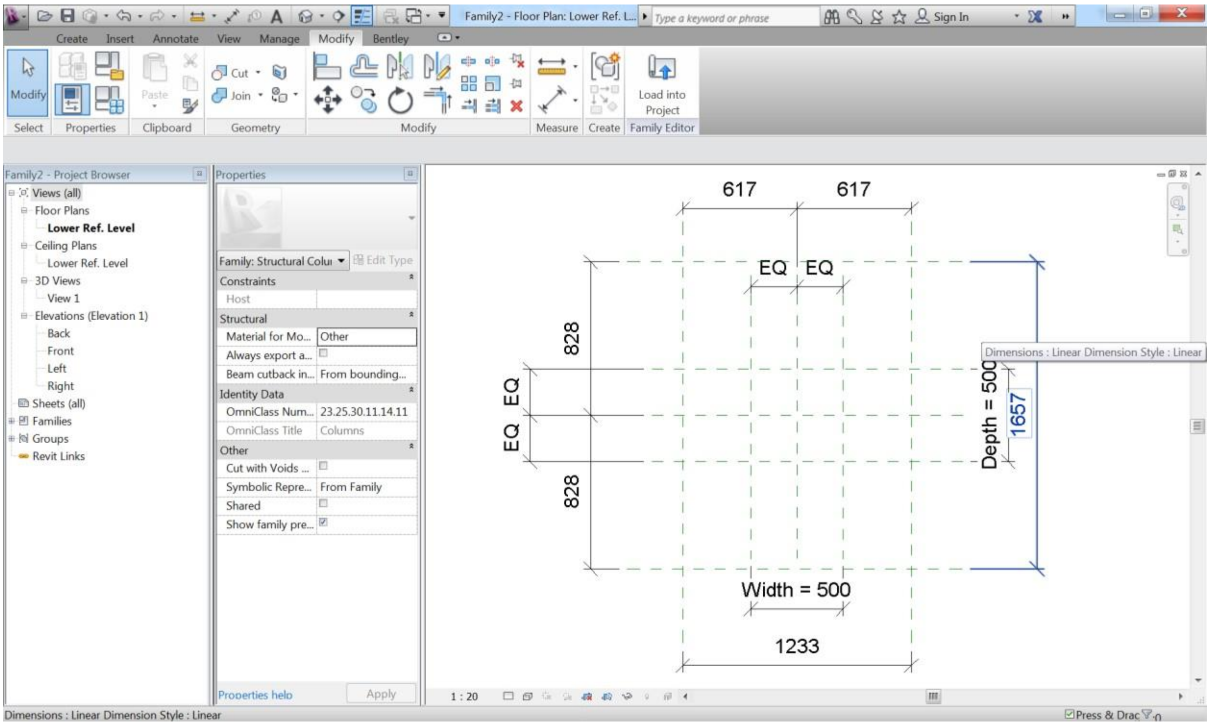


Mirror both reference planes as the image below shows.

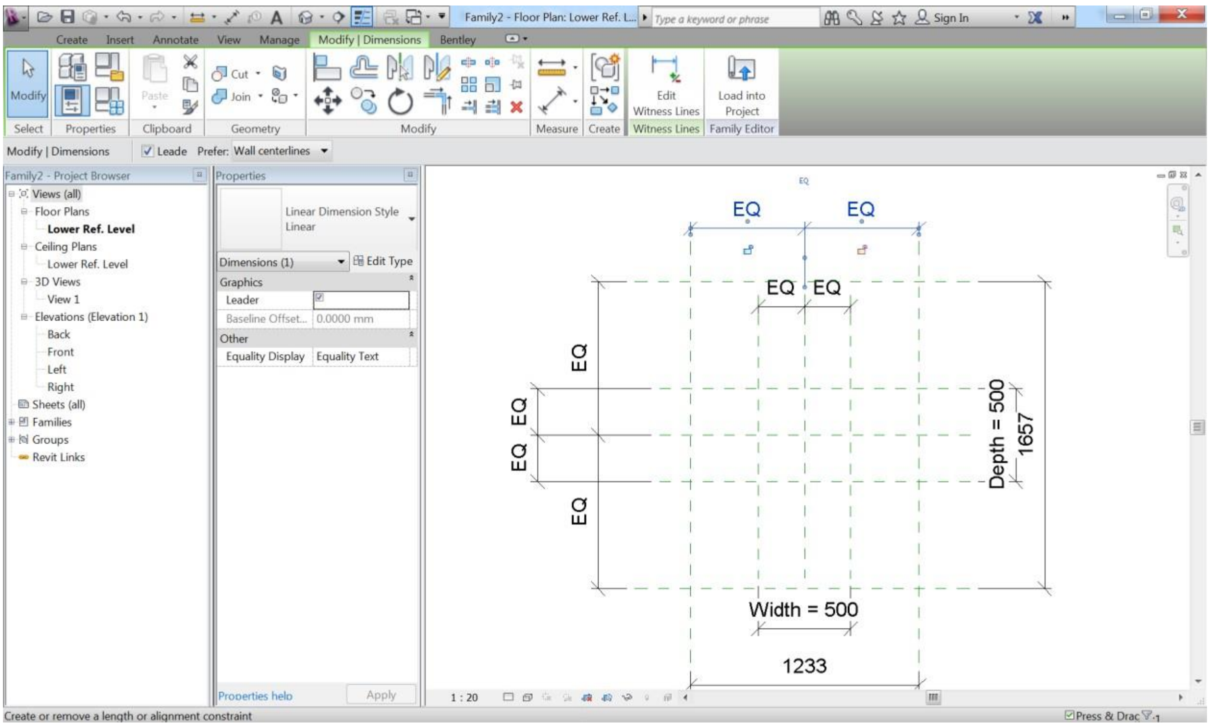


13.3 Parametric Dimensions

Add dimensions as the image below shows.

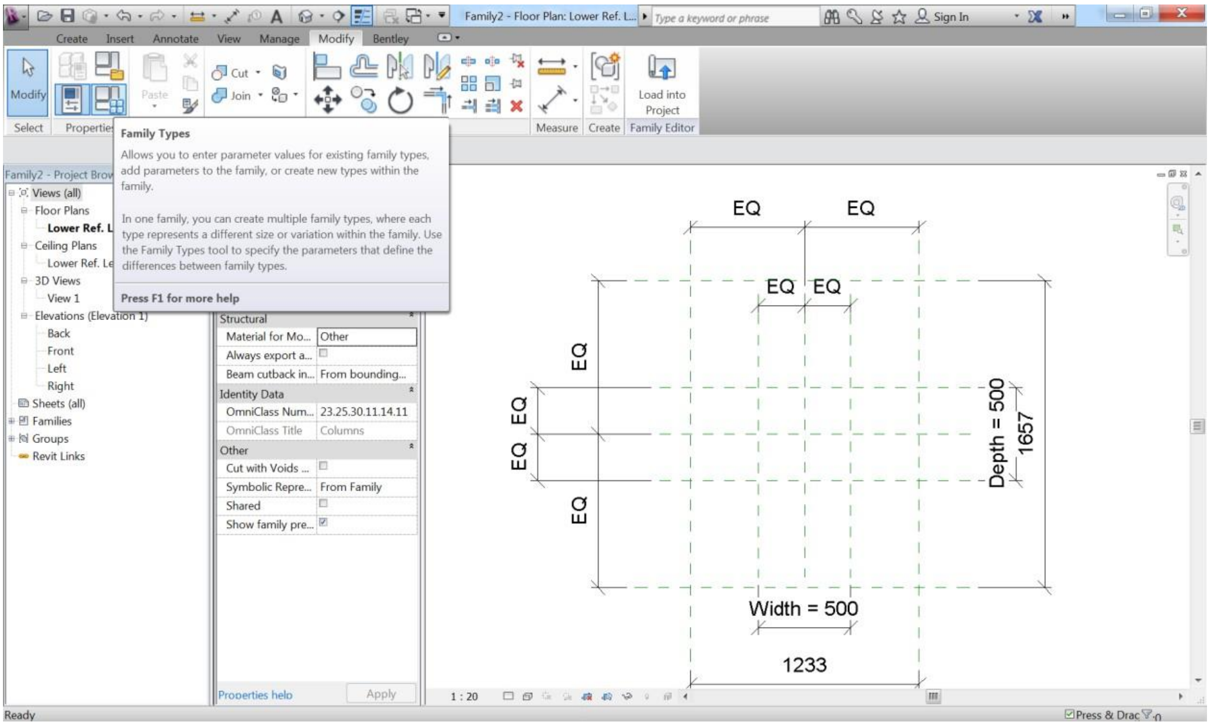


Then turn two of the dimensions to "EQ" dimension.

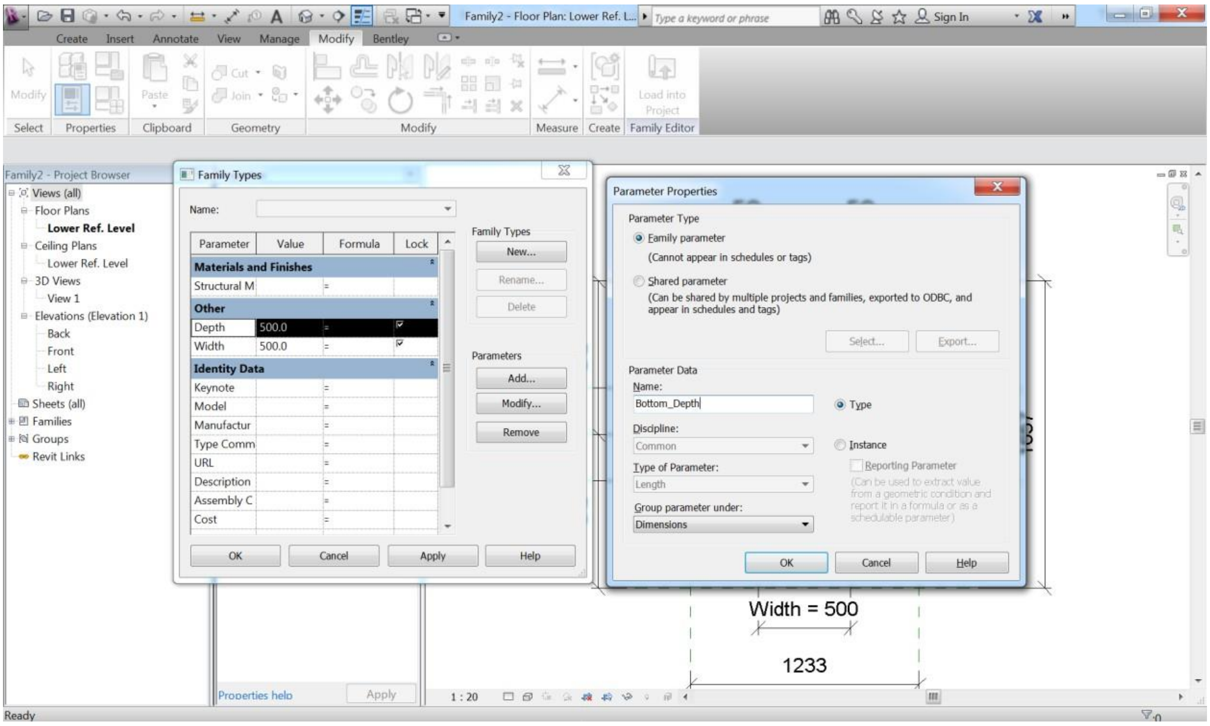


13.4 Creating Parameters

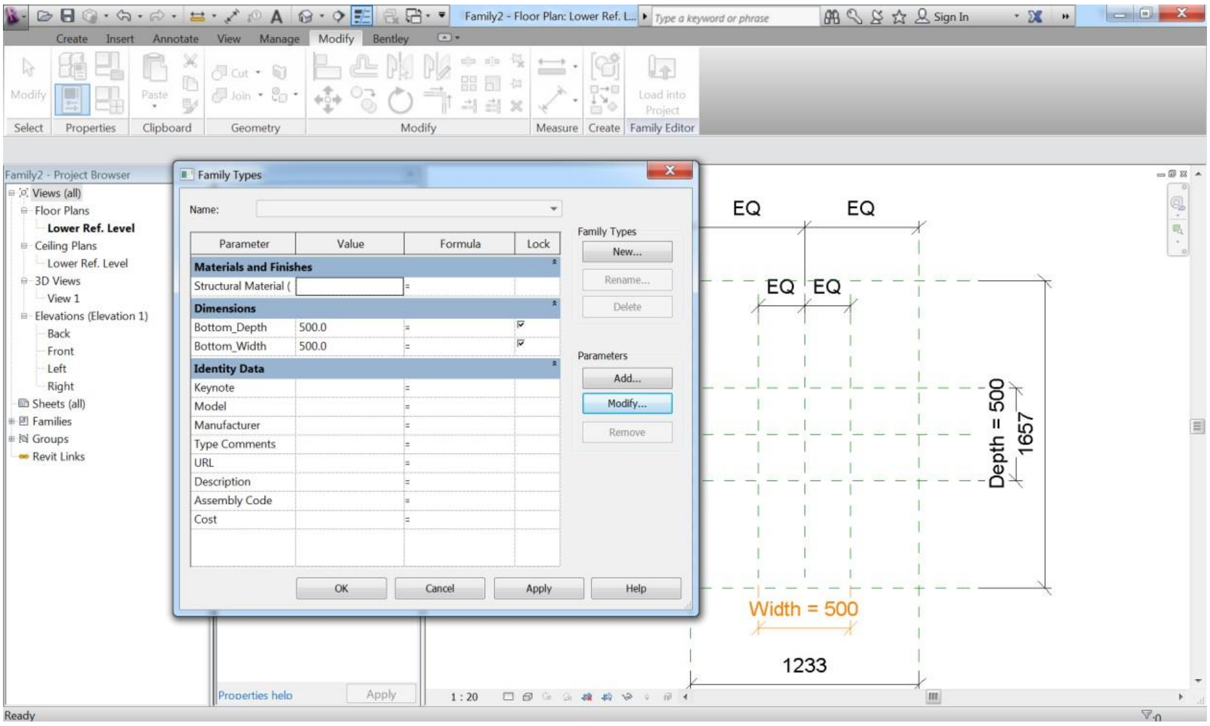
Click "Family Types" button to edit and add parameters.



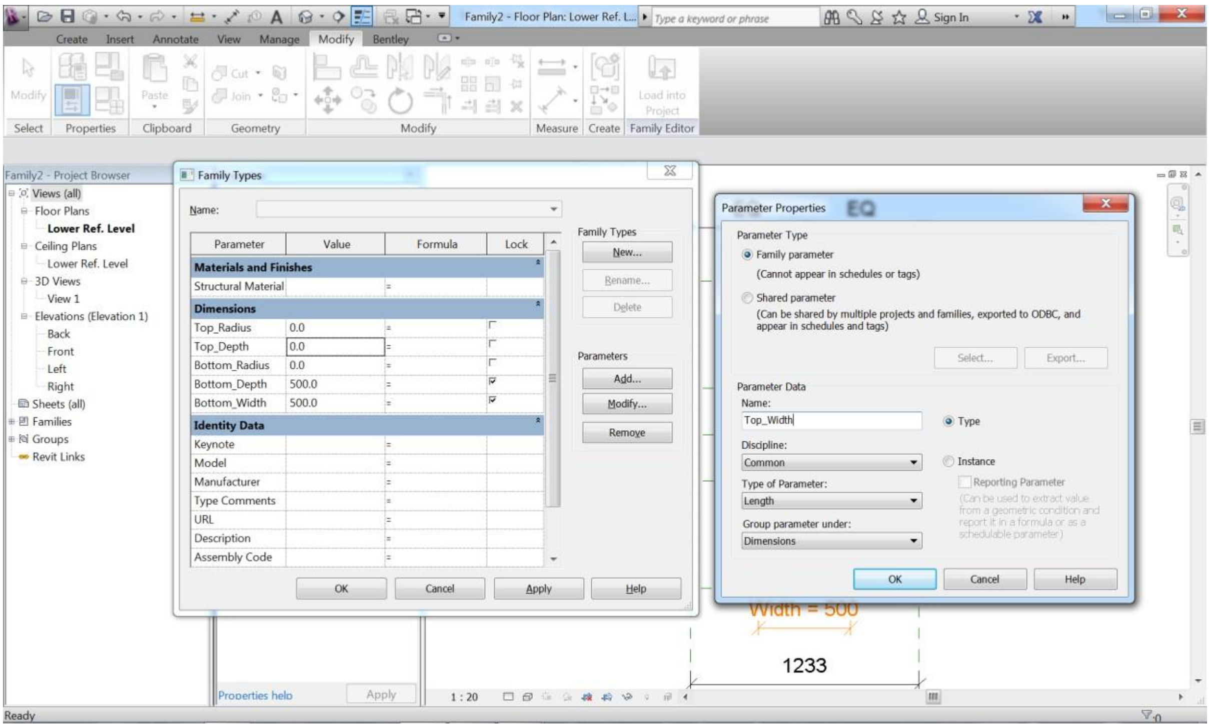
From the appeared window, select "Depth" parameter and click "Modify" button under "Parameters". Change the name to "Bottom_Depth" and "Group~" to "Dimension".



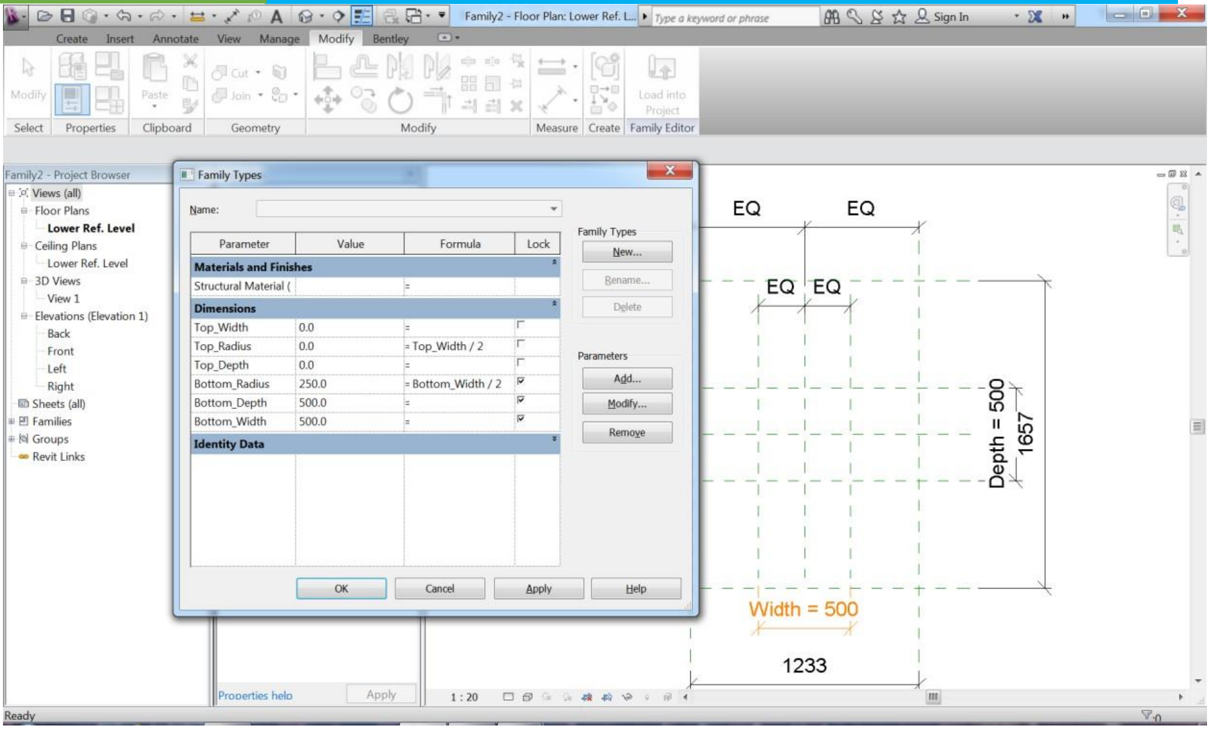
Repeat the same step to the other parameter.



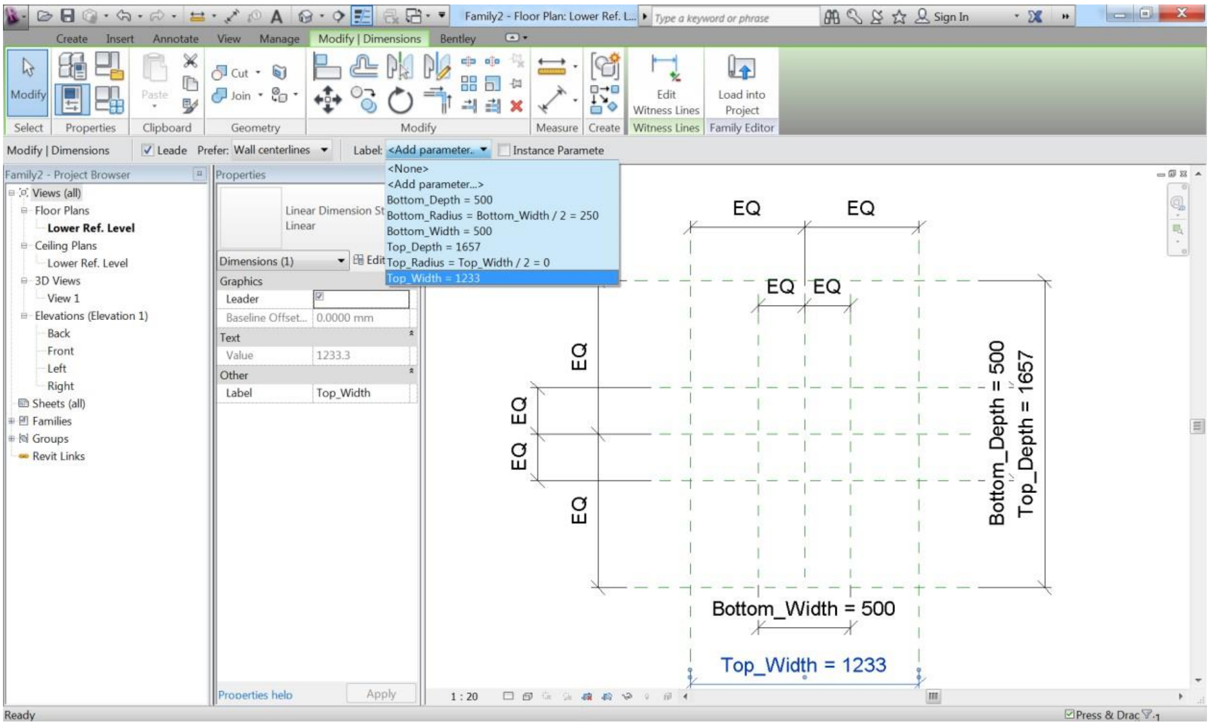
Add "Bottom_Radius", "Top_Radius", "Top_Depth", and "Top_Width" parameters to the family. All parameters are "Common" and "Length" under "Dimensions" group.



For "Formular" of "Top_Radius", type in "Top_Width/2" and for that of "Bottom_Radius", type in "Bottom_Width/2".

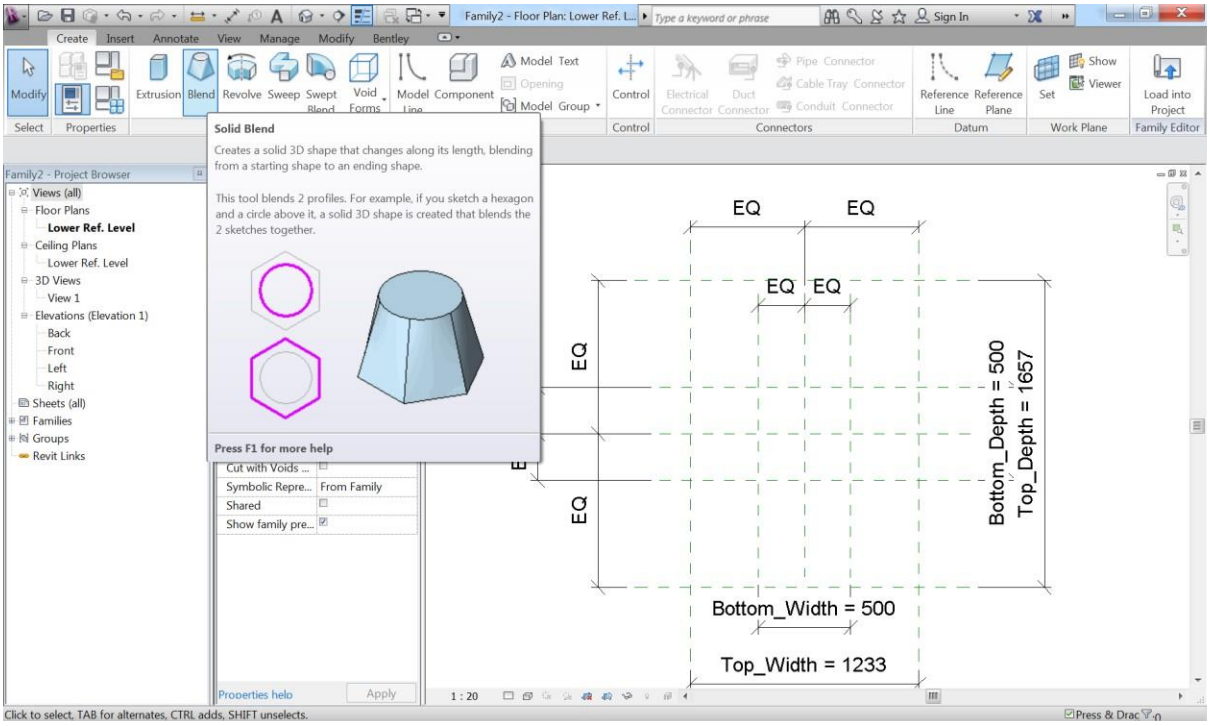


Associate the parameters to dimensions as the image below shows.

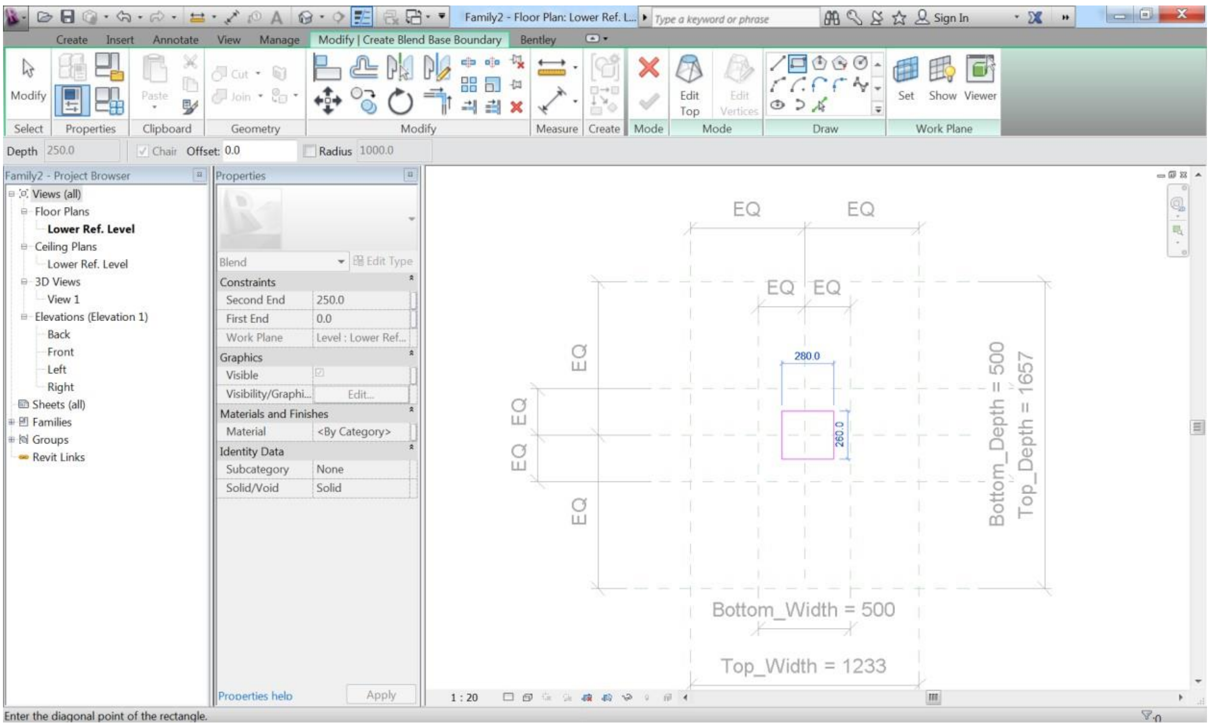


13.5 Geometry – Solid Blend

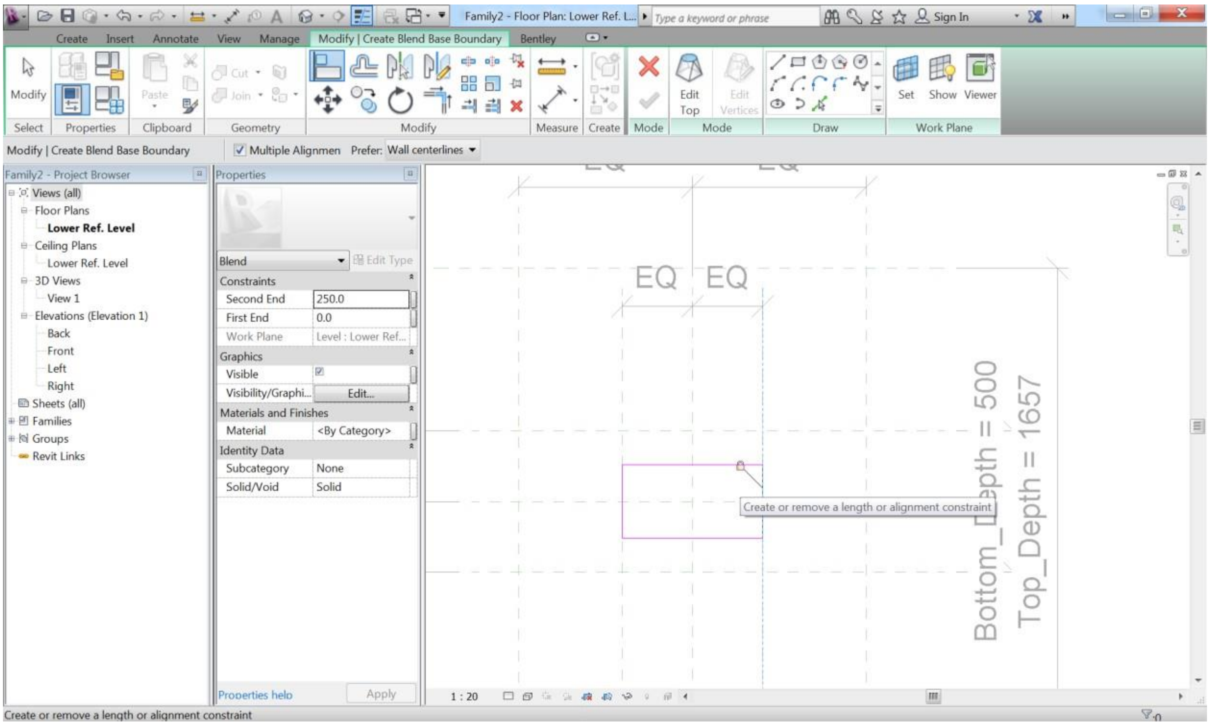
To create a solid with different profiles on the top and at the bottom, you have to use "Blend" function under "Create" tab.



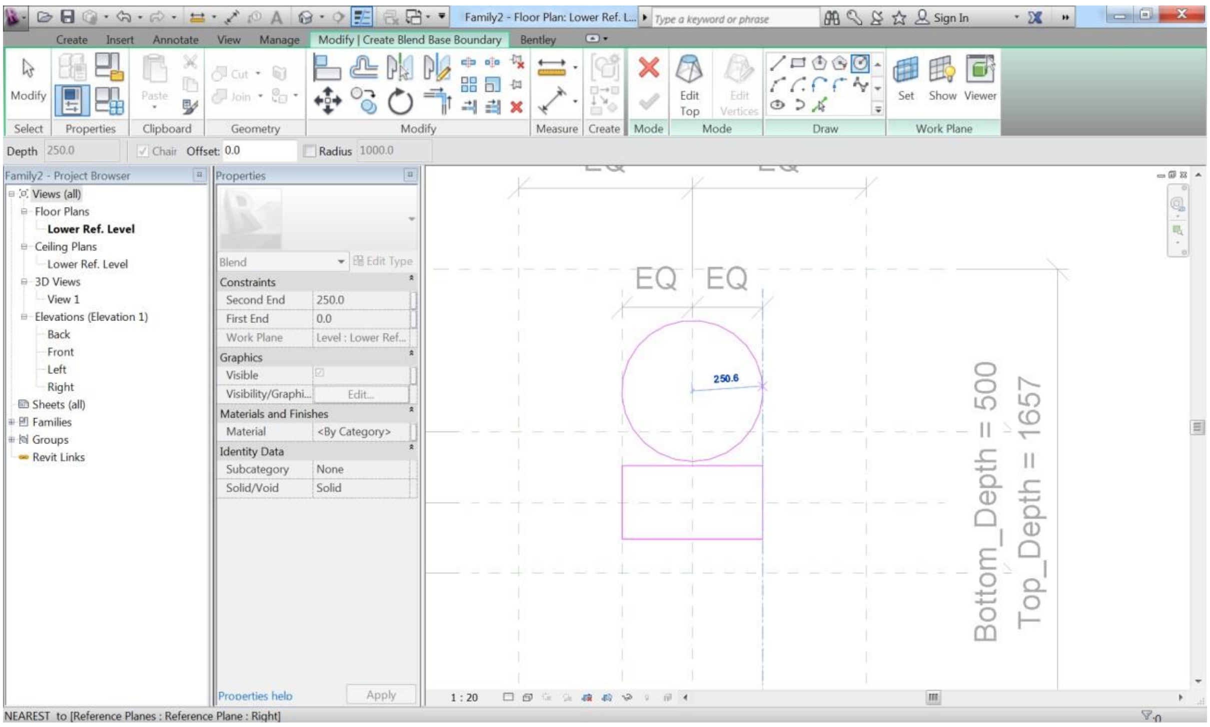
Once you run the function, you will be in the sketch mode for the bottom profile. Select "Rectangle" in the "Draw" group and draw a rectangle in the middle.



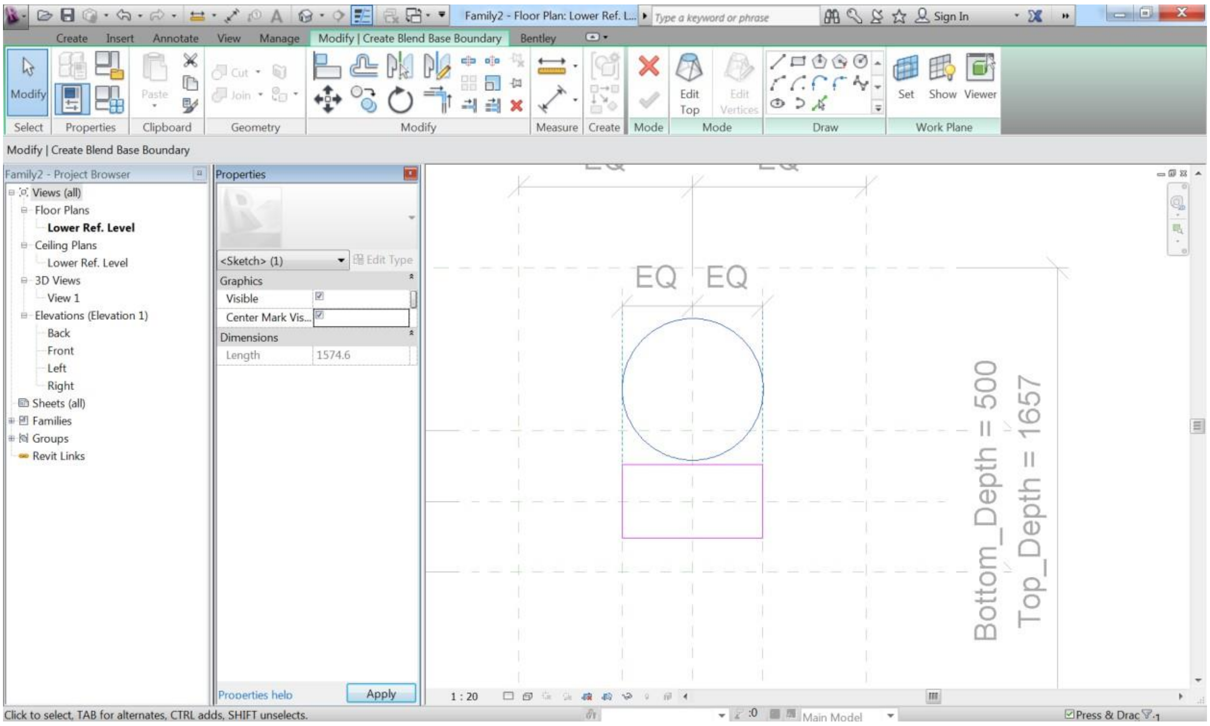
"Align" and constrain the left edge and the right edge to each corresponding reference planes.



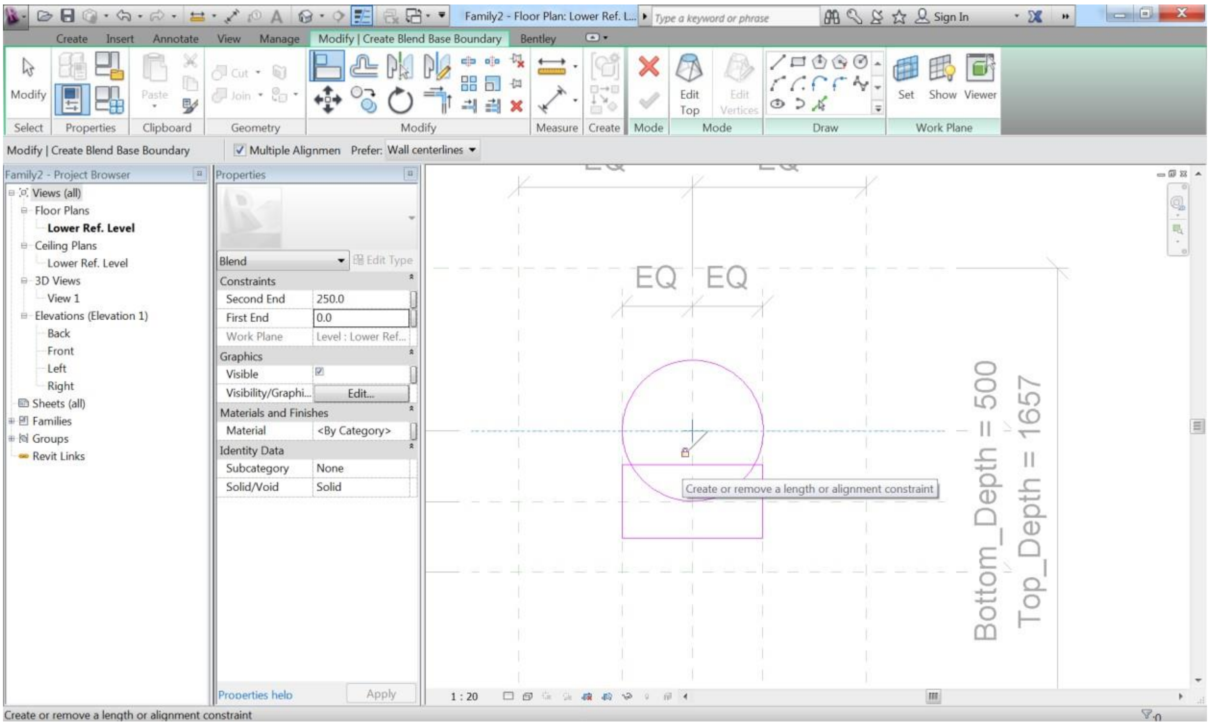
Select "Circle" in the "Draw" group and draw a circle. click on the vertical center reference plane for the center of the circle.



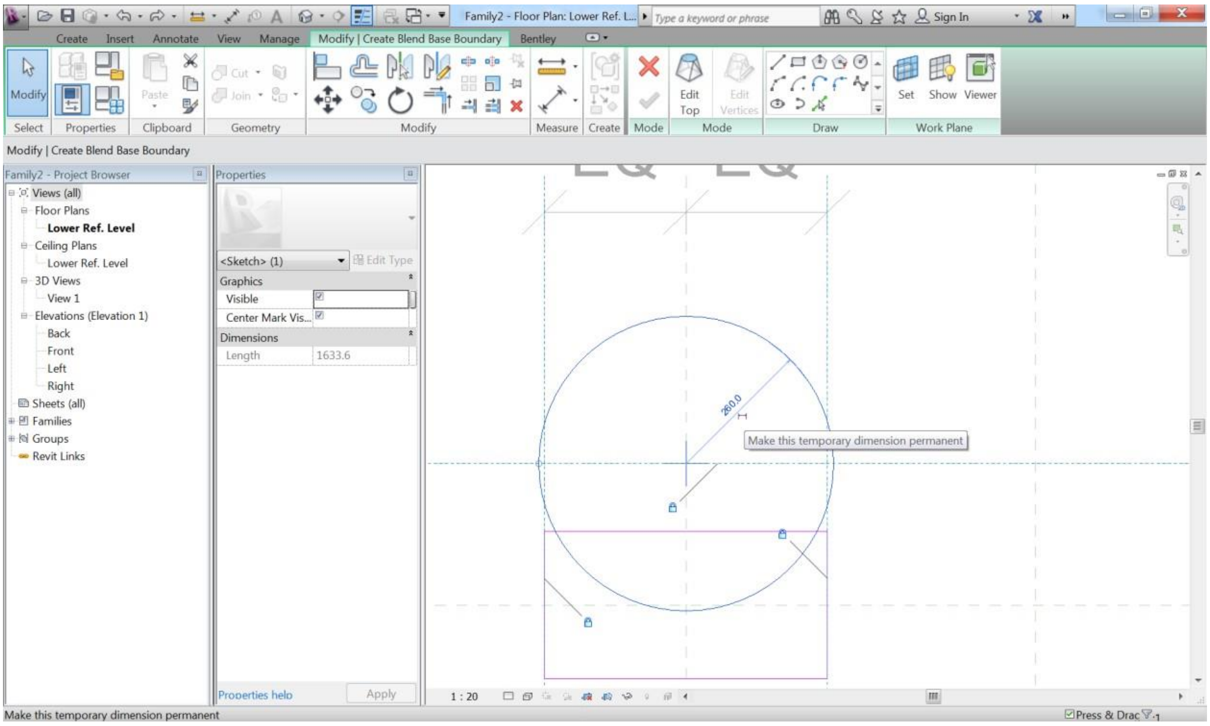
Select the circle and go to the properties to check “Center Mark Visible” option.



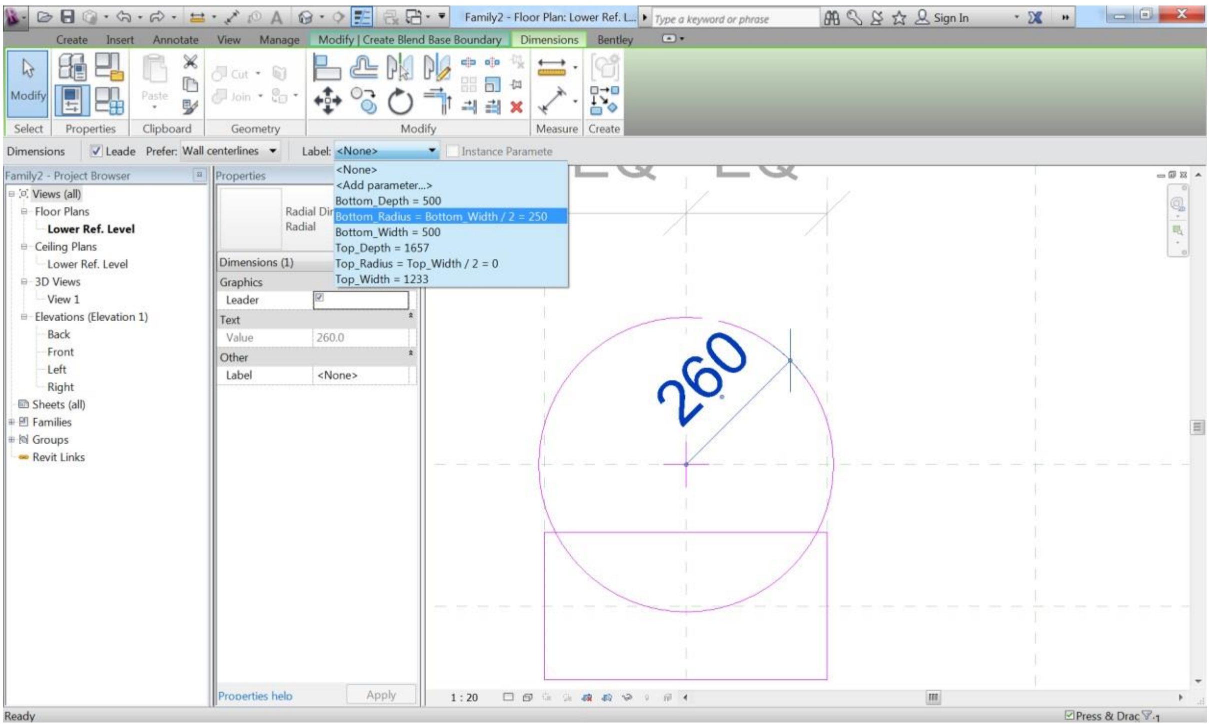
“Align” and constrain the center to the upper horizontal reference plane - one of the preset reference plane.



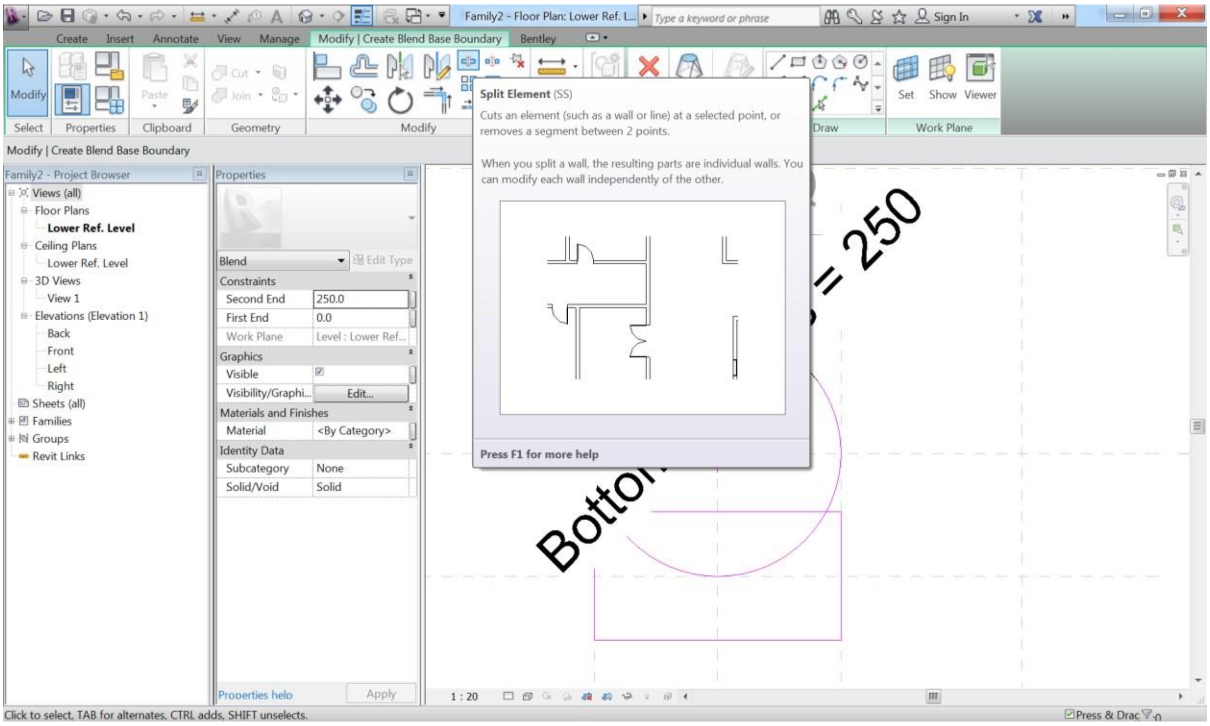
Select the circle to make the temporary dimension visible.



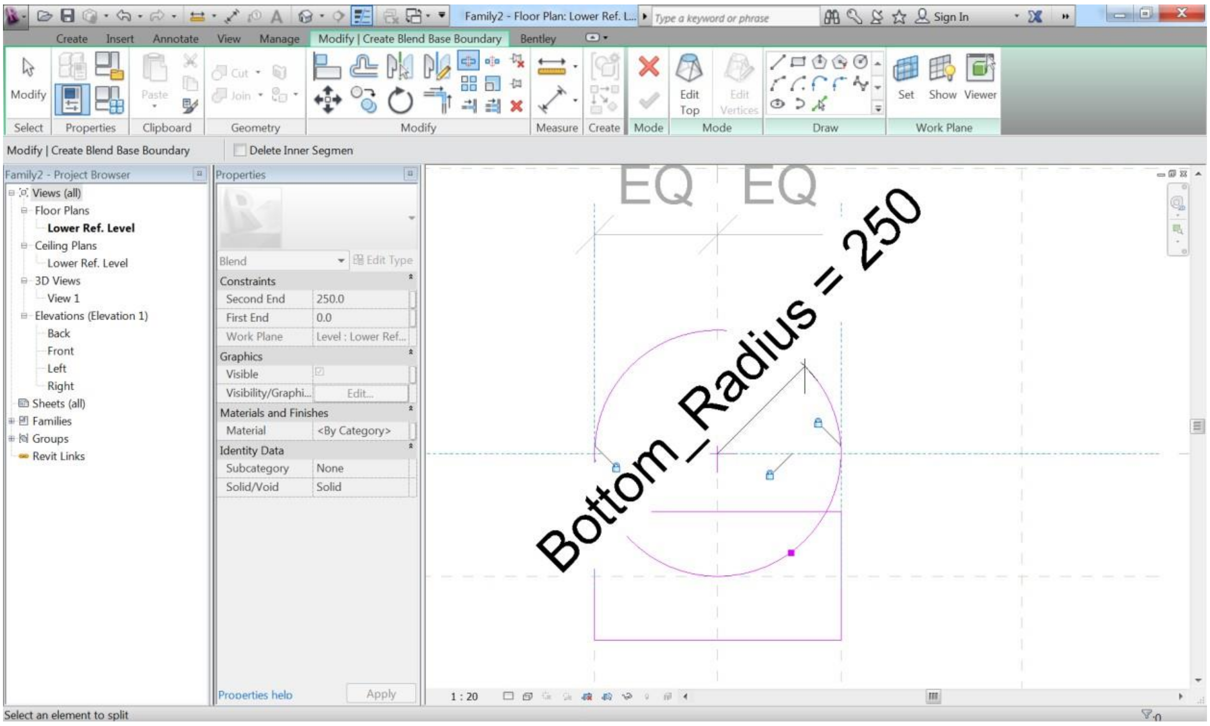
Click the dimension icon to make the temporary dimension permanent. Relate the dimension to "Bottom_Radius" parameter.



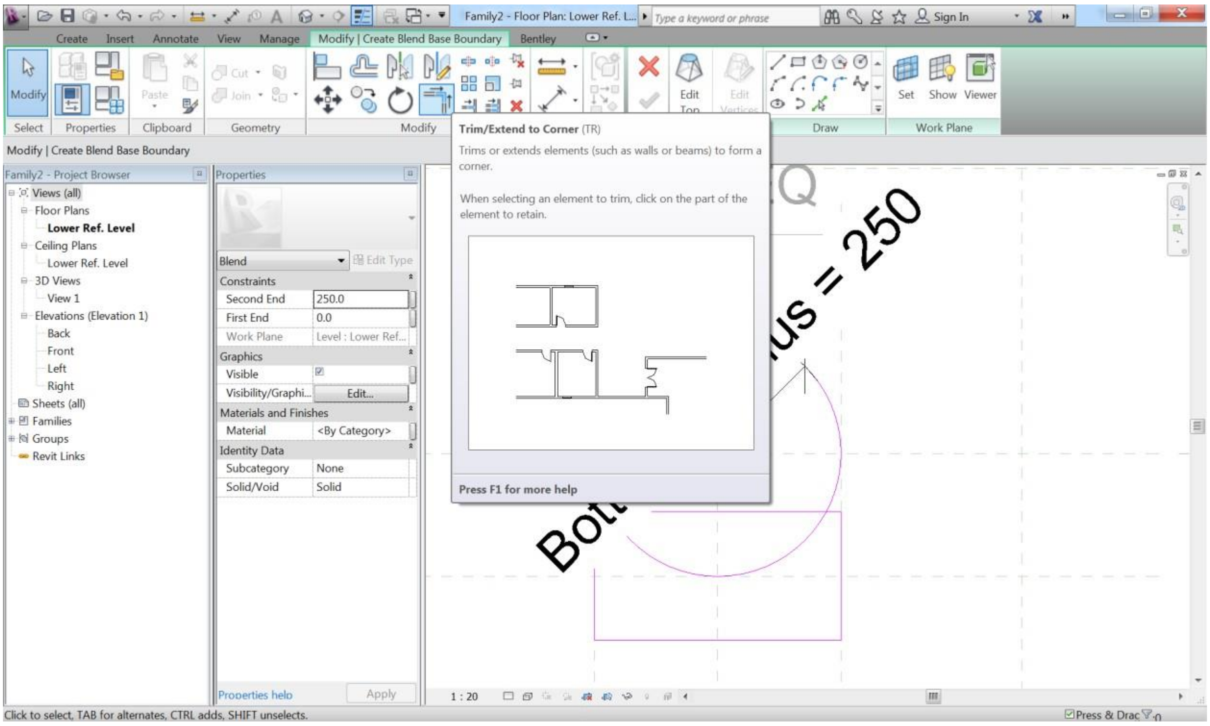
Go to “Modify” tab and click “Split Element” button.



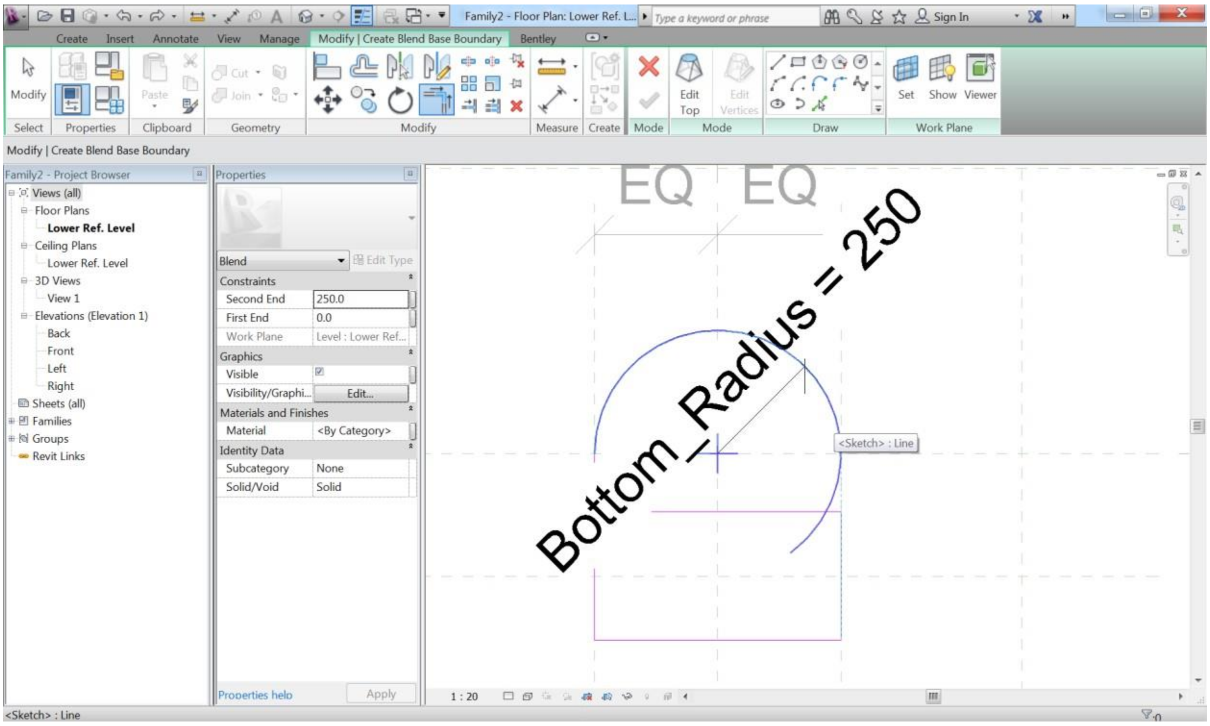
Click the lower part of the circle to put splitting point.



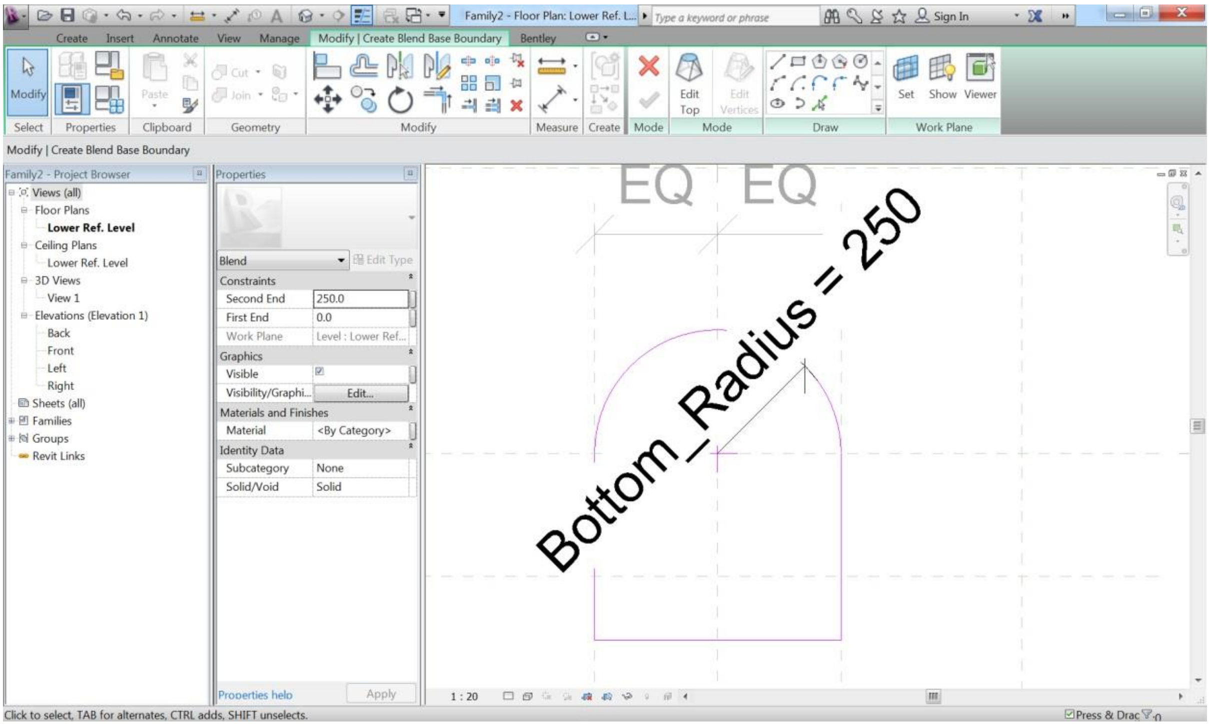
Then click "Trim/Extend to Corner" button from the "Modify" tab.



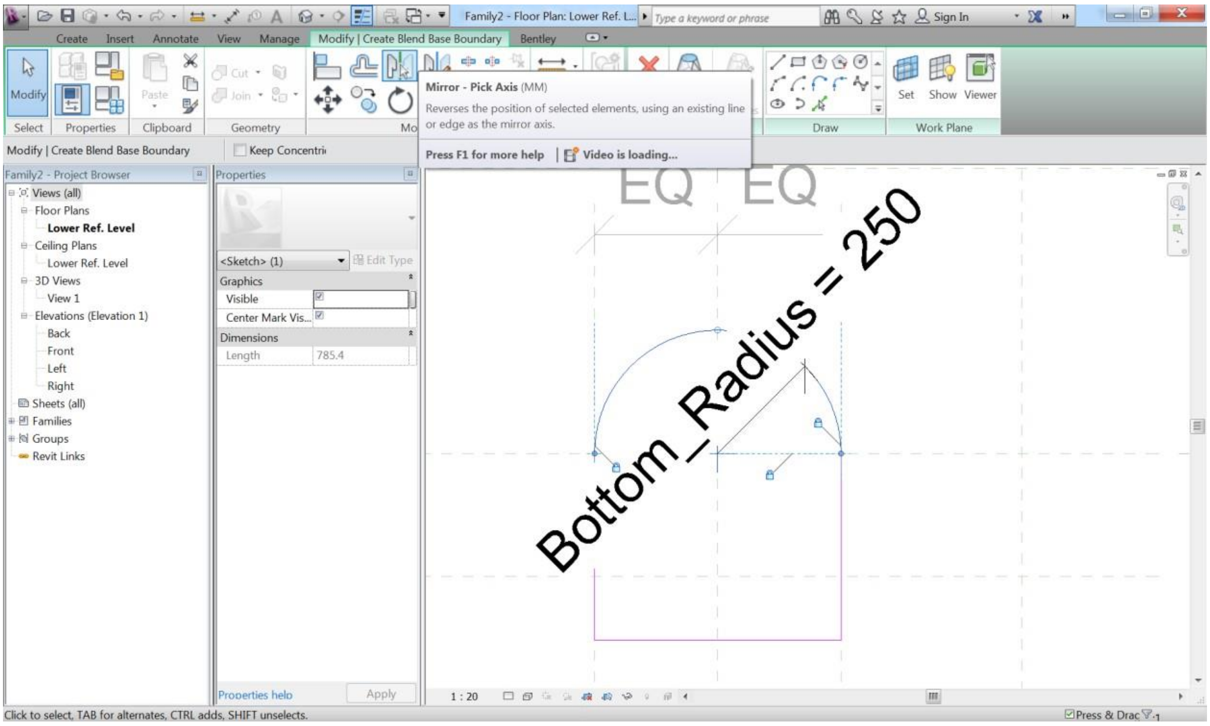
"Trim" each side and the upper part of the split circle.



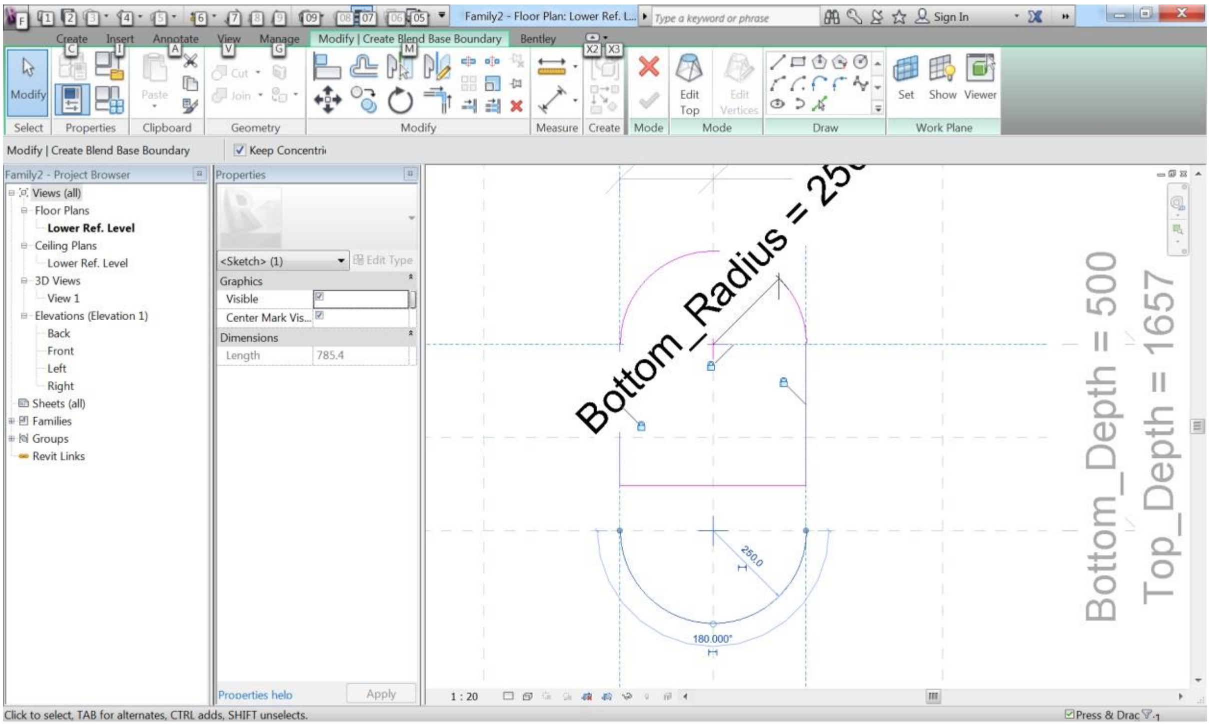
The trimmed profile should look like the image below.



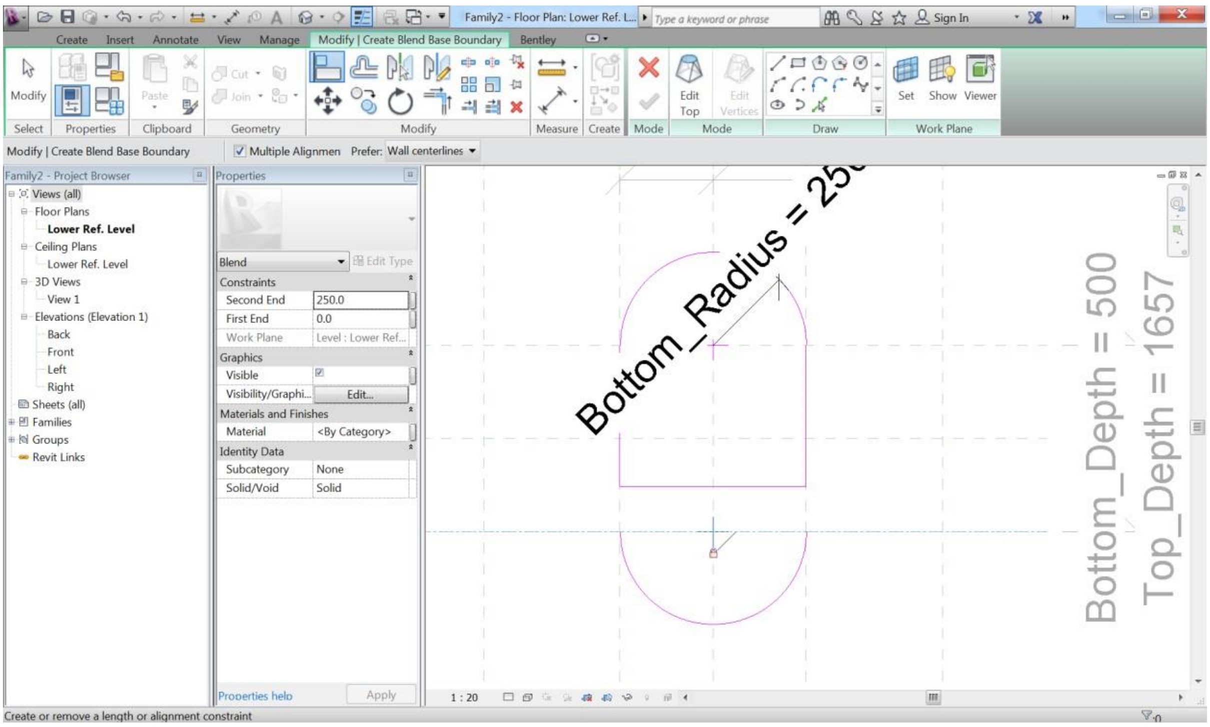
Select the semi-circle and then go to “Modify” tab and click “Mirror – Pick Axis” button.



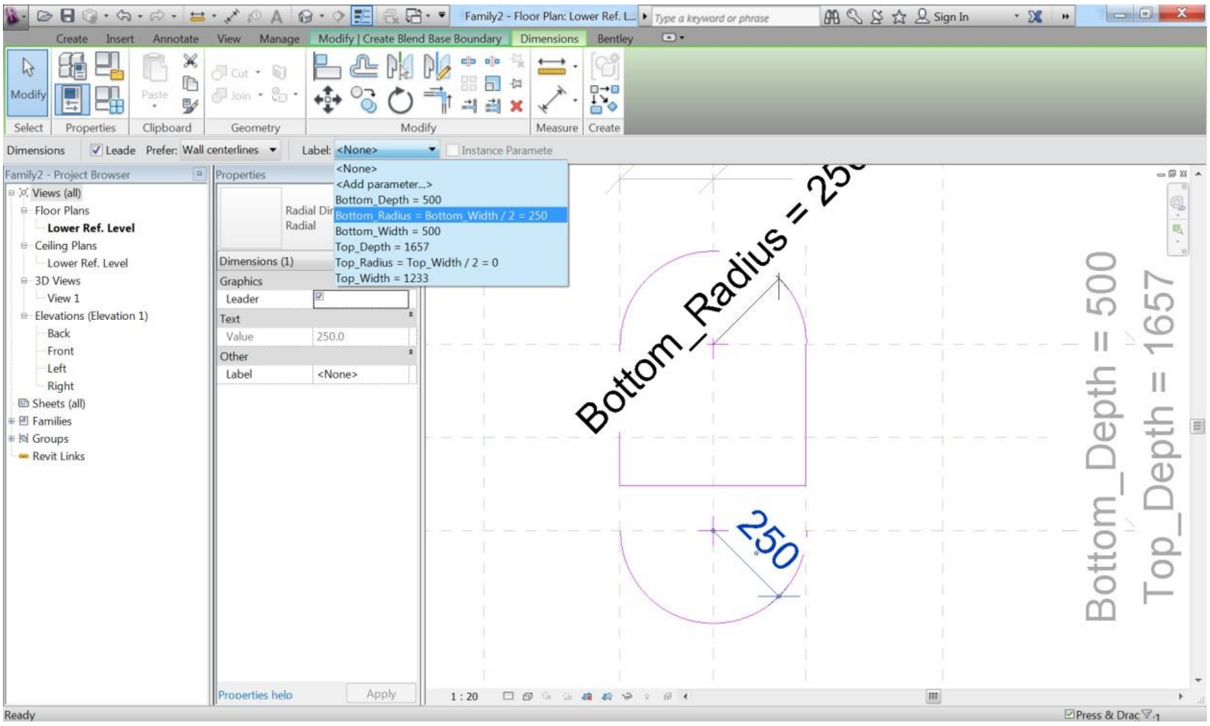
Click the horizontal center as the axis.



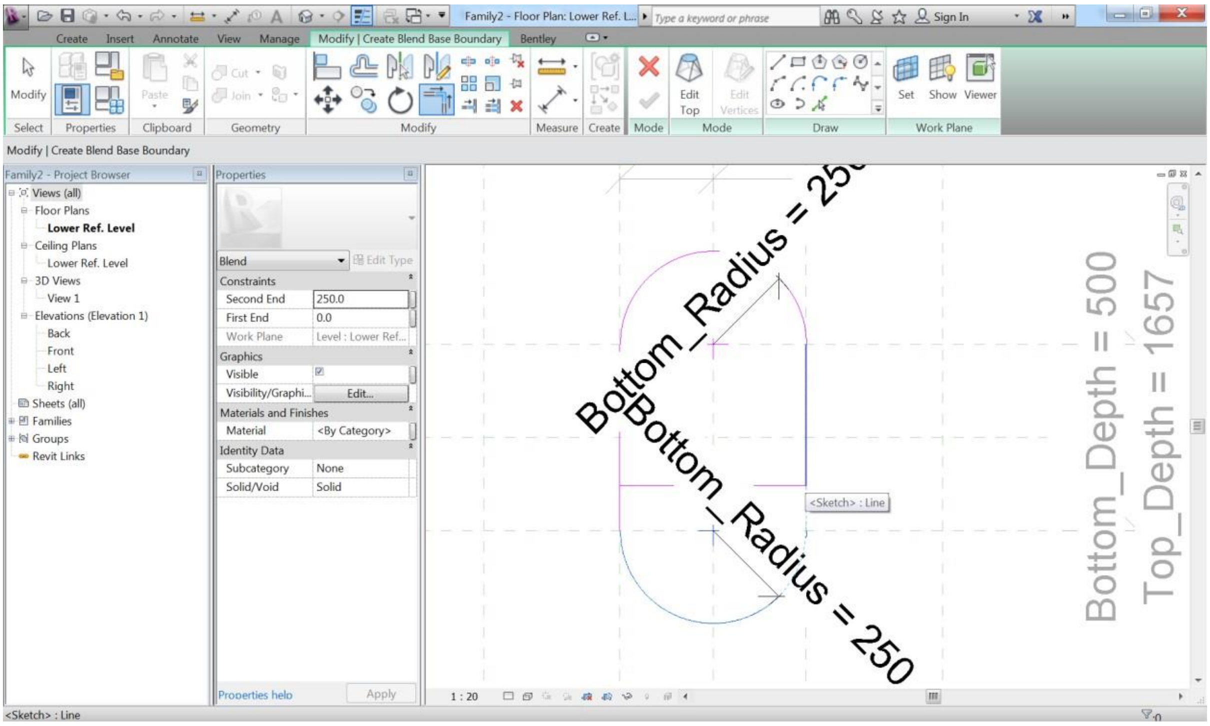
Then align the center of the semi-circle to the lower horizontal reference plane to constrain the center to the reference plane.



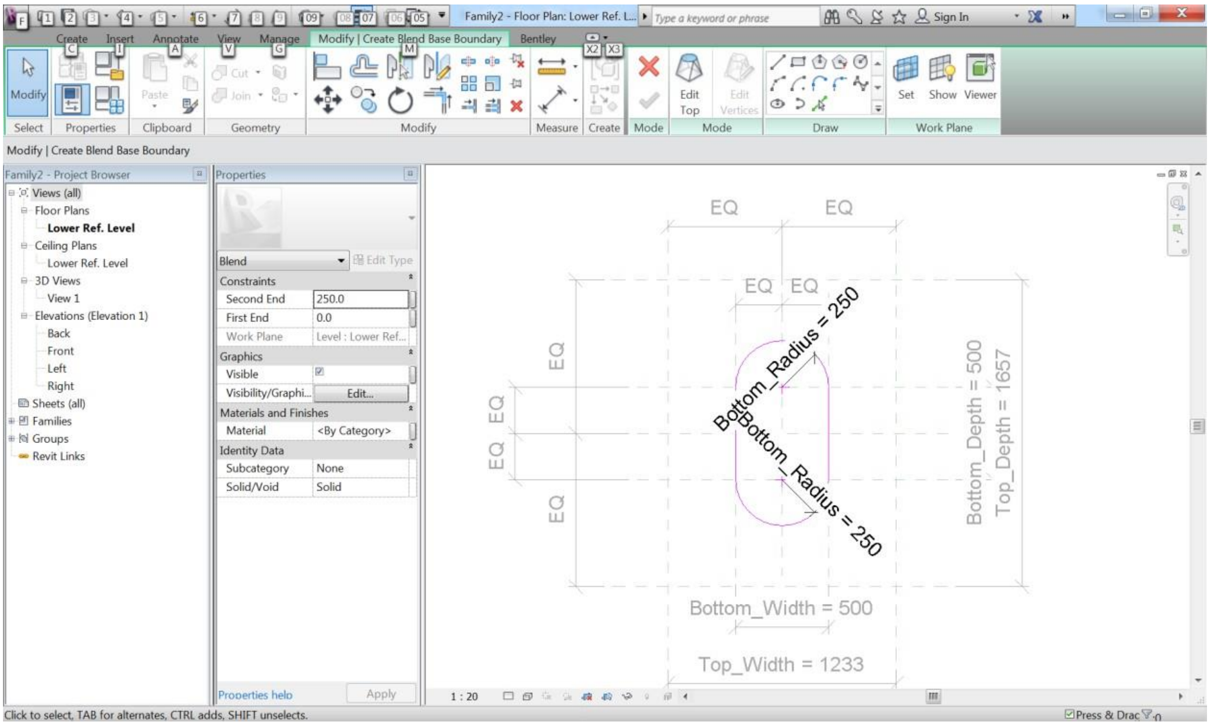
To parameterize the radius, repeat the same steps to the mirrored semi-circle.



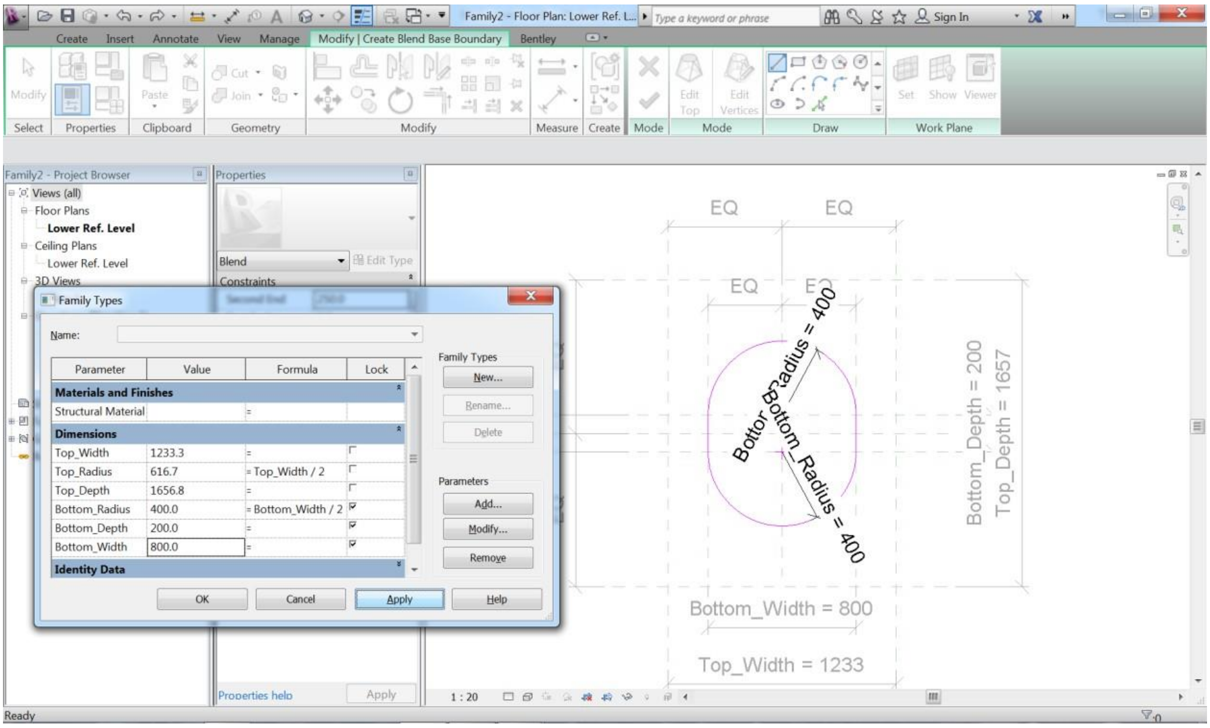
Trim each side and the semi-circle.



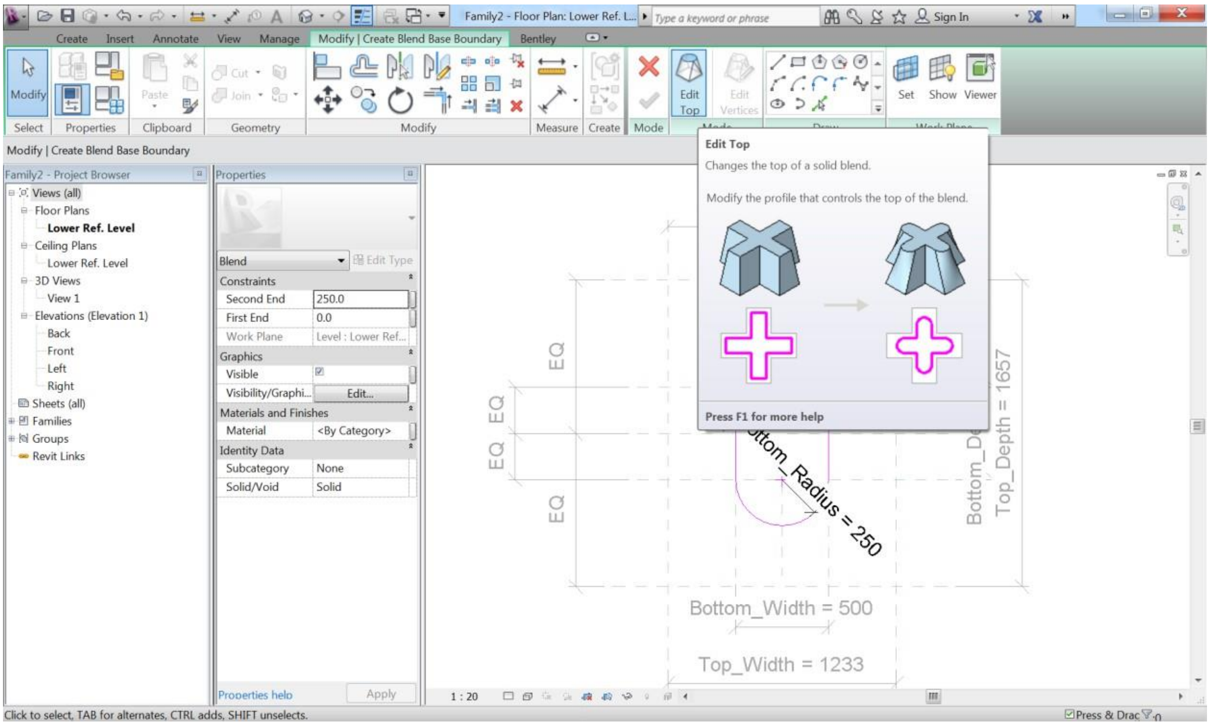
The profile the bottom face is done.



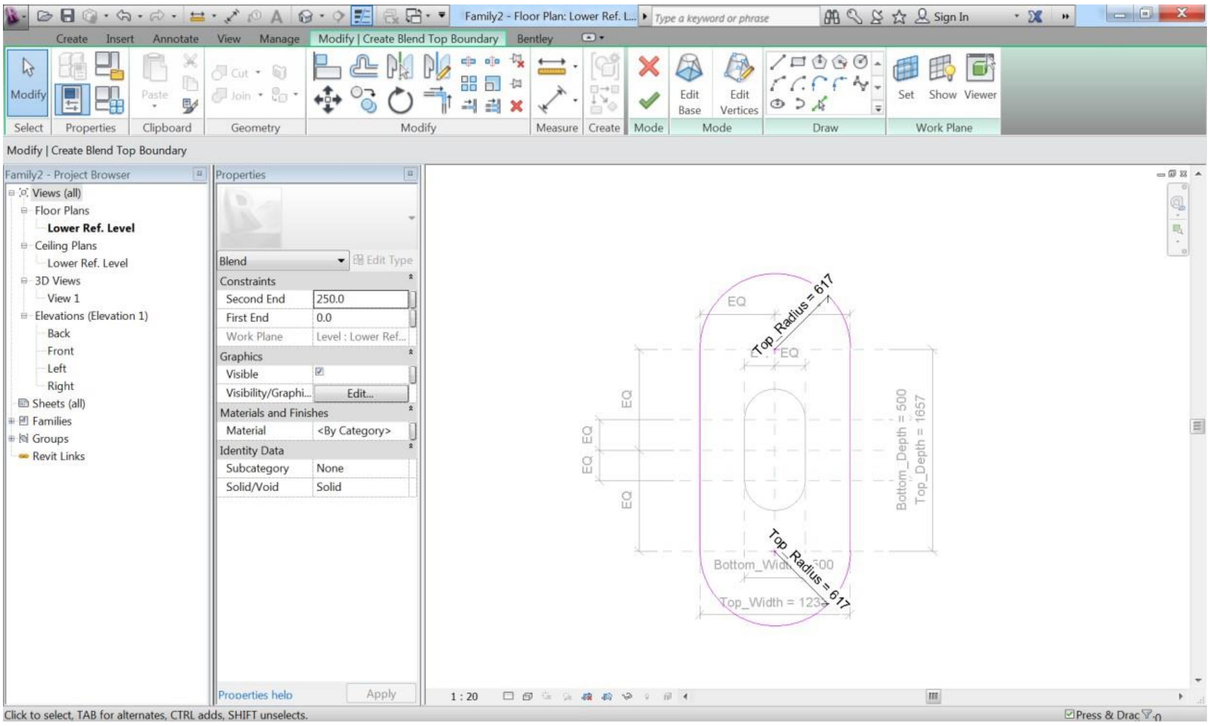
Before moving on to the top profile, test the parametric frameworks. Open “Family Types” and type in different values to “Bottom Depth” and “Bottom_Width”.



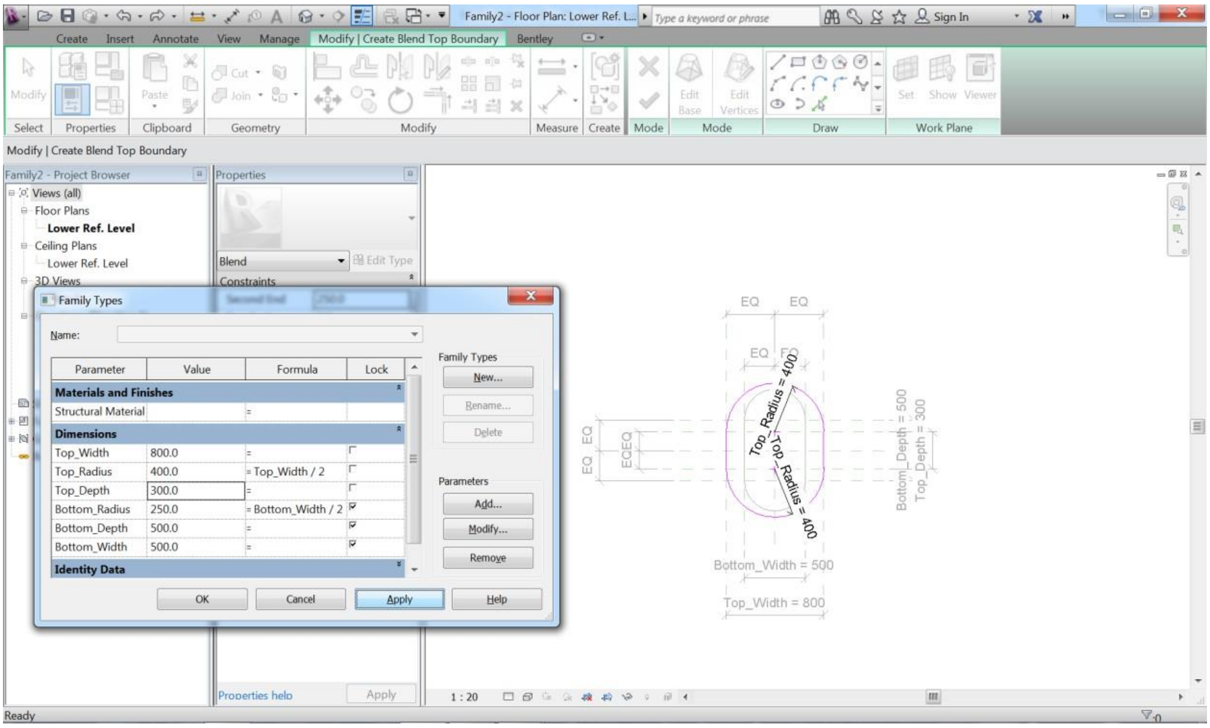
If the parametric frameworks work file, click "Edit Top" button.



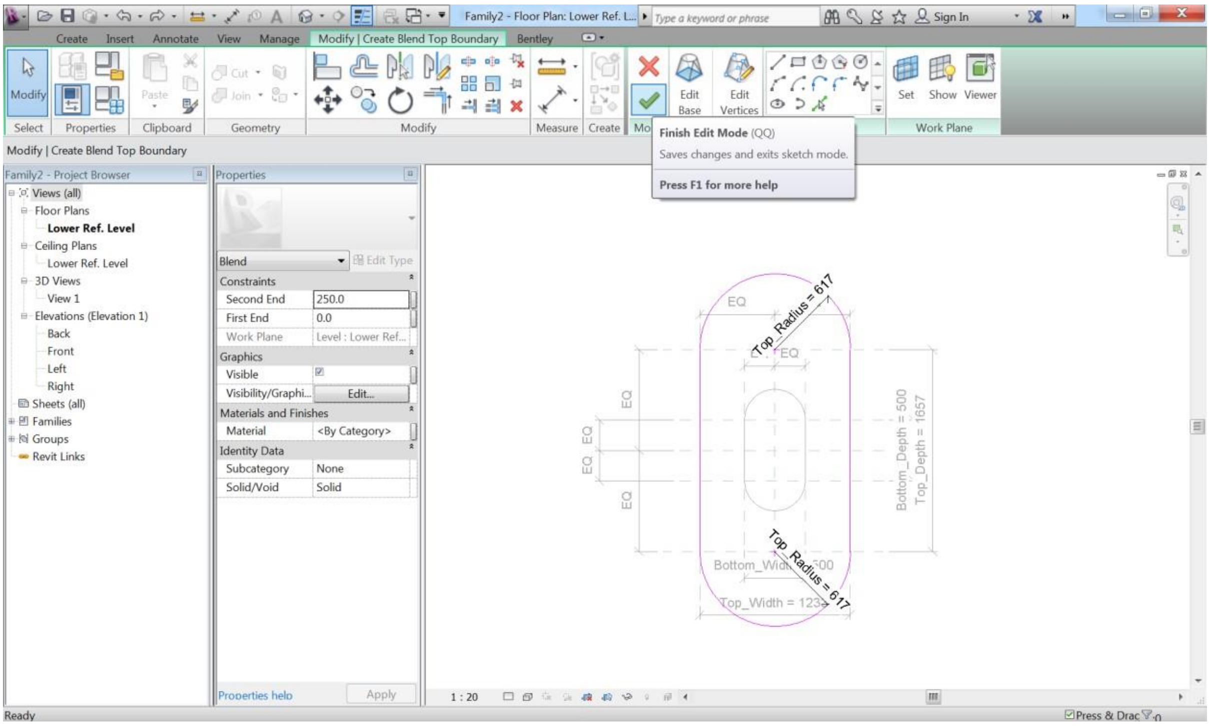
Repeat the same steps with "Top_………" parameters and reference planes that you added.



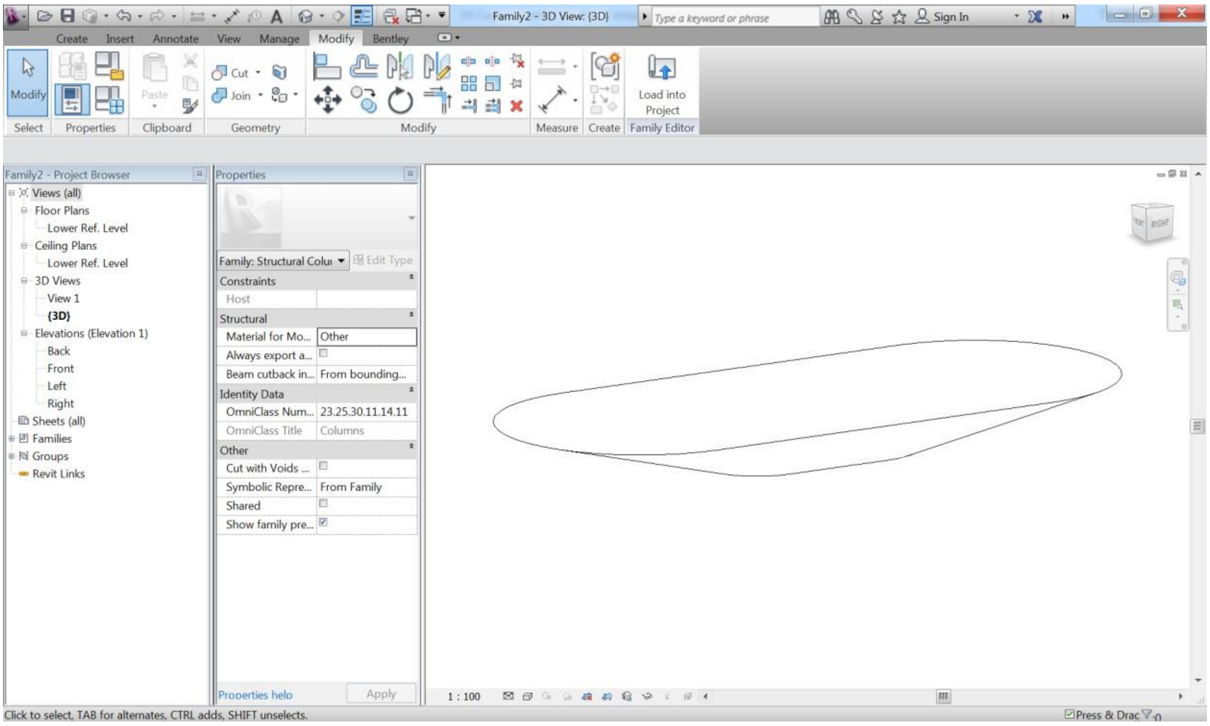
Before finishing the sketch mode, test the top profile as well.



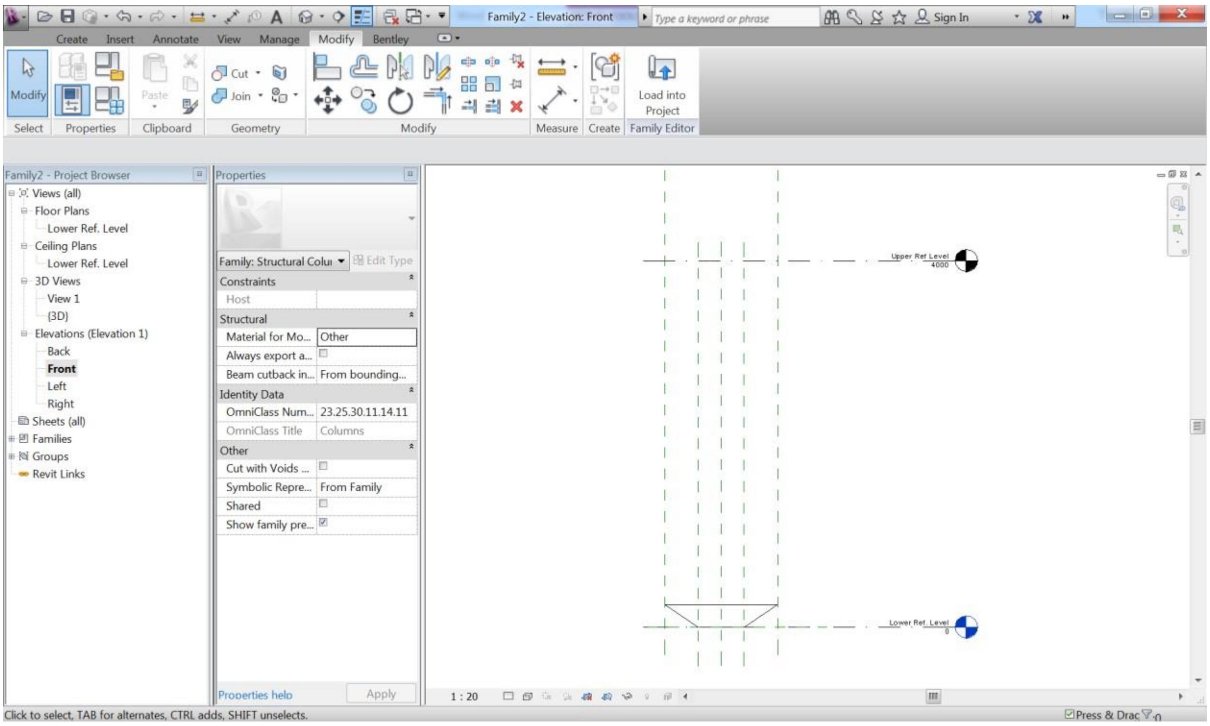
The click "Finish Edit Mode" button to finish the blend function.



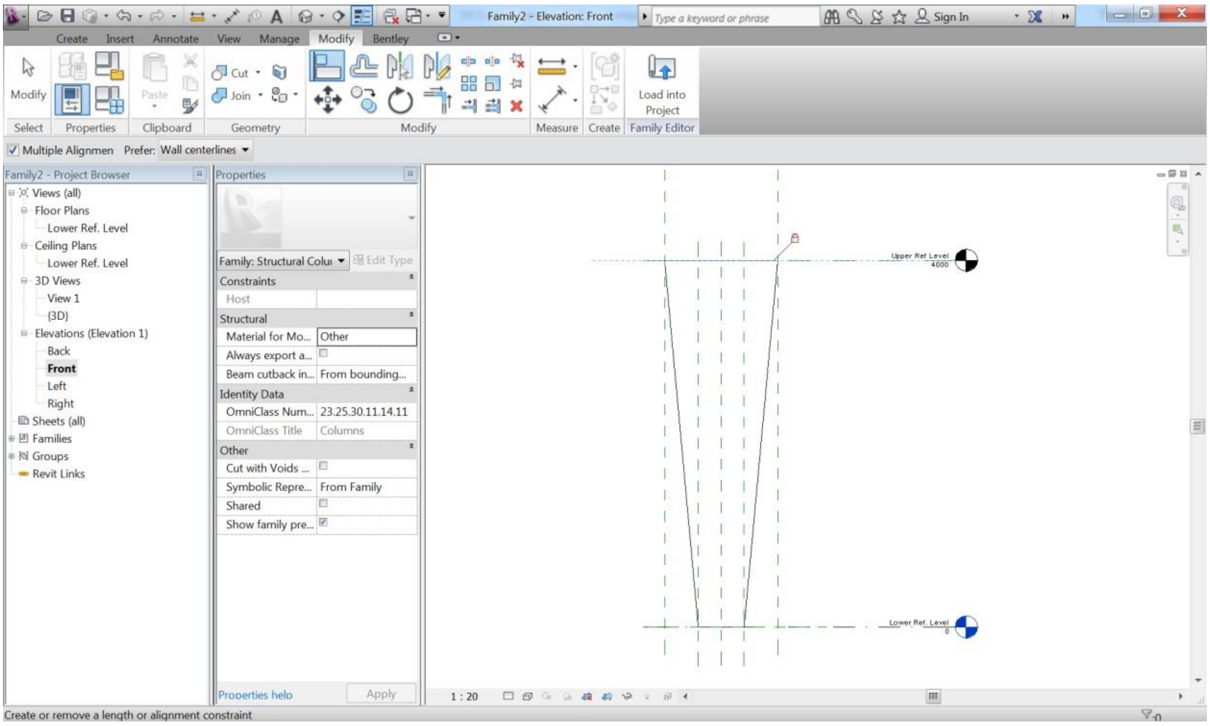
Go to 3D view and then, your geometry should look like that of the image below.



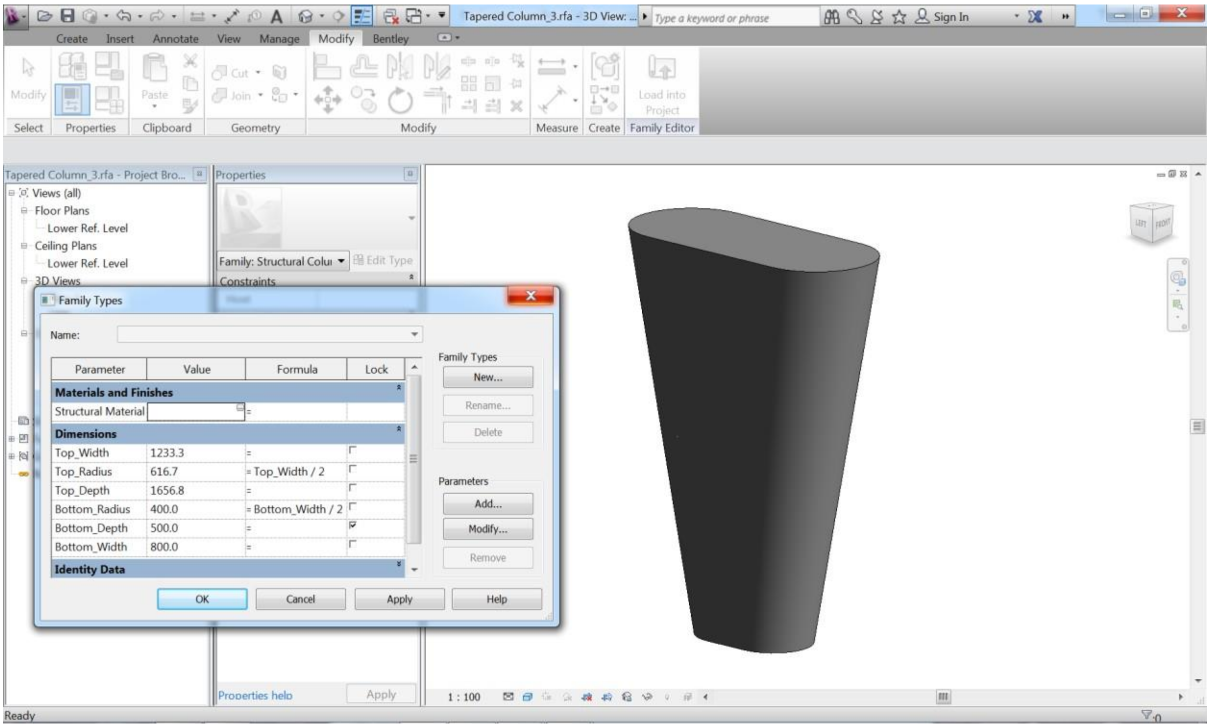
Go to “Front” elevation view.



"Align" and constrain the top face of the solid to the "Upper Ref. Level" annotation.



add more types to the family. You can try it with the catalogue function.



Disclaimer: The process described in this tutorial is only to help the readers understand the concept of the topic and how to use related functions. This tutorial does not offer the optimized way to implement a project in Revit.

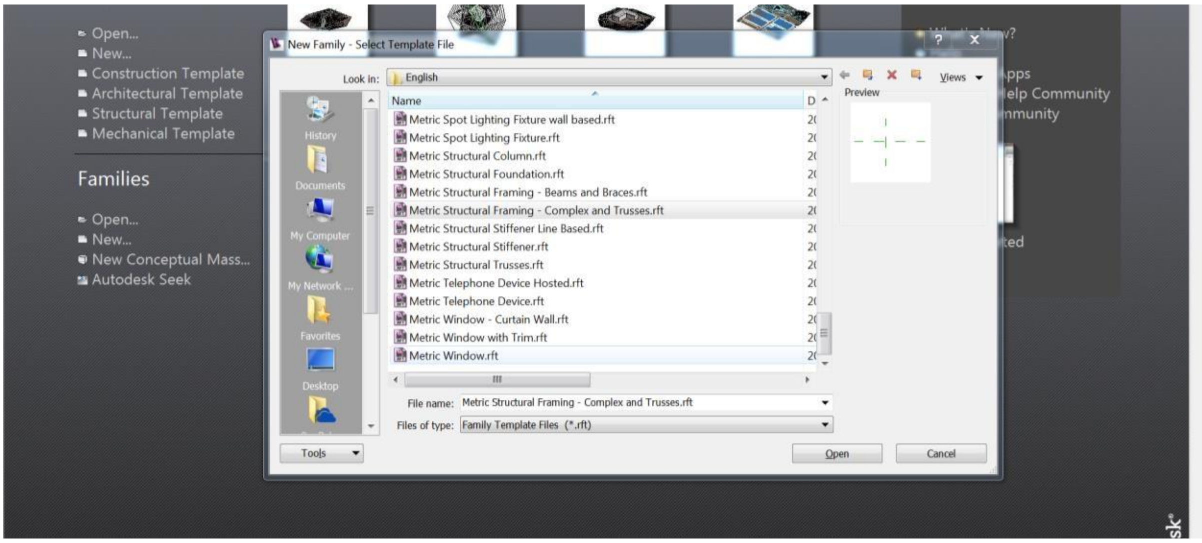
CHAPTER 14 - Customize Beam Family

In this section, you will learn

- How to create a Cranked Beam Family

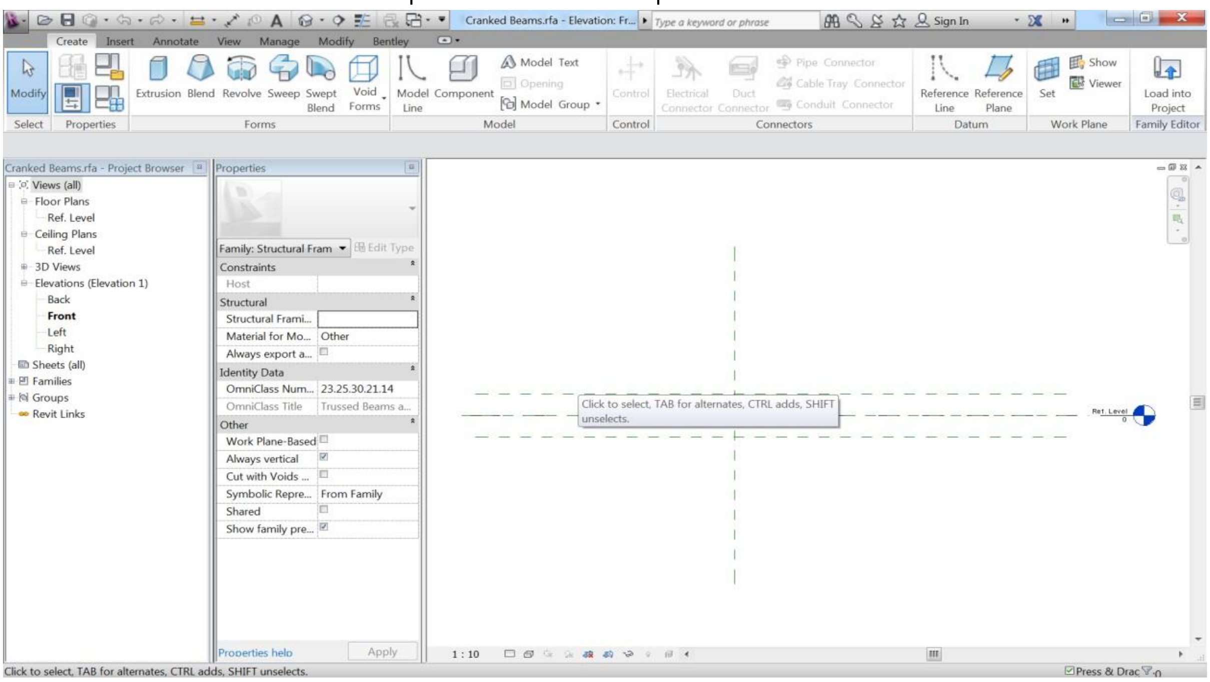
14.1 Template

Start a family editor and select “Metric Structural Framing - Complex and Trusses.rft” for the template.

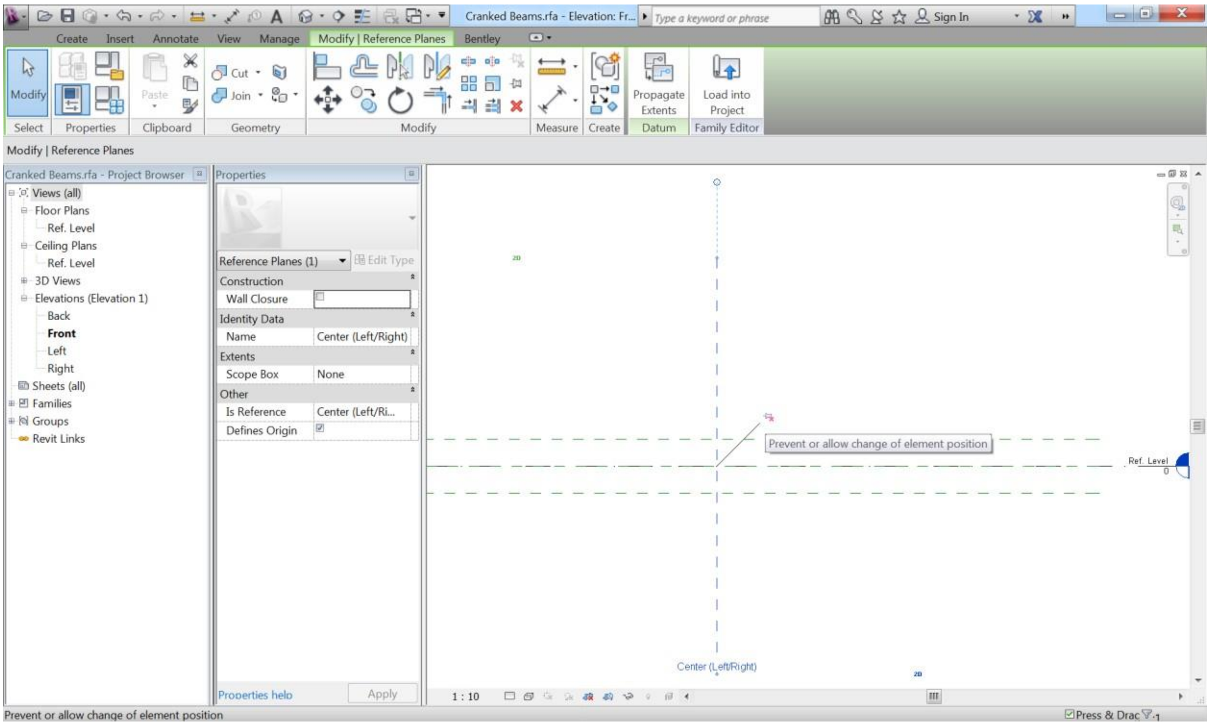


14.2 Reference Planes

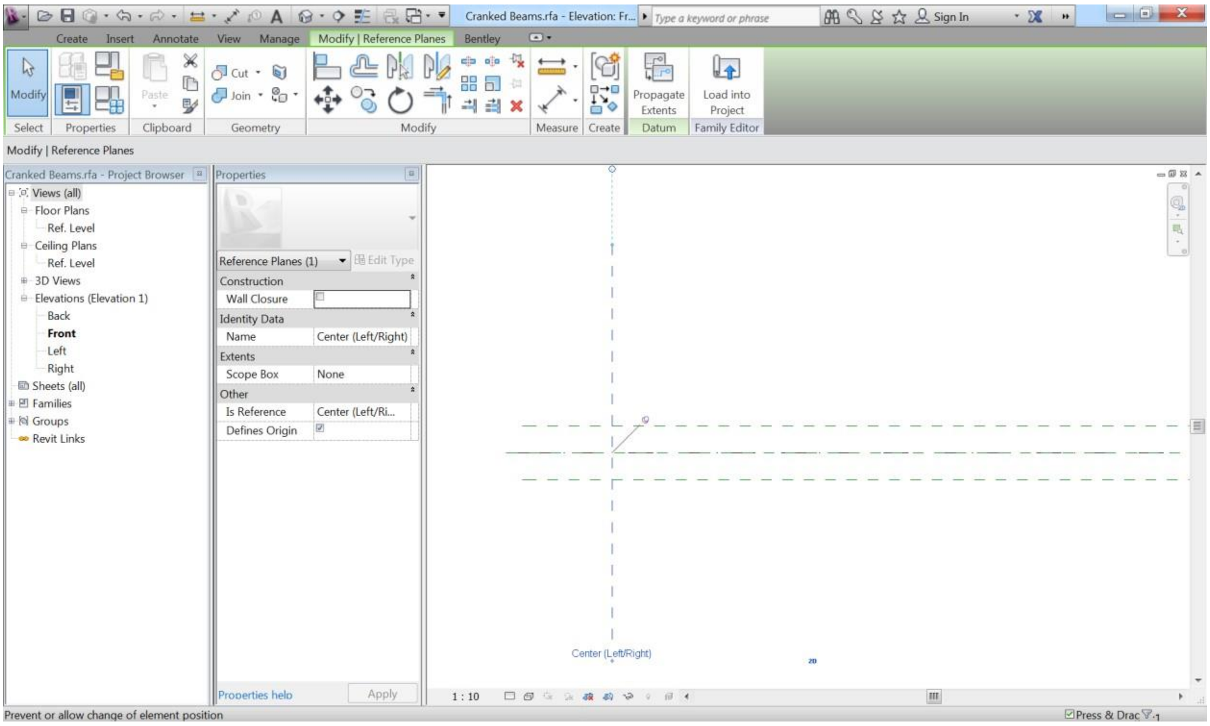
Go to “Front” elevation view and place two reference planes on both sides of the “Ref. Level”.



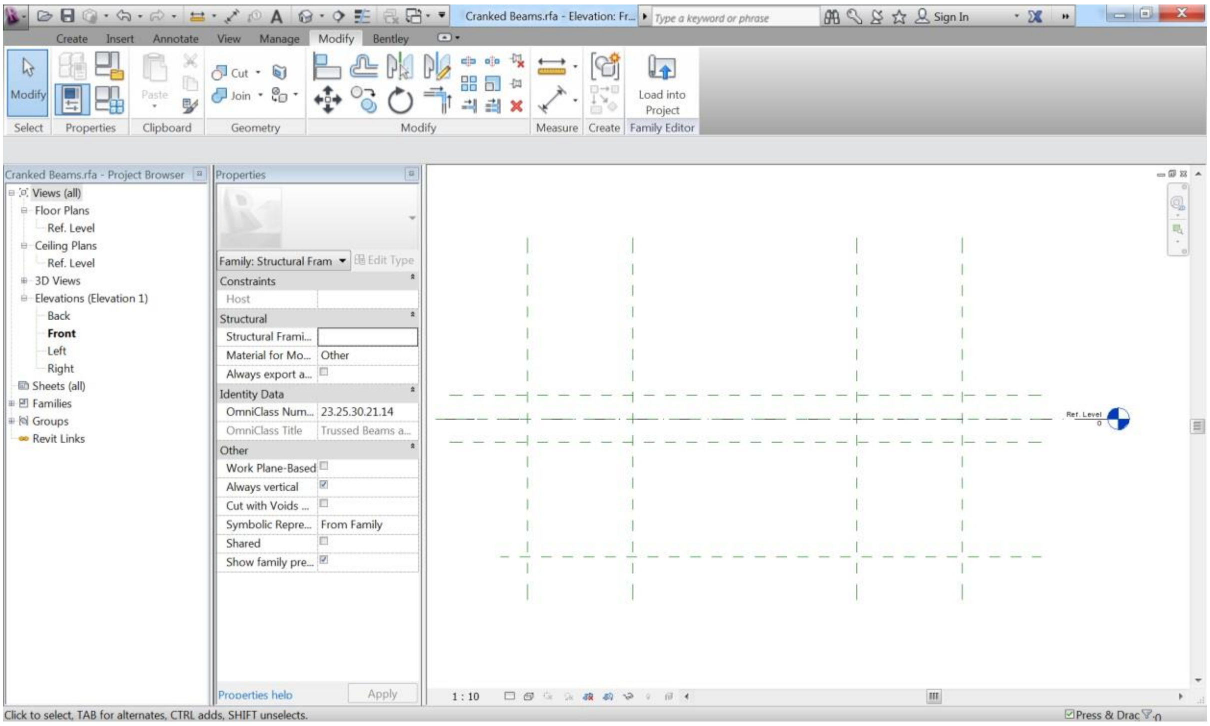
Then select the vertical center. Click appeared "Pin" icon to unpin the object.



Move it to left side and pin the object again.

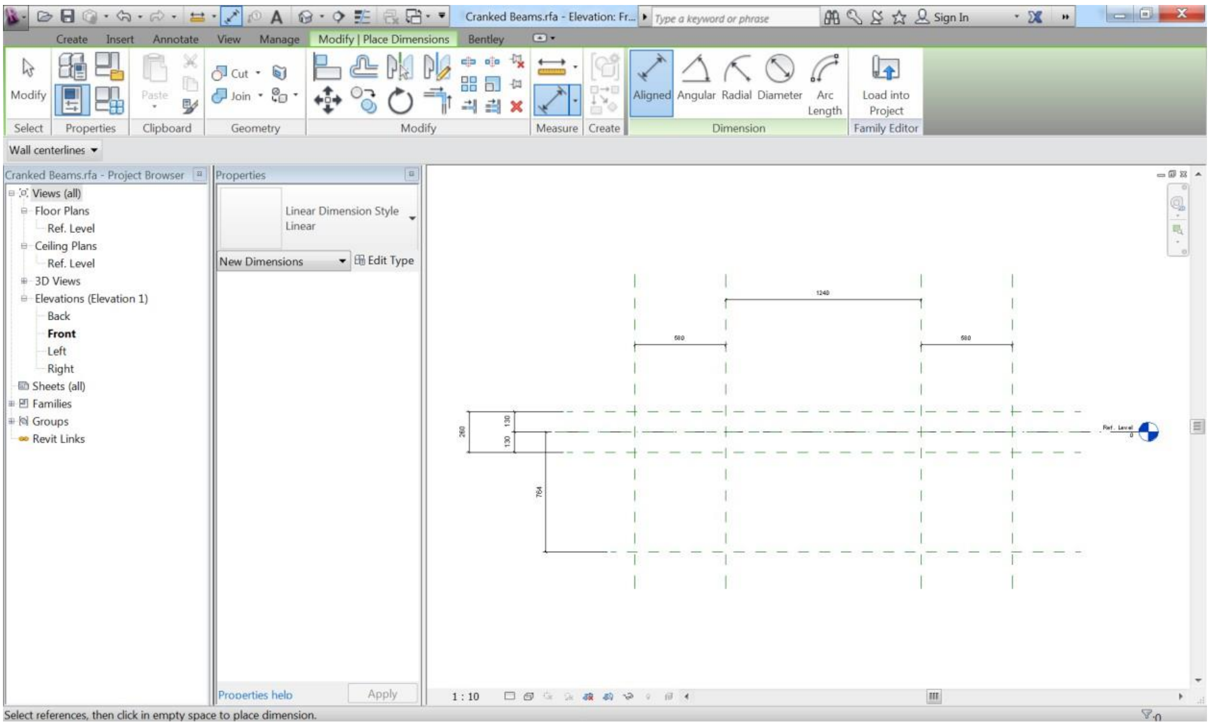


Add one more reference plane in a horizontal way, and place four vertical reference planes. refer to the image below.

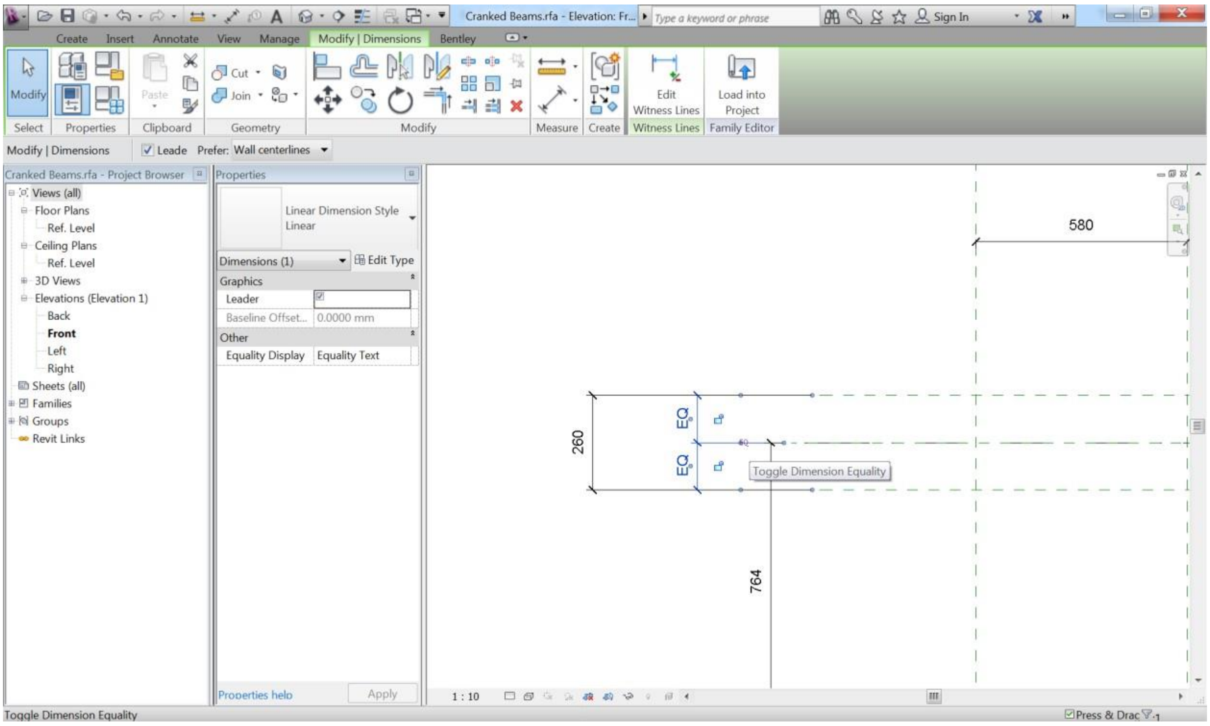


14.3 Parametric Dimension

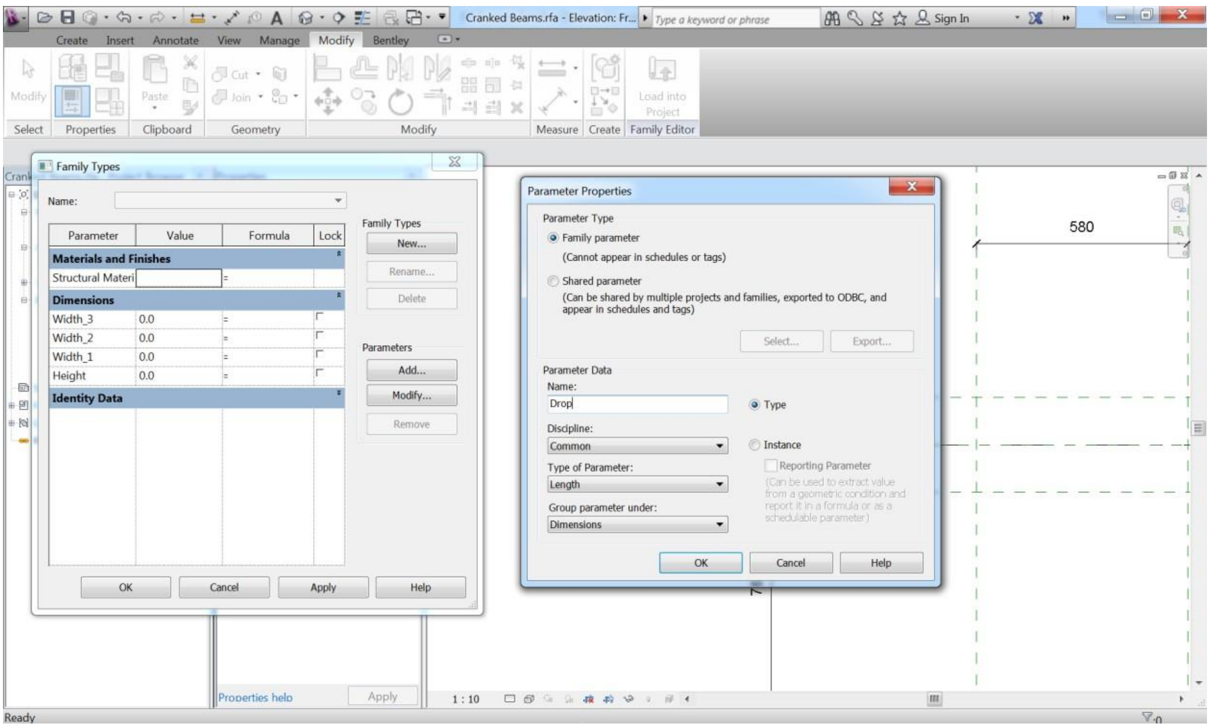
refer to the image below and place dimensions to the reference planes.



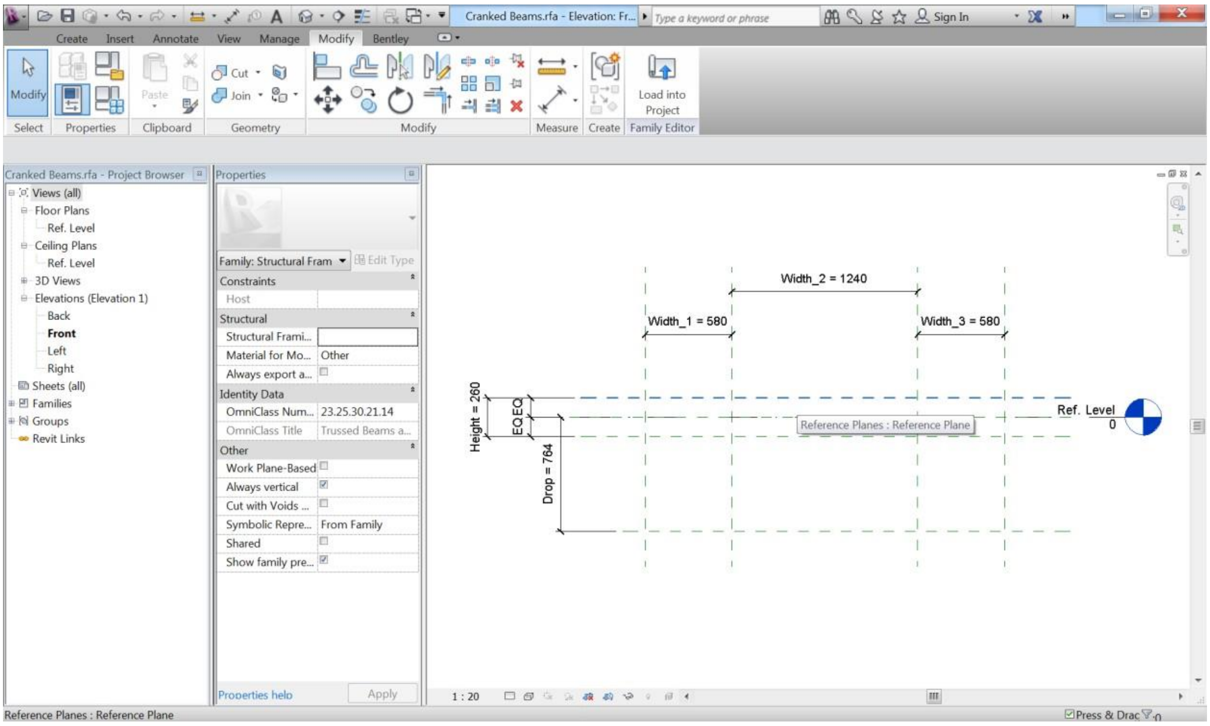
Turn one of the dimensions into "EQ" dimension.



Add "Width_1", "Width_2", "Width_3", "Height", and "Drop" parameters with "Common", "Length", "Dimension", and "Type" options.

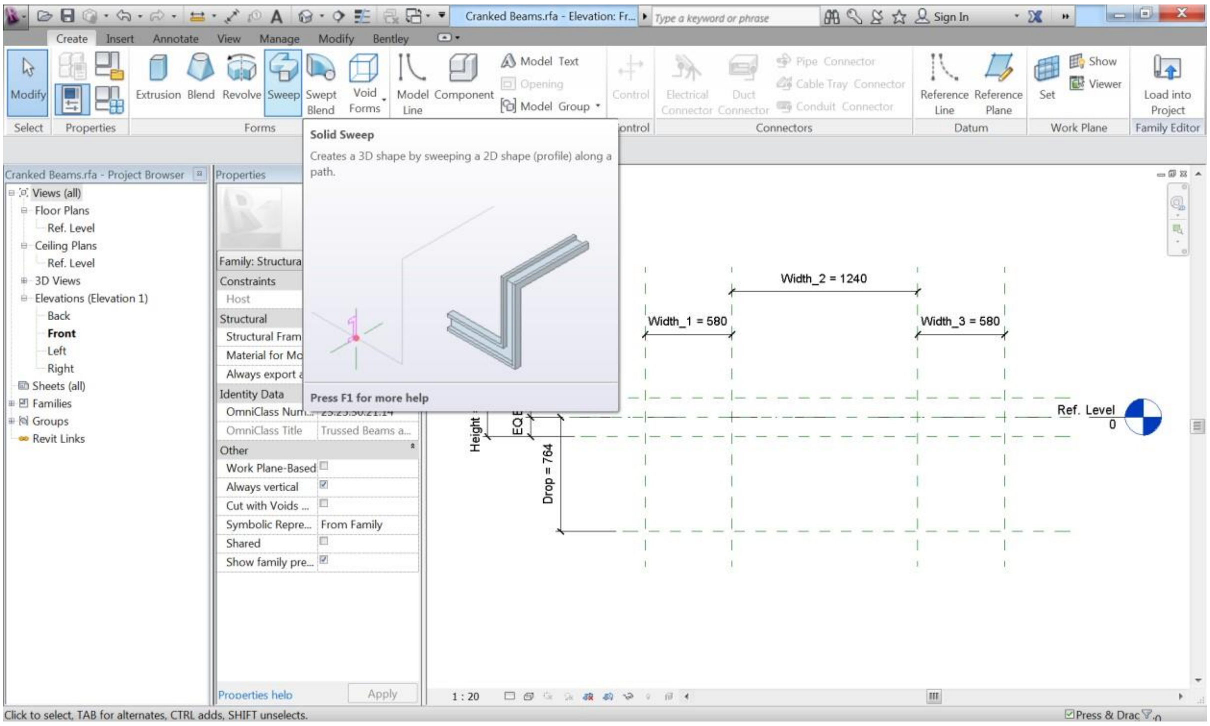


Refer to the image below and associate the parameters to the dimensions.

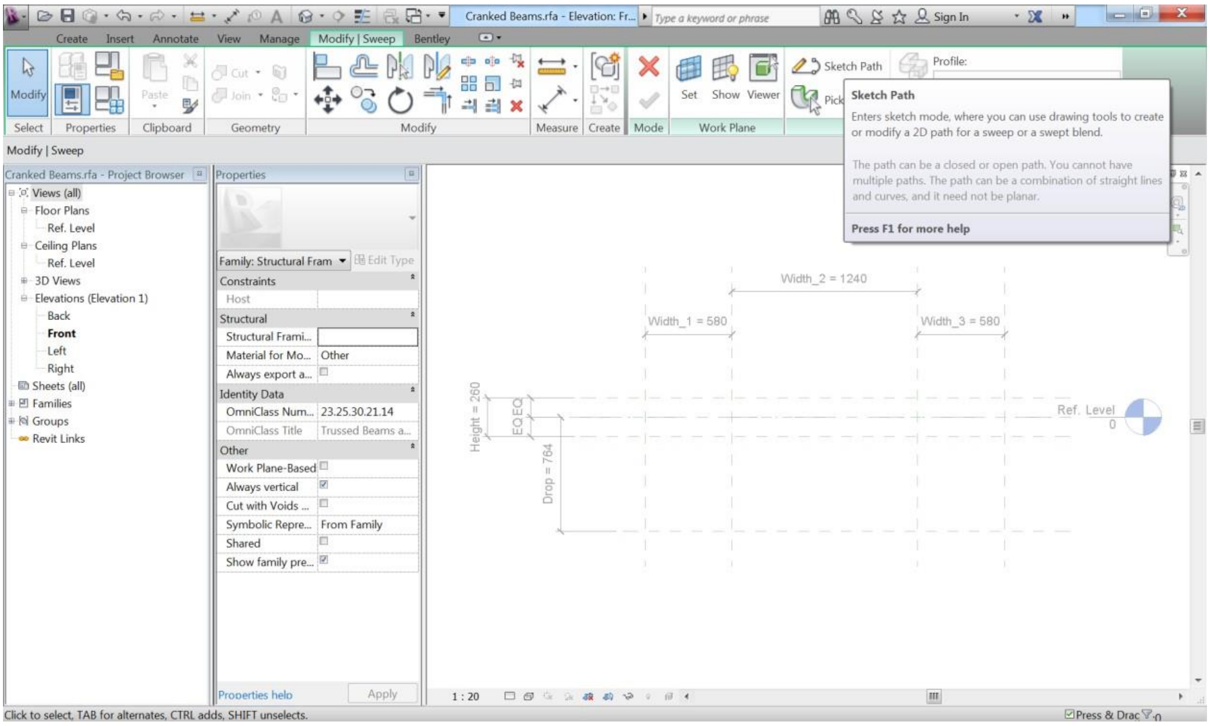


14.4 Sweep

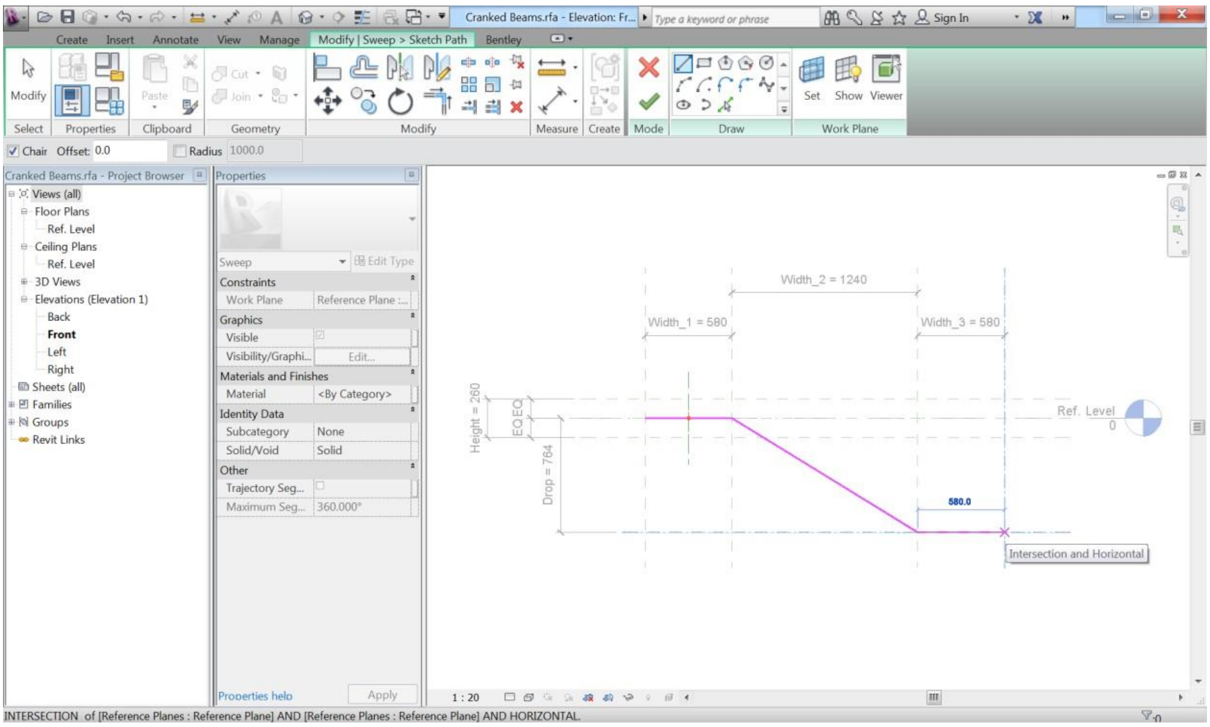
Go to "Create" tab and click "Sweep" button.



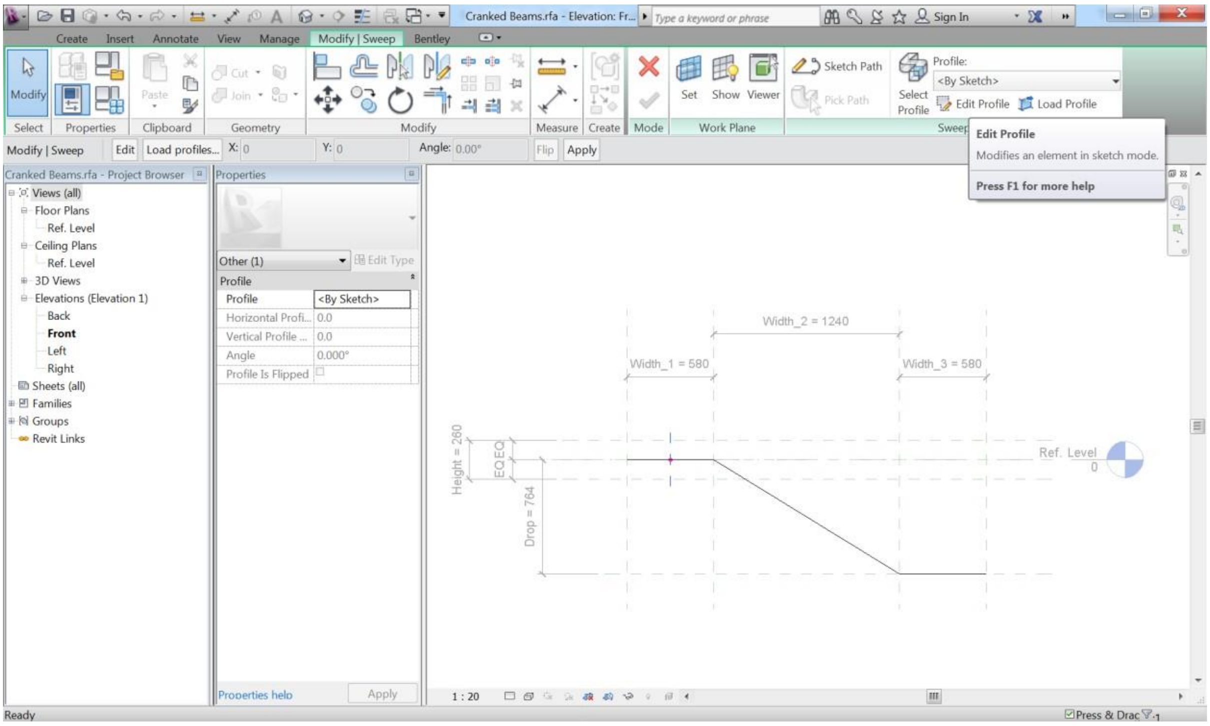
Click “Sketch Path” button from the ribbon menu.



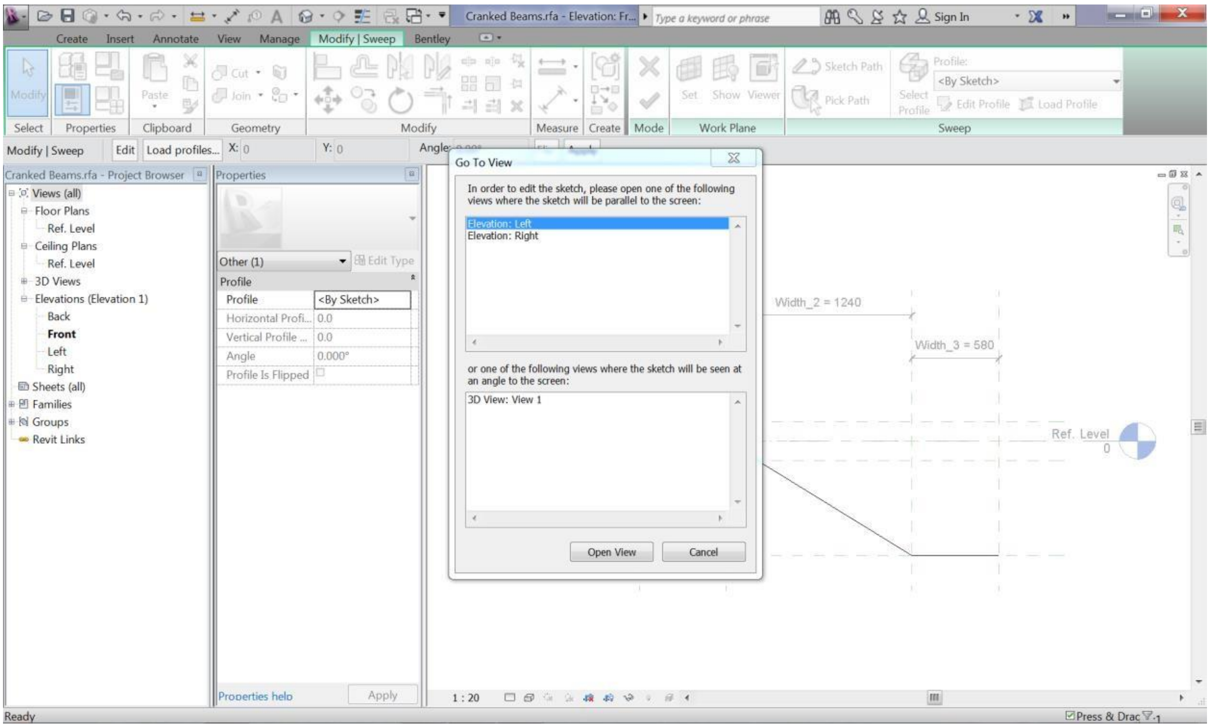
draw a line that go through the intersecting points of the reference planes. Refer to the image below.



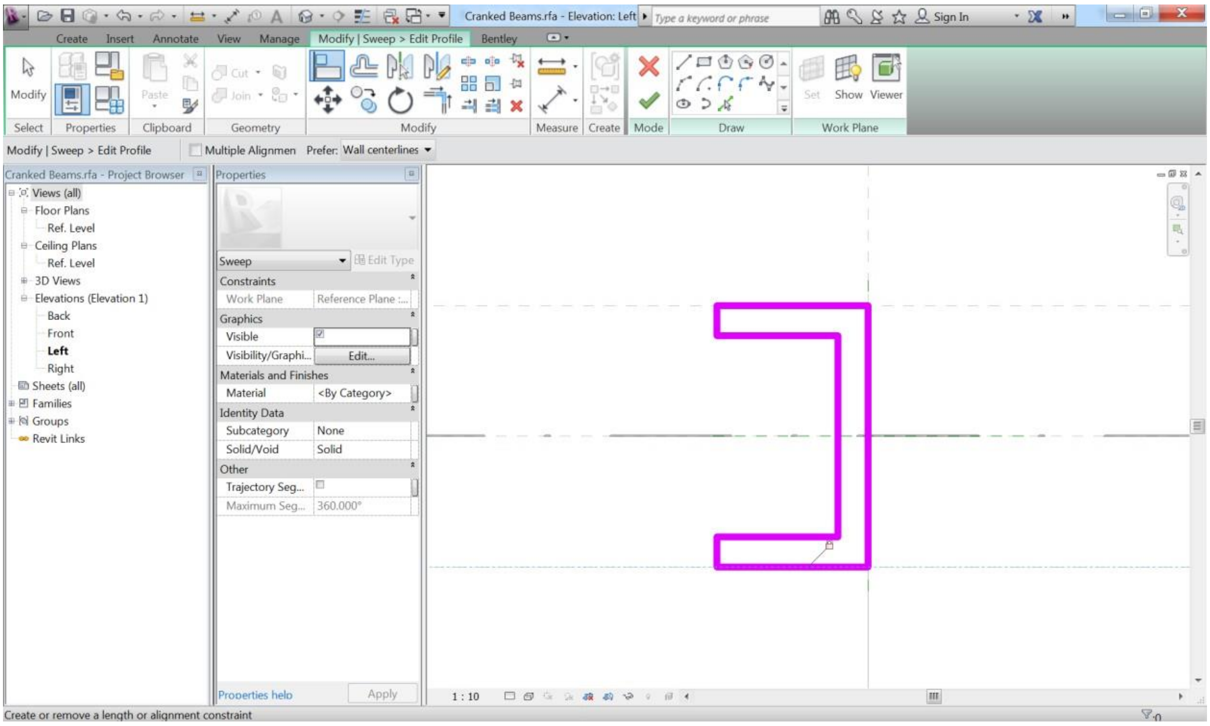
Finish the sketch mode and click "Edit Profile" button.



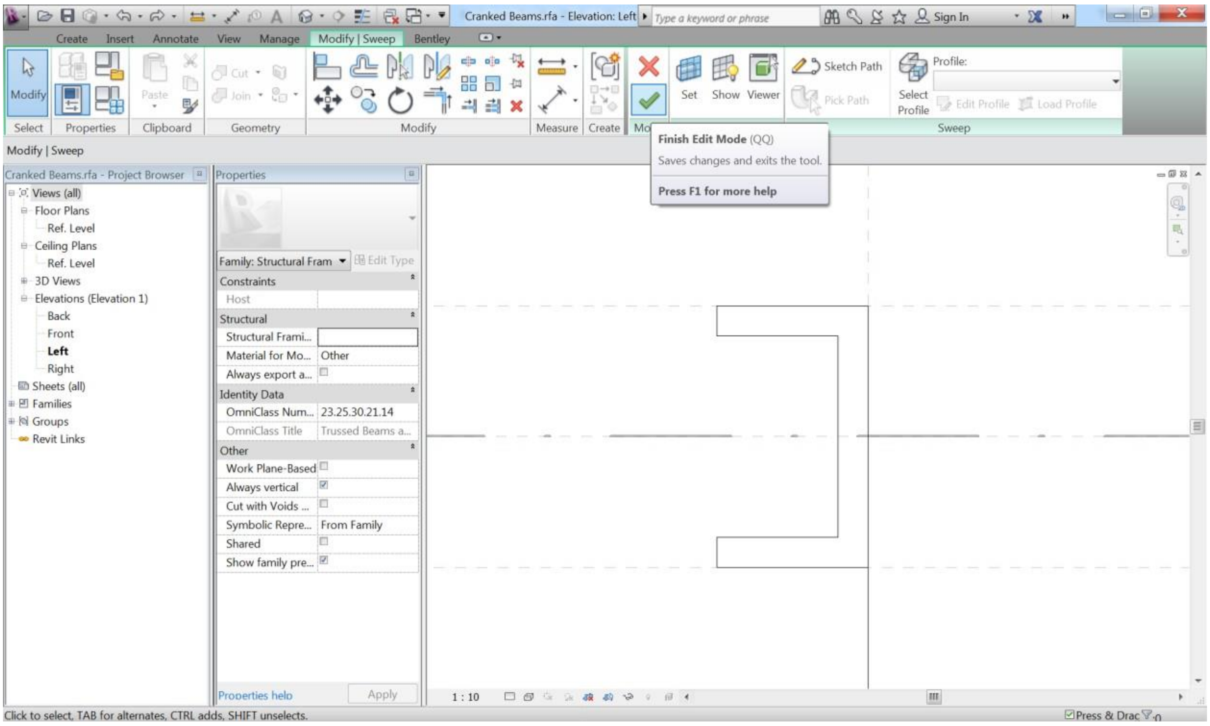
Select "Left" elevation view for the profile.



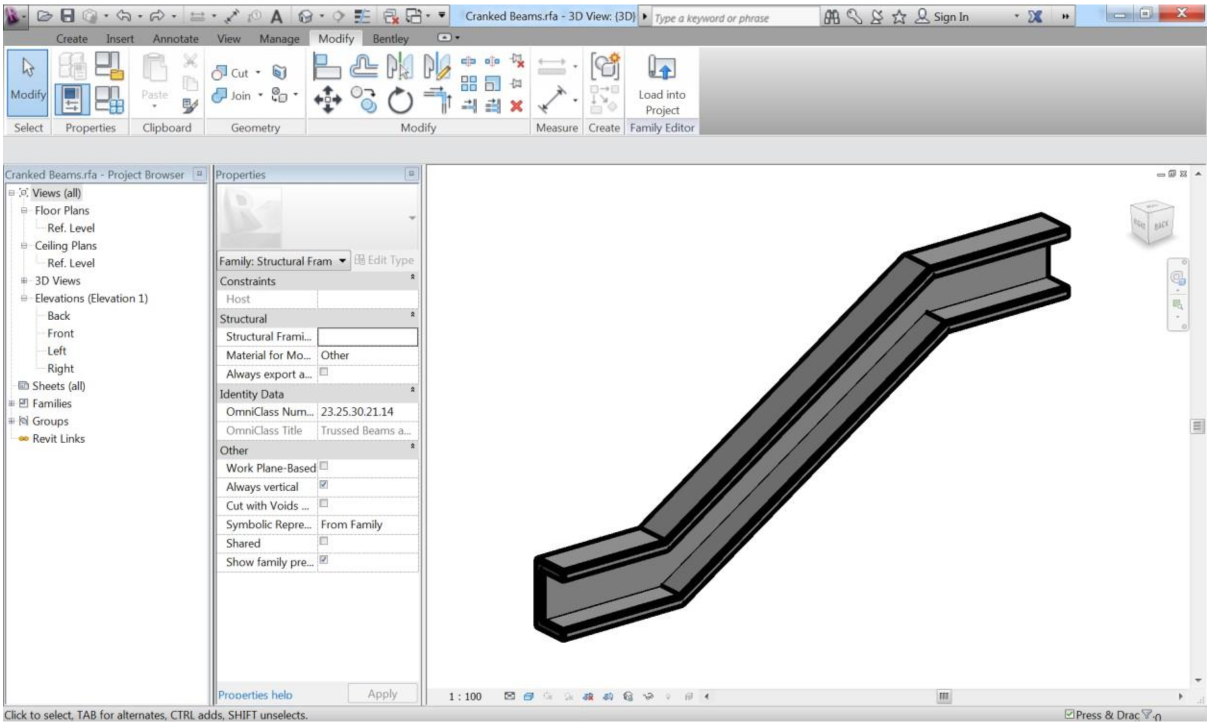
Draw a profile for a c-channel in the view. align the top and bottom line segments to the reference planes.



Finish the sketch mode for the profile. Then finish the sketch mode for sweep.

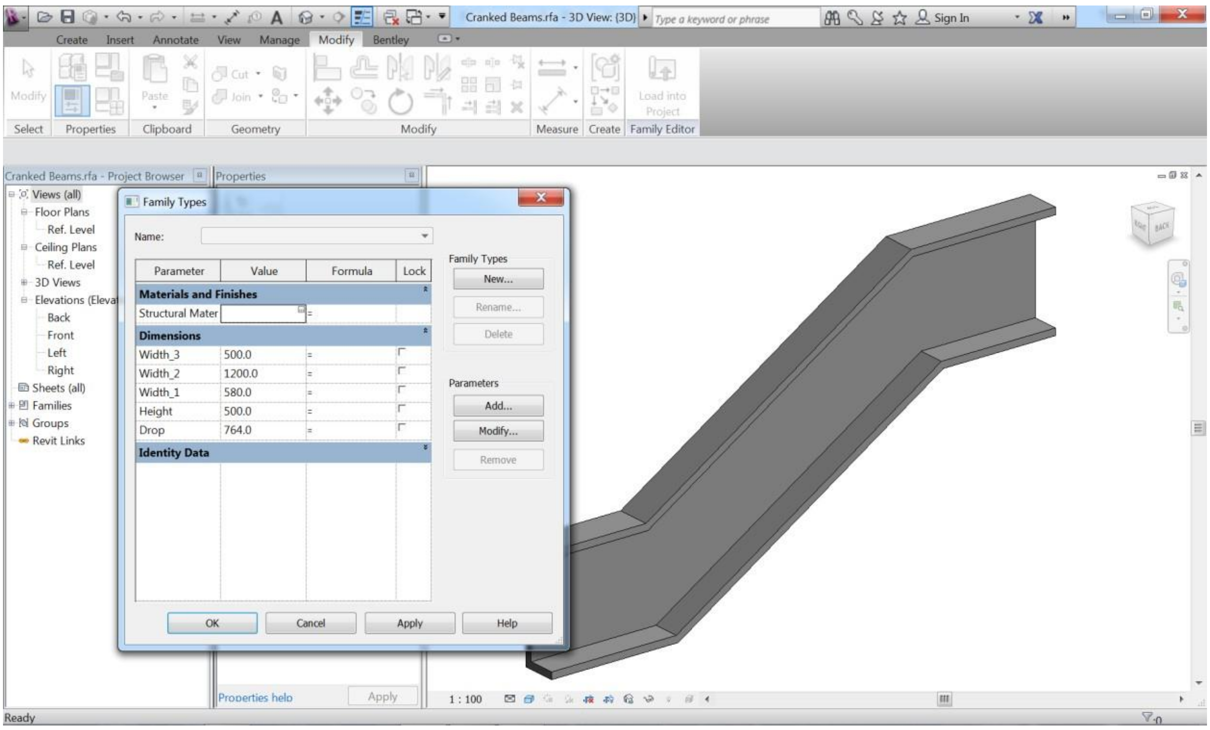


Go to 3D view, then you can see the geometry of your cranked beam is there.



14.5 Test

Change the values to see if the parametric frameworks work as intended.



Disclaimer: The process described in this tutorial is only to help the readers understand the concept of the topic and how to use related functions. This tutorial does not offer the optimized way to implement a project in Revit.

CHAPTER 15 - PRECAST HOLLOW CORE SLABS

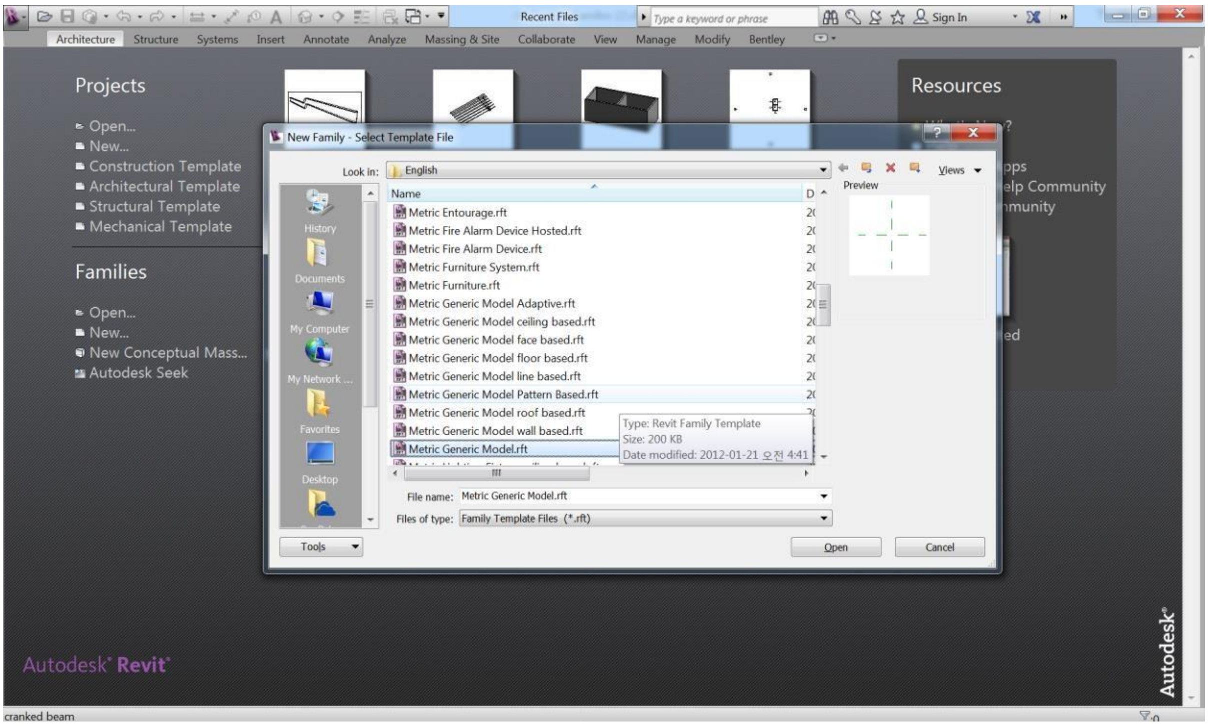
In this section, you will learn

- How to create precast hollow core slabs structural families in Revit

<image from http://www.bison.co.uk/>

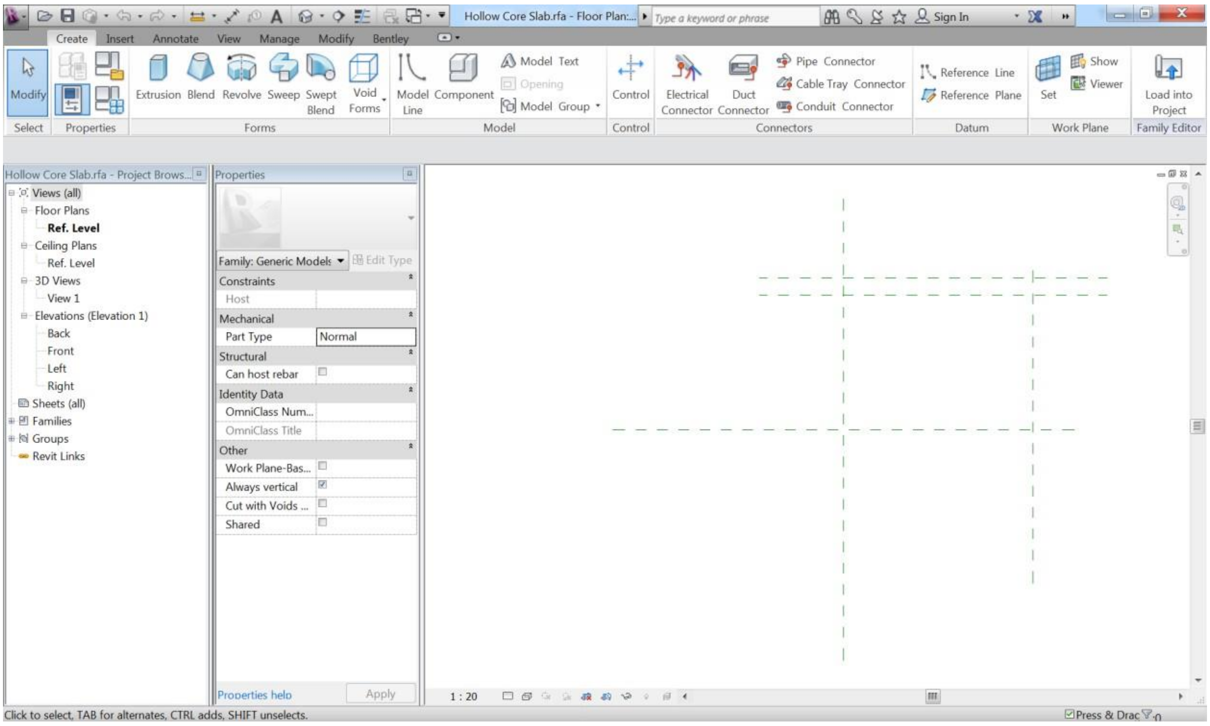
15.1 Template

First of all, we are going to create a basic geometry of the hollow core slab. Start a family editor with "Metric Generic Model".

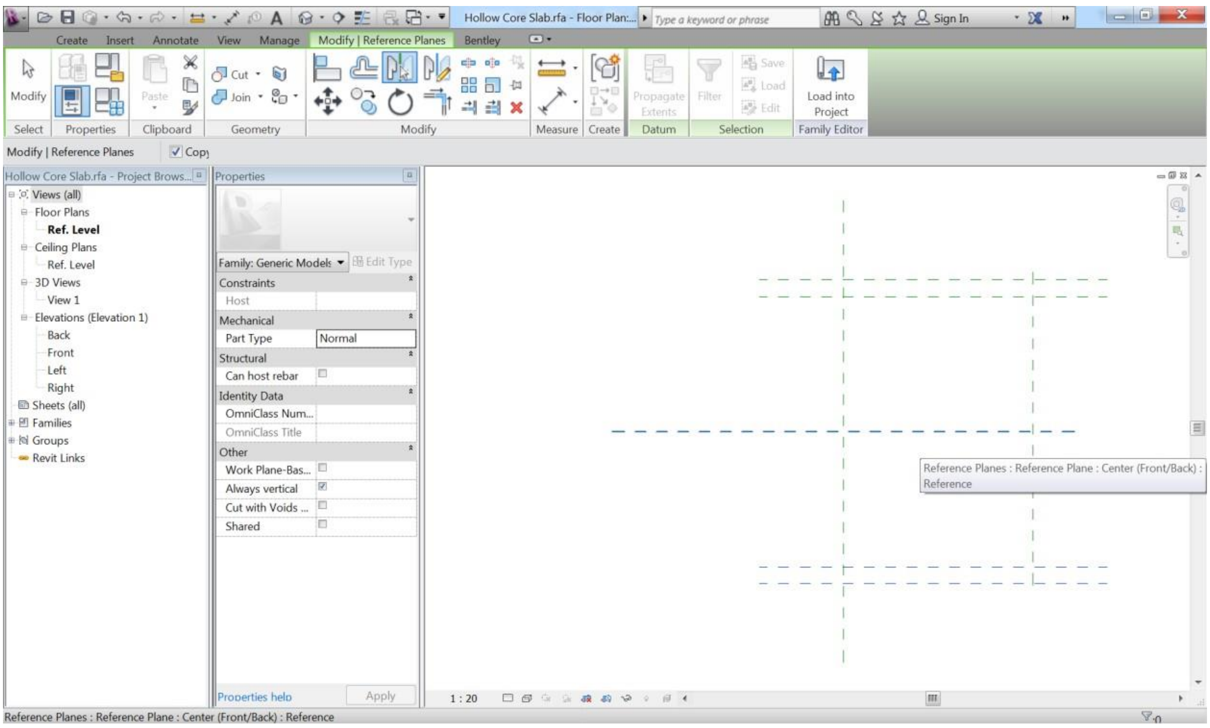


15.2 Reference planes

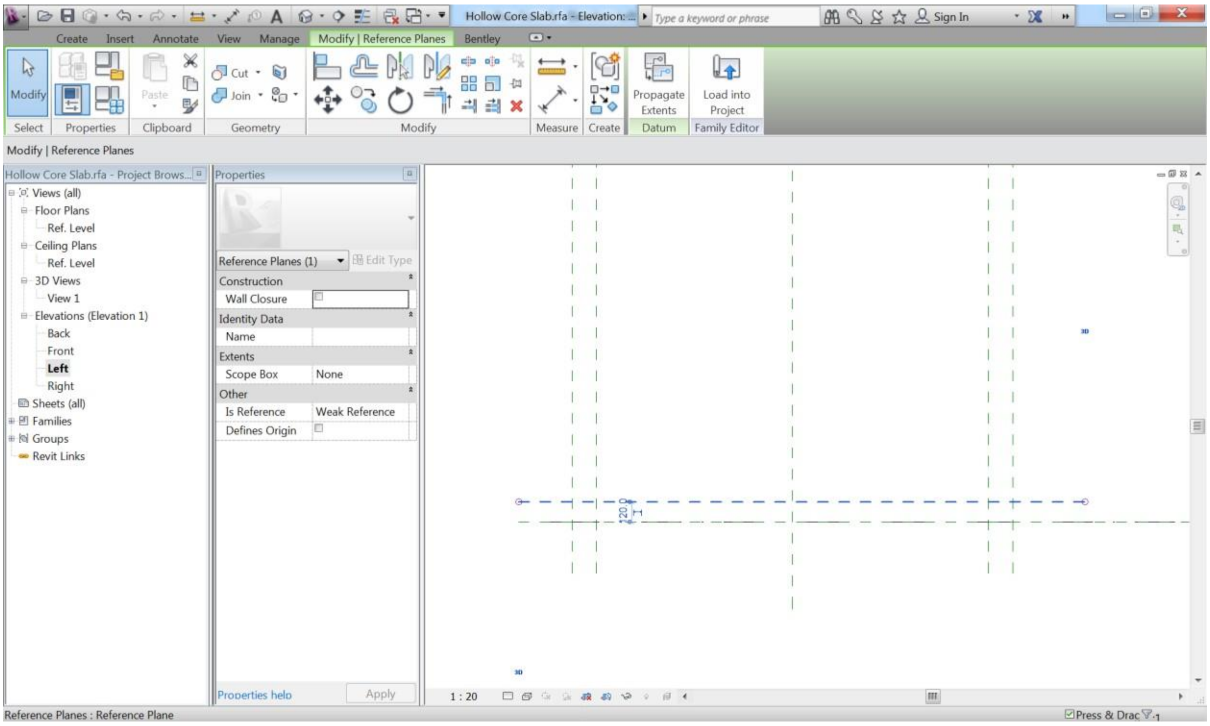
Once the family editor begins, add reference planes as the image below shows. - Go to "Create" tab and click "Reference Plane" button.



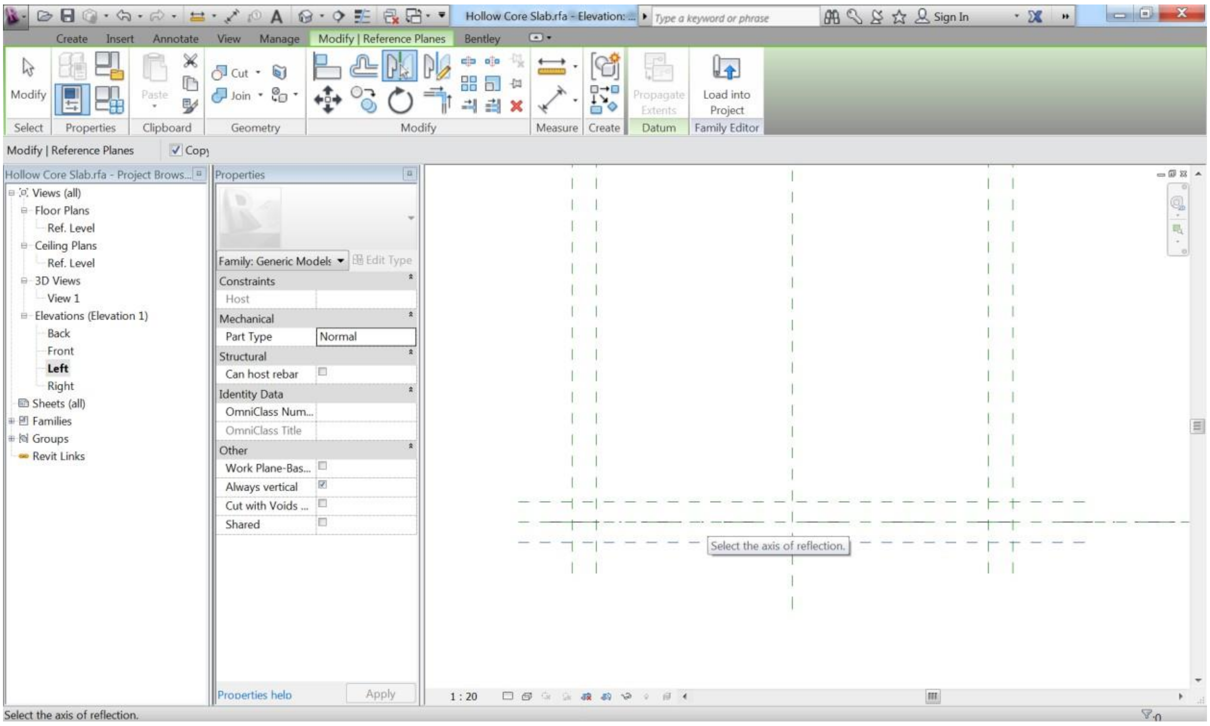
Then mirror the upper 2 reference planes to the other side.



Go to “Left” view, add a reference plane on either side of the horizontal center reference plane.

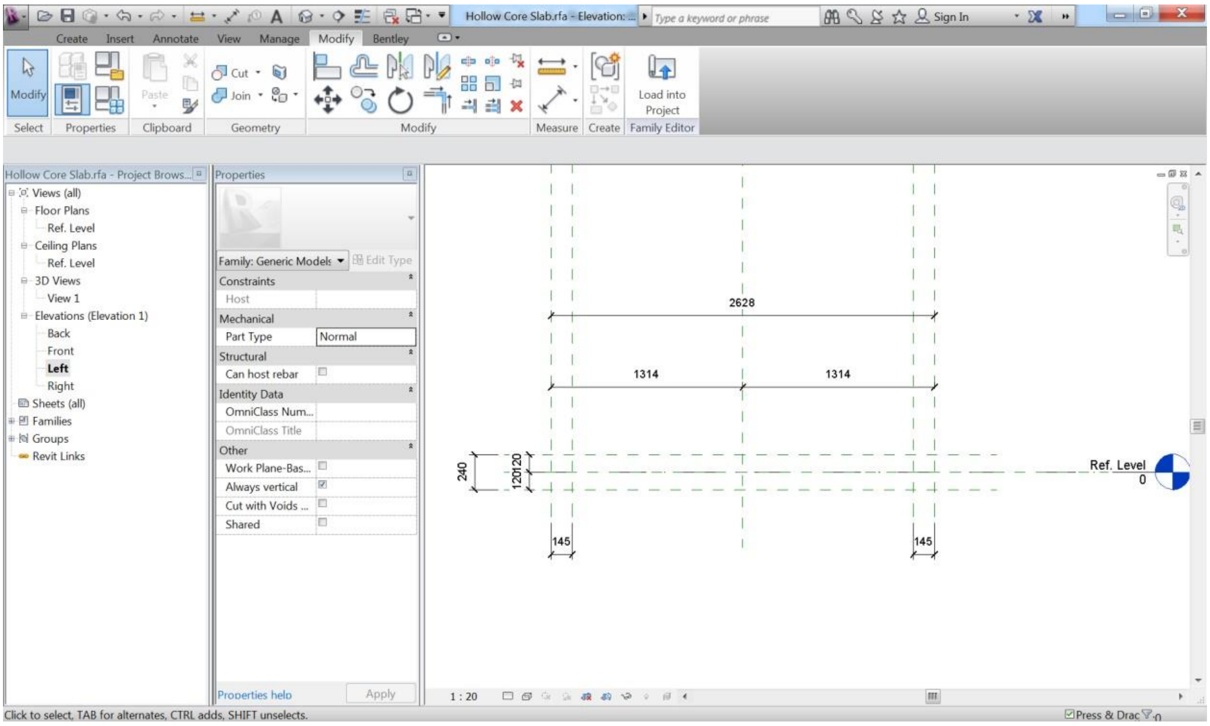


Then mirror the added reference plane to the other side of the horizontal center reference plane.

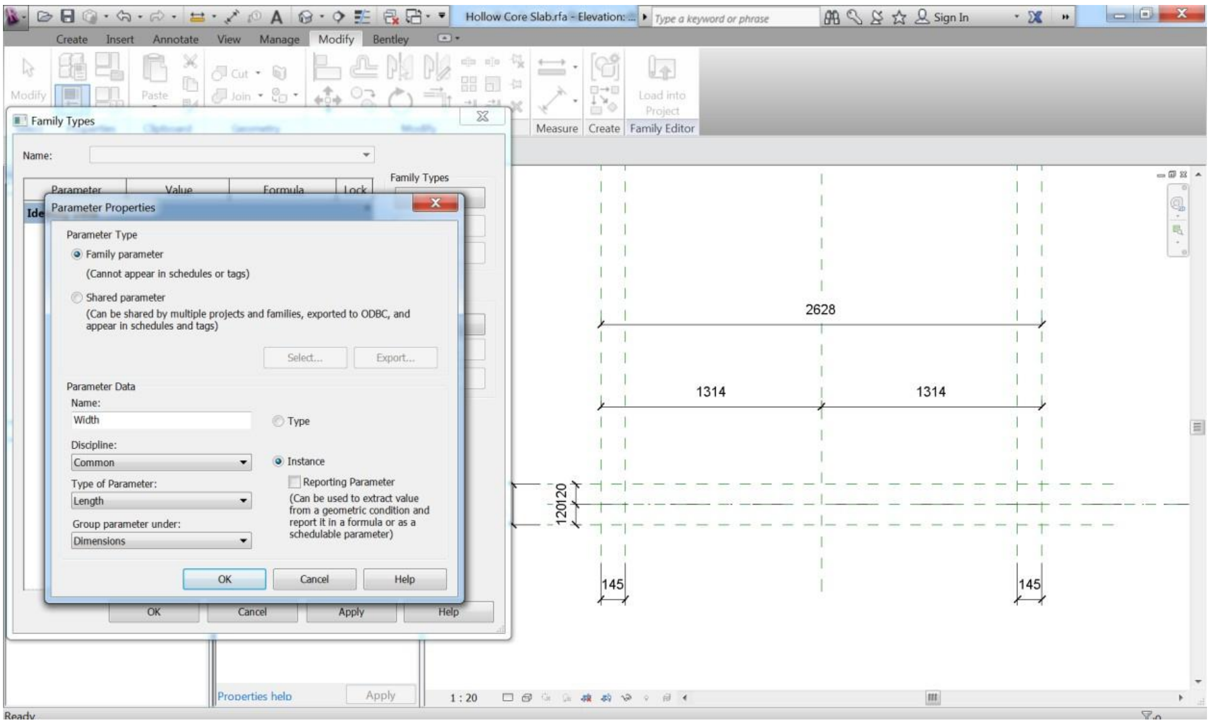


15.3 Parametric Dimension

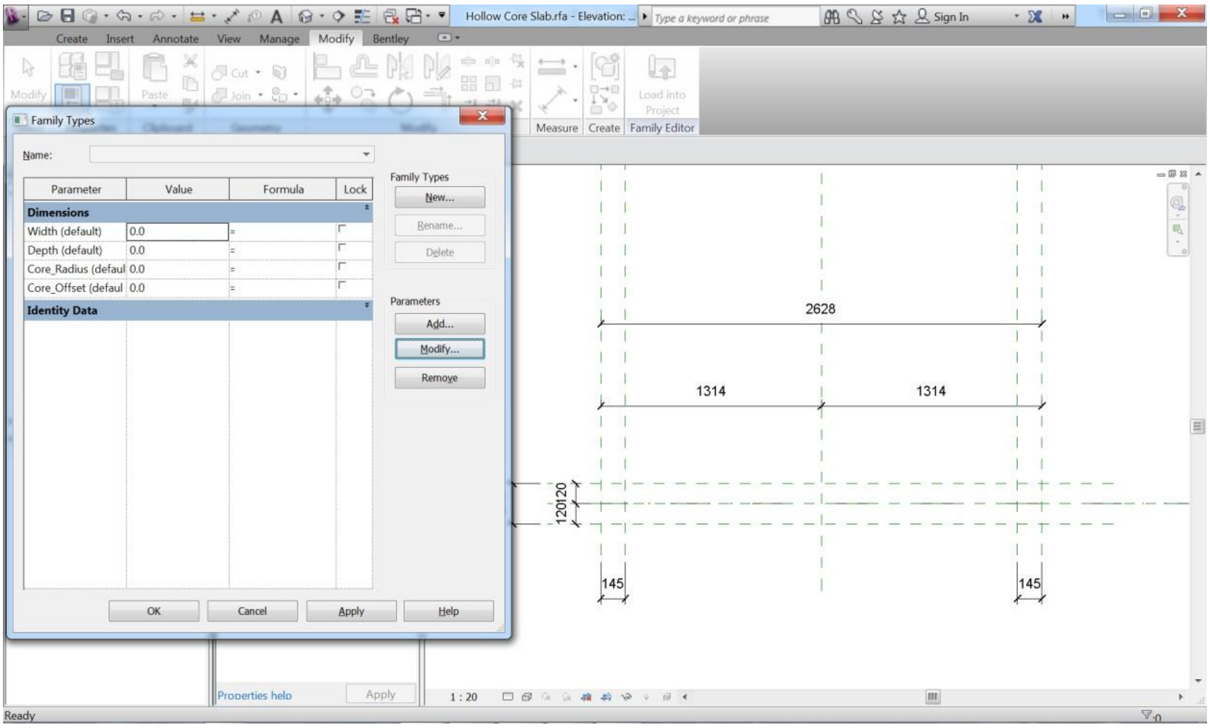
In the “Left” elevation view, add dimensions as the image below shows.



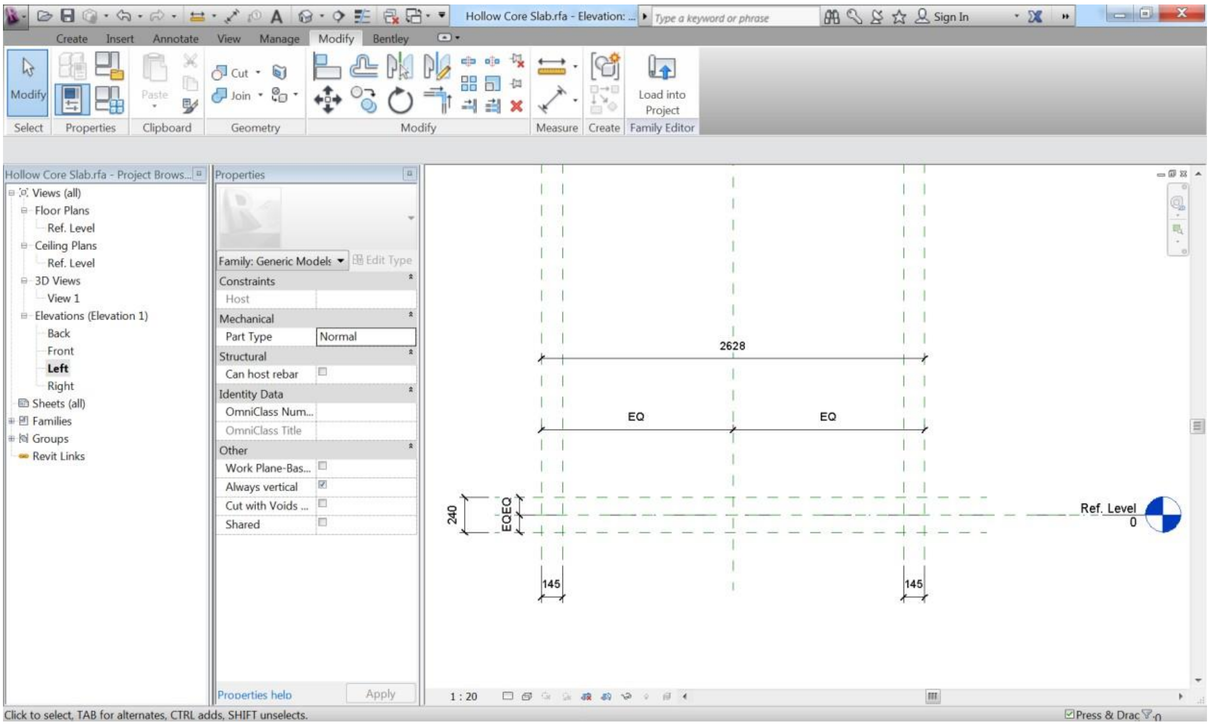
Go to “Family Types” window and add a parameter called “Width”. set the options to “Common”, “Length”, “Dimensions”, and “Instance”.



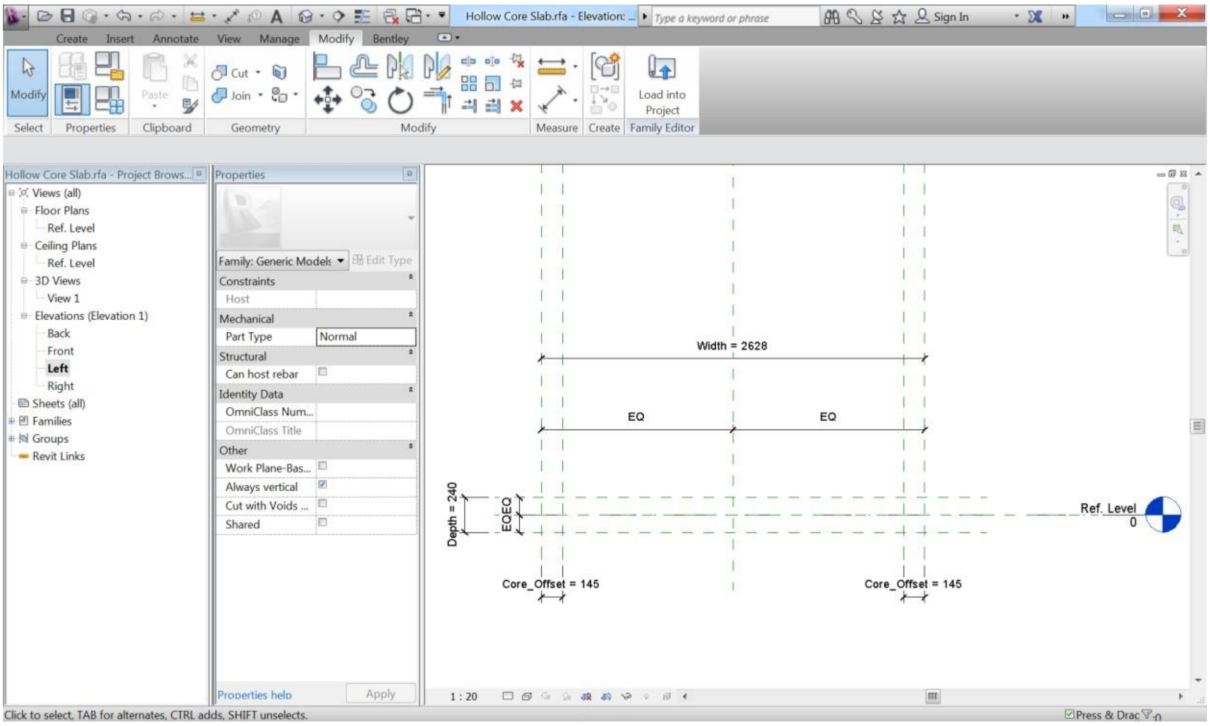
With the same settings, add "Depth", "Core_Radius", and "Core_Offset".



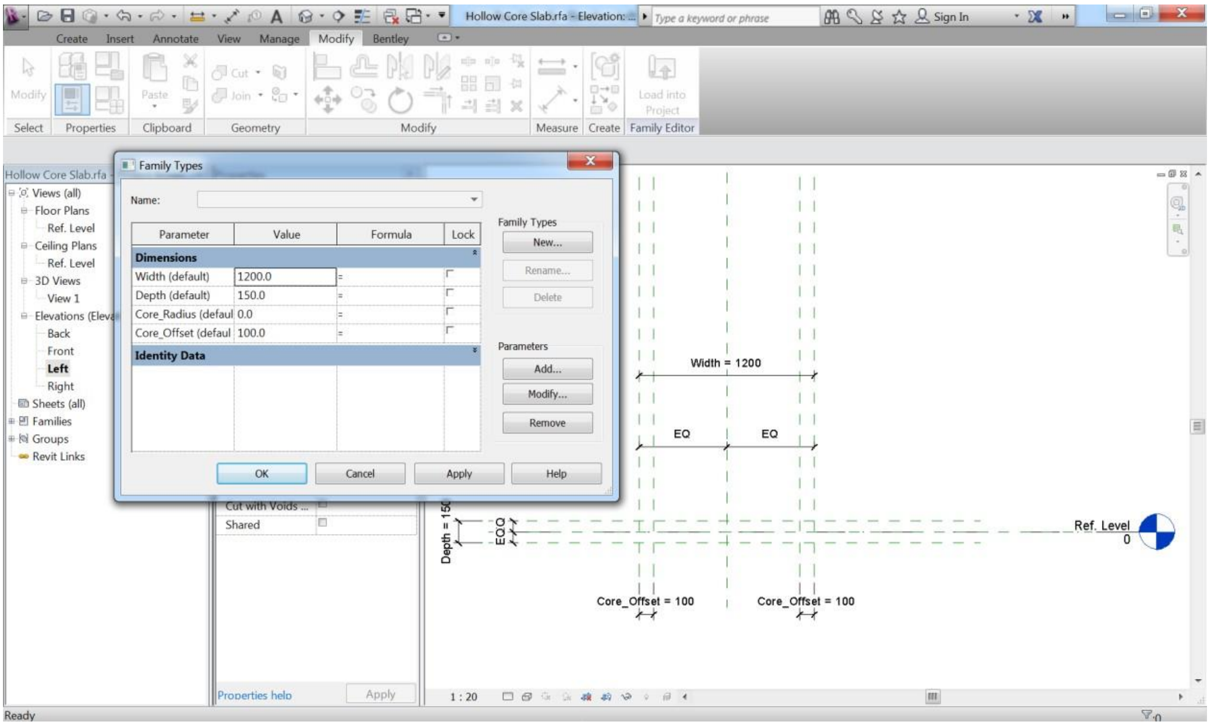
Go back to the view, and turn the two sets of dimensions to "EQ" dimension.



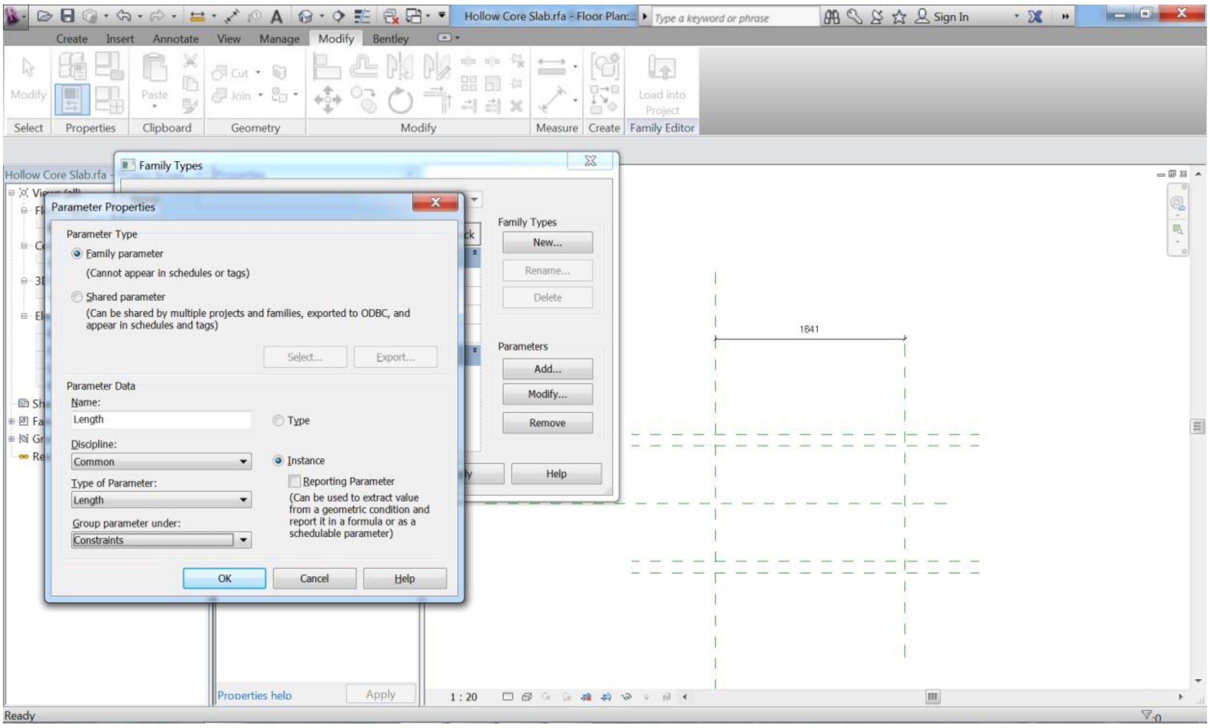
Associate "Core_Offset", "Depth" and "Width" parameters to the dimensions as below.



Set the value of "Width" to "1200", "Depth's" to "150", and "Core_Offset's" to "100".



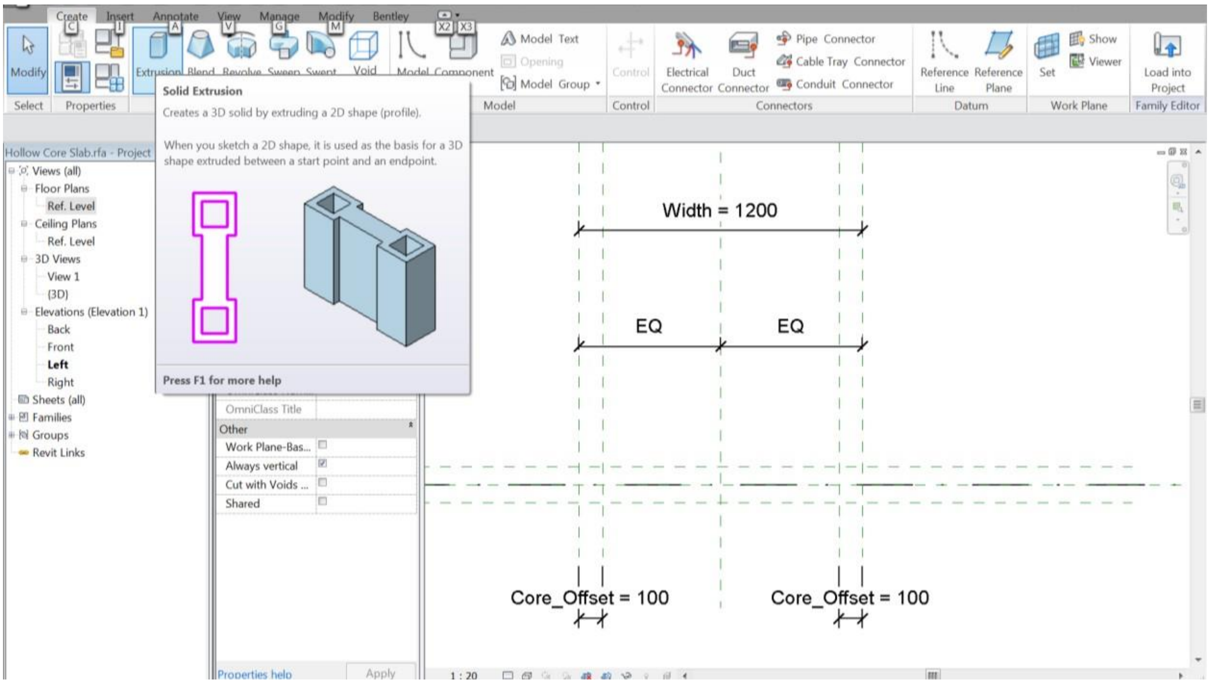
Go to the floor plan view and add a dimension between the vertical center and the paralleling reference plane. Label the dimension with a new parameter called “Length”. The group needs to be “Constraints”.



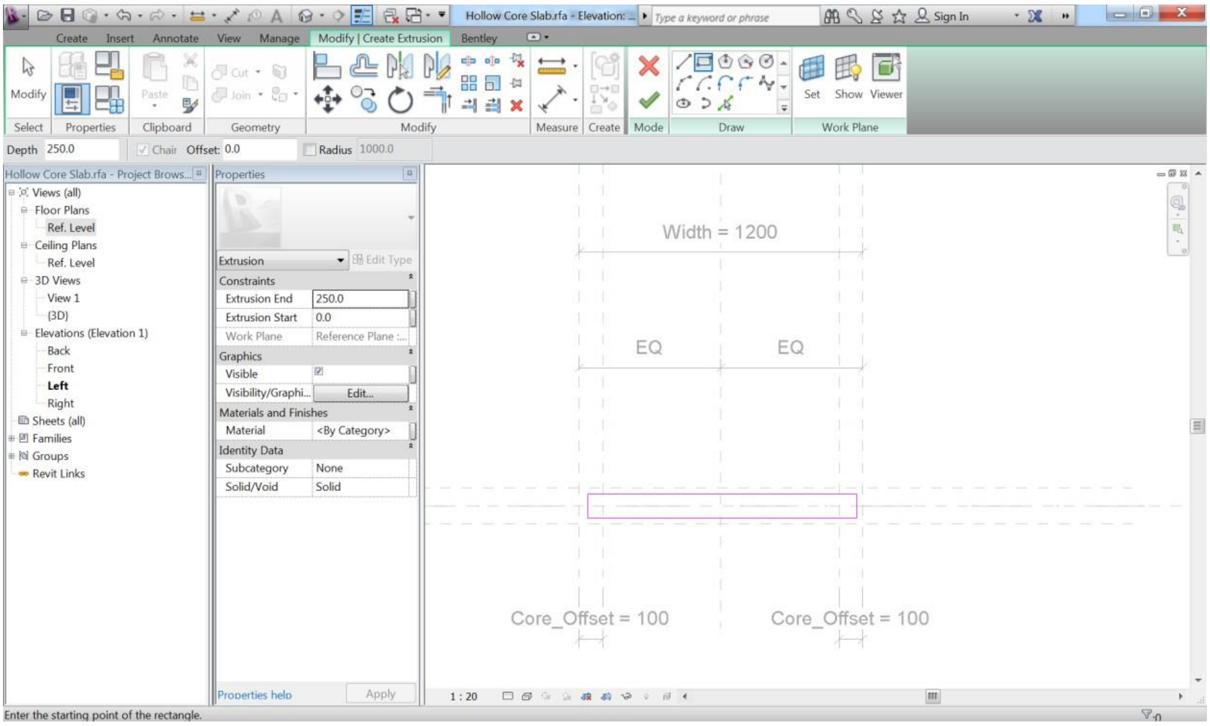
15.4 Solid Geometry

Now we are going to create a solid geometry for the family. Go to “Left” elevation view and click

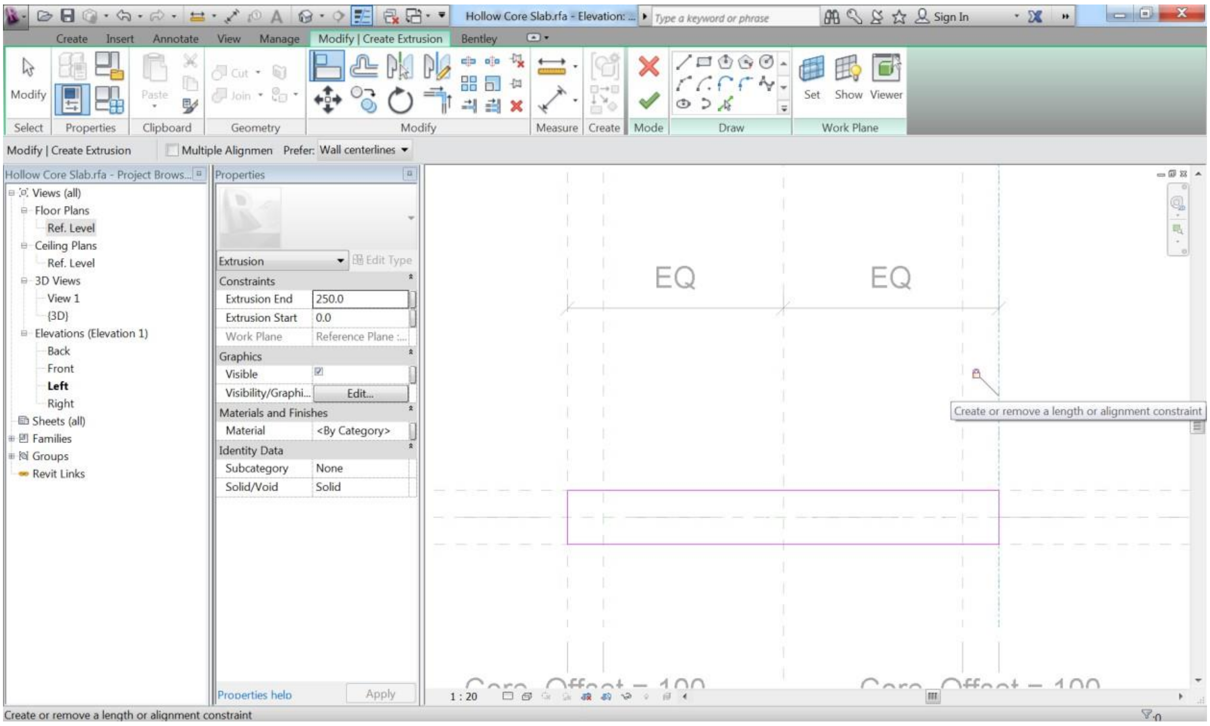
“Extrusion” button.



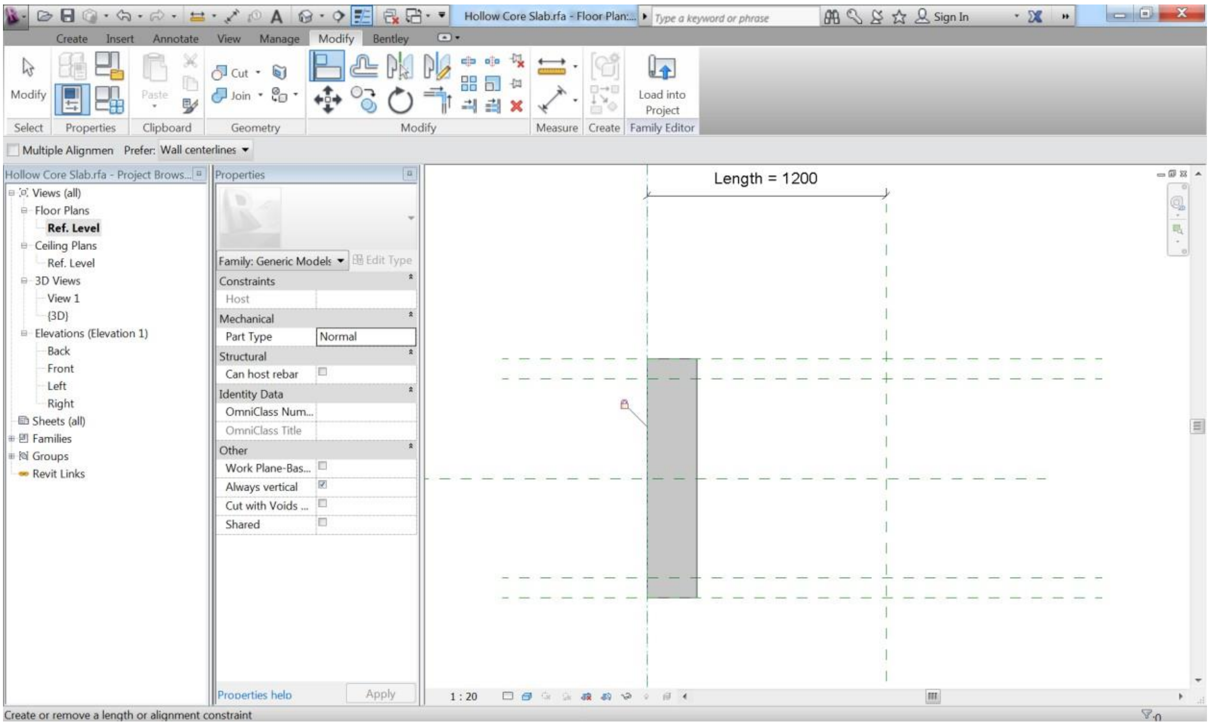
Place a rectangle within the reference planes.

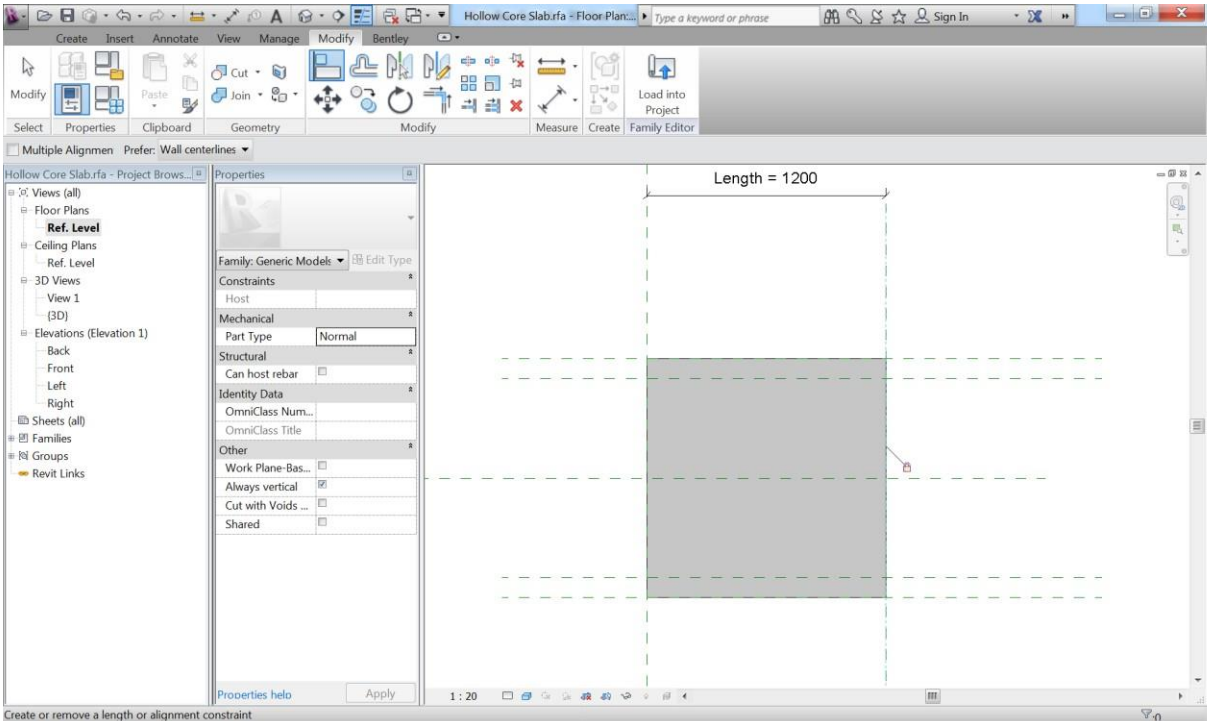


Align and constrain each edge of the rectangle to the corresponding reference planes.



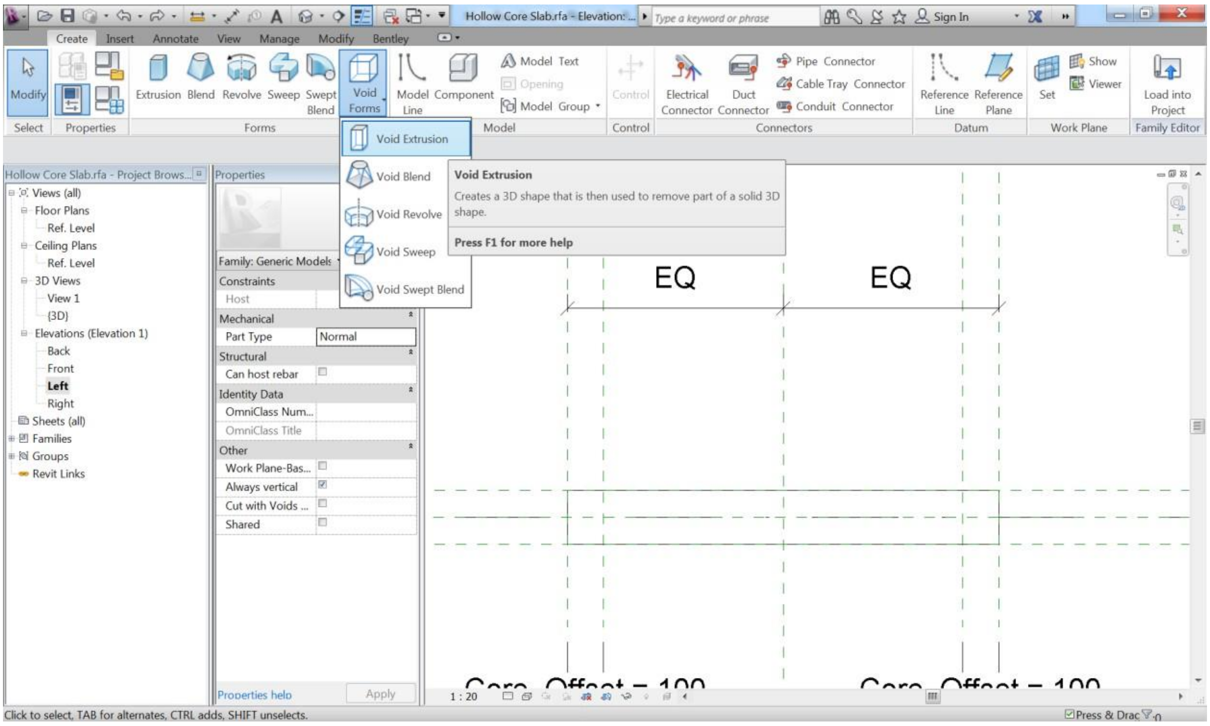
Go to the top view and align and constrain the two faces of the extrusion to each reference plane.



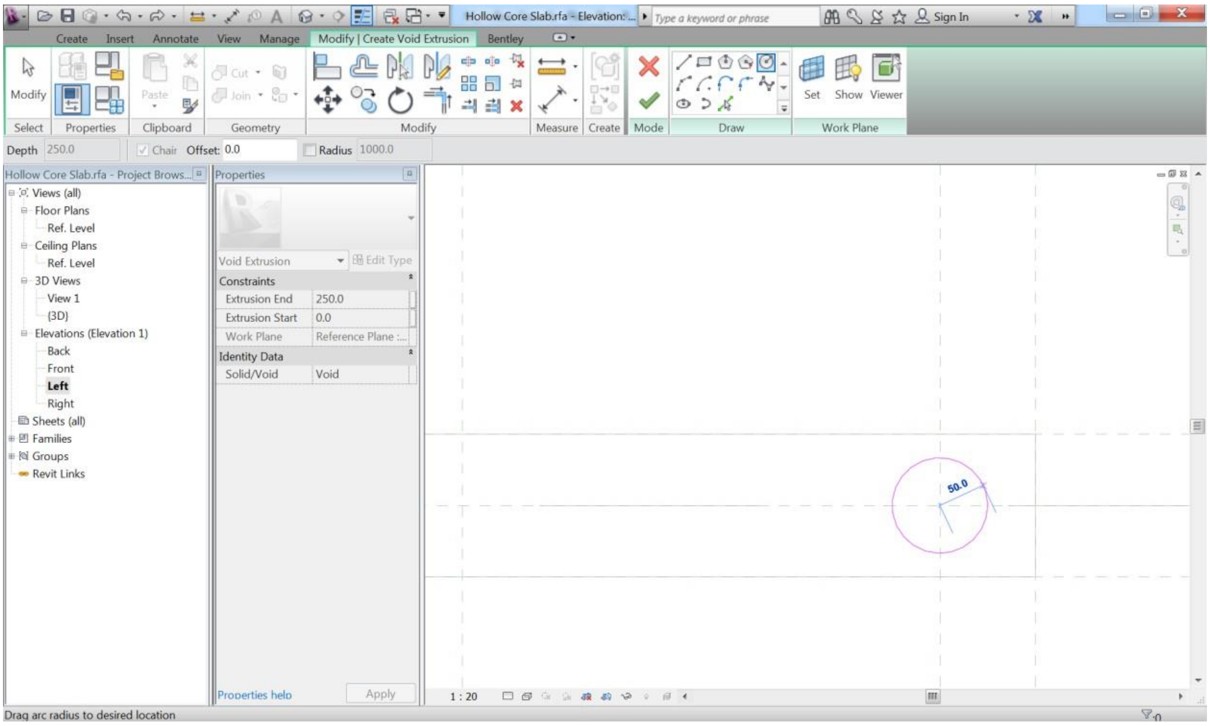


15.5 Void Geometry

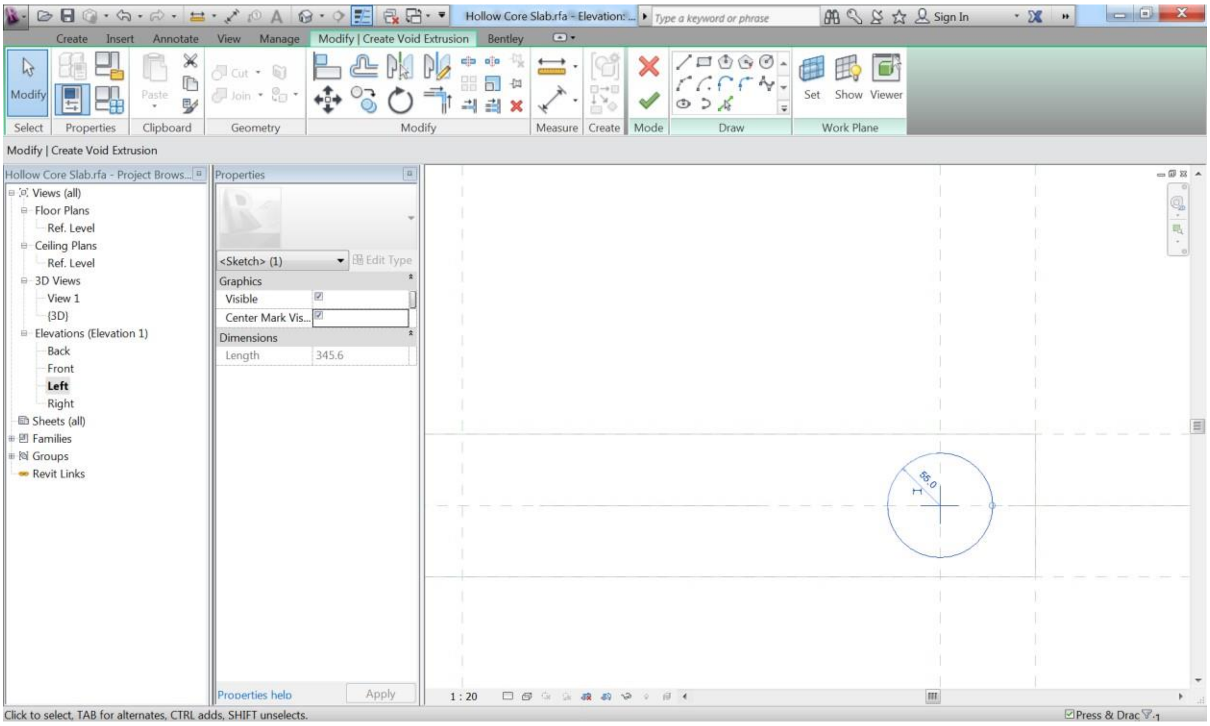
Now we need to add void extrusion to the solid extrusion. Go to “Create” tab and click “Void Extrusion” button.



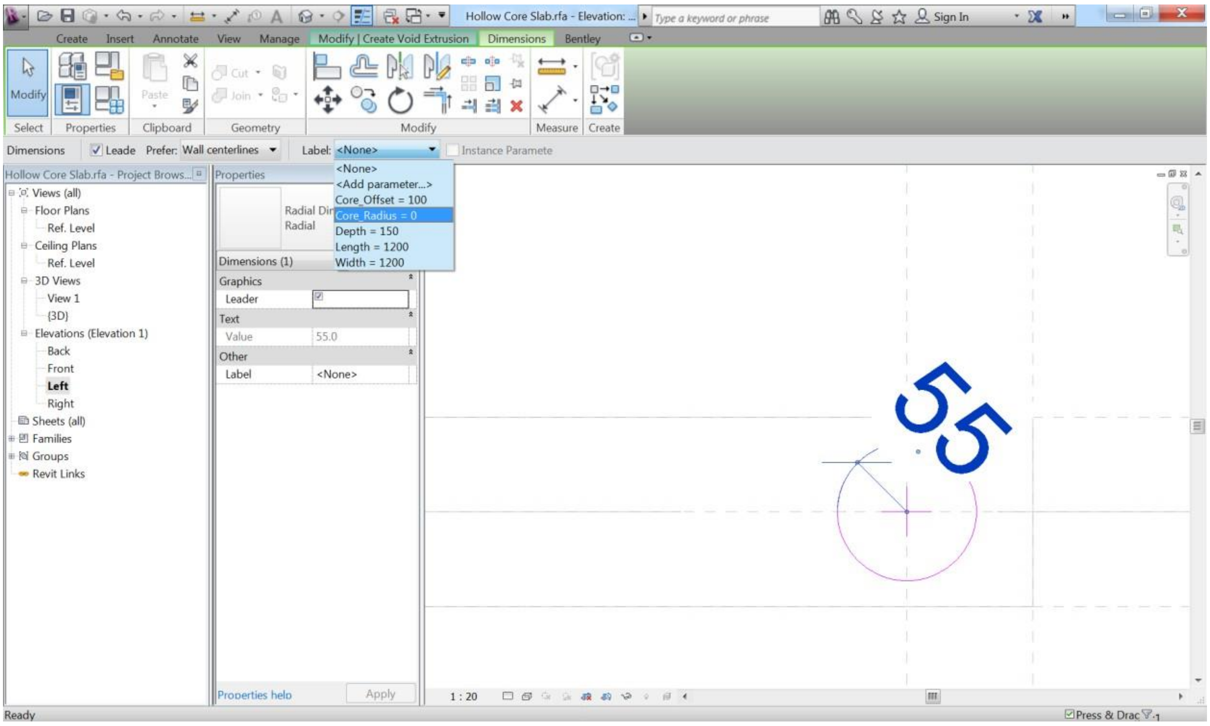
Place a Circle on the right side of the extrusion.



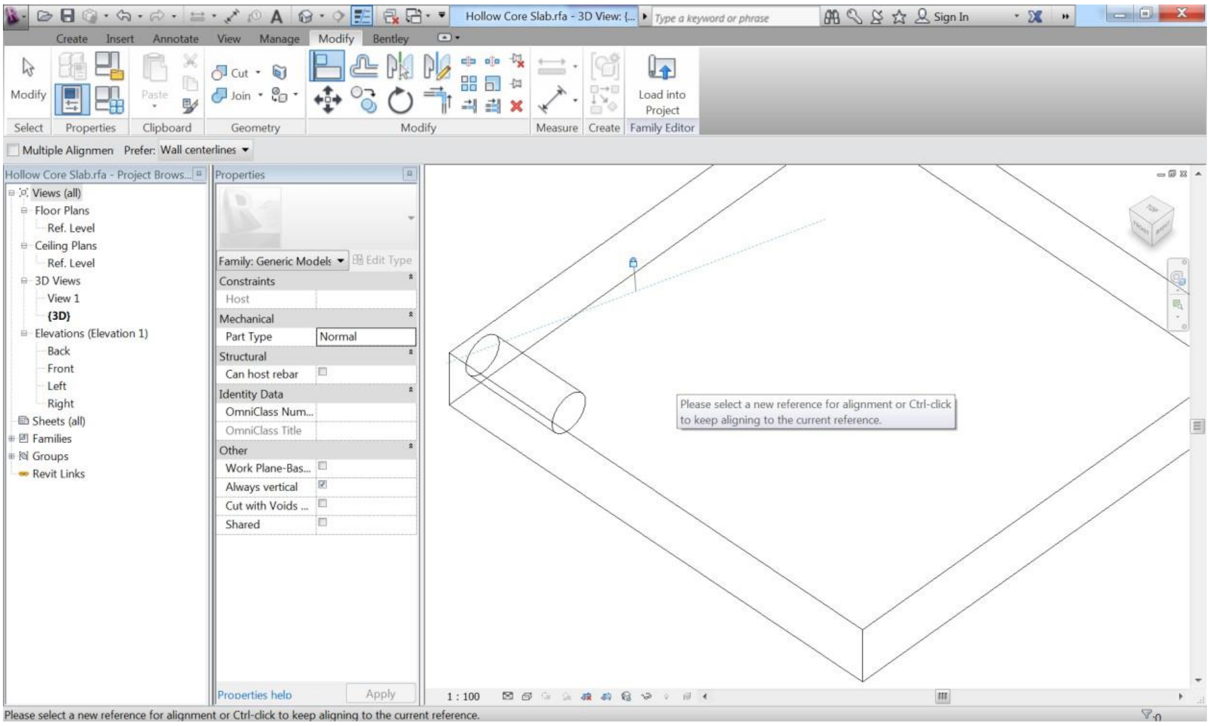
check the “Center Mark Visible” option to align the circle to the reference planes.



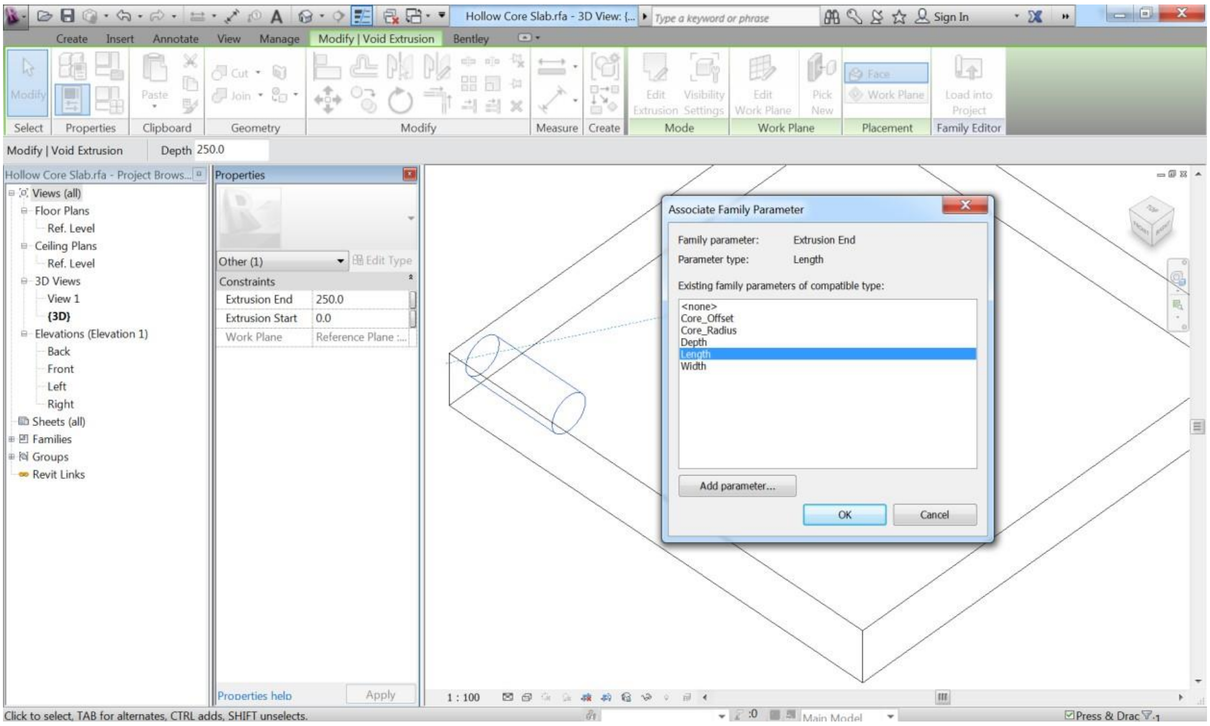
Turn the temporary radius dimension into permanent and label it with “Core_Radius” parameter.



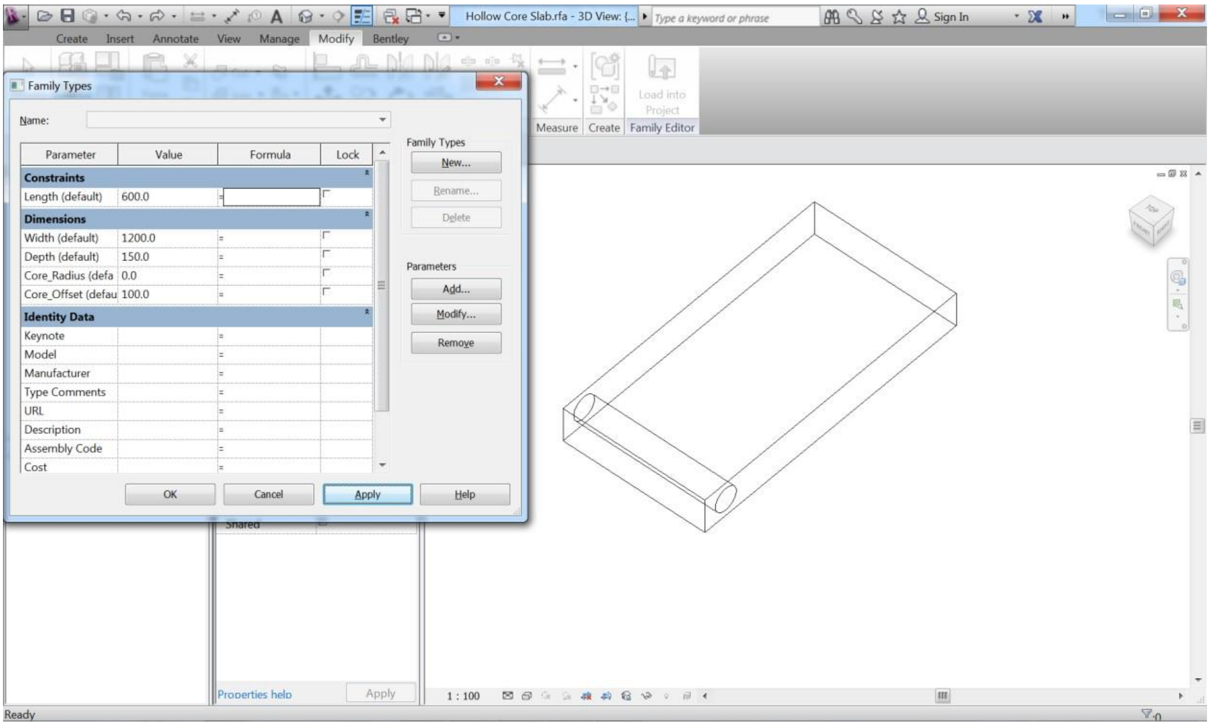
Finish the sketch mode and go to 3D view. Align and constrain the starting face of the void extrusion to that of the solid extrusion.



Select the void and associate its “Extrusion End” parameter to “Length” family parameter.

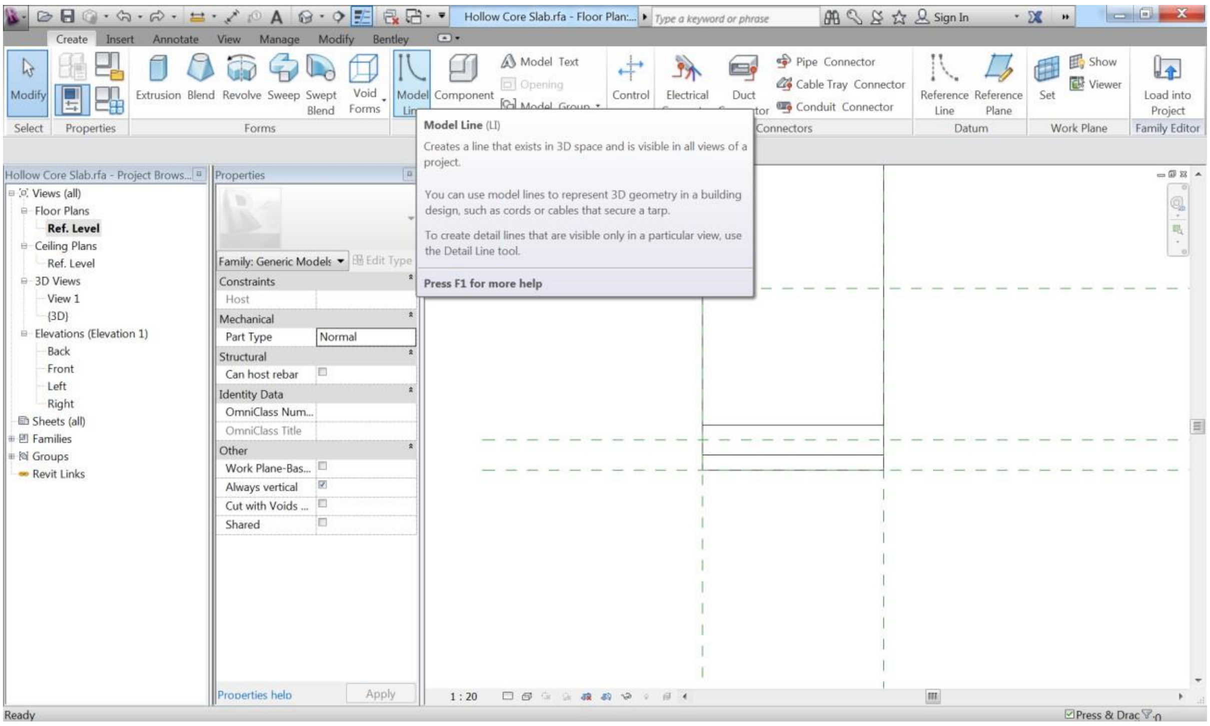


Before moving on, open "Family Type" and change the "Length" value to test the constraint.

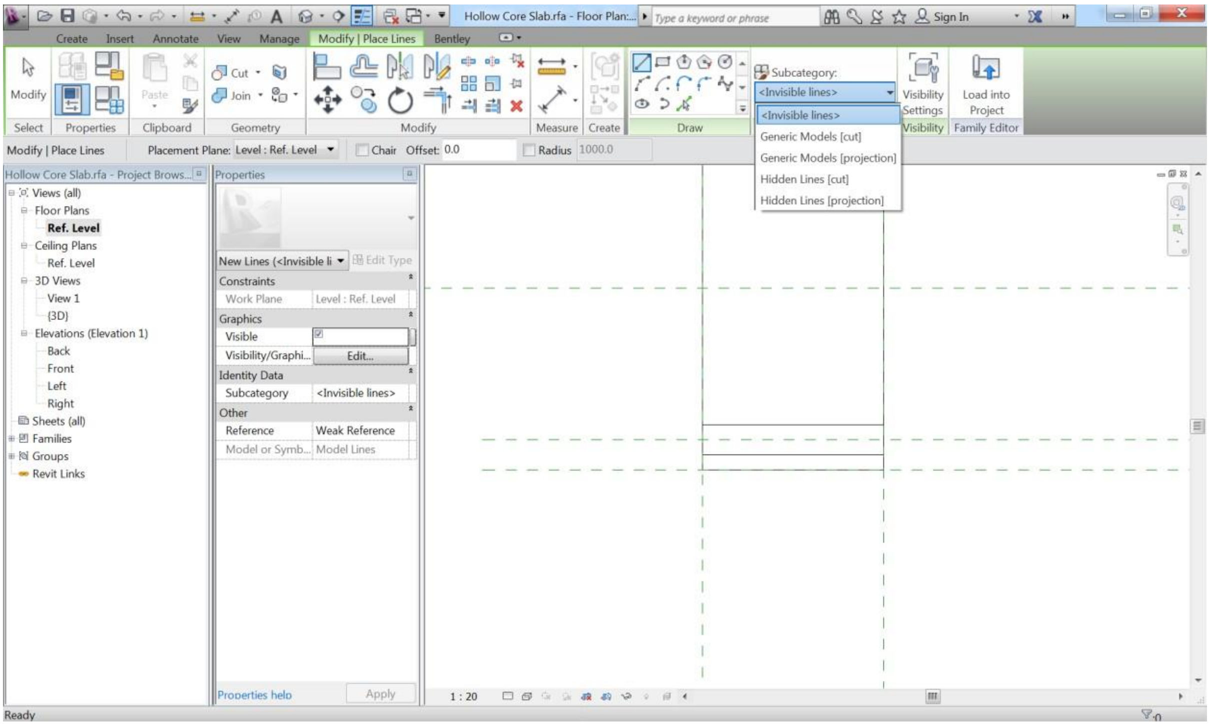


15.6 Group

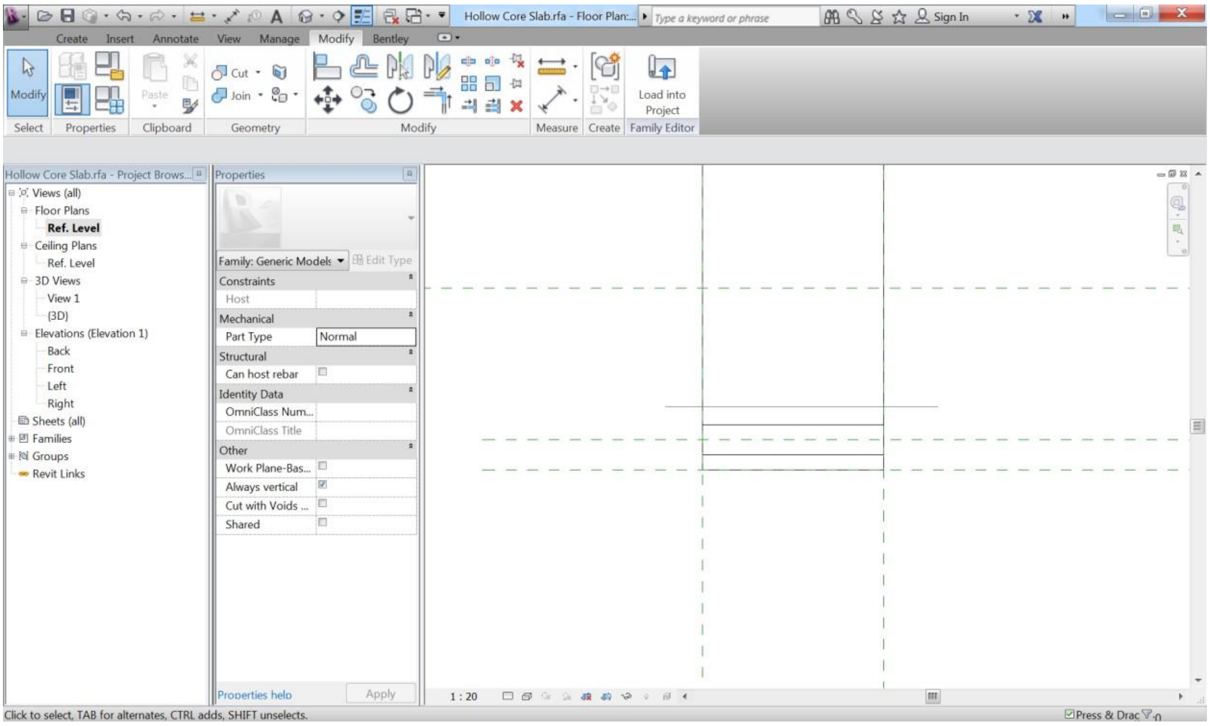
Go to the "Ref. Level" view click 'Model line" under "Create" tab.



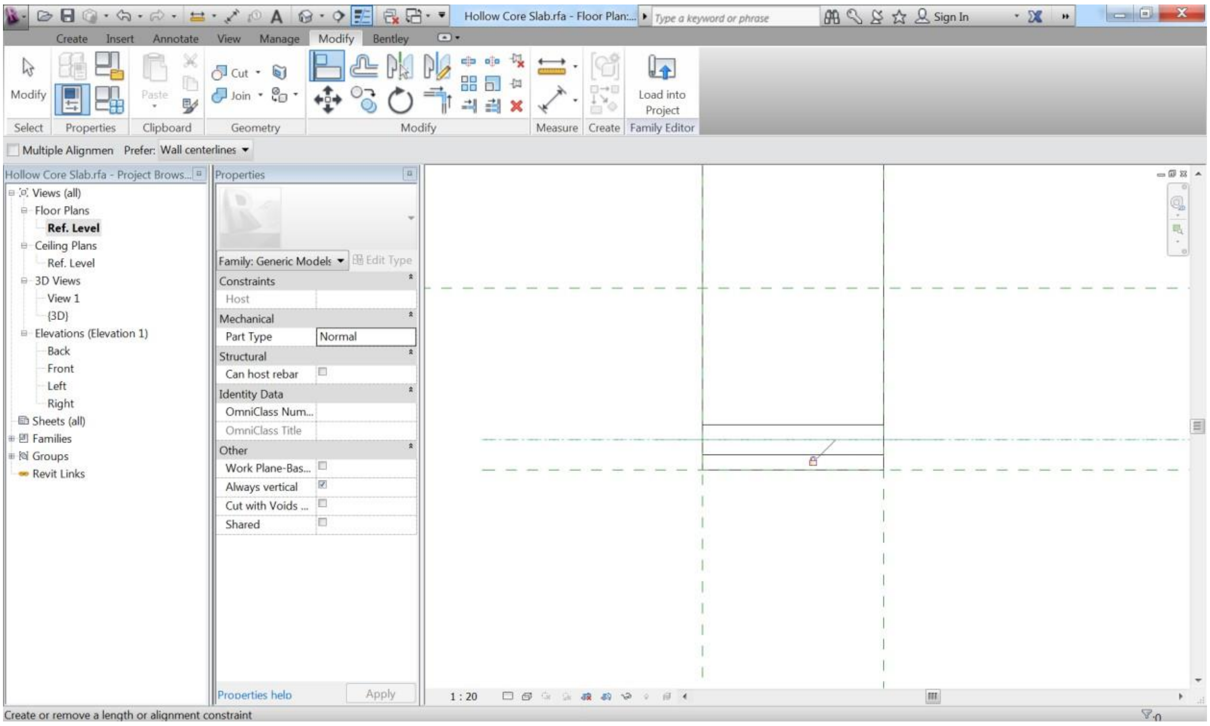
From “Subcategory”, select “<Invisible lines>”.



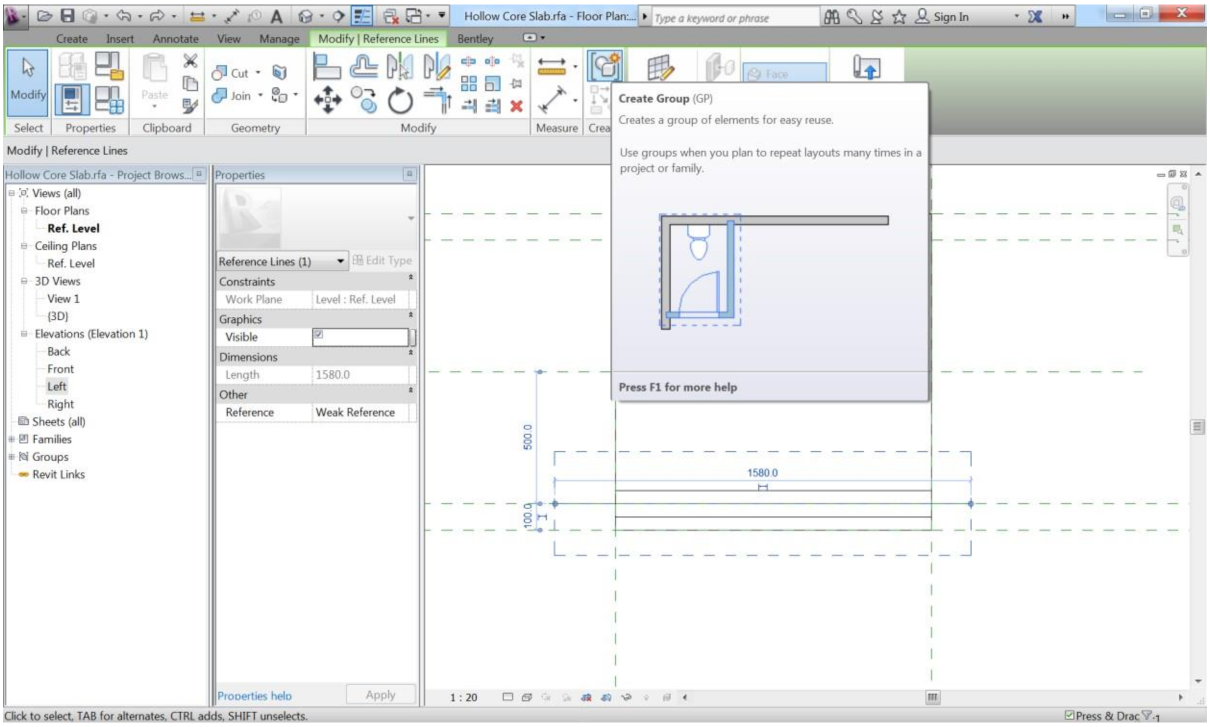
Add an invisible line around the void extrusion.



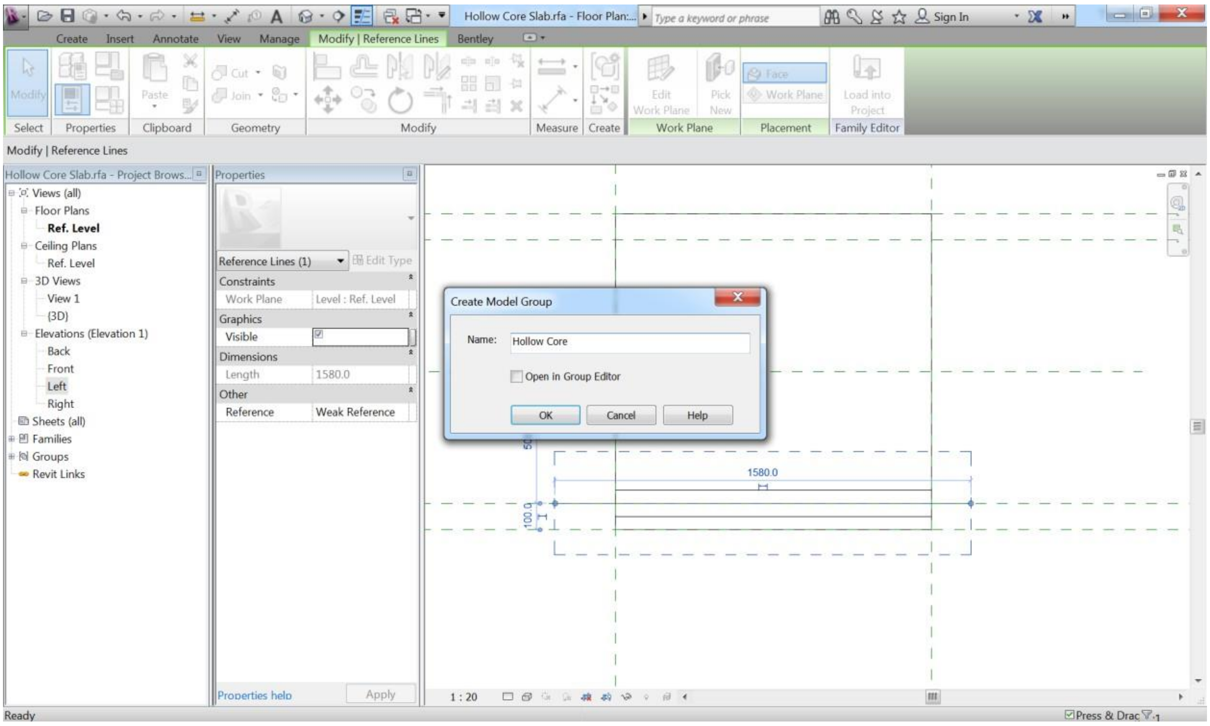
Align the invisible line to the reference plane that go through the center of the void extrusion.



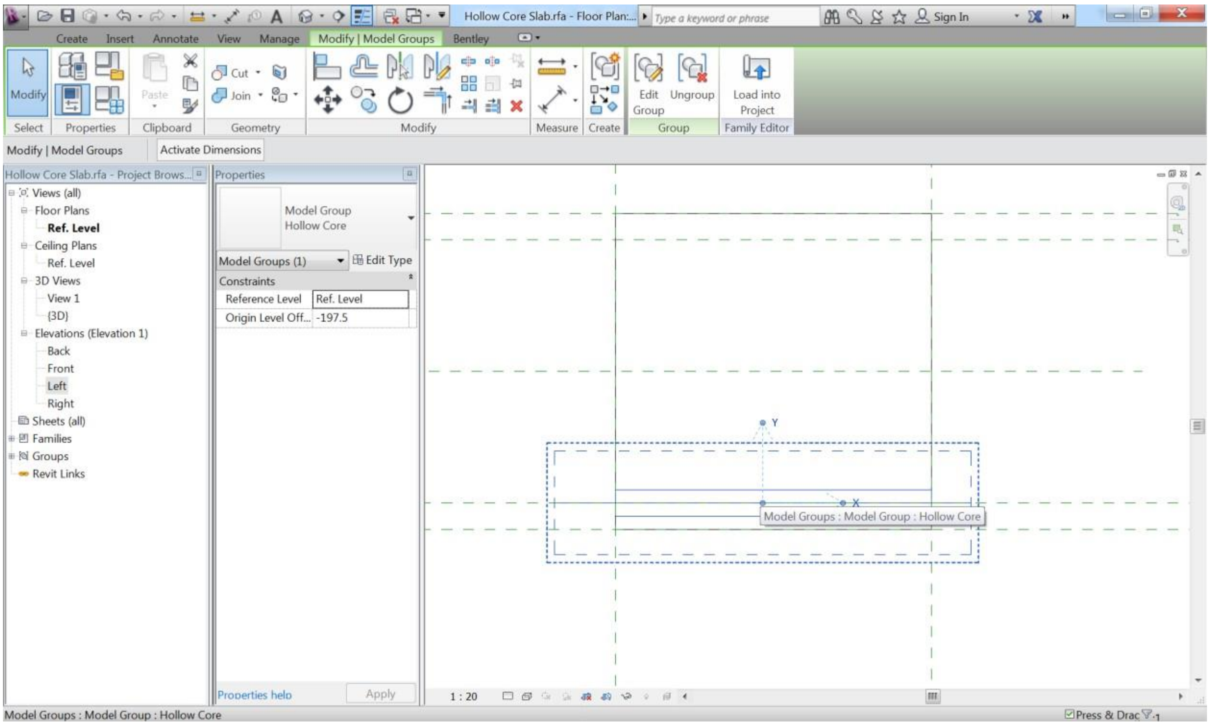
Select the invisible line and the void extrusion and click “Group” button under “Modify” tab.



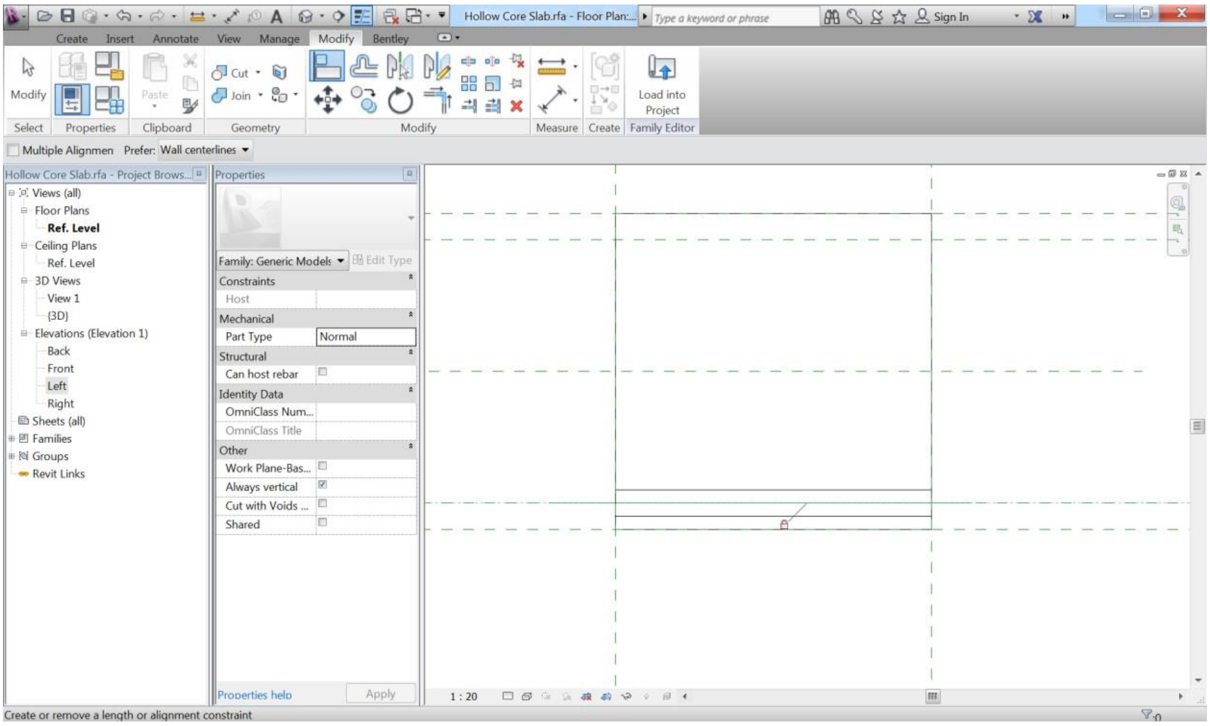
Type in "Hollow Core" for the name.



Select the group, then it should look like the one in the image below.

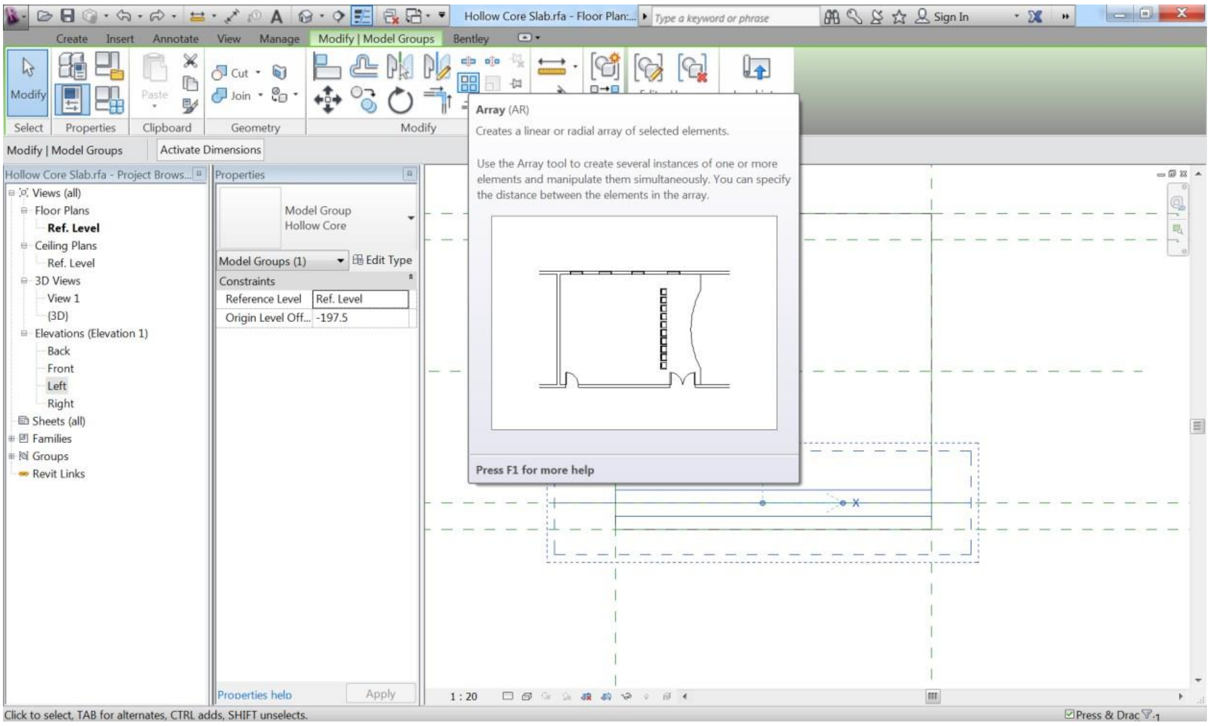


Align and constrain the invisible line in the group to the reference plane.

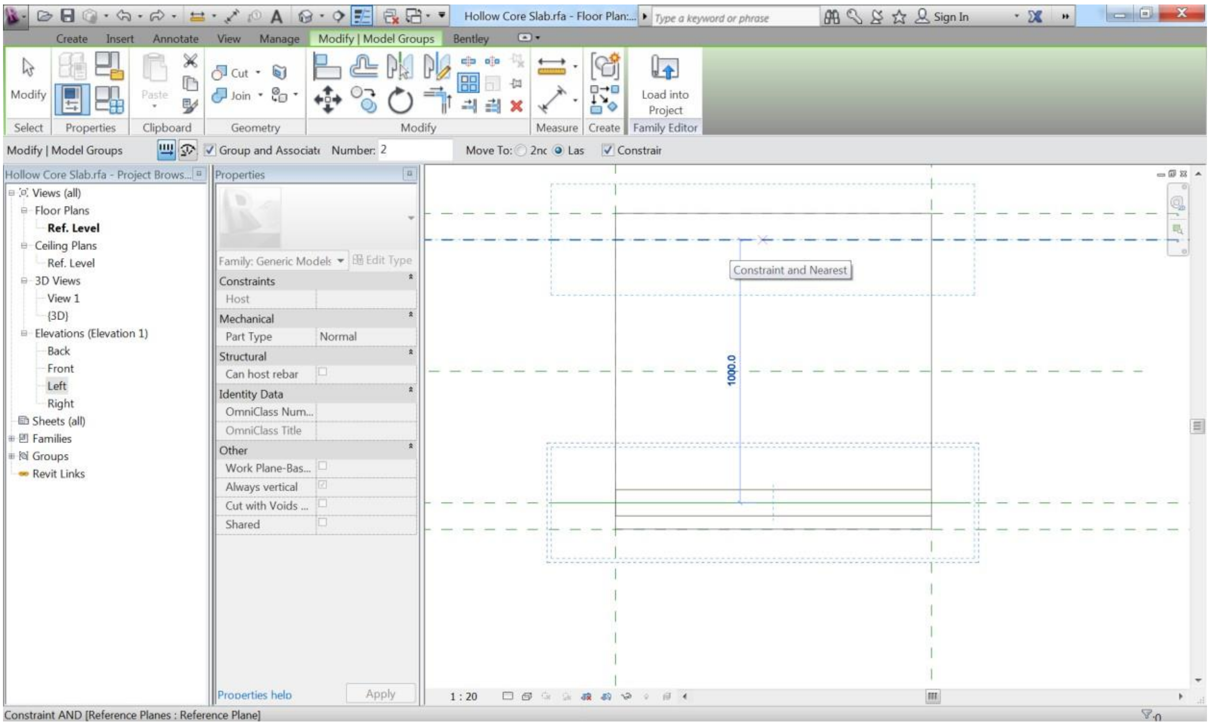


15.7 Array

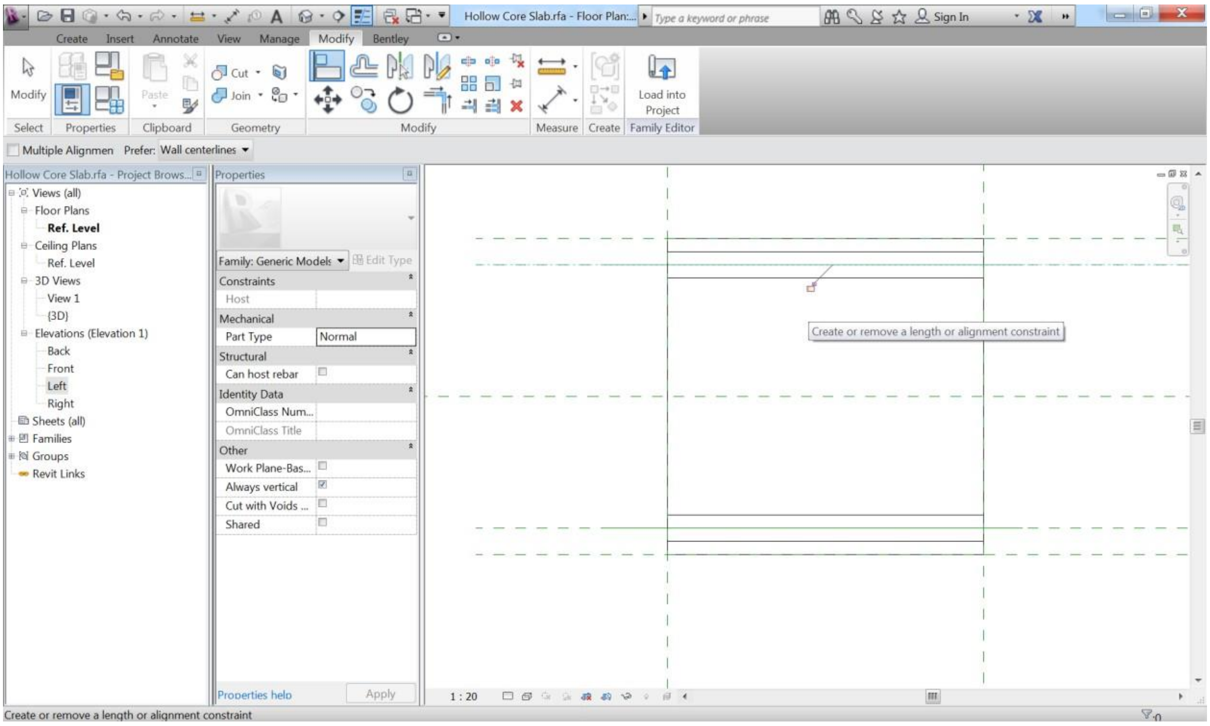
Select the group and go to "Modify" tab and select "Array" function.



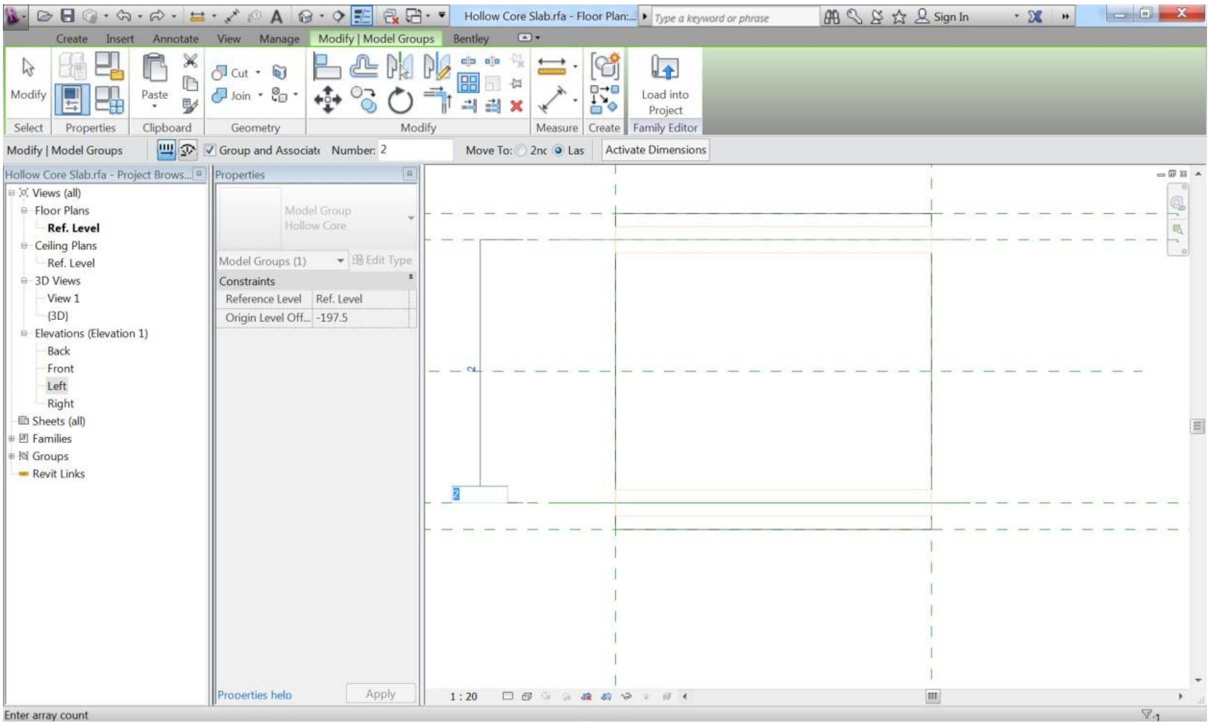
Set the "Move to" to "Last" from the option bar and click then standard point and target point for the array. At this point, you don't need to change the number of the array.



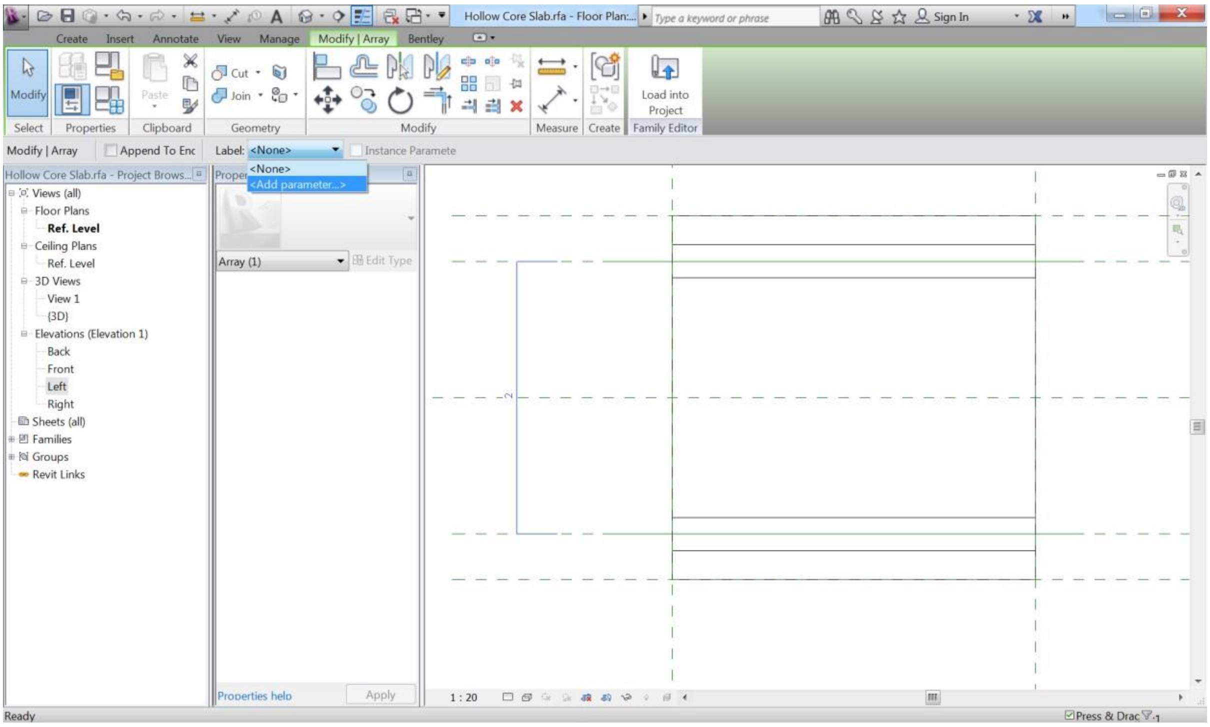
Once array is done, align and constrain the arrayed group to the reference plane. refer to the image below.



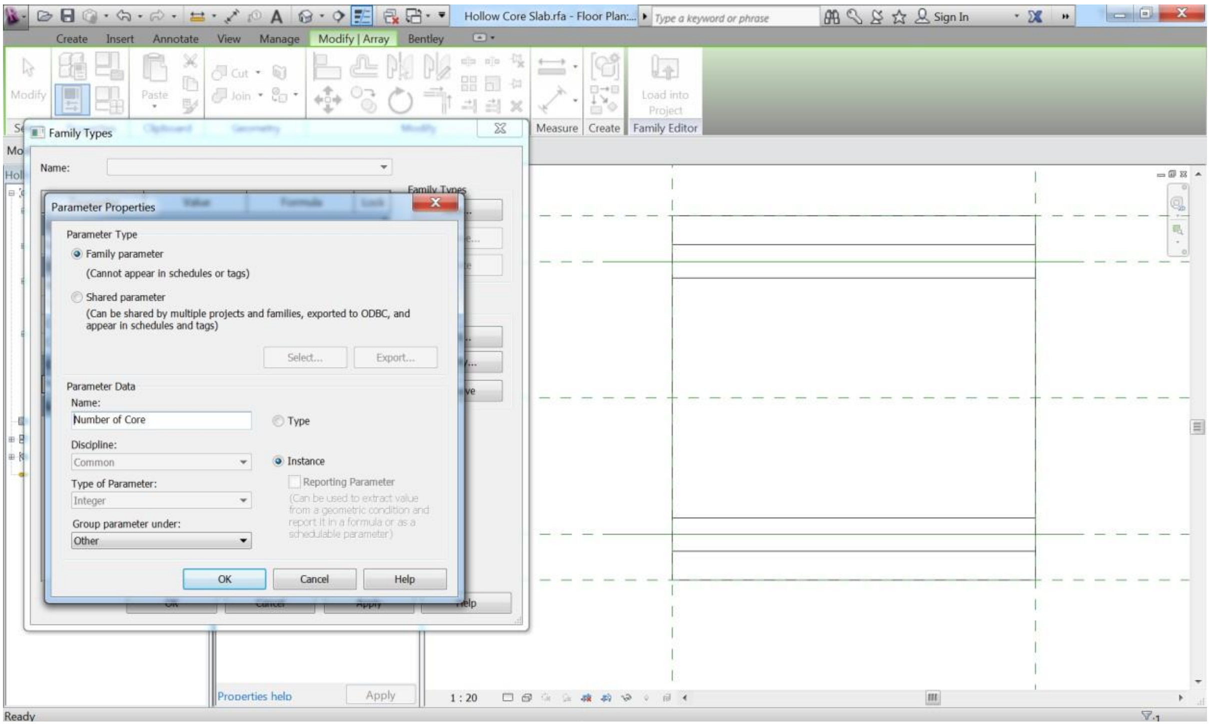
Now select the two group to make the number annotation visible.



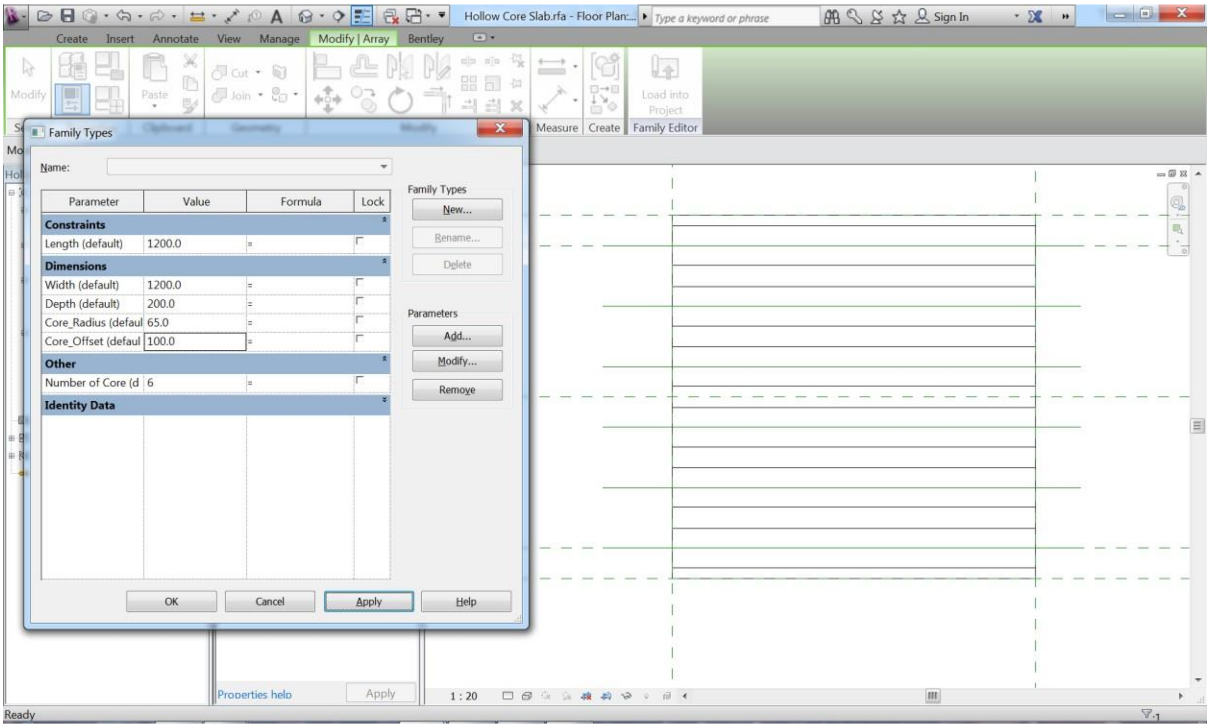
Select the number annotation and add a parameter to it.



Type in "Number of Core" for the name of the parameter.

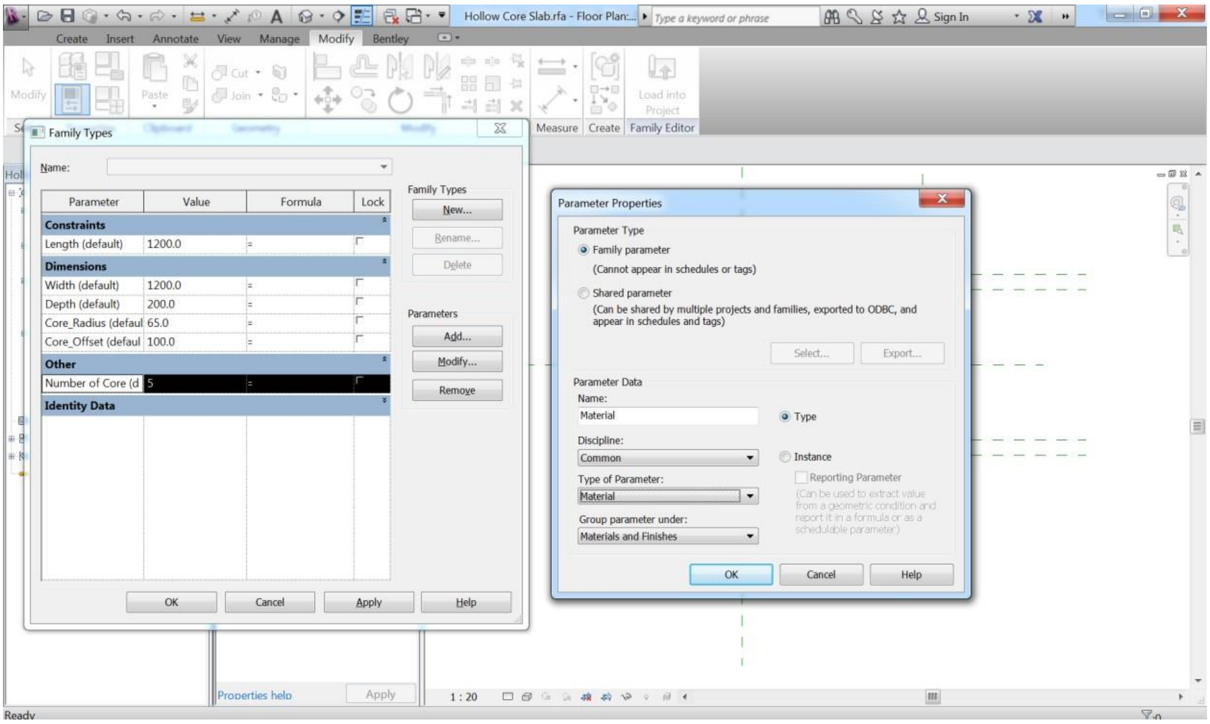


Open "Family Types" window and change the values to test your parametric frameworks. note that in this kind of complicated processes, constraints can be lost. make sure this works properly. Otherwise, identify where constraints are lost and fix the problem.

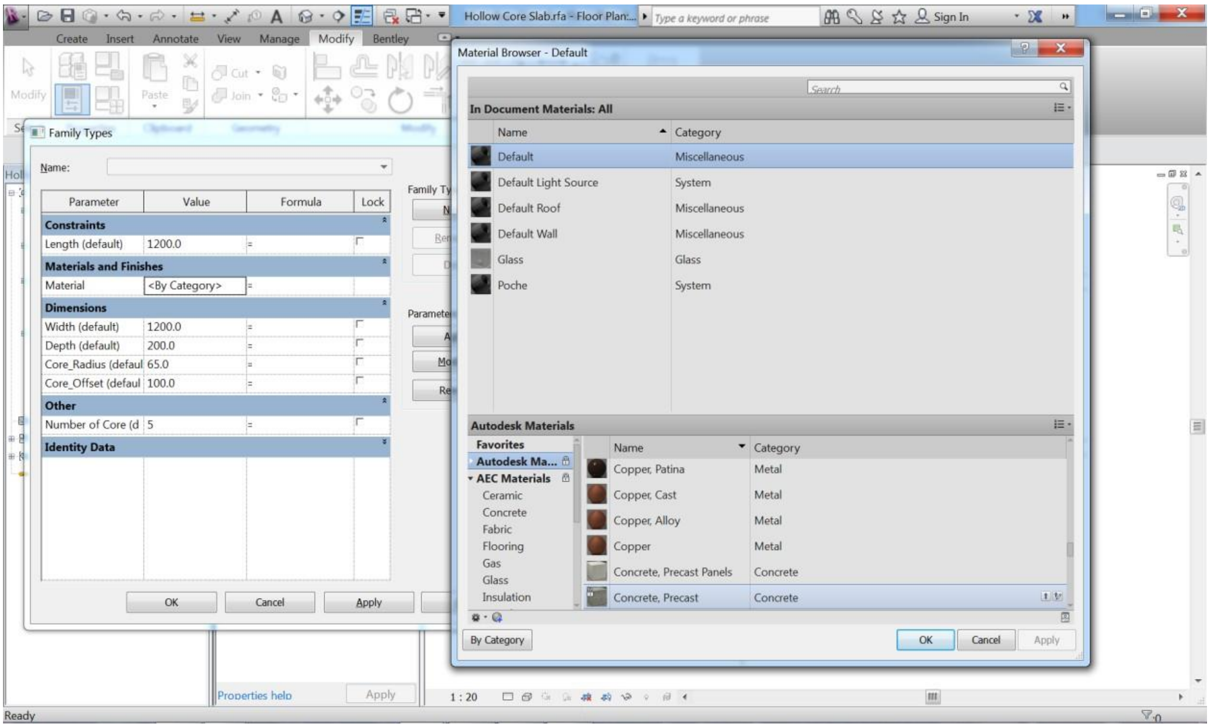


15.8 Material

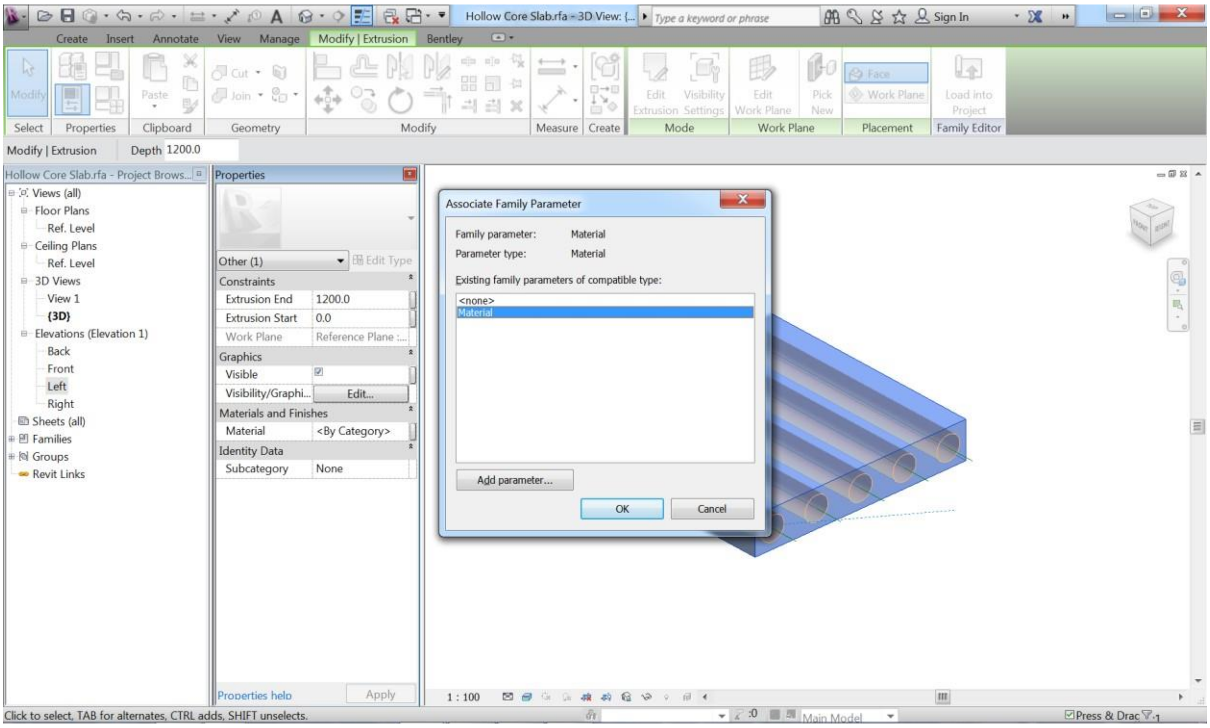
Open the "Family Types" window and click add a parameter called "Material". Set the options to "Common", "Material", "Materials and Finishes", and "Type".



From the Autodesk Materials, find "Concrete, Precast" and add the material to the document.

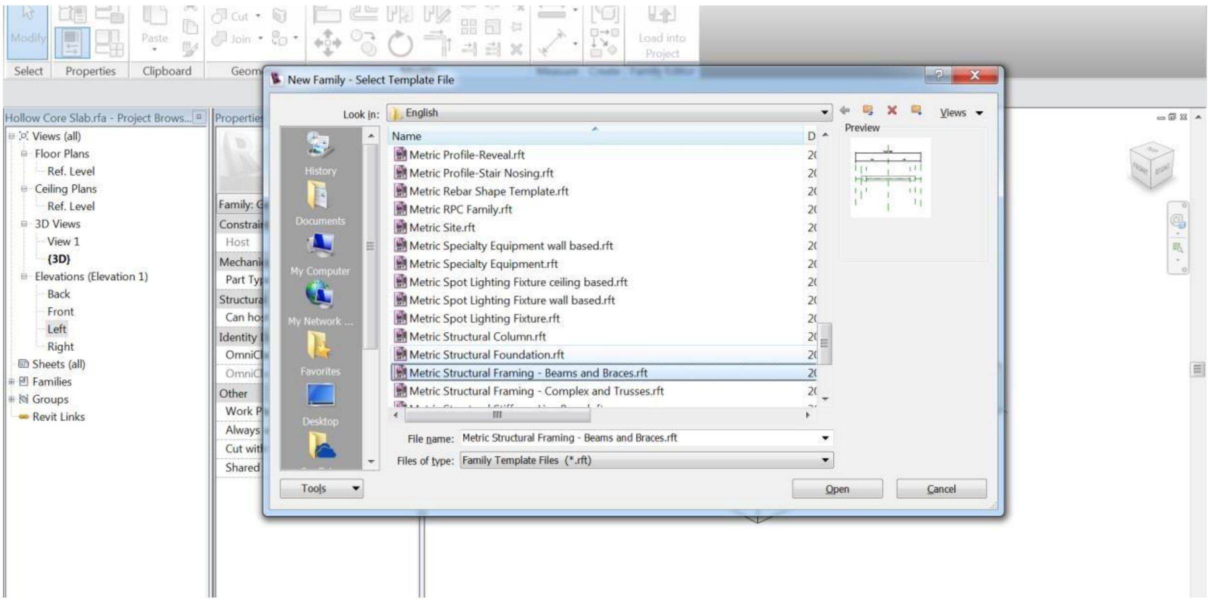


Go to the 3D view and select the extrusion. From its properties, relate "Material" property to "Material" parameter. save your geometry.

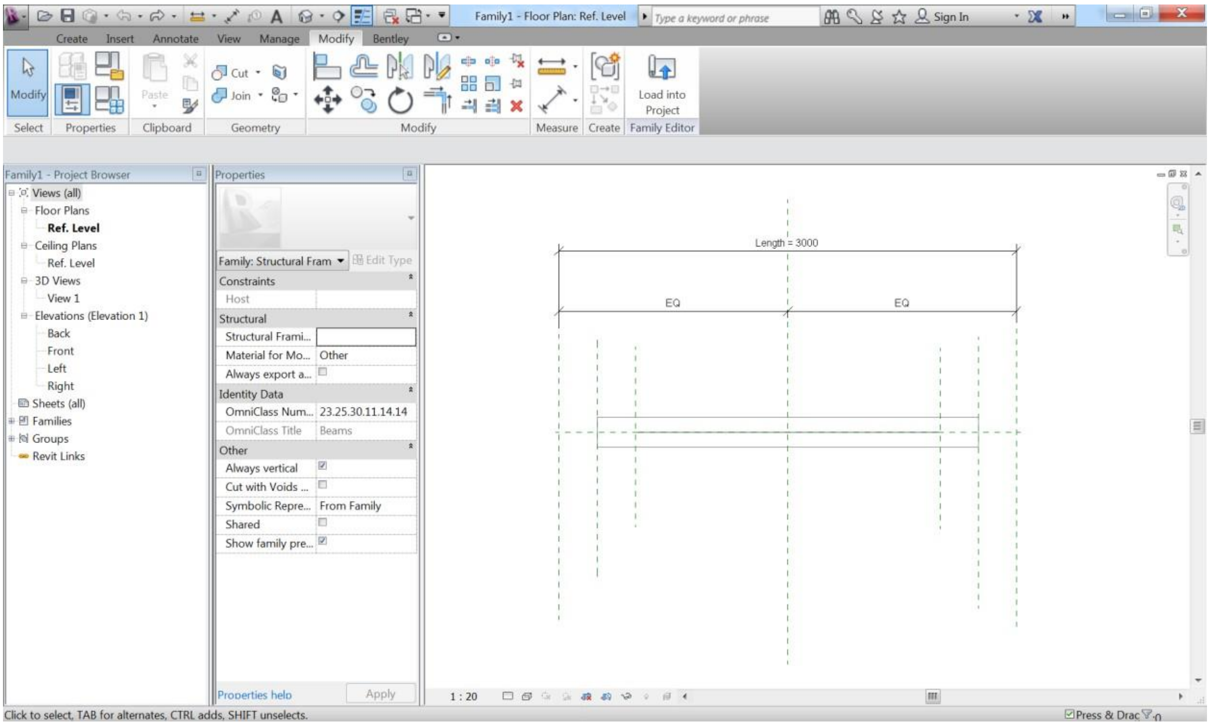


15.9 Nesting

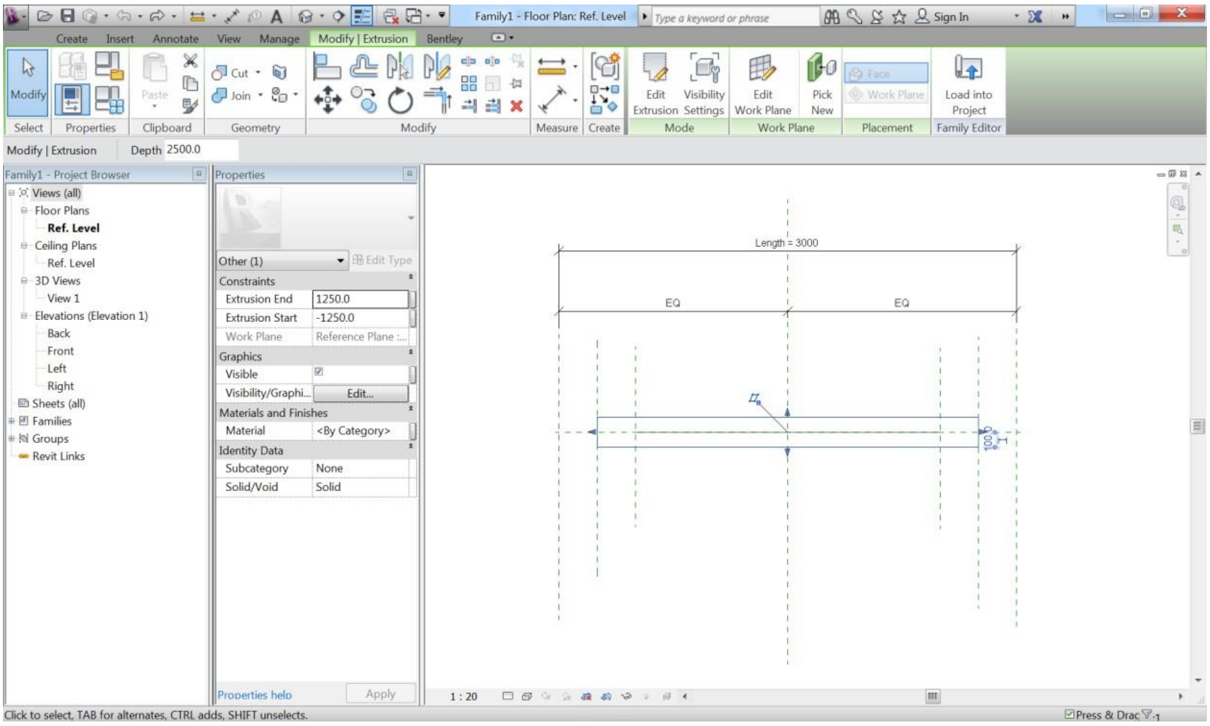
Now you can load what you have done into a project. Or you can nest it into another family to make it work like other structural elements. Start another family editor with "Metric Structural Framing – Beams and Braces". You also can use "Metric Structural Framing – Complex and Trusses" based on your project requirements.



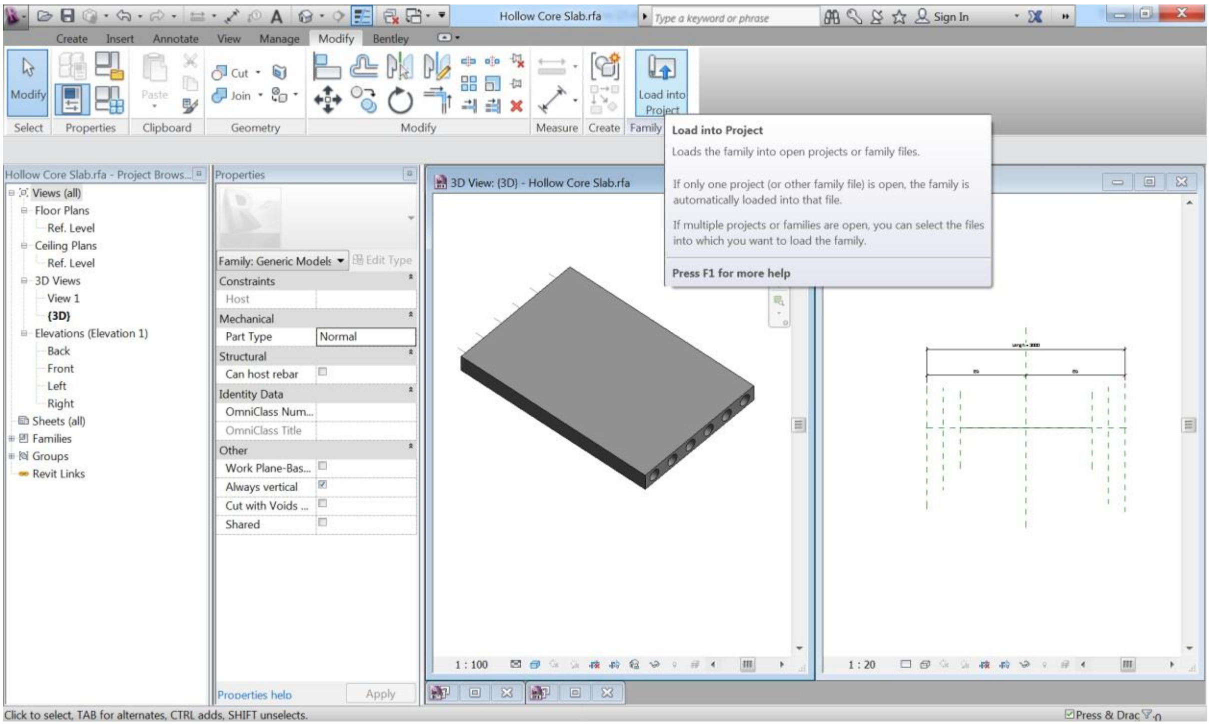
You can see that there are preset geometric objects, an extrusion and a line.



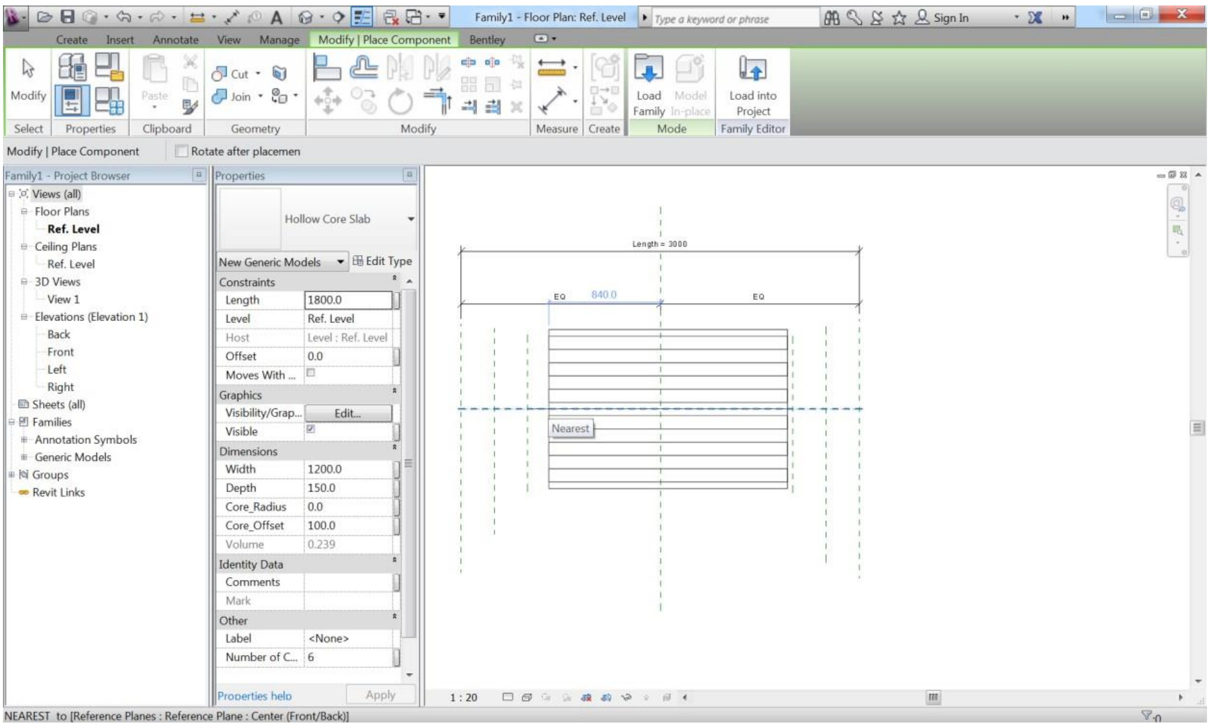
Select the geometric objects and erase them.



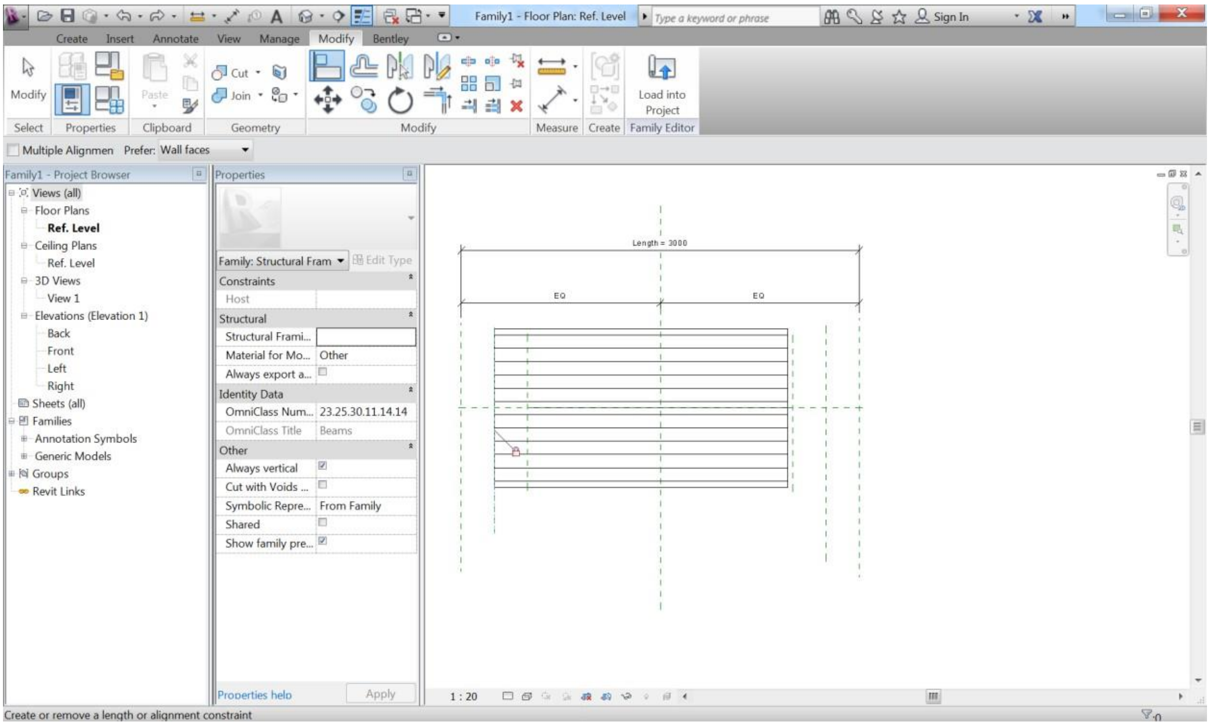
Switch back to the generic mass model and load it into the new family editor.



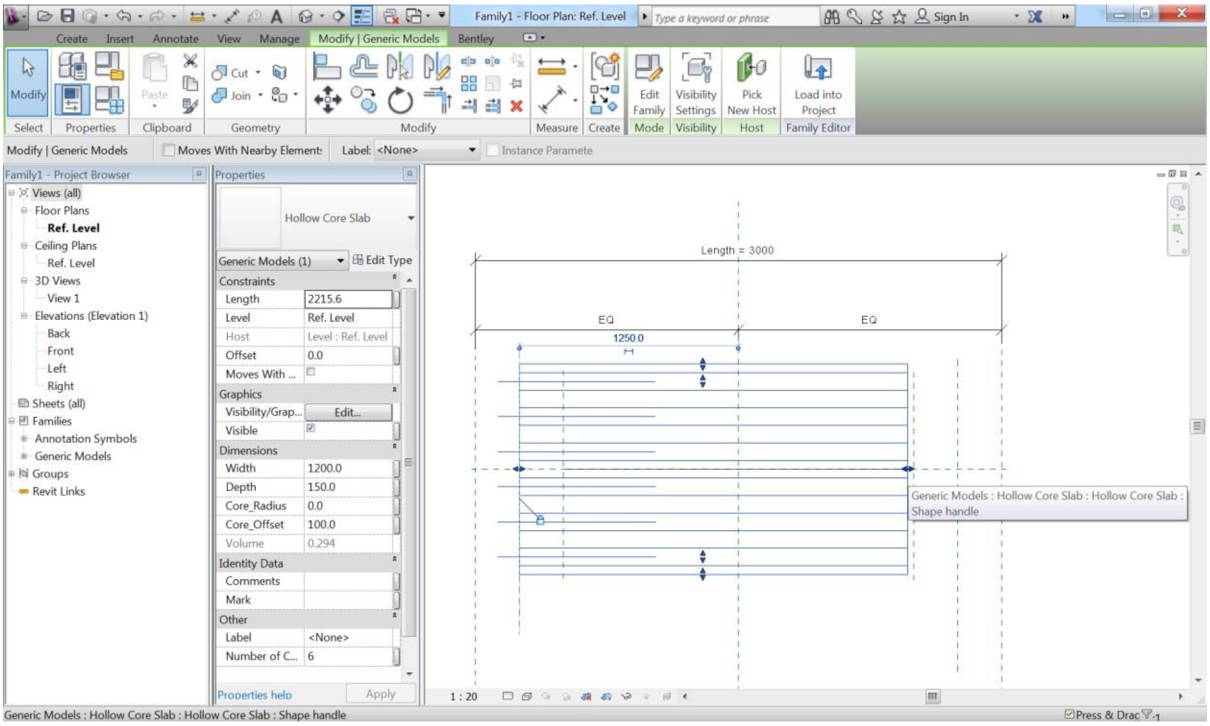
Place it along the horizontal center reference plane. align the center of the imported mass to that of the horizontal center.



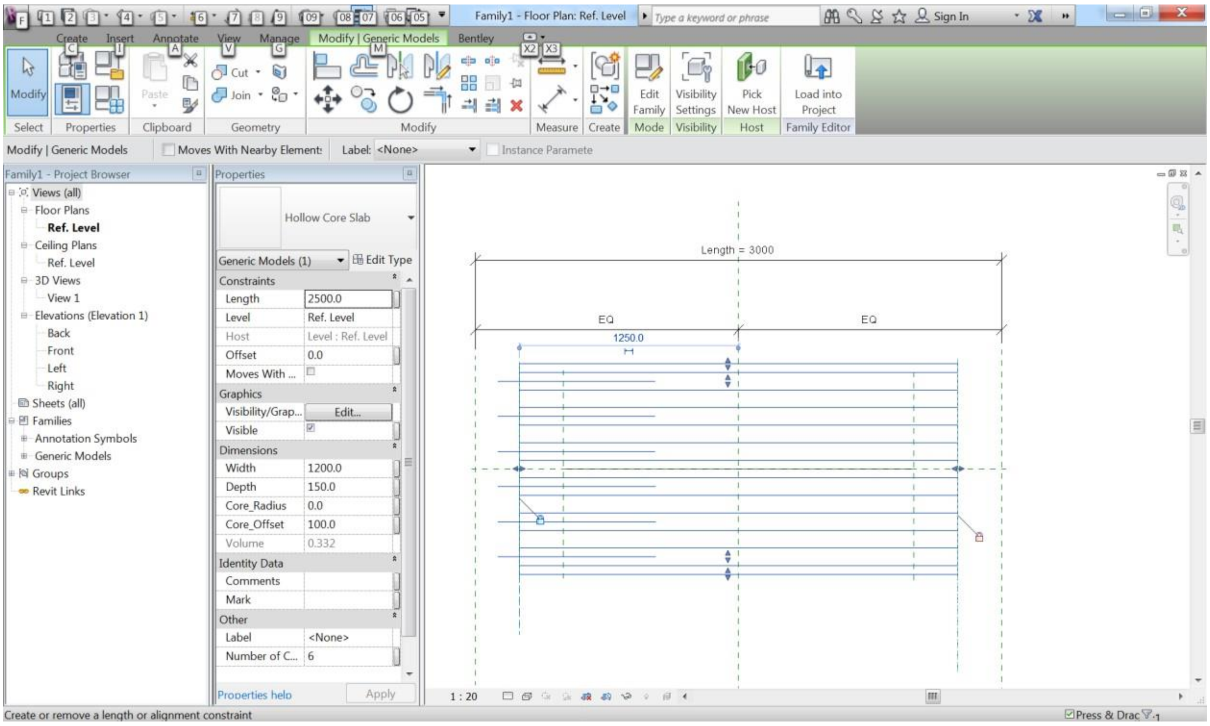
Align the left side the second left reference plane - where the preset extrusion solid was aligned.



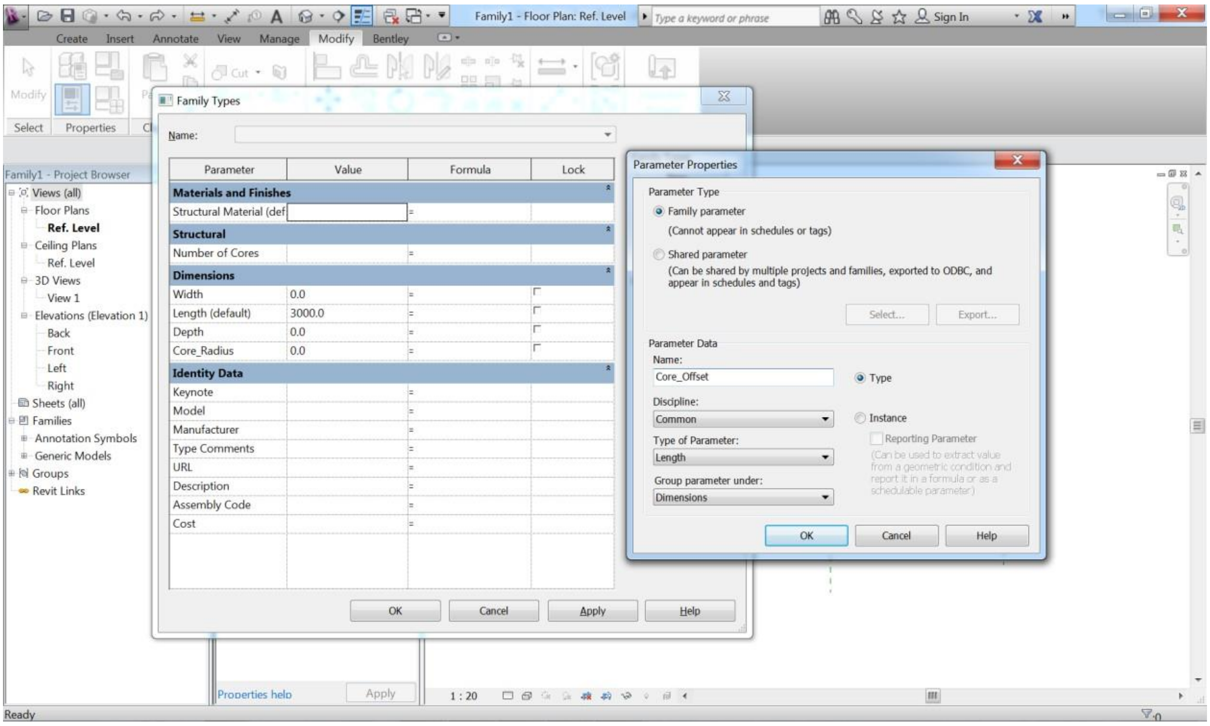
Select the geometric object, then arrow heads will apper on the four sides.



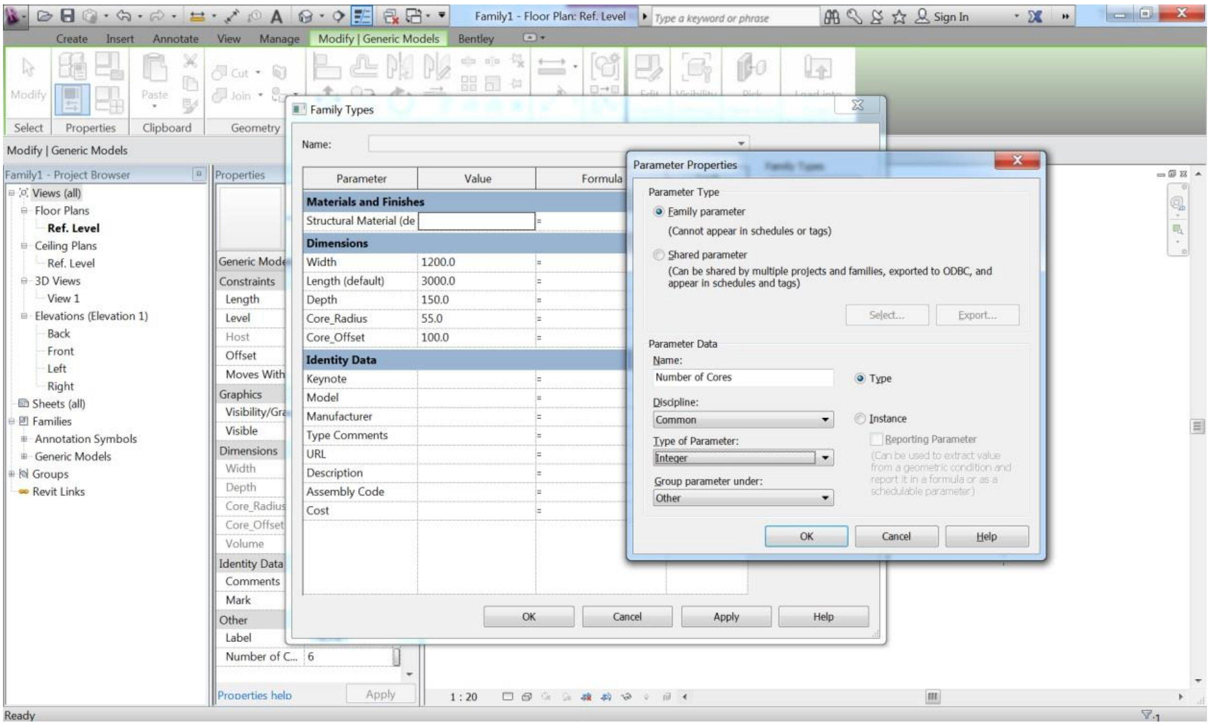
Drag the arrow head on the right side and algin it to the second right reference line.



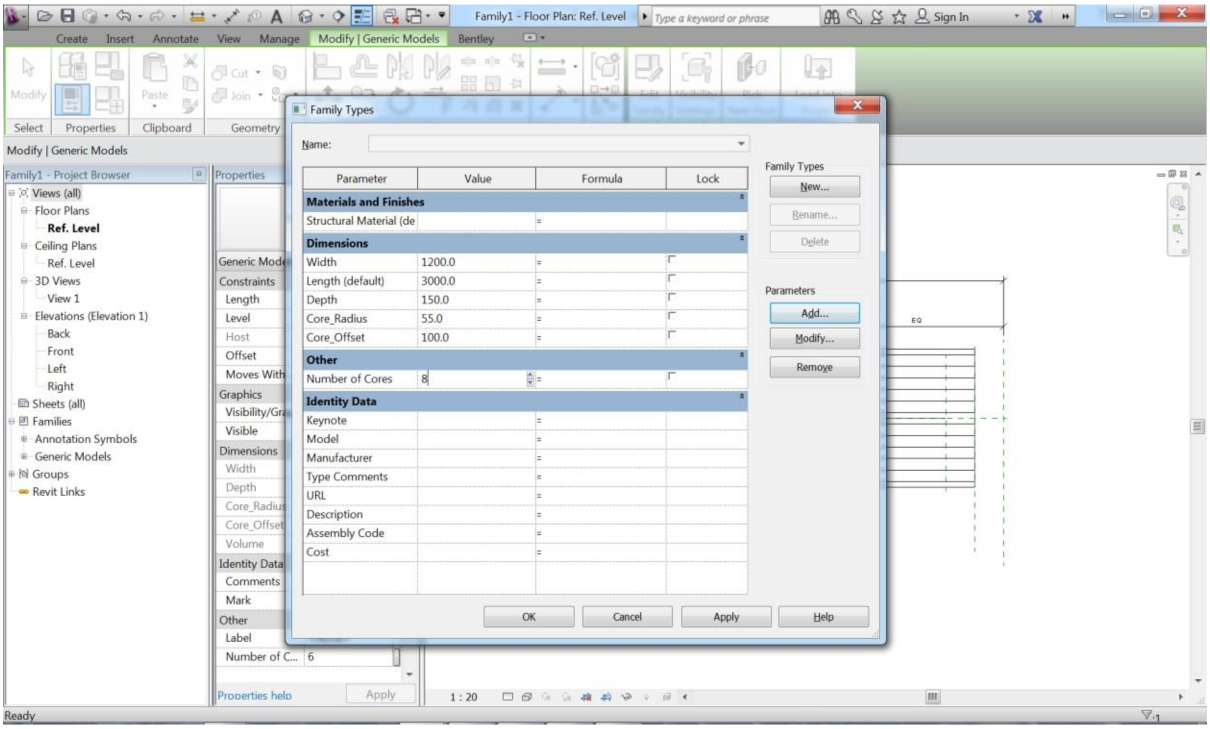
add “Width”, “Depth”, “Core_Radius”, and “Core_Offset”. set the options to “Common”, “Length”, “Dimensions”, and “Type”.



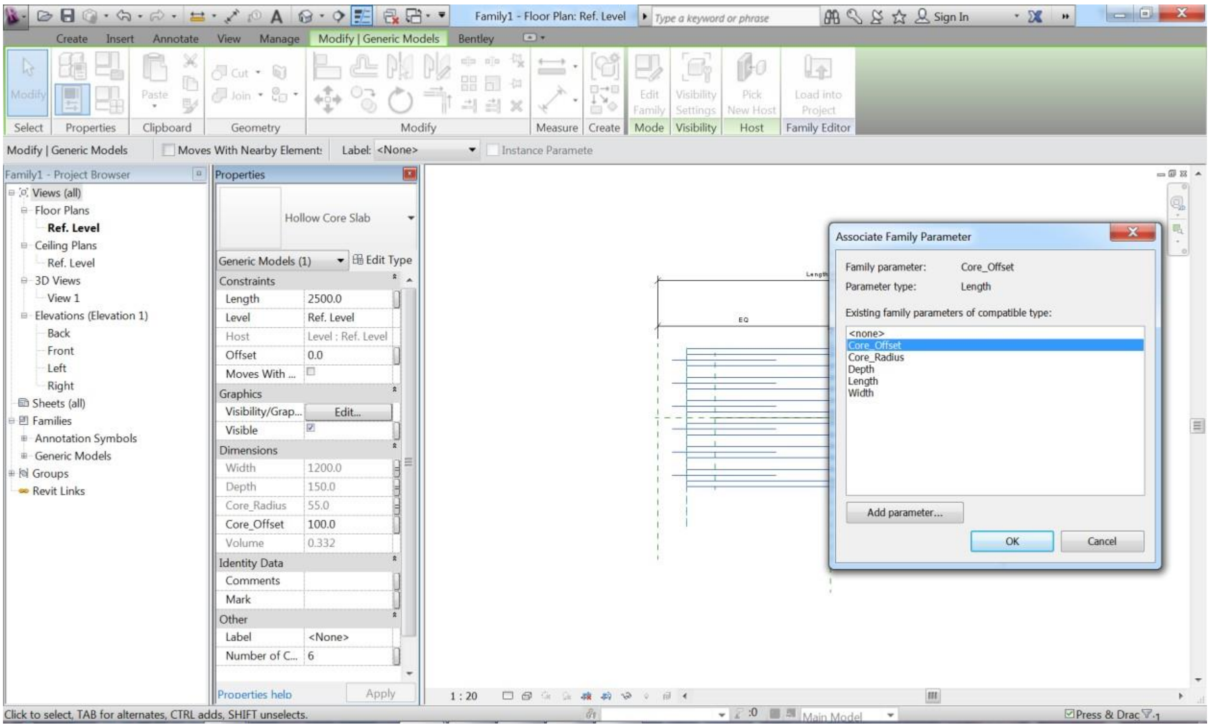
Lastly, add “Number of Cores” parameter” with “Common”, “Integer”, “Other”, and “Type”.



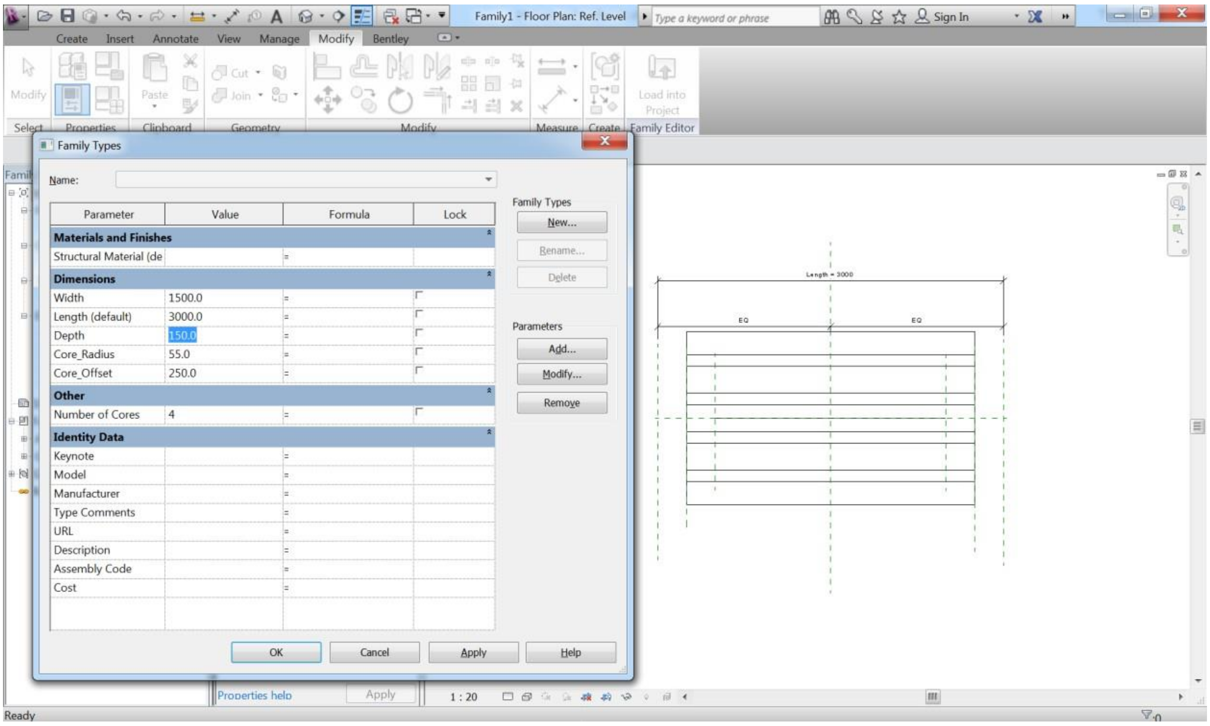
Set the value of parameters as the below image shows.



Then go back to the view and select the loaded mass. From its properties, associate "Width", "Depth", "Core_Radius", "Core_Offset", and "Number of Cores" to the family parameters.

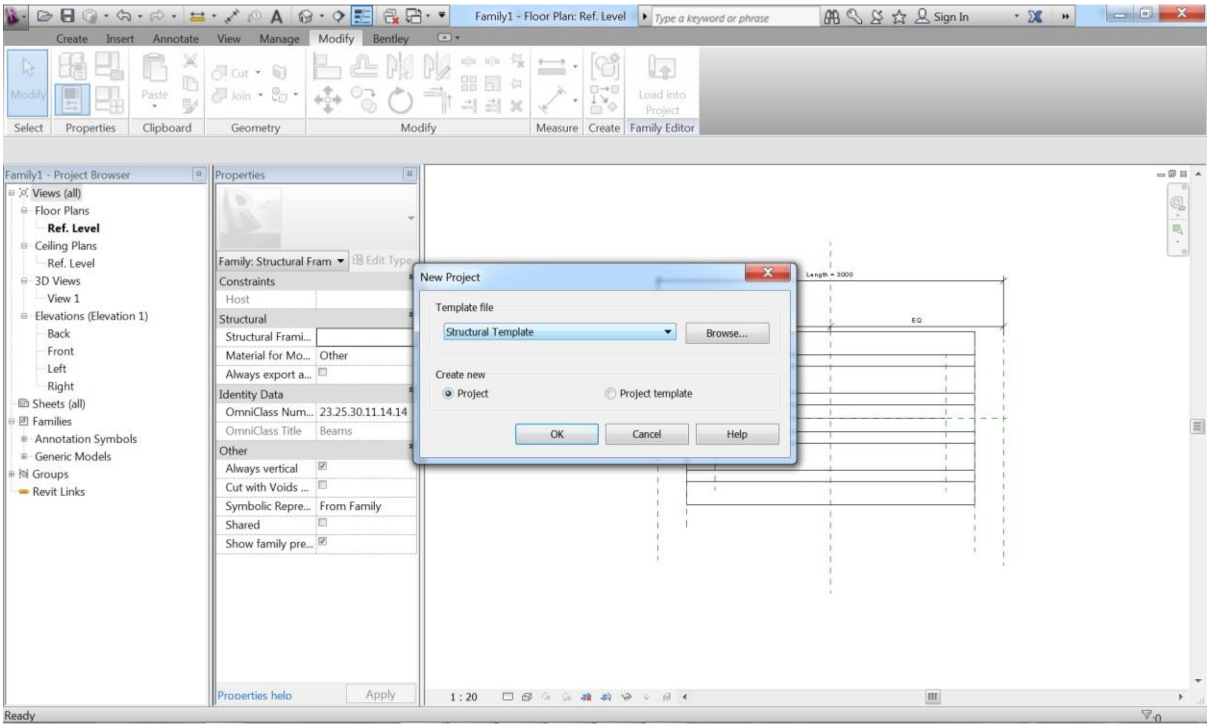


open "Family Types" window and change the values to test the parametric frameworks. You can try to add types by using catalogue. Save the family with the name of "Hollow Core Slab_Nested.rfa".

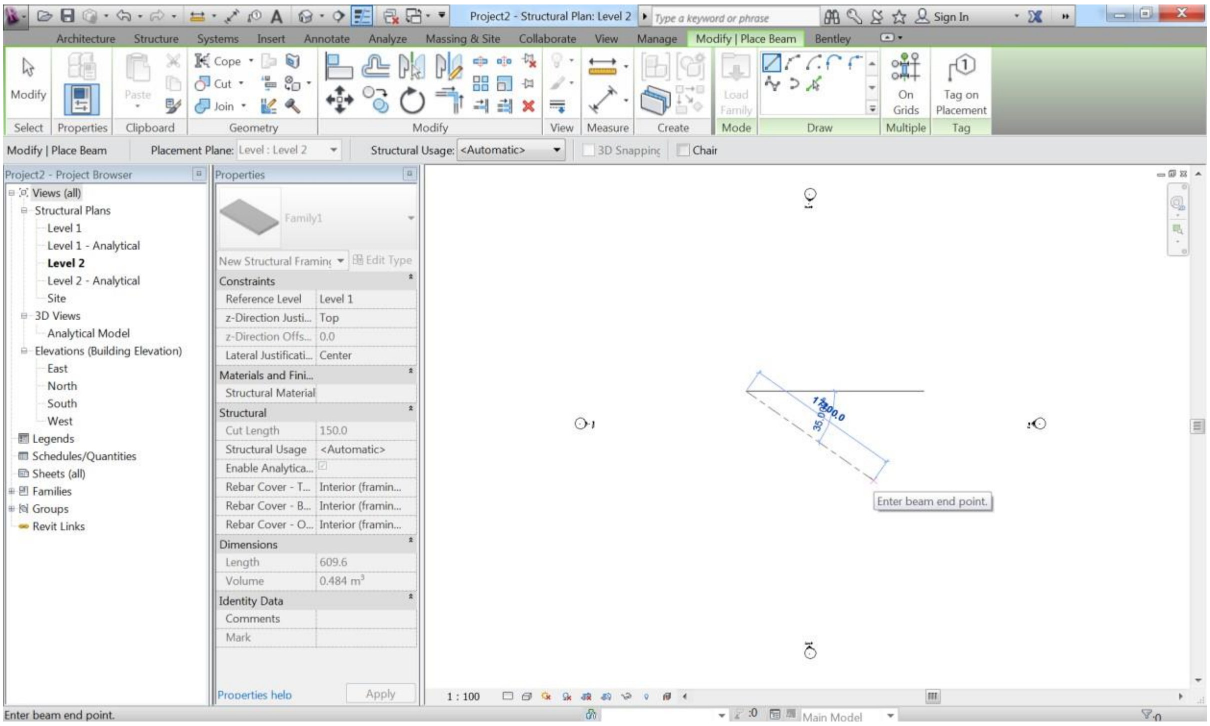


15.10 Load into Project

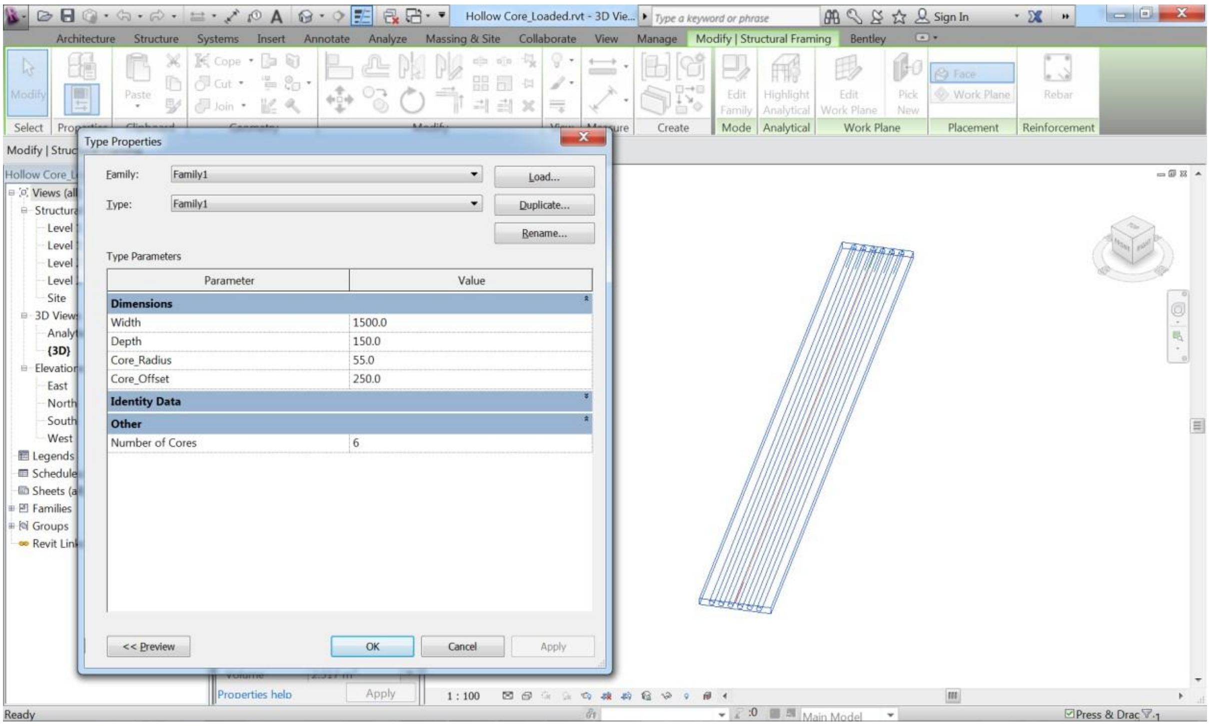
Start a new project with “Structural Template”.



Switch back to the family and load it into the project to draw a family instance in the project.



Select the added object and try to adjust the value of the family parameters.



Disclaimer: The process described in this tutorial is only to help the readers understand the concept of the topic and how to use related functions. This tutorial does not offer the optimized way to implement a project in Revit.

Table of Contents

CHAPTER 16 - STRUCTURAL ANALYSIS IN REVIT

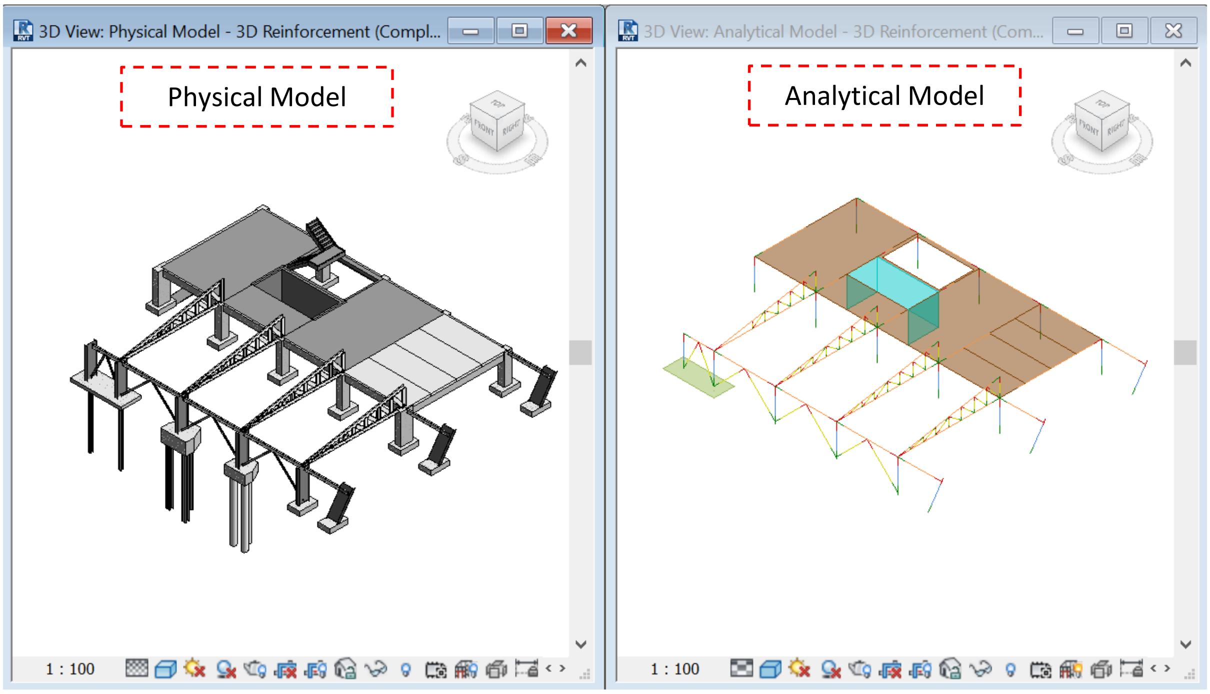
In actual project settings, Revit is used for model development, coordination and documentation. Developed models can then be used for structural analysis. The two most common solutions are "Robot Structural Analysis Professional" (RSA) and cloud based "Revit Structural Analysis Toolkit". In this section, you will learn

- How to prepare structural (analytical) model for analysis.
- Options in structural modelling for analysis.
- Structural analysis using RSA.
- How to view analysis result in Revit.
- The 2-way linkage between Revit and common structural analysis program such as ETABS.

16.1 Structural Analytical Model in Revit

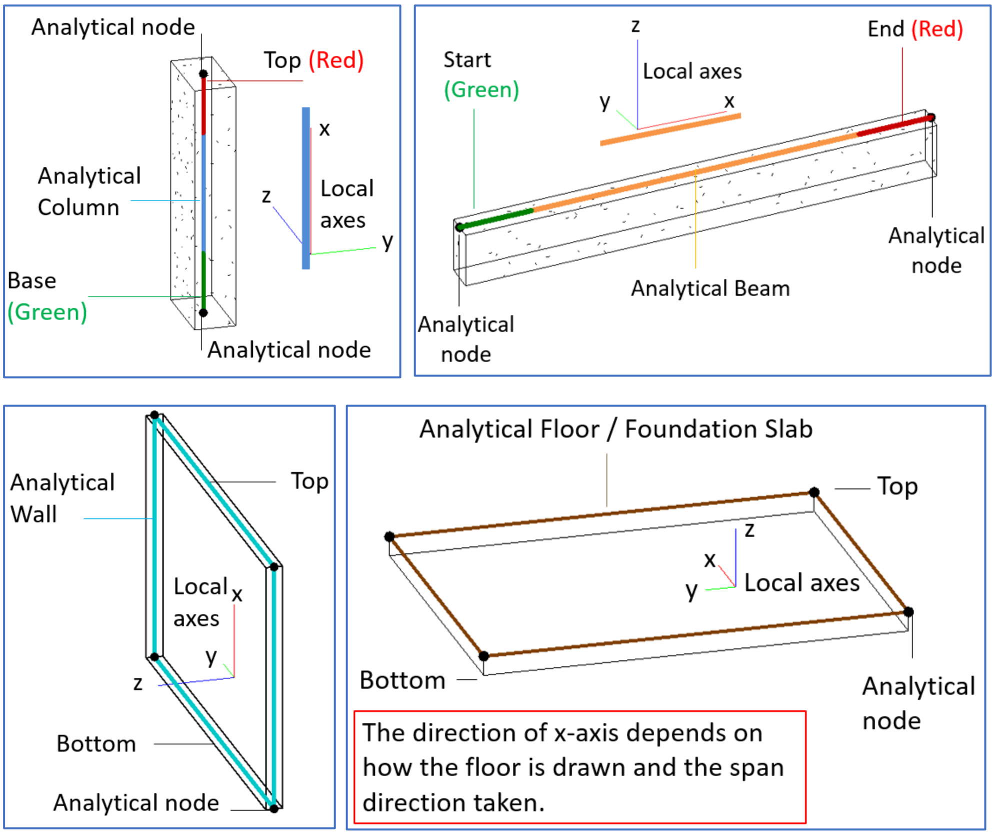
A Structural Analytical Model comprised of a set of structural member analytical models. The following structural elements have structural member analytical models: Structural Columns, Structural Framing elements (such as beams and braces), Structural Floors, Structural Walls, and Structural Foundation elements.

Analytical model is created automatically when you create the physical model and can be exported to analysis and design applications.



16.1.1 Structural Member Analytical Models

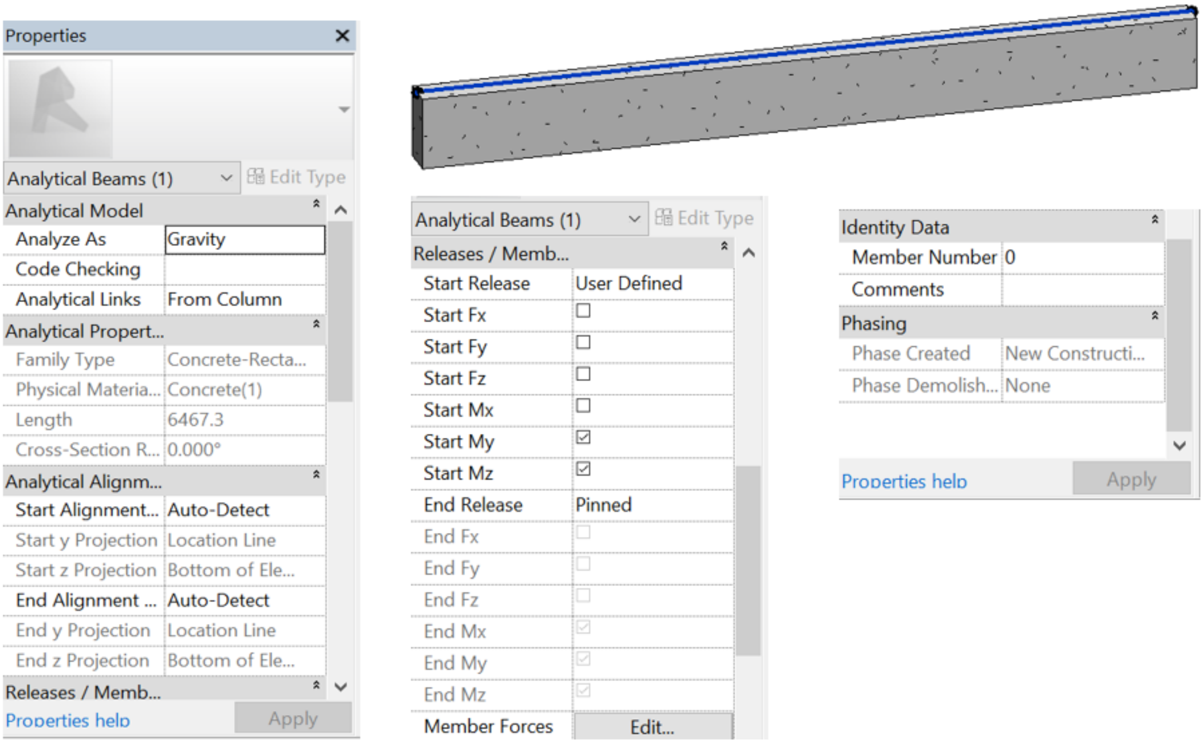
Structural member analytical models are represented by lines joining their analytical nodes. Apart from analytical properties, they are characterized by their local axes. The following figures depicted how the member analytical models' local axes are defined.



The analytical model of any one structural element has:

- Instance parameters
- Physical material properties
- Default position relative to the structural member itself
- Location with respect to a projection plane, either as placed or as adjusted

The properties of an analytical beam are listed below as an example. You can go to Autodesk Knowledge Network for more details of other structural member analytical models.



(source: Autodesk Knowledge Network)

https://knowledge.autodesk.com/support/revit-products/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/Revit-Model/files/GUID-0AF3EAB9-D369-4C07-BAAD-CB1EE2DEC7B2-htm.html

Name	**Description**
Analytical Model	
Analyze as	Used by external analysis applications to determine whether beam conditions contribute to lateral analysis.
Analytical Links	Either Yes, No, or From Column. A frame element that can be set to fully rigid or released in global directions. When Analytical Links are enabled, an additional analytical segment is engaged in the model between the end of the analytical model of a beam and the analytical model of a column. From Column defines the analytical link of the beam to that of its connected column. See Analytical Link.
Approximate Curve	Applies to curved beams only. Select to create segment based on both the values for Maximum discretized offset and Use hard-points. See Segmented Analytical Model Parameters.
Maximum discretized offset	Applies to curved beams only. Limits the distance between a smooth curve and a line segment when the Approximate curve parameter is selected. See Segmented Analytical Model Parameters.

Name	Description
Use hard-points	Applies to curved beams only. Forces the segmented analytical model to have nodal points at the ends of the members attached to the curved beam when the Approximate curve parameter is selected. See Segmented Analytical Model Parameters.
Analytical Properties	
Family Type	The family type of the element.
Physical Material Asset	The name of a physical asset assigned to the beam material (read-only).
Length	The length of the analytical model.
Cross-Section Rotation	Displays the cross-section rotation of analytical members. The angle of rotation is measured from the work plane of the member and the direction of the center reference plane.
Analytical Alignment	
Start Alignment Method	Either Auto-detect or Projection. Specifies if the horizontal and vertical references of the beam start position are automatically justified or have a defined projection.
Start y Projection	Either Location Line, Left of Element, Center of Element, Right of Element, <Grids...>, or <Reference Planes...>. Available when Start Alignment Method is specified as Projection. Specifies the horizontal location of the analytical model at the start of the beam.
Start z Projection	Either Location Line, Top of Element, Center of Element, Bottom of Element, <Level...>, <Reference Level...>, or <Reference Planes...>. Available when Start Alignment Method is specified as Projection. Specifies the vertical location of the analytical model at the start of the beam.
End Alignment Method	Either Auto-detect or Projection. Specifies if the horizontal and vertical references of the beam end position are automatically justified or have a defined projection.
End y Projection	Either Location Line, Left of Element, Center of Element, Right of Element, <Grids...>, or <Reference Planes...>. Available when End Alignment Method is specified as Projection.

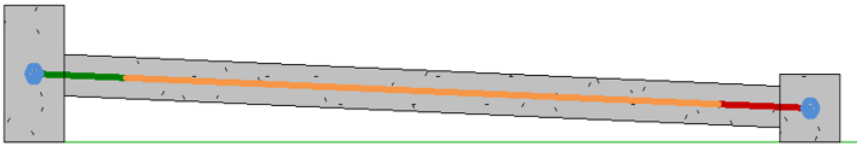
Name	Description
	Specifies the horizontal location of the analytical model at the end of the beam.
End z Projection	Either Location Line, Top of Element, Center of Element, Bottom of Element, <Level...>, <Reference Level...>, or <Reference Planes...>. Available when End Alignment Method is specified as Projection. Specifies the vertical location of the analytical model at the end of the beam.
Structural Analysis (See About the Structural Analytical Model.)	
Start Release	Specifies the end release condition: either Fixed, Pinned, Bending Moment or User Defined. User Defined allows you to enable/disable each of the end release conditions.
Start Fx	The translational release at the start end of the brace along the local x-axis. When selected the degree of freedom is released and the force is not available.
Start Fy	The translational release at the start end of the brace along the local y-axis. When selected the degree of freedom is released and the force is not available.
Start Fz	The translational release at the start end of the brace along the local z-axis. When selected the degree of freedom is released and the force is not available.
Start Mx	The rotational release at the start end of the brace along the local x-axis. When selected the degree of freedom is released and the moment is not available.
Start My	The rotational release at the start end of the brace along the local y-axis. When selected the degree of freedom is released and the moment is not available.
Start Mz	The rotational release at the start end of the brace along the local z-axis. When selected the degree of freedom is released and the moment is not available.
End Release	Specifies the end release condition: either Fixed, Pinned, Bending Moment or User Defined. User Defined allows you to enable/disable each of the end release conditions.
End Fx	The translational release at the end of the brace along the local x-axis. When selected the degree of freedom is released and the force is not available.
End Fy	The translational release at the end of the brace along the local y-axis. When selected the degree of freedom is released and the force is not available.
End Fz	The translational release at the end of the brace along the local z-axis. When selected the degree of freedom is released and the force is not available.
End Mx	The rotational release at the end of the brace along the local x-axis. When selected the degree of freedom is released and the moment is not available.

Name	Description
End My	The rotational release at the end of the brace along the local y-axis. When selected the degree of freedom is released and the moment is not available.
End Mz	The rotational release at the end of the brace along the local z-axis. When selected the degree of freedom is released and the moment is not available.
Member Forces	Specifies the internal forces at the start and end of the analytical beams and braces. This data communicates load information for both documentation and connection fabrication. Click Edit to enter user-defined force and moment values.
Identity Data	
Member Number	An identifier created for the analytical member. This value should be unique across Analytical Beams, Analytical Braces and Analytical Columns in a project. Revit warns you if the number is already used but allows you to continue using it. You can see the warning using the Review Warnings tool. See Review Warning Messages.
Comments	User comments.
Phasing	
Phase Created	Indicates in which phase the beam was created. See Project Phasing.
Phase Demolished	Indicates in which phase the beam was demolished. See Project Phasing.

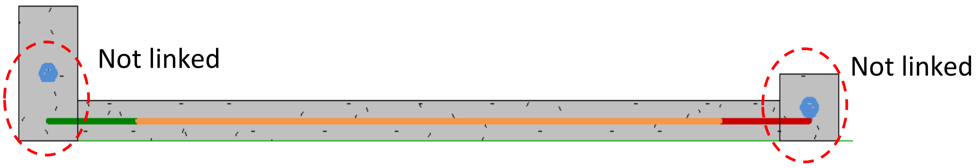
16.1.2 Analytical Links

When a beam is not directly support at the centre of a column, an analytical link is needed to correctly model the effect of such eccentricity for structural analysis. Note that an analytical beam starts and ends where you picked the physical beam.

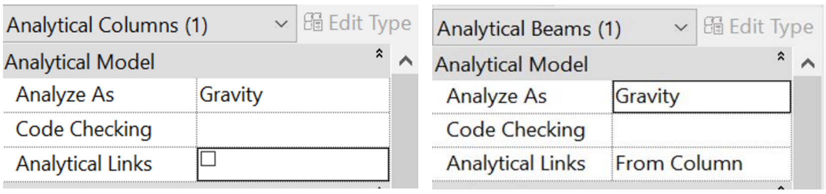
Usually you will pick the centre of the supporting columns and the analytical beam will follow.



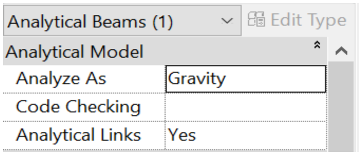
However, if the beam is an edge beam then it should be shifted to align with the edge.

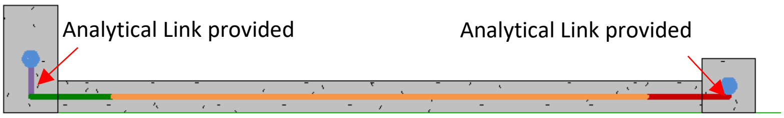


When the beam is not supported directly at the centre of the columns, there is no analytical links provided. This is because, by default, beams' analytical links are from columns, but analytical links are not enabled in analytical columns.



If you want Revit to provide analytical links for the beam at its ends, you should set the analytical links option to “Yes” for the analytical beam.

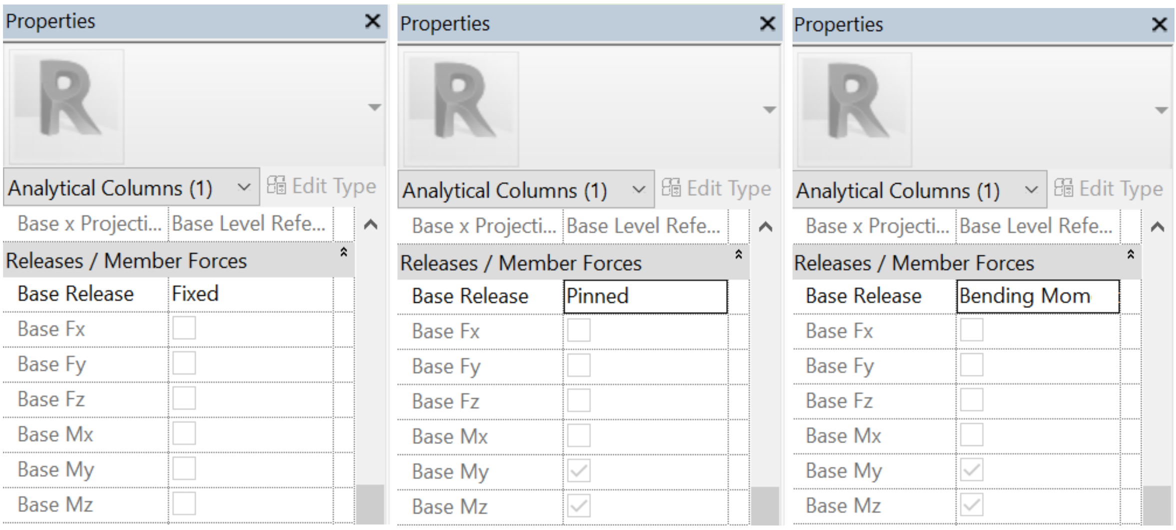




However, if you only want Revit to provide analytical link at one end of the beam, you should set the analytical links option to “From Column” for the analytical beam and enable the analytical link option in that supporting analytical column.

16.1.3 Member End Releases

There are 6 degrees of freedom (DOF) in each node of a member analytical model. These degrees of freedom are called “Releases” in Revit. You can set the releases as predefined options (Fixed / Pinned / Bending Moment) or “User Defined” that you can release any of the DOF. A check in the DOF, Base My for example, releases the rotation degree of freedom about (local) member y-axis and there will be no moment My.



Note that you should check and set the member releases to ensure it is analytically correct for structural analysis. For example, Revit does not automatically set the member releases at cantilever end to “Free” (i.e. all 6 DOFs released)!

16.1.4 Boundary Conditions

You must impose/add Boundary Conditions to support your structure or it will be unstable for analysis. Boundary conditions are usually added at the base of columns and walls.

Notes:

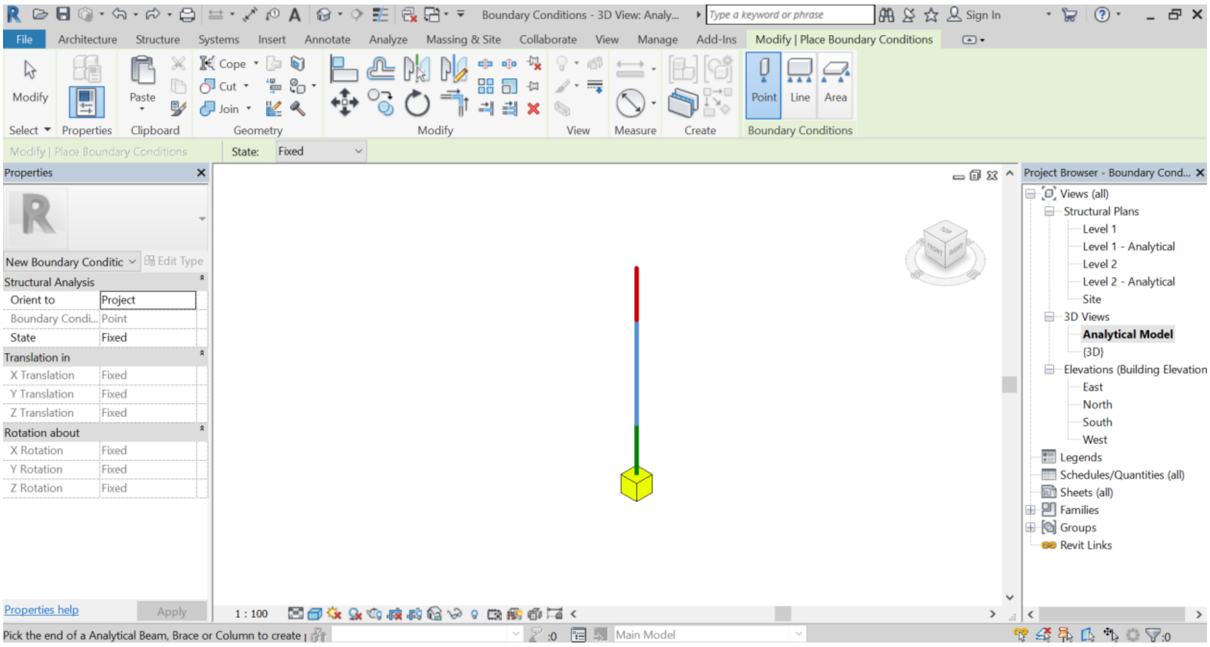
- "Fixed" boundary condition is automatically set for Isolated Foundation. You do not need to add a boundary condition to it.
- For Foundation Slabs, you will need to add boundary conditions or it will be free to move.

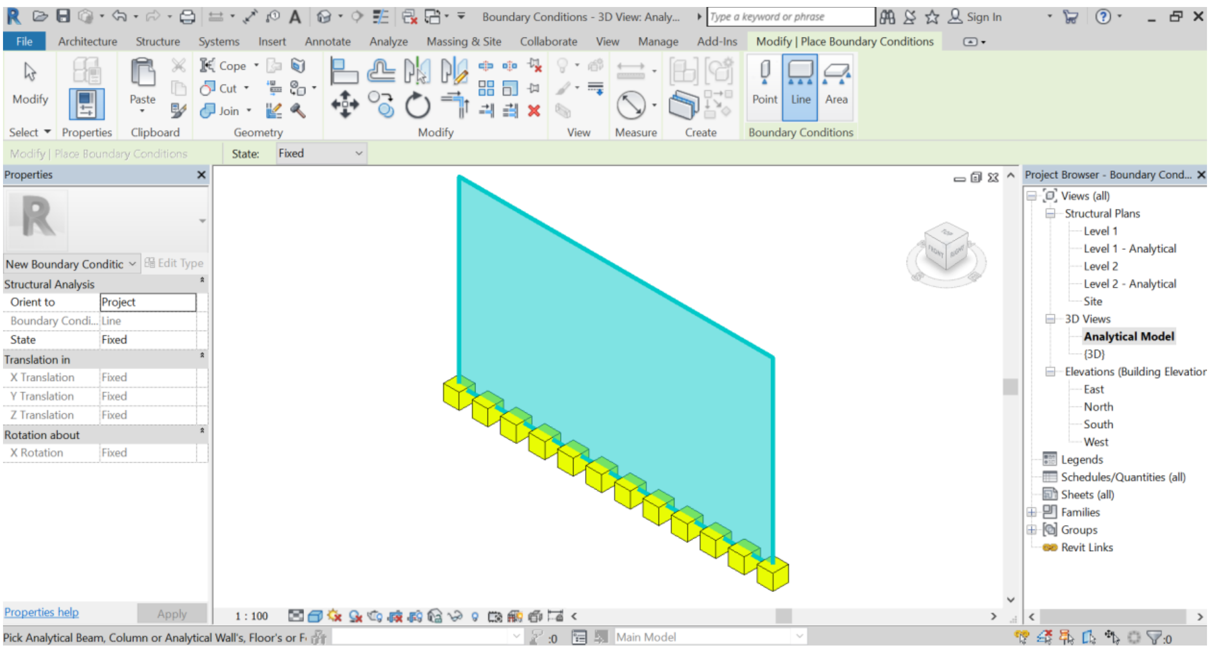
To add boundary conditions,

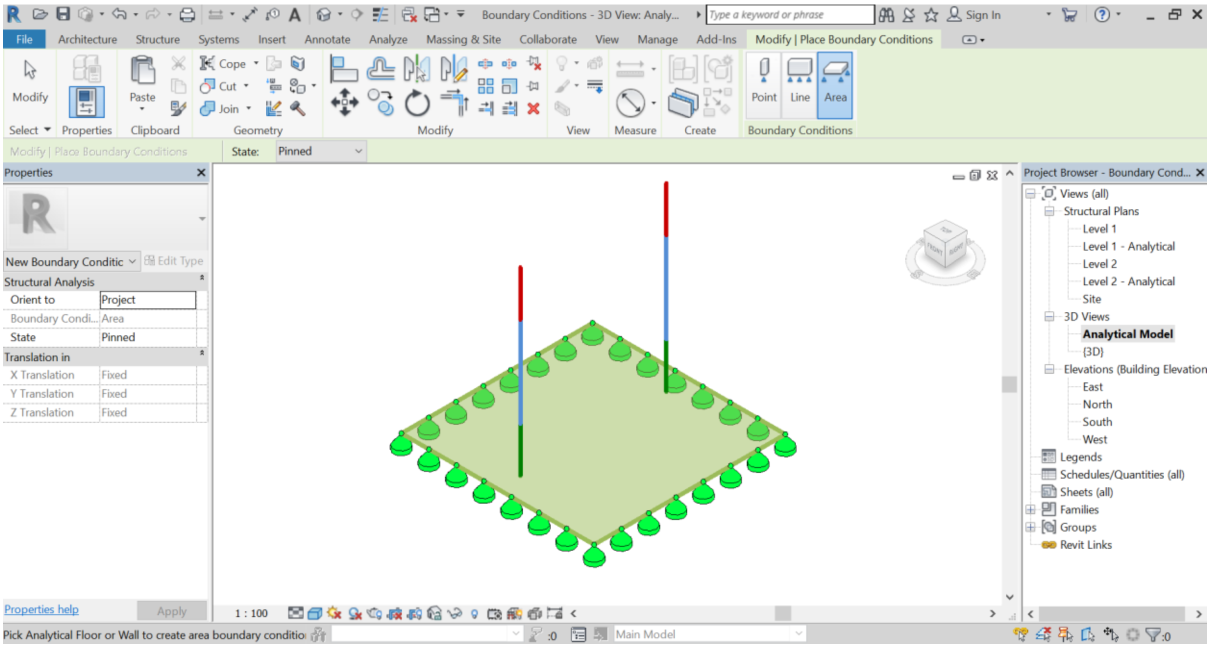
1 Click Analyze tab ➢ Analytical Model panel ➢ Boundary Conditions.

2 Select "Point" / "Line" / "Area" Boundary Conditions.

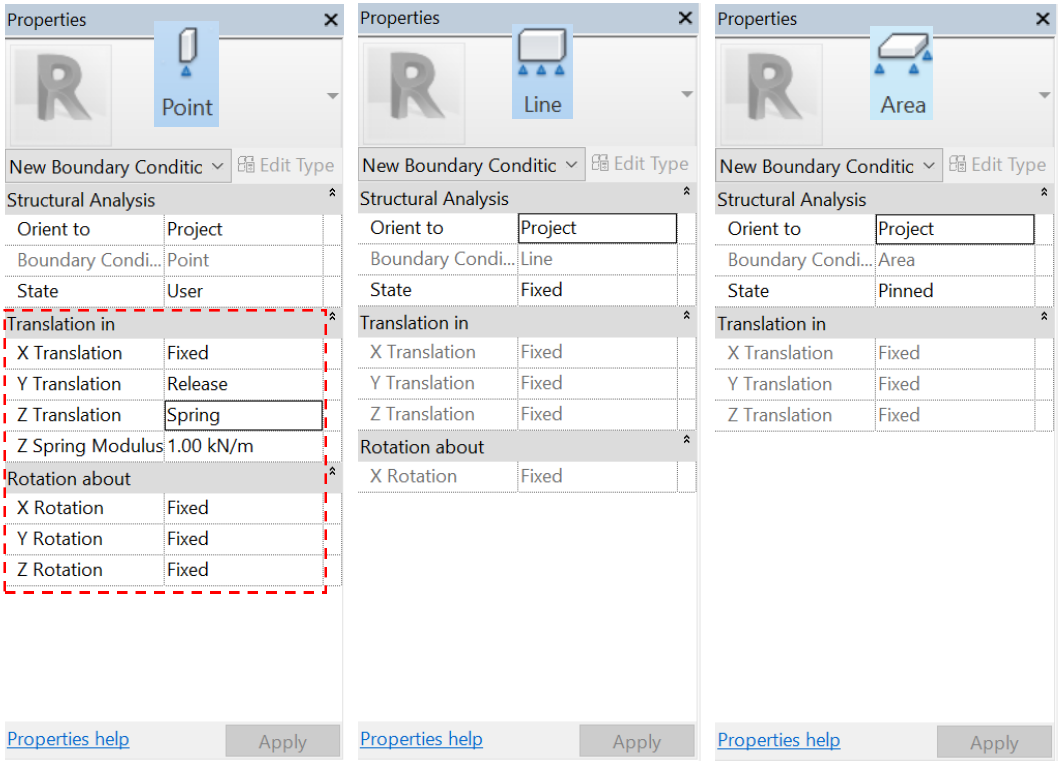
3 Click at the base of a column / wall / foundation slab to place the Point / Line / Area boundary conditions.

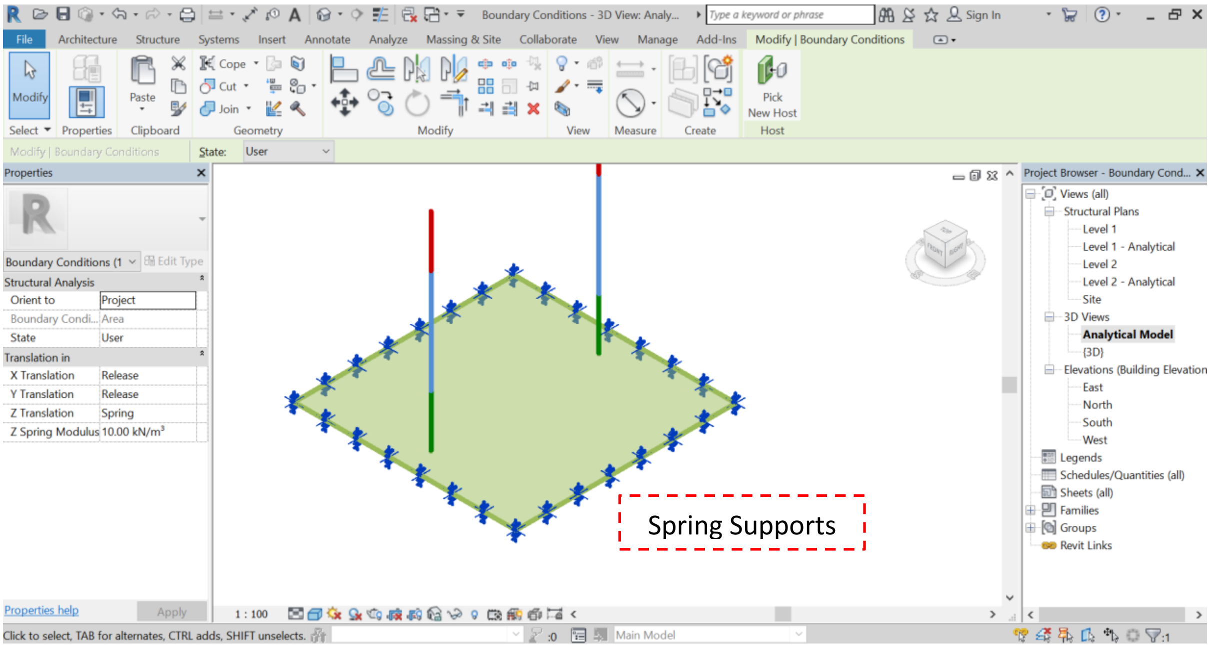






The number of DOFs is different for Point, Line and Area boundary conditions. For Point boundary conditions, all 6 DOFs can be set. Each DOF has three options: Fixed / Release / Spring (with user input spring modulus value).

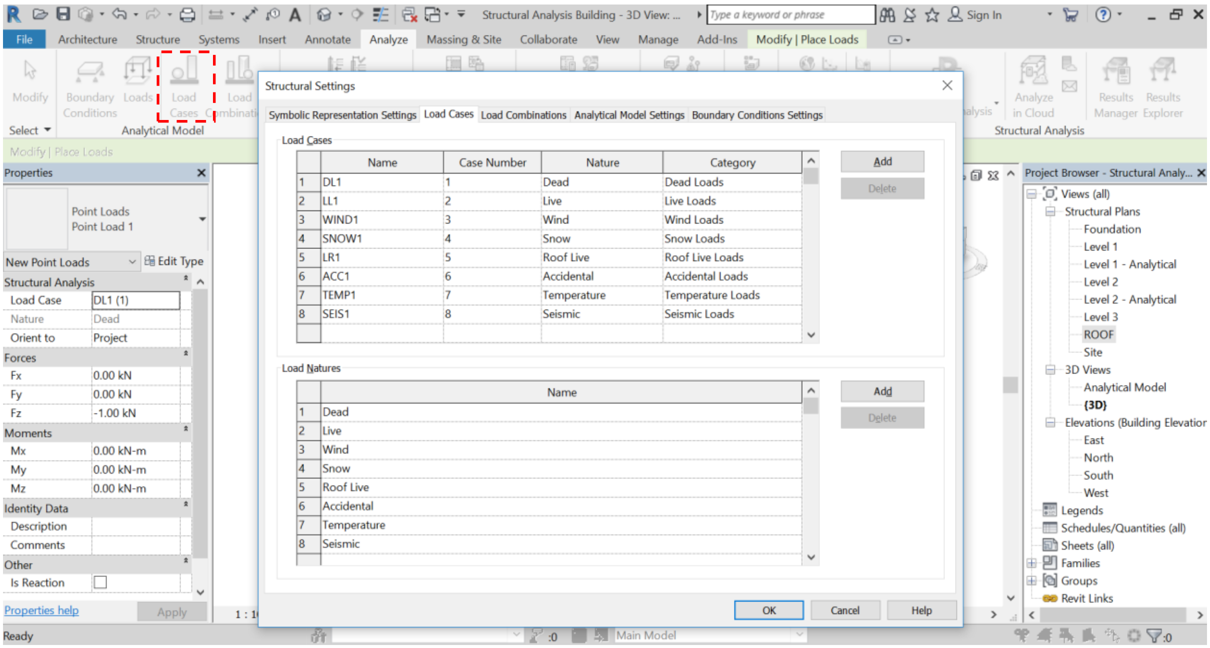




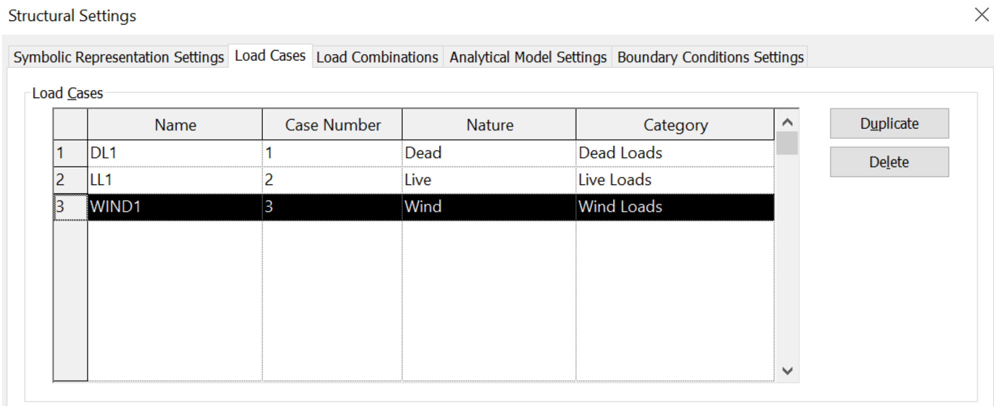
16.1.5 Load Cases

Open from the course folder the "Structural Analysis Building.rvt" file.

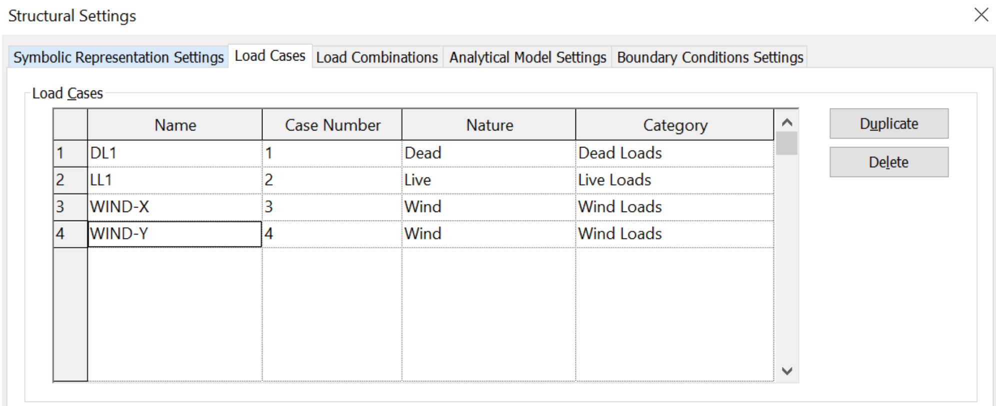
To set Load Cases, click Analyze tab ➢ Analytical Model panel ➢ Load Cases



There are 8 load cases automatically set in Revit if you use its Structural Template. Some load cases are not commonly used in Hong Kong and you can select and delete them. For example, SNOW1, LR1, ACC1, TEMP1 and SEIS1 can be deleted.



You can add a load case by duplicating an existing one and rename it. Usually you will need more than one wind case. Add WIND-Y by duplicating WIND1. Also rename WIND1 to WIND-X.

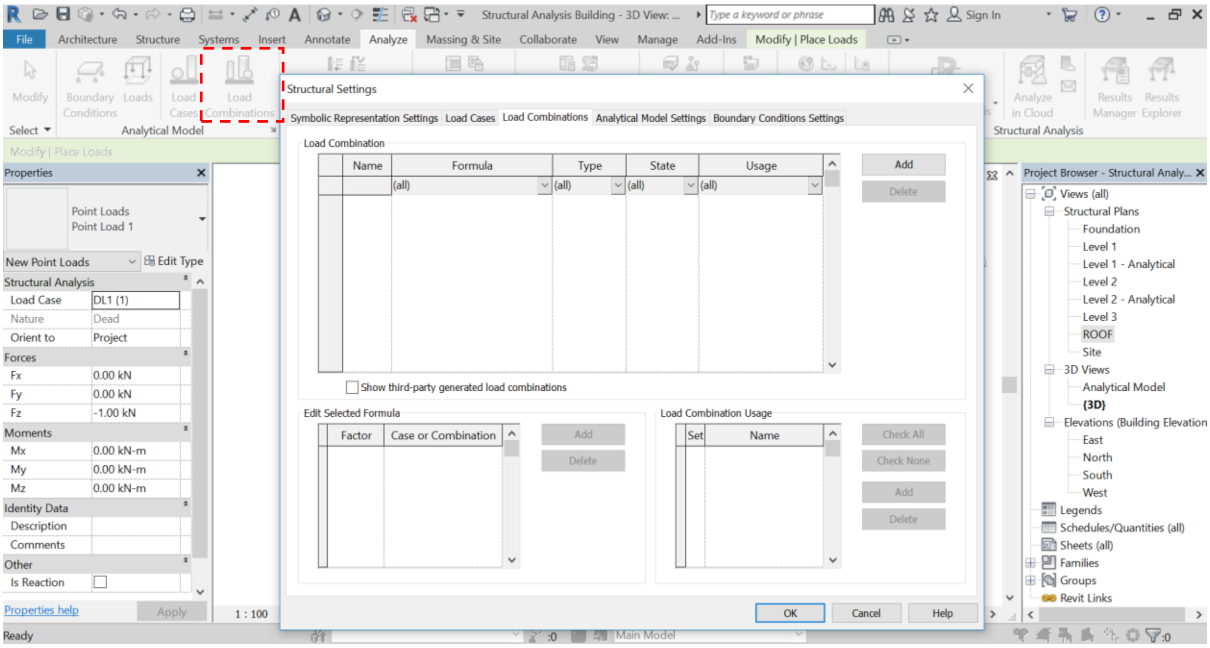


With all the load cases created, you can next set the load combinations for your structural analysis.

16.1.6 Load Combinations

Continue with the "Structural Analysis Building.rvt" model and set Load Combinations.

Click Analyze tab ➢ Analytical Model panel ➢ Load Combinations

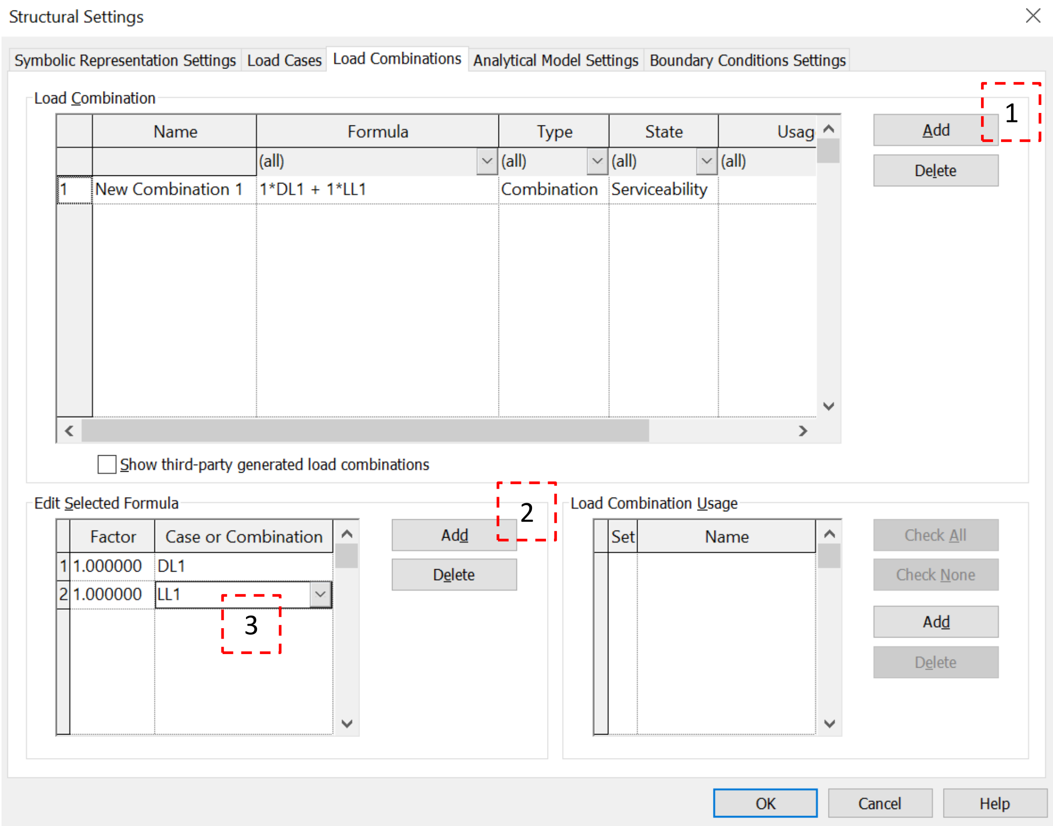


To add a load combination,

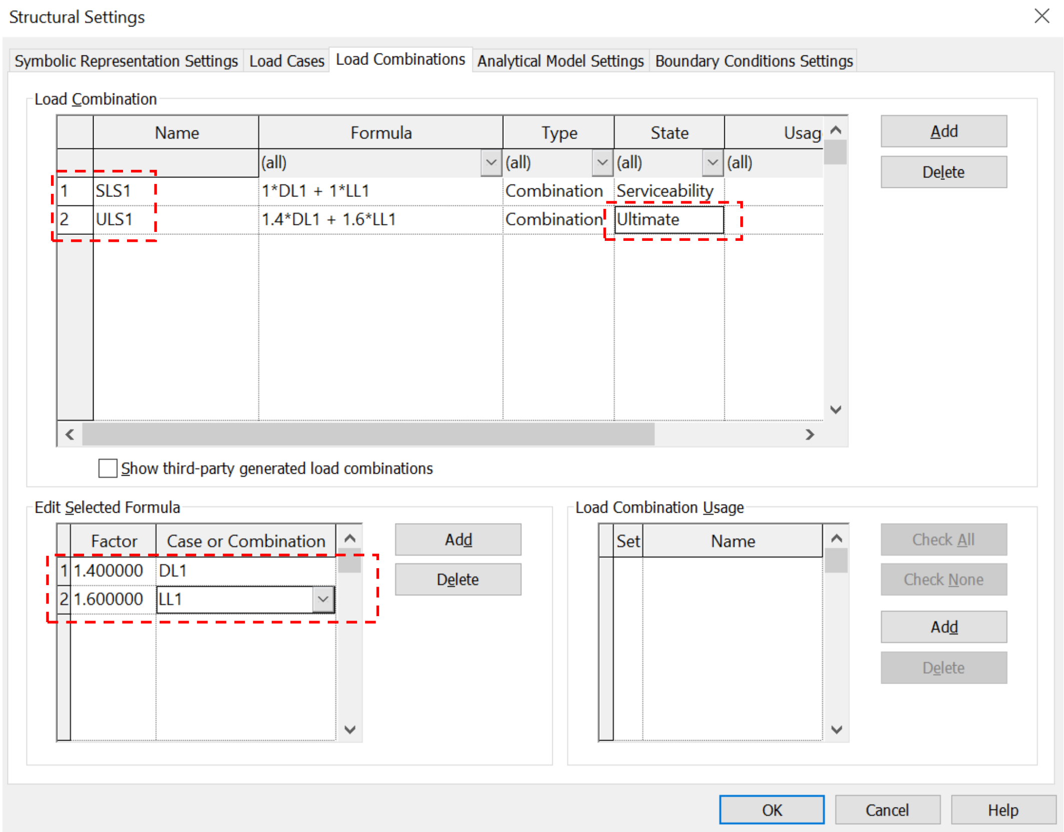
1 Click Add.

2 Click Add in Edit_Selected Formula to add load cases for the load combination.

3 Change the Factor and Load Case as necessary.



Add another load combination with formula 1.4*DL1 + 1.6*LL1 and set the State to “Ultimate”. Then rename this load combination to “ULS1” and the first load combination to “SLS1”.



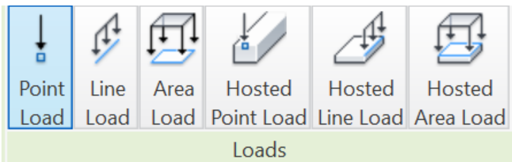
Next you can apply loads to the structure.

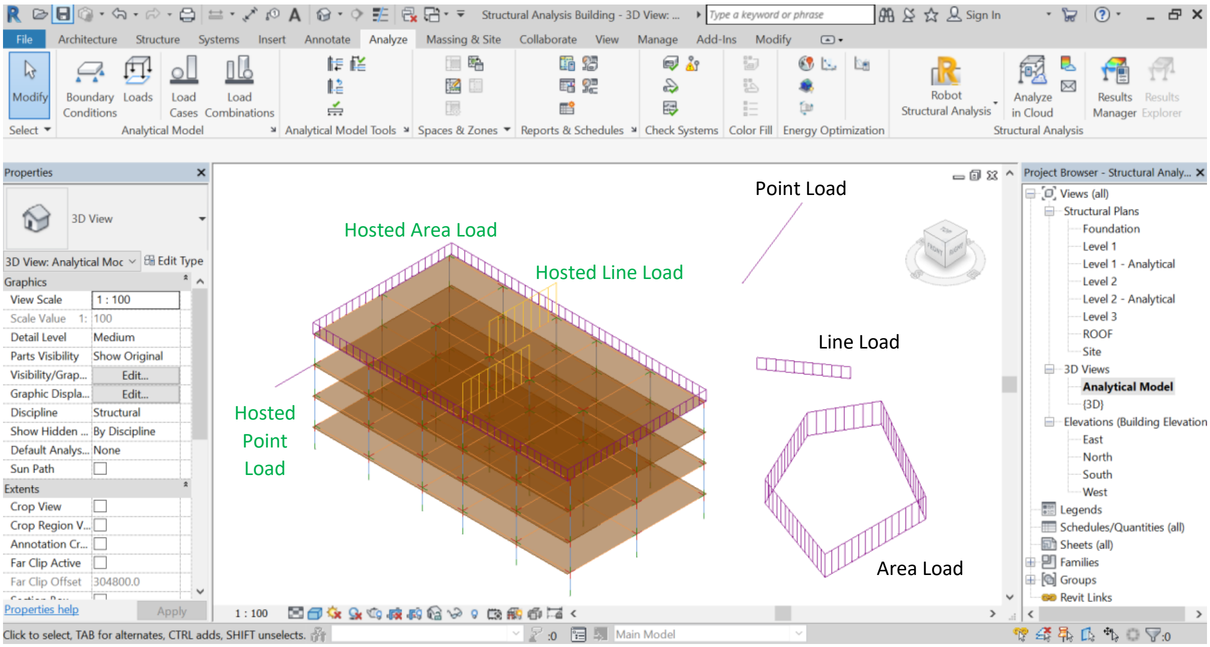
16.1.7 Applying Loads

Continue with the “Structural Analysis Building.rvt” model and apply loads to the structure.

Click Analyze tab ➢ Analytical Model panel ➢ Loads

There are Point Load / Line Load / Area Load / Hosted Point Load / Hosted Line Load / Hosted Area Load that you can apply to the structure. Hosted loads require structural elements to host them while other loads can be applied at anywhere.



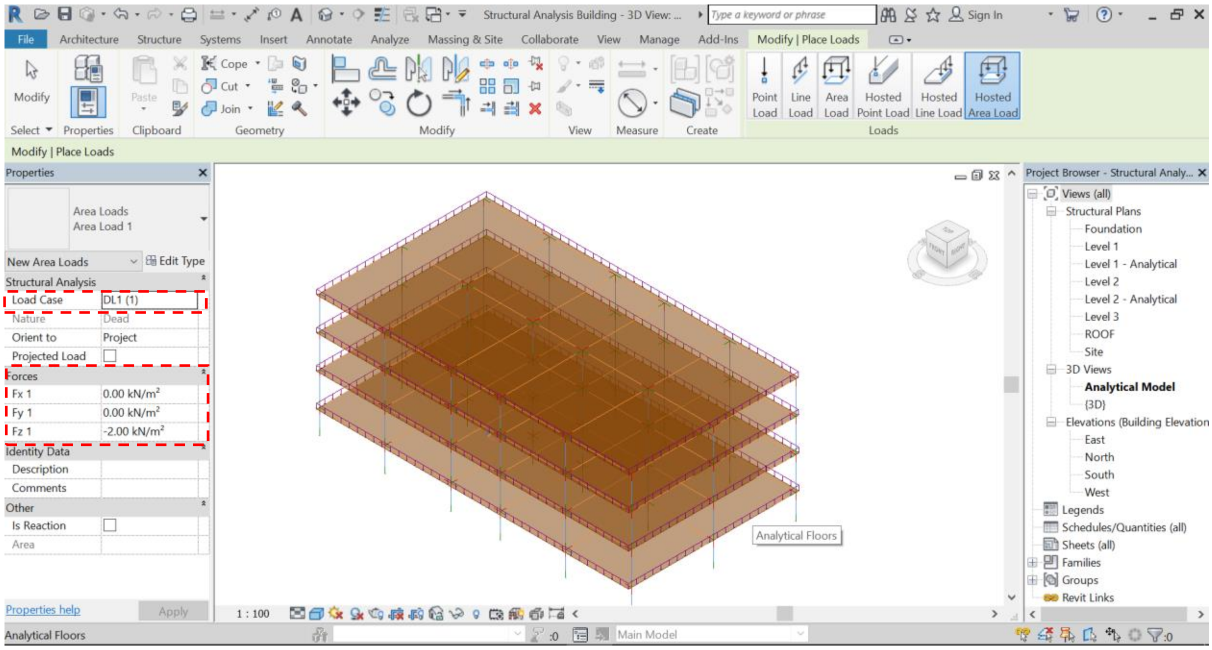


Now apply the following loads to the structure ("Structural Analysis Building.rvt").

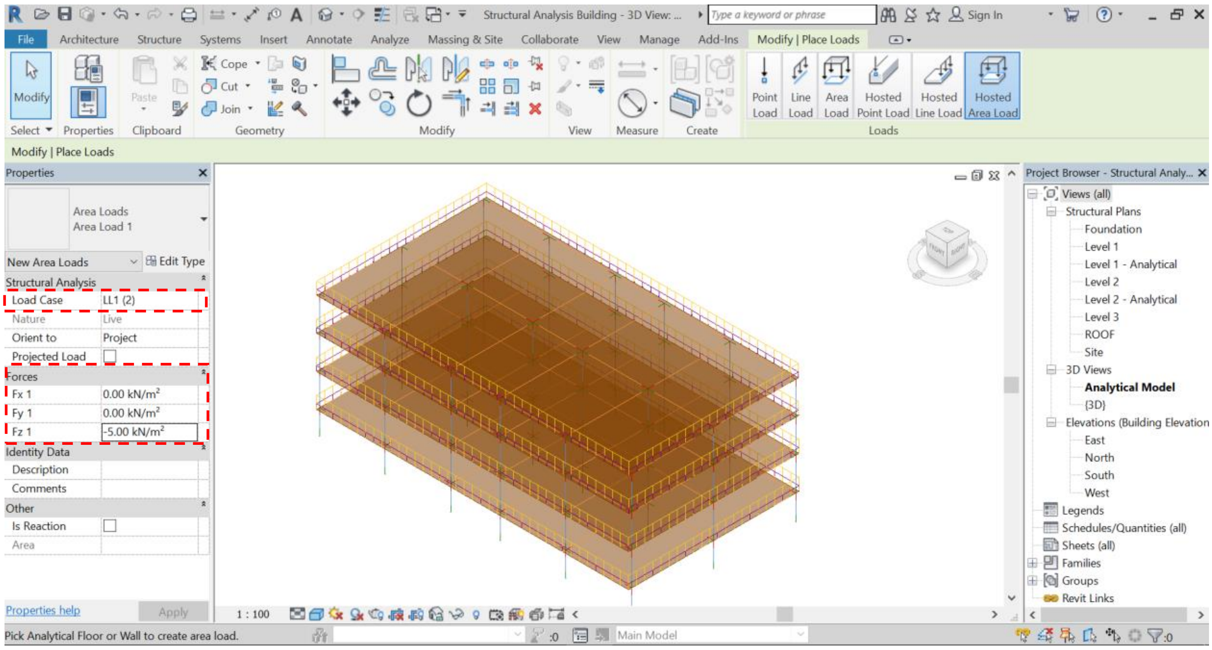
1 Go to Analytical Model view then click Analyze tab ➢ Analytical Model panel ➢ Loads ➢ Hosted Area Load.

2 First, apply some Hosted Area Load as Dead Load to the structure. Set the Load Case to DL1 (1) and enter -2.00 kN/m2 for Force Fz 1 (other forces set to zero value). (Note that positive z-direction is upward. Therefore, gravity load should be input as negative value.)

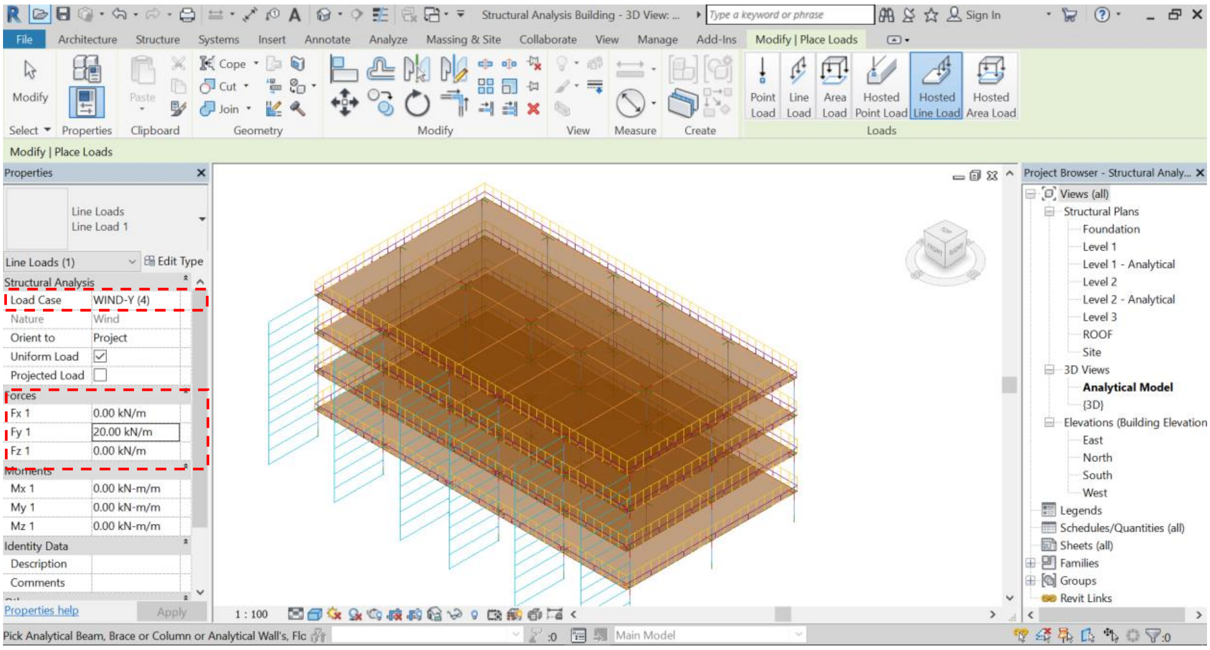
3 Hover over to the roof floor and add the hosted area load. Add this load to the other floors.



5 Next apply some Hosted Area Load as Live Load to the structure. Set the Load Case to LL1 (2) and enter -5.00 kN/m^2 for Force Fz 1 (other forces set to zero value). Then apply this load to all floors.

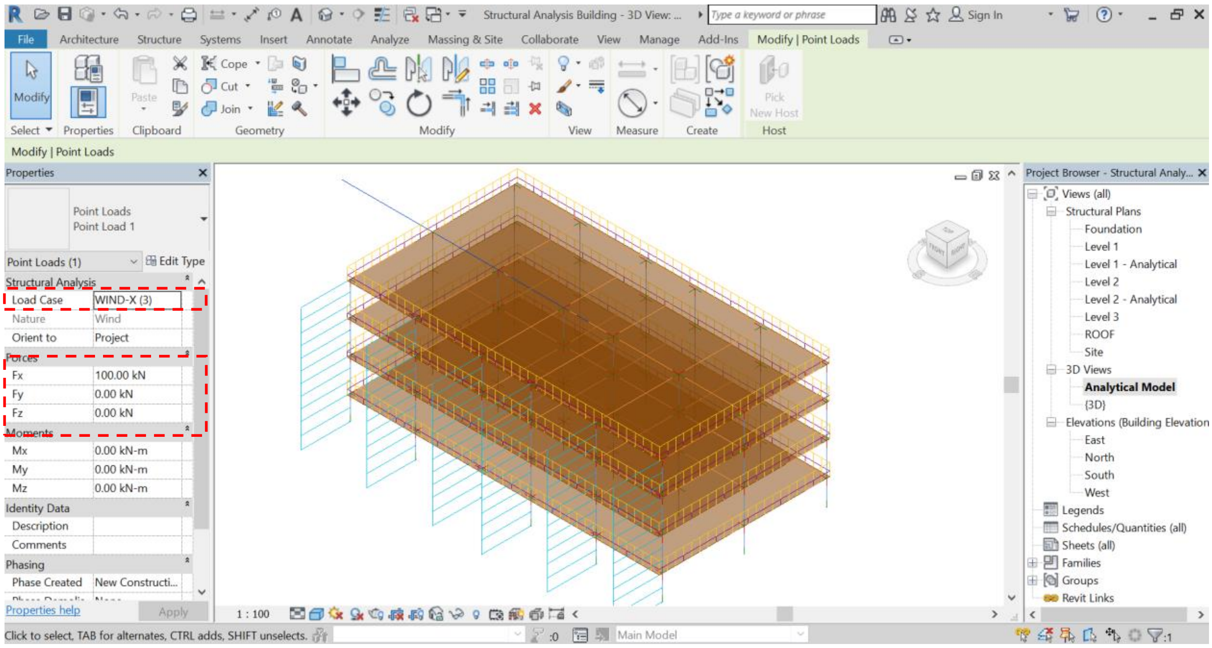


6 Next apply some Hosted Line Load as Wind Load to the structure. Select Hosted Line Load from the Load panel. Then set the Load Case to WIND-Y (4) and enter 20.00 kN for Force Fy 1 (other forces set to zero value). Then apply this load to all the columns along Grid-C as shown below.



7 Lastly, apply a Point Load as Wind Load to the structure. Go to ROOF plan view. Click Analyze tab ➢ Analytical Model panel ➢ Loads ➢ Point Load. Then set the Load Case to WIND-X (3) and enter 100.00 kN for Force Fx 1 (other forces set to zero value). Then apply this load to the center of the floor.

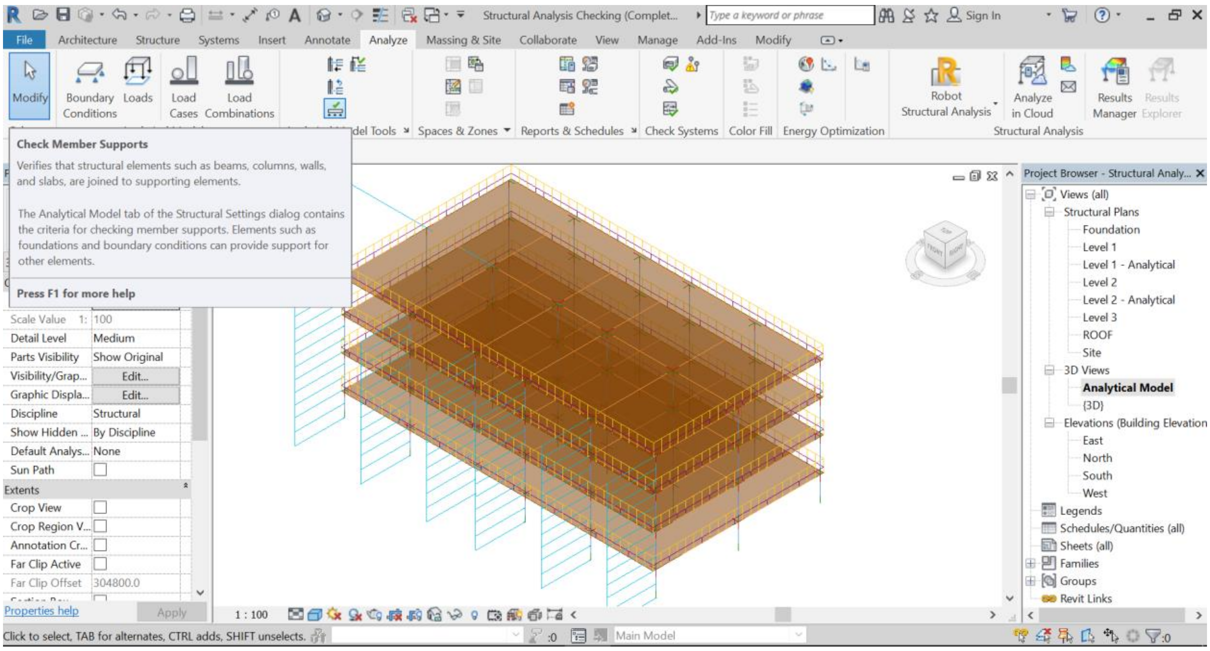
8 You may not be able to see the load in plan view. Go to Analytical Model view to check your input. Alternatively, you can set to view Structural Load in plan view by going to Visibility/Graphics (type “vv” for shortcut) and turn on Structural Load.

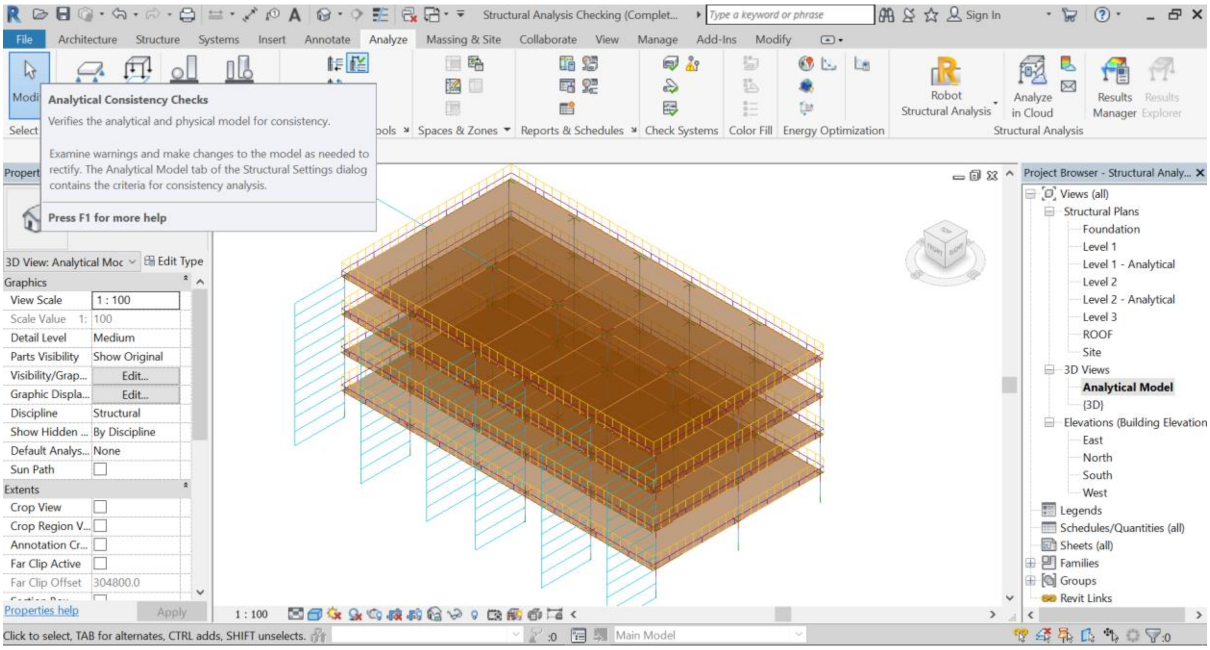


Now that the structure has all the ingredient and it is ready to be analysed.

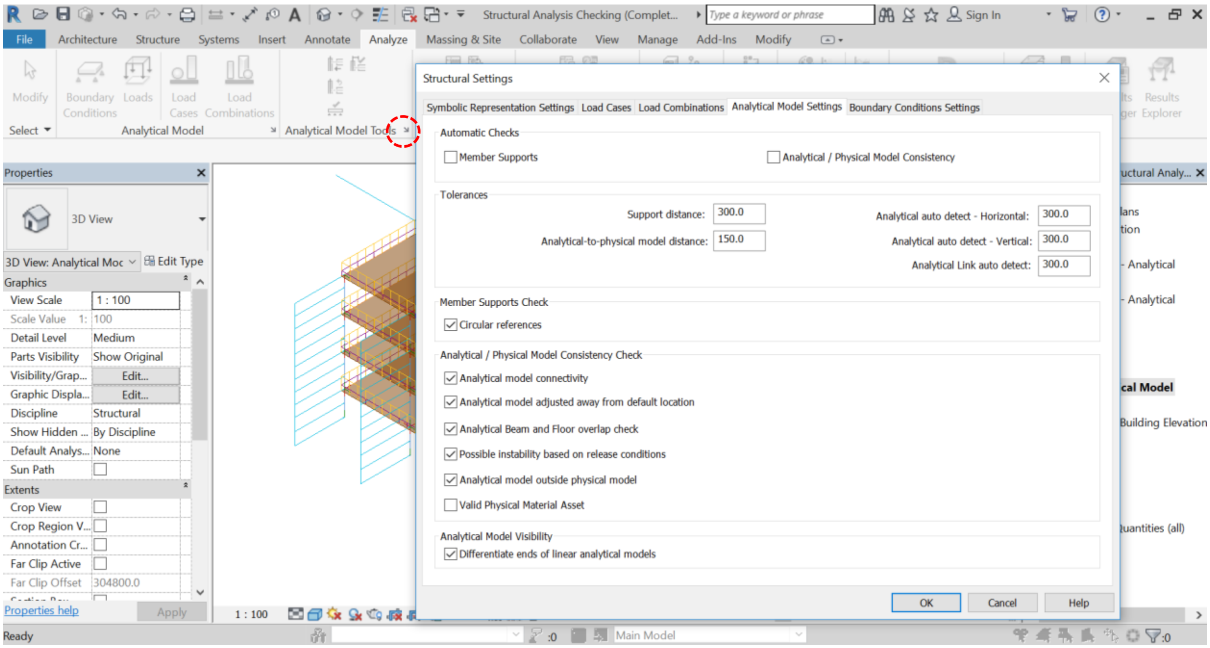
16.1.8 Check Supports and Consistency Check

Revit provides two checking options for you to make sure the structural model is good without missing supports and that it is analytically consistent. These options are under the Analyze tab.

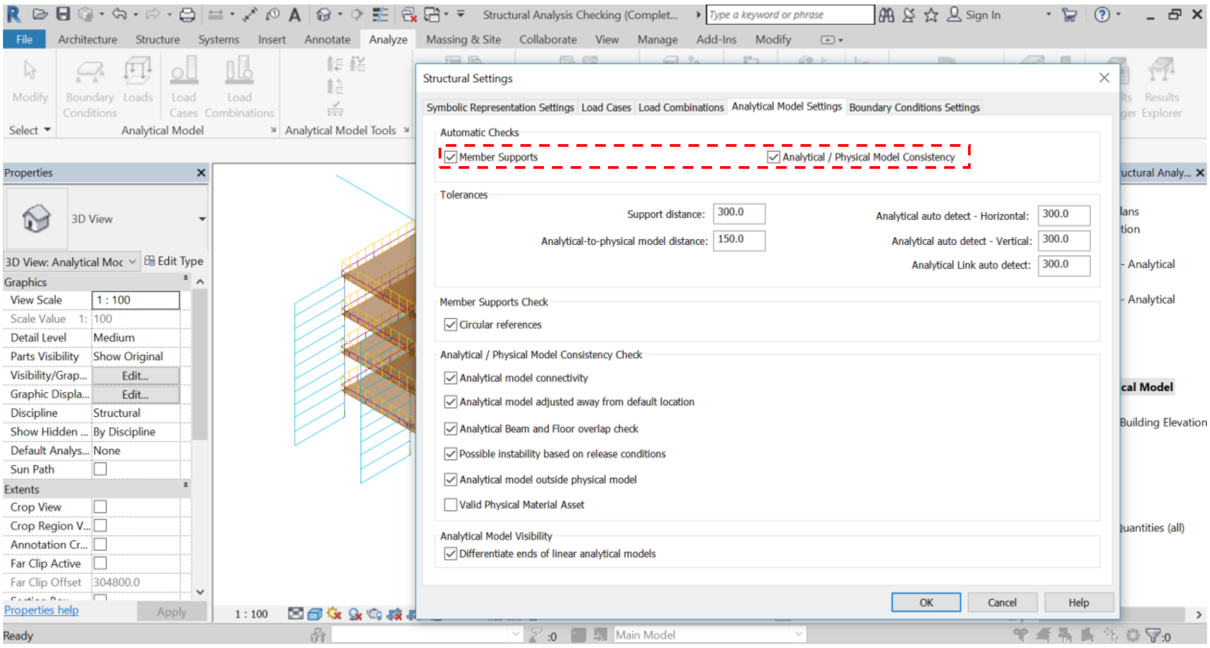




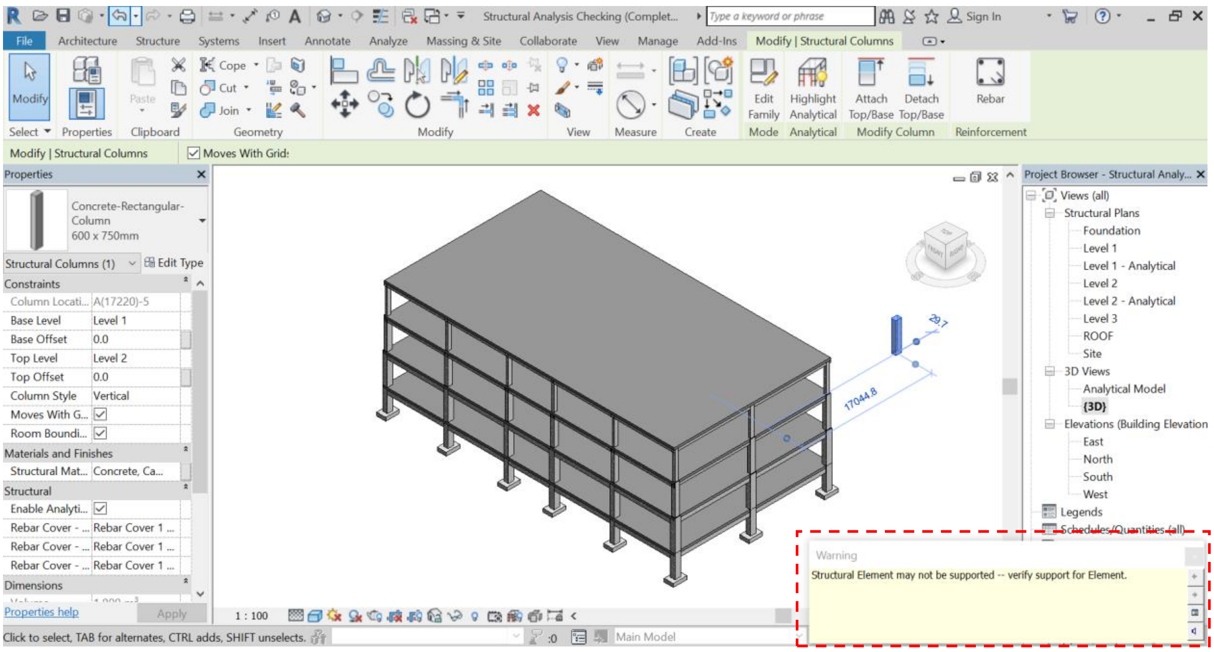
You can view and set the checking criteria by going to the Analytical Model Settings.



Now, turn on both the Member Supports and Analytical / Physical Model Consistency checks on under Automatic Checks and see what will happen if you try adding something unsupported to the model.

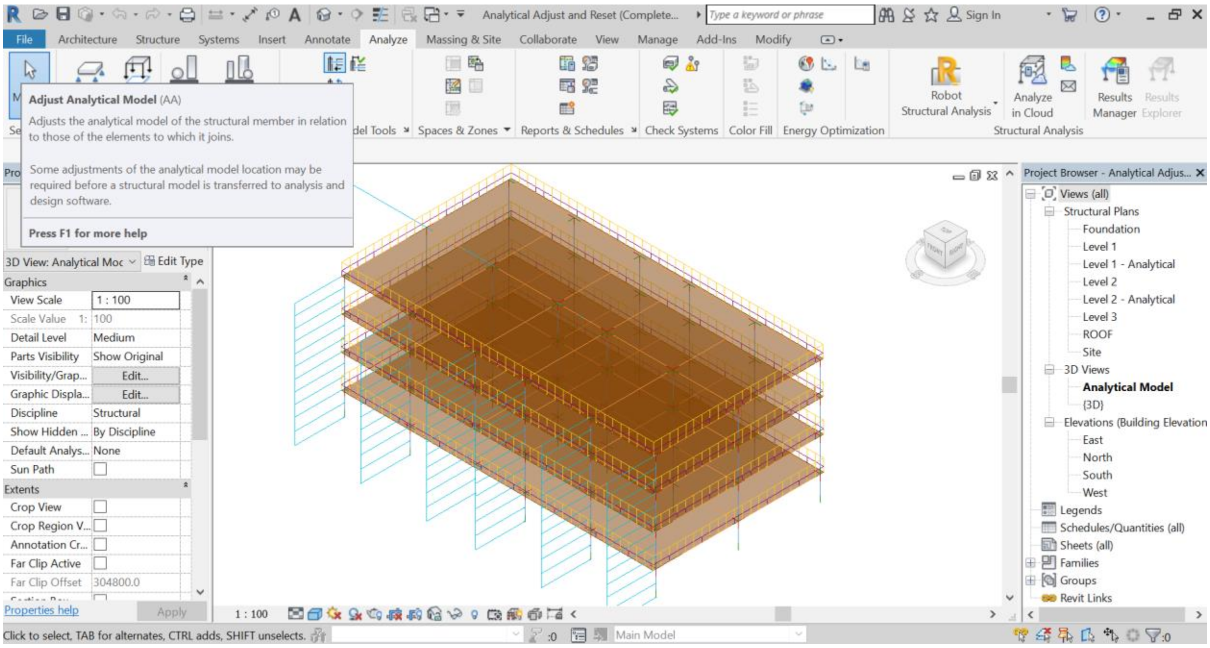


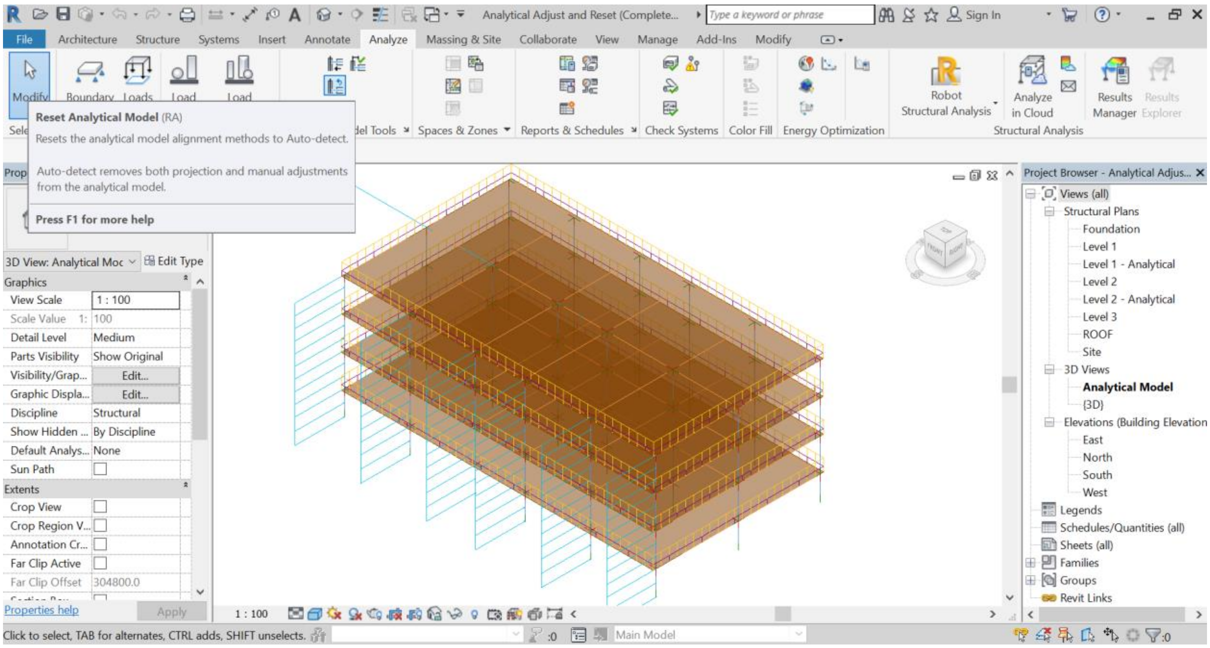
For example, if you copy a column to a random spot in 3D then you will get a warning. The warning is "Structural element may not be supported – verify support for element.". You can continue to build your model and view all these warnings later by going to Manage tab ➢ Inquiry panel ➢ Review Warnings .



16.1.9 Analytical Adjust and Analytical Reset

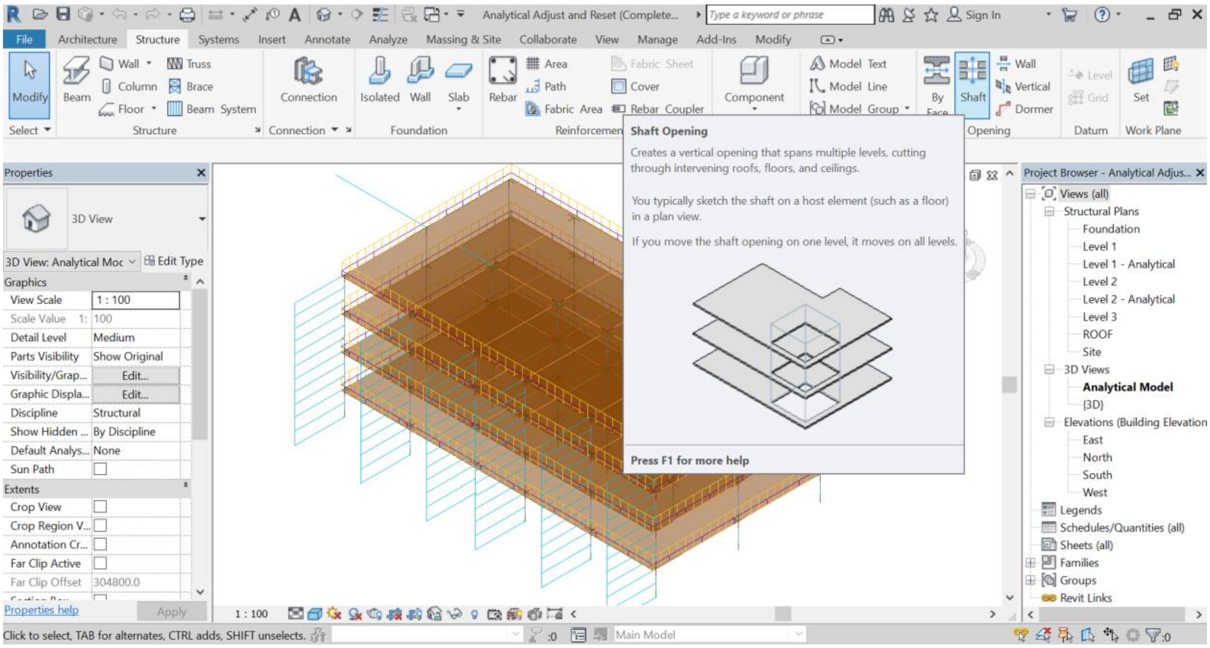
You can adjust the analytical model using the Analytical Adjust tool. Revit also provides an Analytical Reset option to reset the analytical model of an element to its default position.



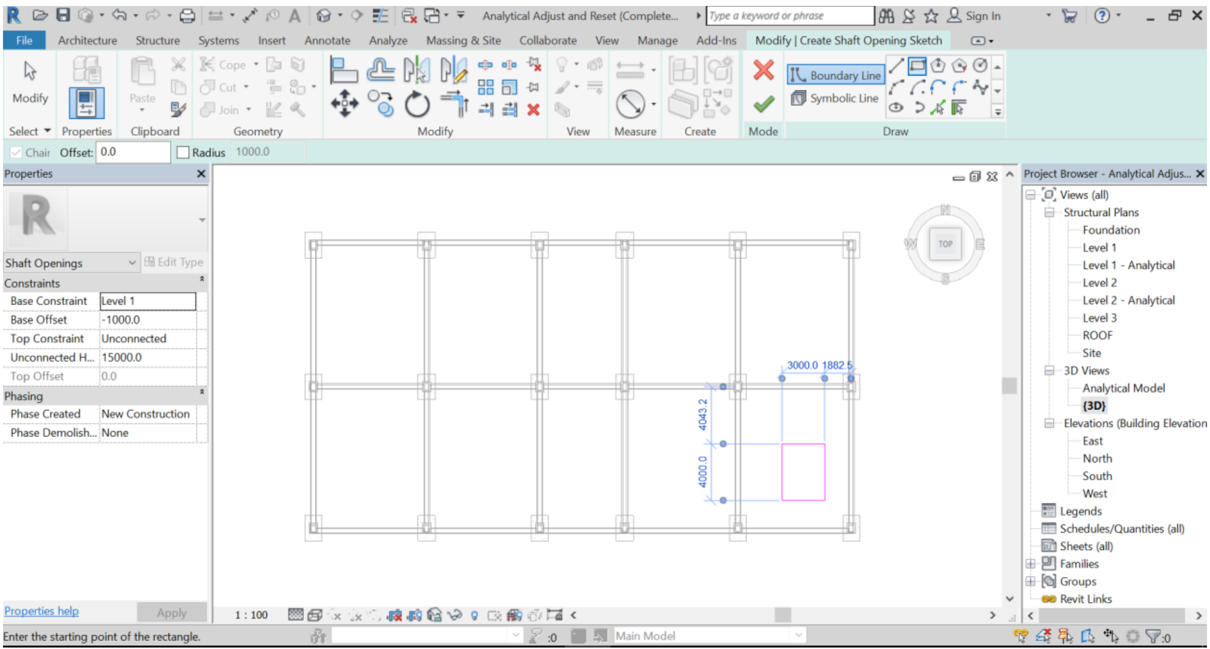


Now, add a shaft opening, some walls and copy a column in the model before you adjust the analytical model. (Note: exact dimensions on plan are not important when placing the elements.)

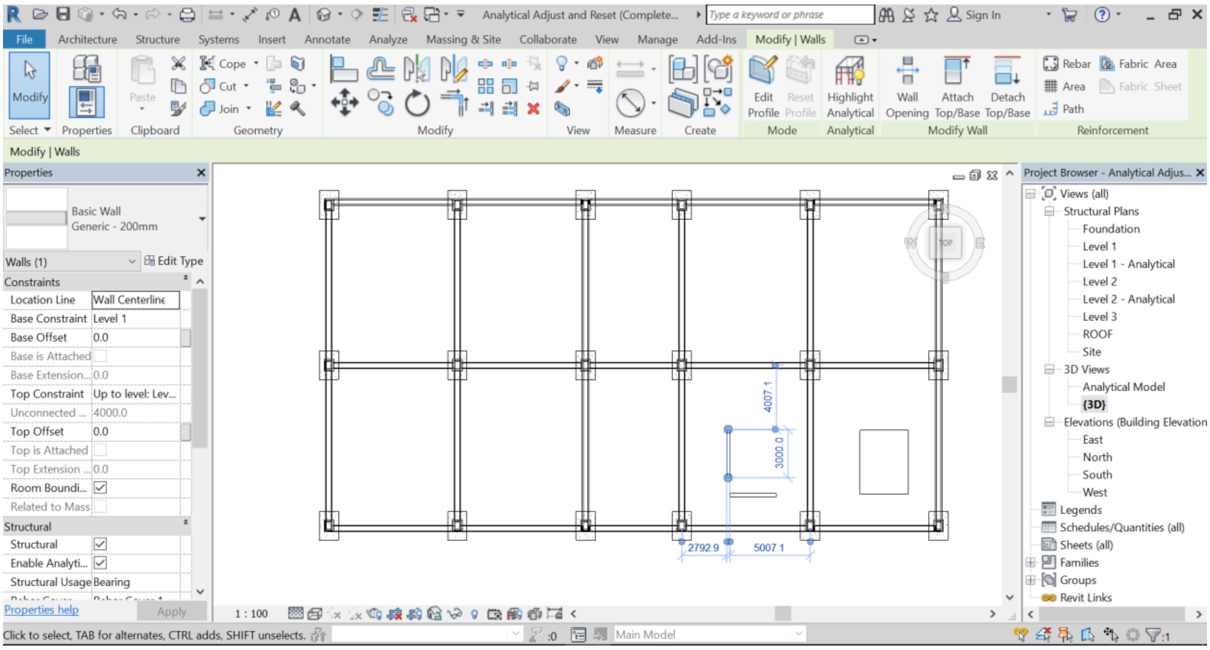
1 Go to 3D view then click Structure tab ➢ Opening panel ➢ Shaft.



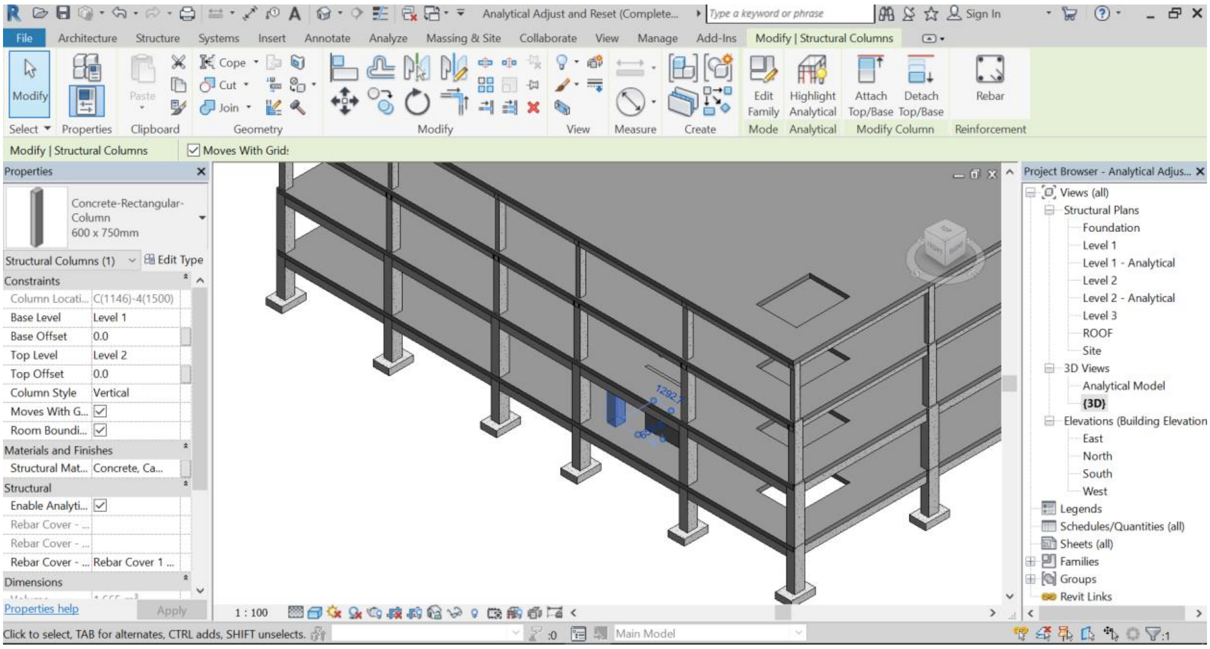
2 Align the view to "Top" and then set the "Base Offset" to "-1000" and "Unconnected Height" to "15000". Sketch a rectangle in the structure as shown below.



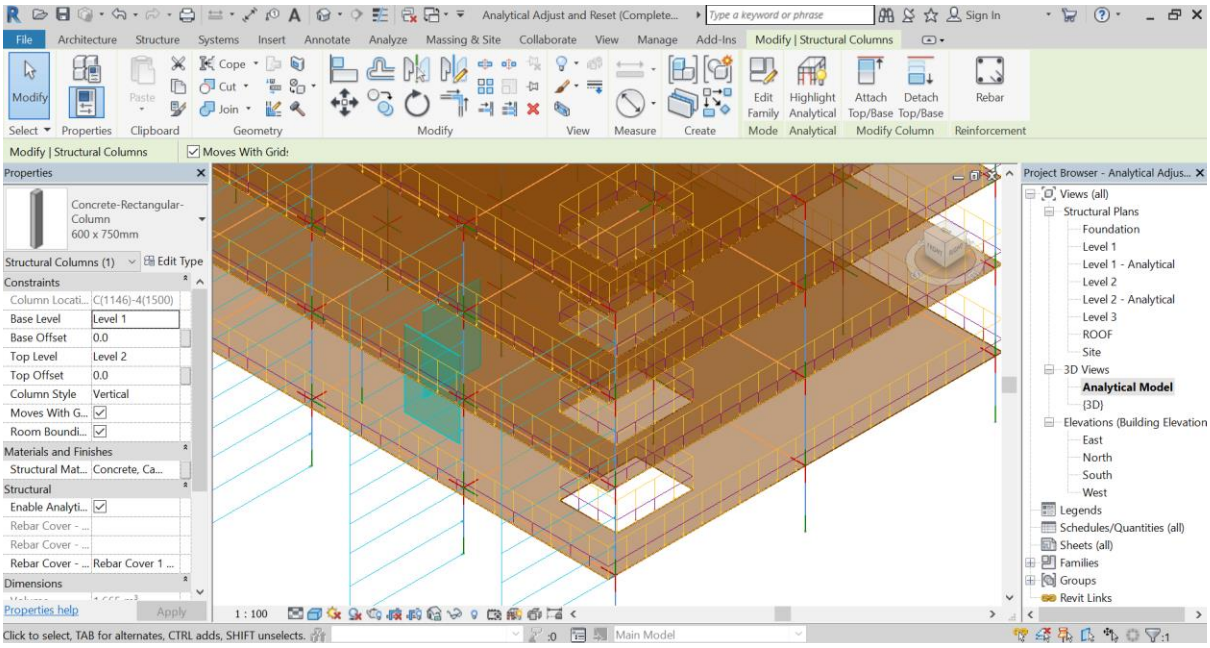
3 Then place two wall objects from level 1 to level 2. Place them separately as shown below.



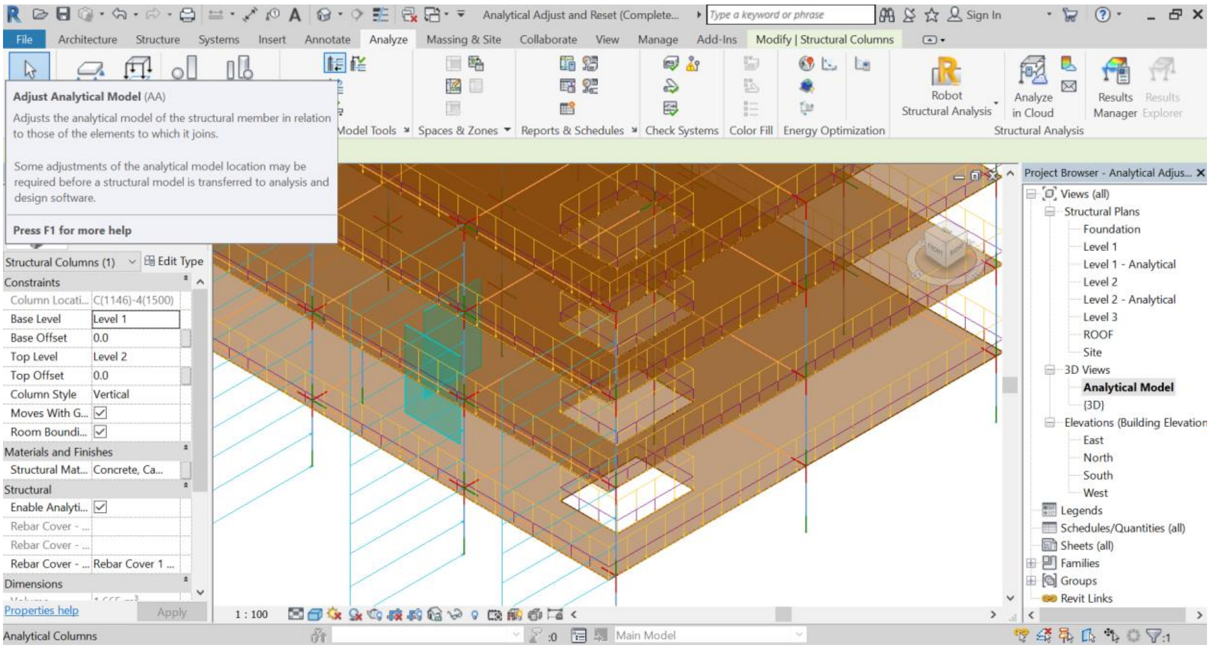
4 Copy a column at Level 1 and place it on level 1 floor as shown below.



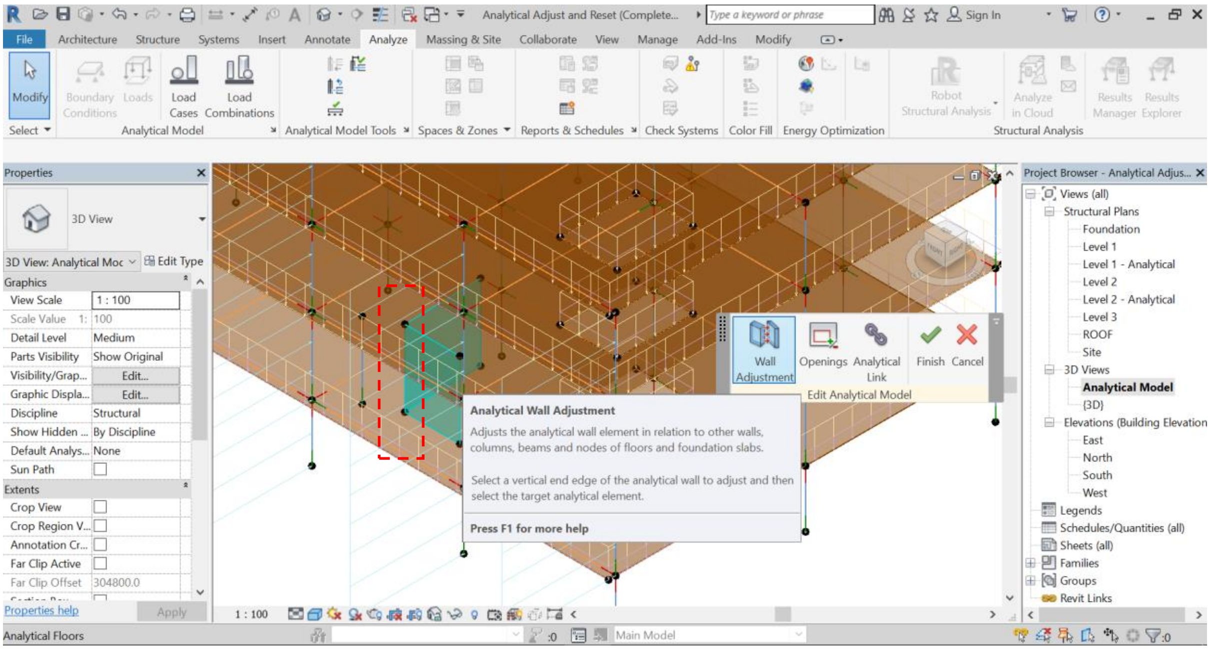
5 Go to the Analytical Model view and you will see the analytical model is updated as well.

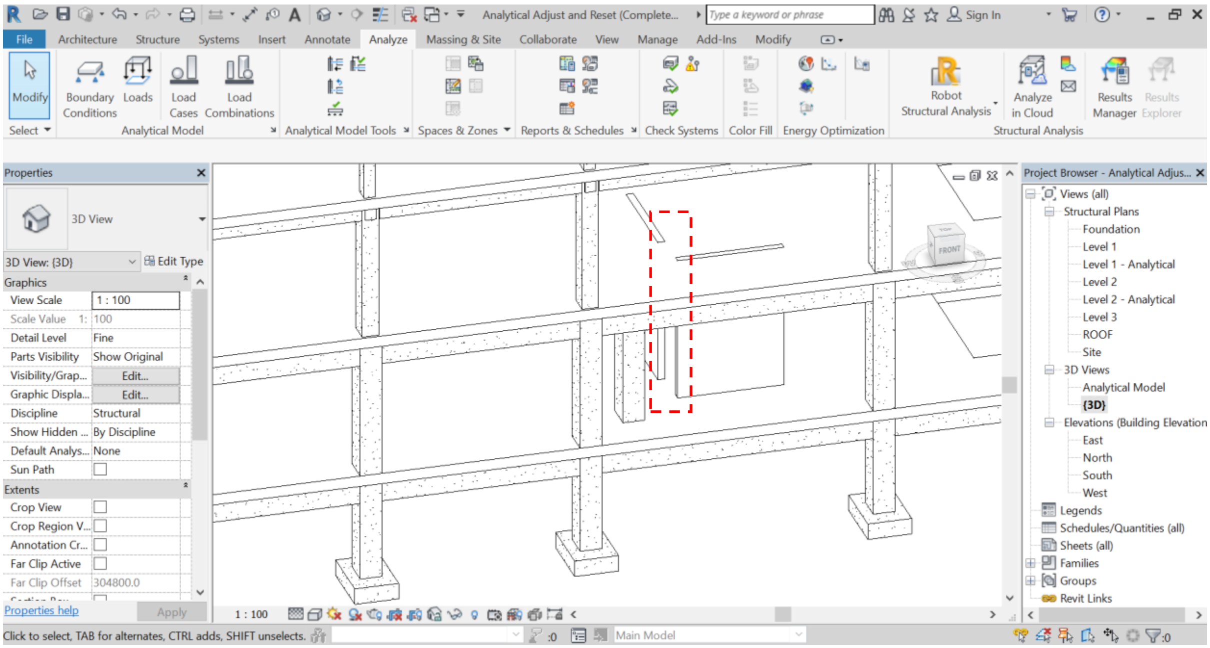


6 Go to “Analyze” tab and click “Analytical Adjust Model”.

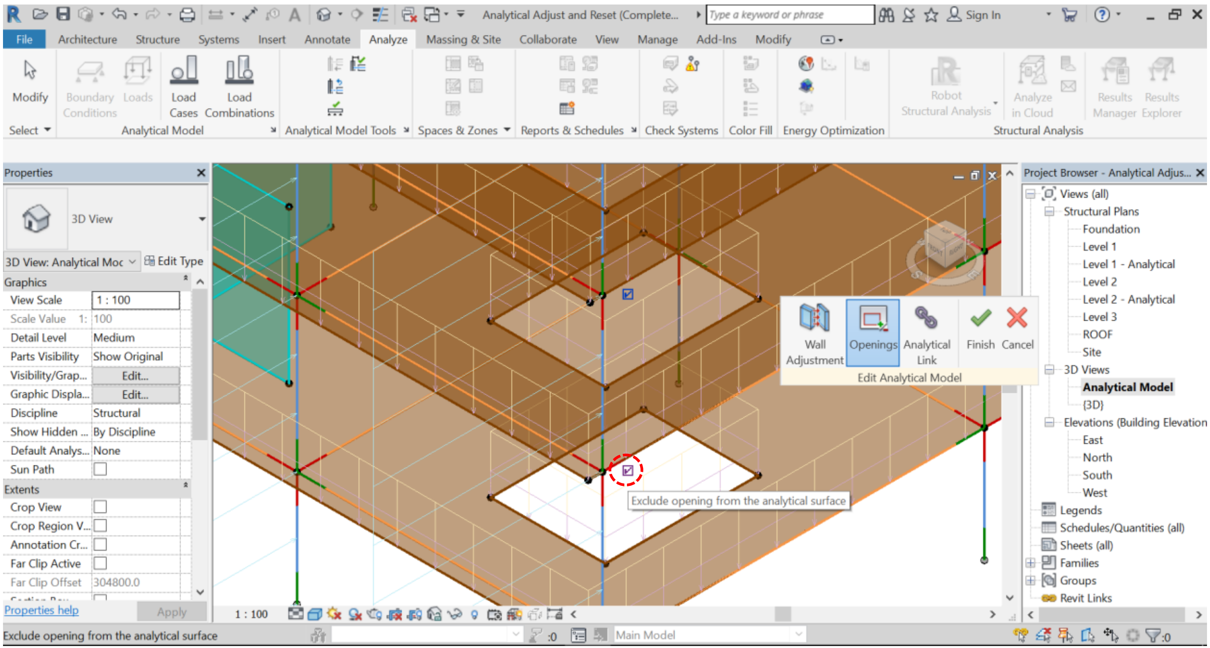


7 Select “Wall Adjustment”. This function modifies the analytical wall element in relation to other walls. Click one edge and then click the target that you want to connect. Note that the analytical walls have been adjusted to connect but the physical model remain unchanged.

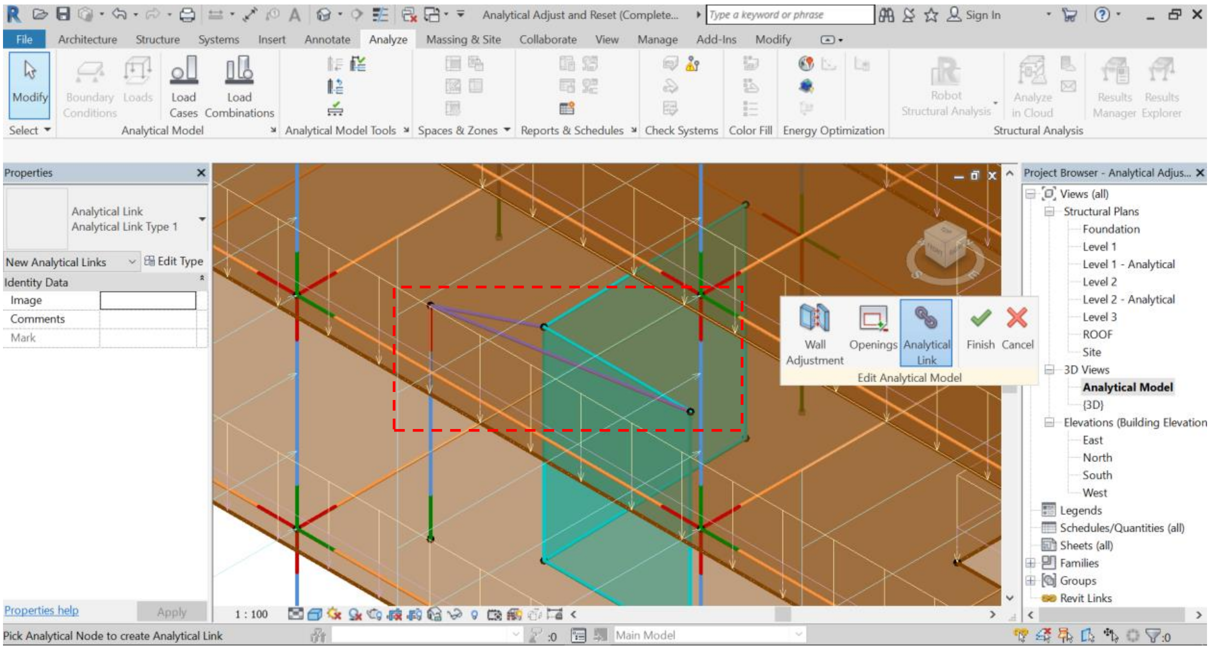




8 With “Openings”, you can modify the analytical model to include / exclude an opening from the surface.

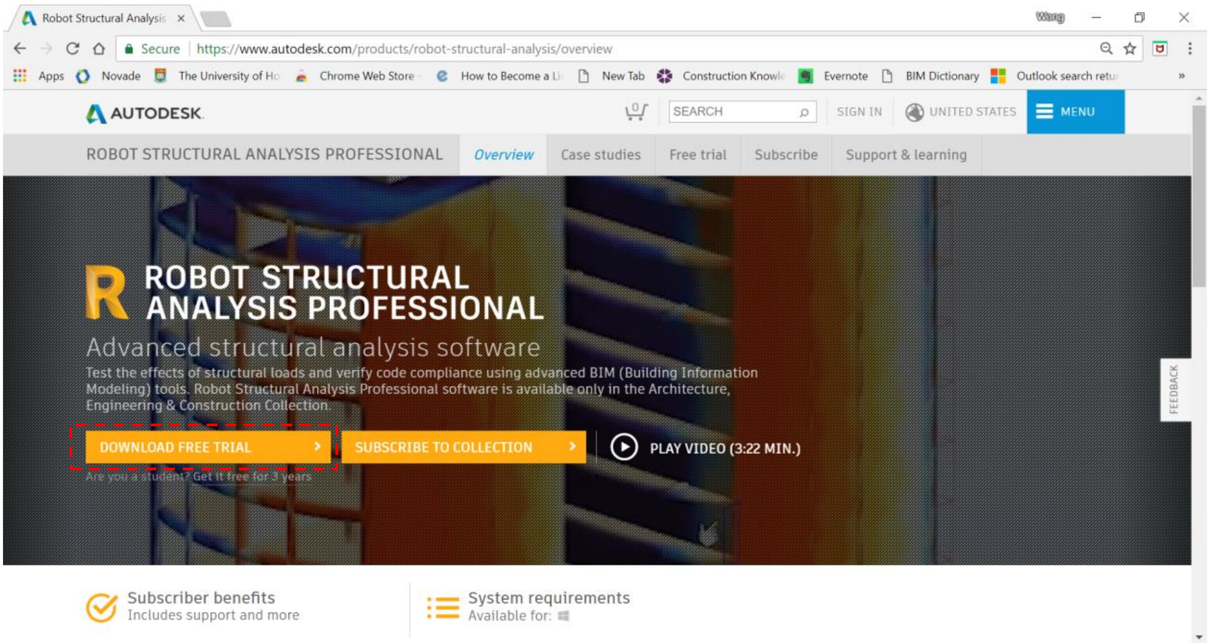


9 With "Analytical Link", you can add an analytical link (like rigid link in common structural analysis) connecting 2 analytical nodes.

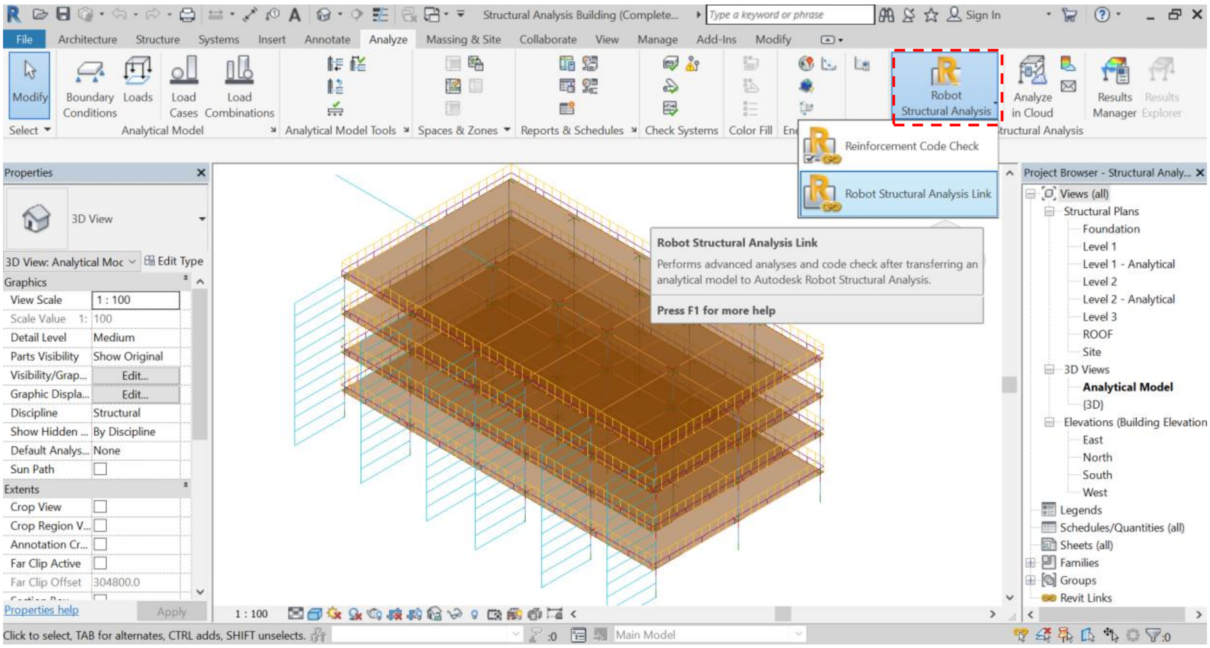


16.2 Structural Analysis using Robot Structural Analysis Professional (RSA)

After checking and adjusting the analytical model, you can now export the structure to RSA for analysis. You can then view the result in RSA or import it back to Revit for viewing and documentation. You can download RSA free trial (30 days) and test it out with your Revit 2016 or above version.



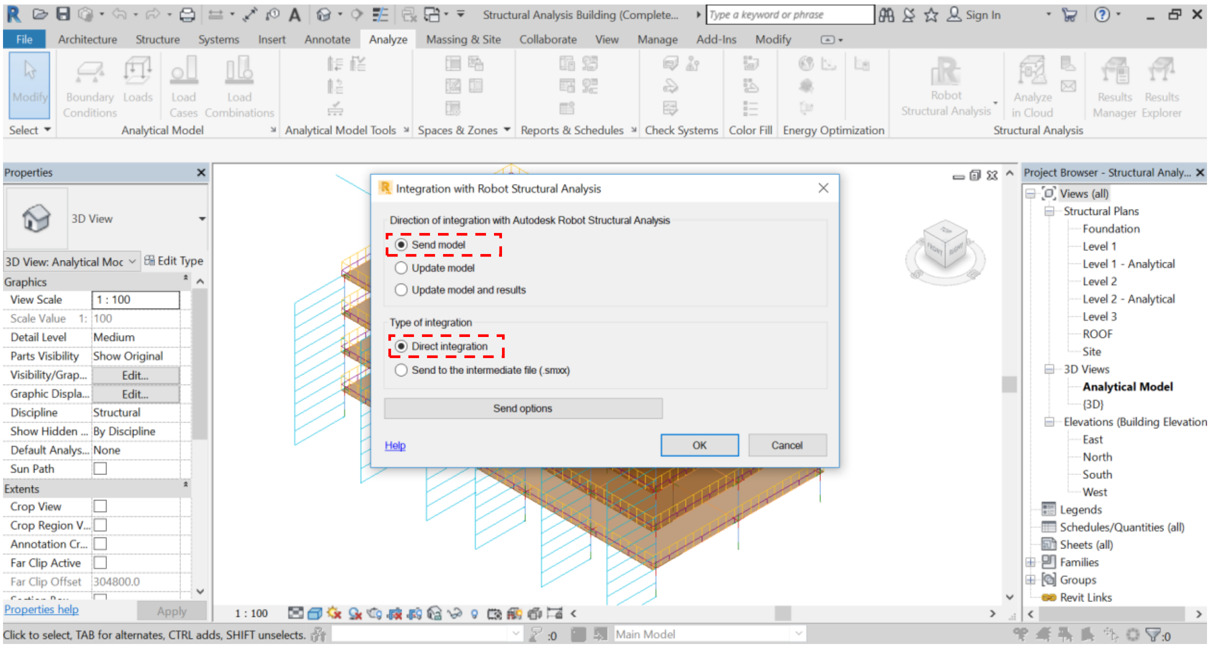
You can see an RSA icon in the “Analyze” tab after installing RSA.



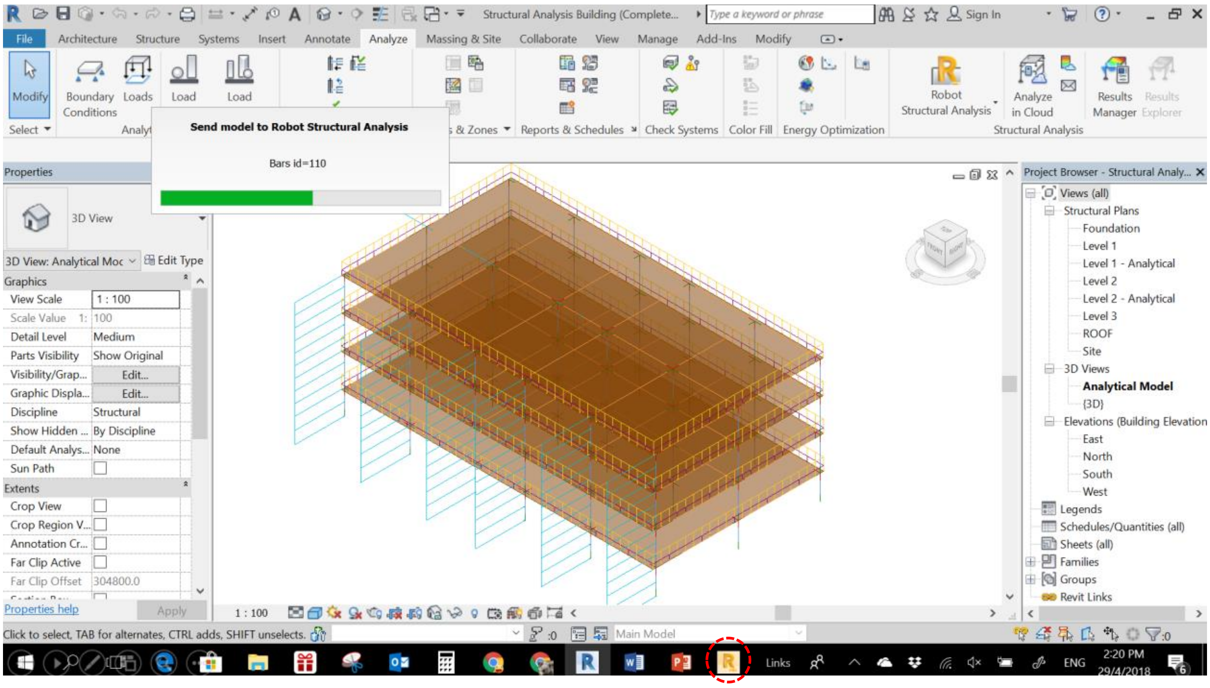
16.2.1 Sending Revit Model to Robot Structural Analysis (RSA)

1 Click Analyze tab ➢ Structural Analysis panel ➢ Robot Structural Analysis Link

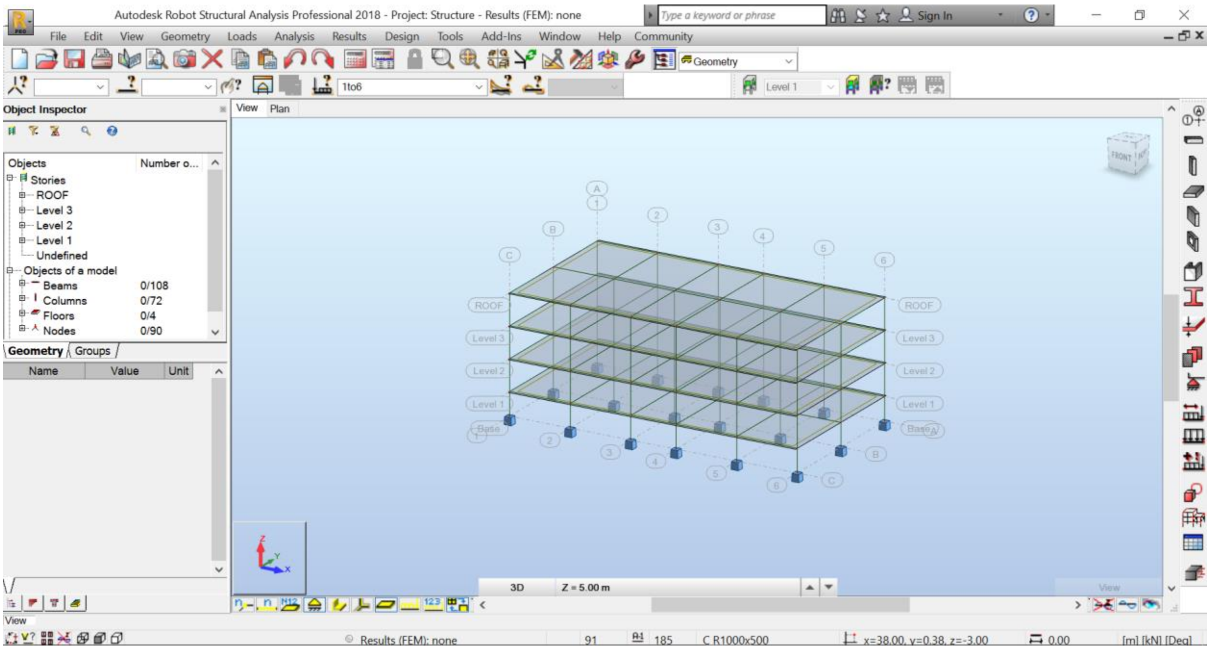
2 Select “Send model” and “Direct integration” from the dialogue box.



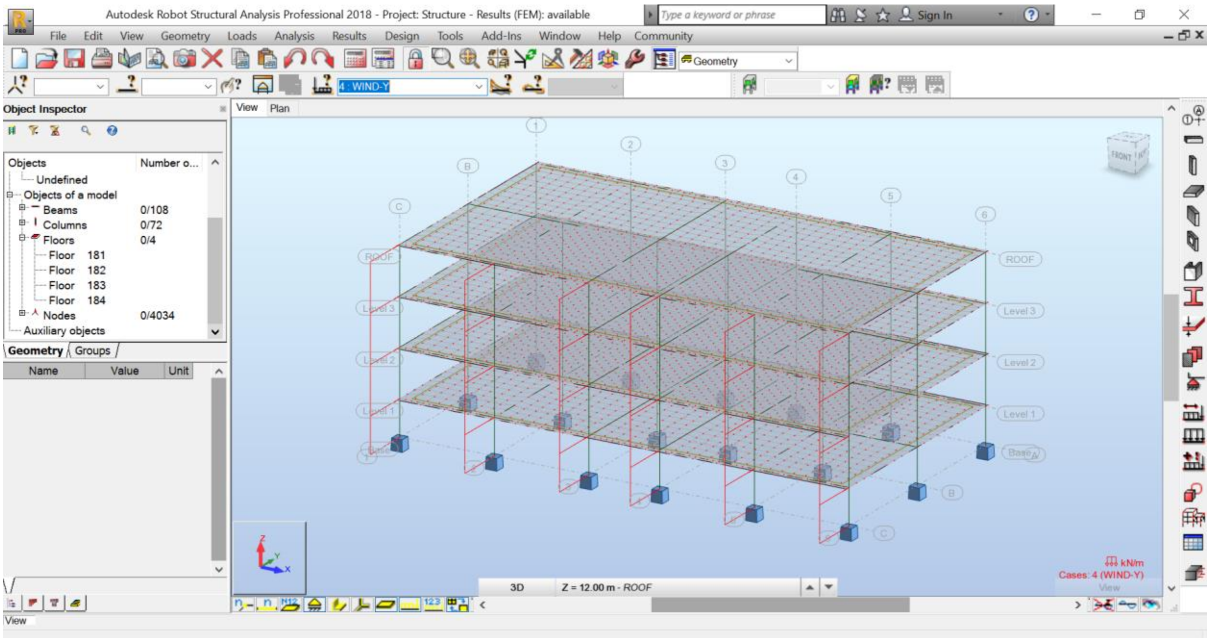
3 Click “OK” to start exporting the analytical model to RSA. RSA will start automatically during the process.



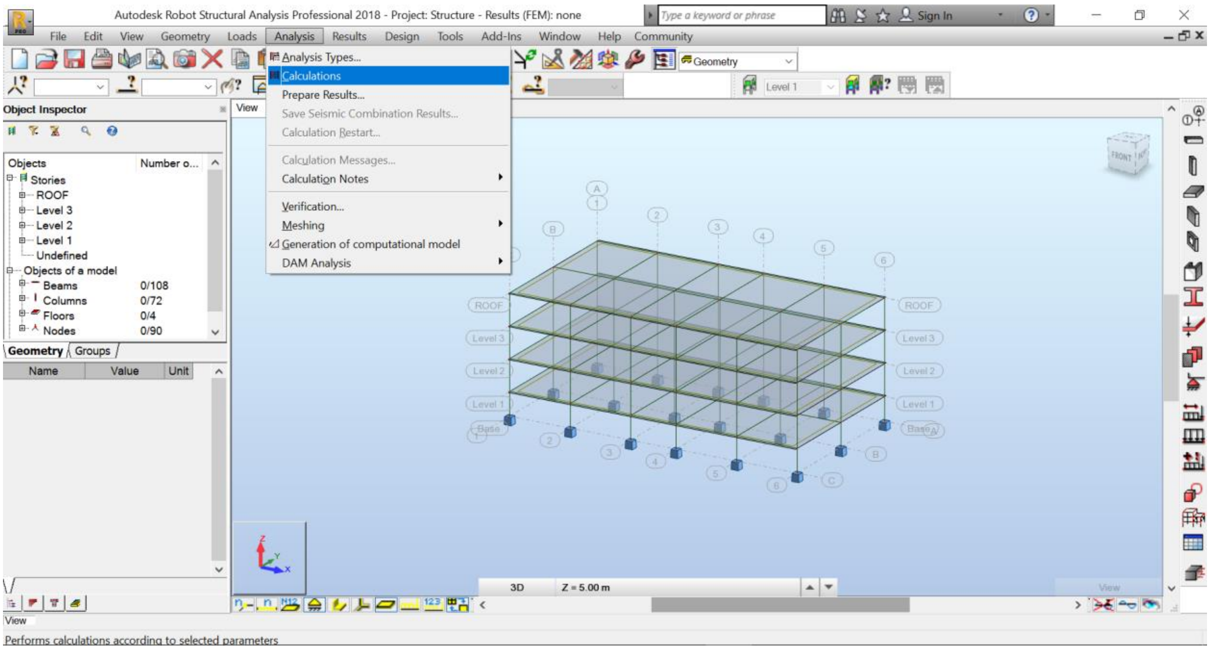
4 Go to RSA and you should see your model in RSA interface.



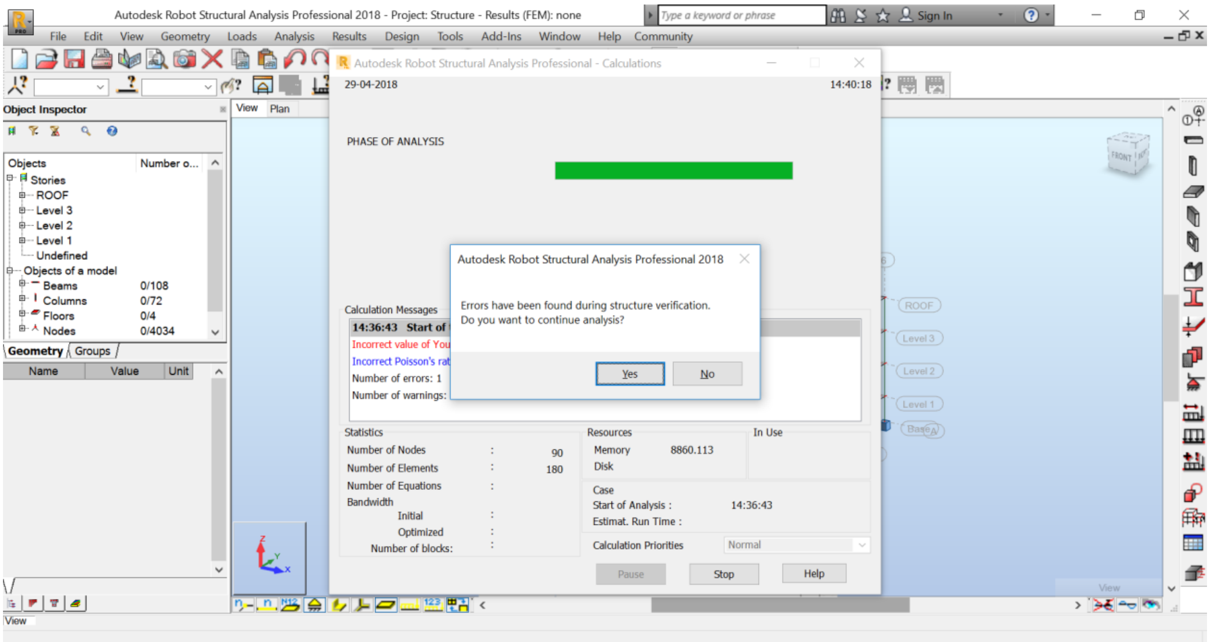
You can turn on the Load symbols to see if the loads are transferred correctly. For example, below screenshot shows the forces applied to the structure for the load case “WIND-Y”.

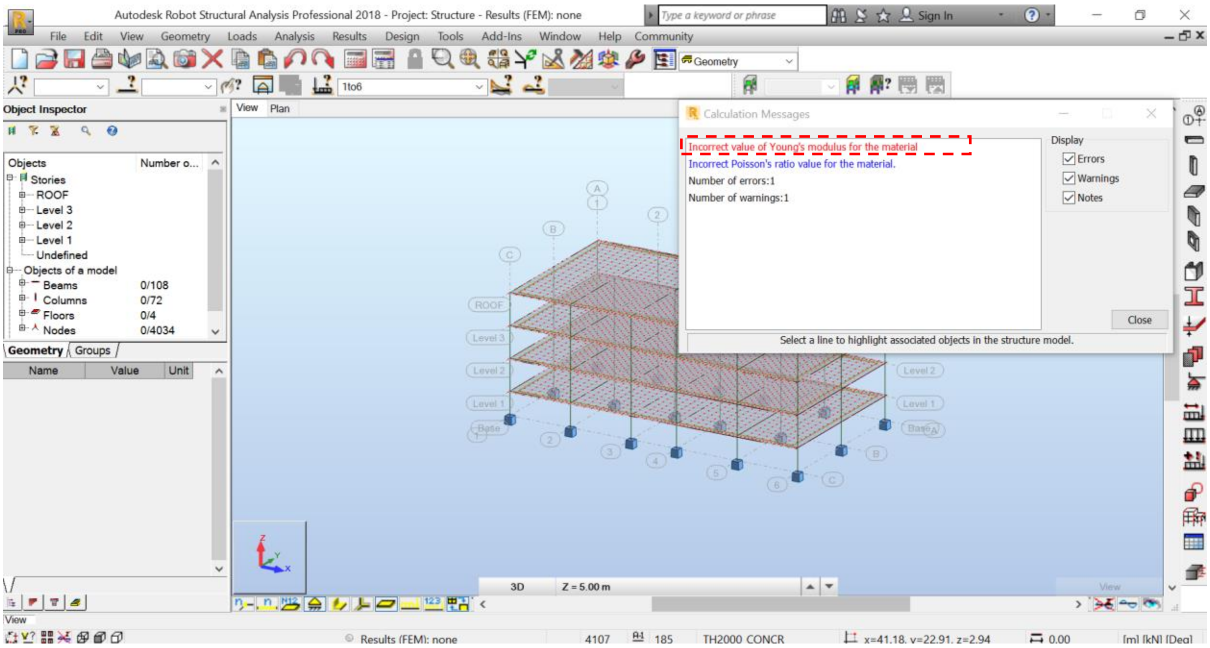


5 Now click Analysis ➢ Calculation to analysis the structure.

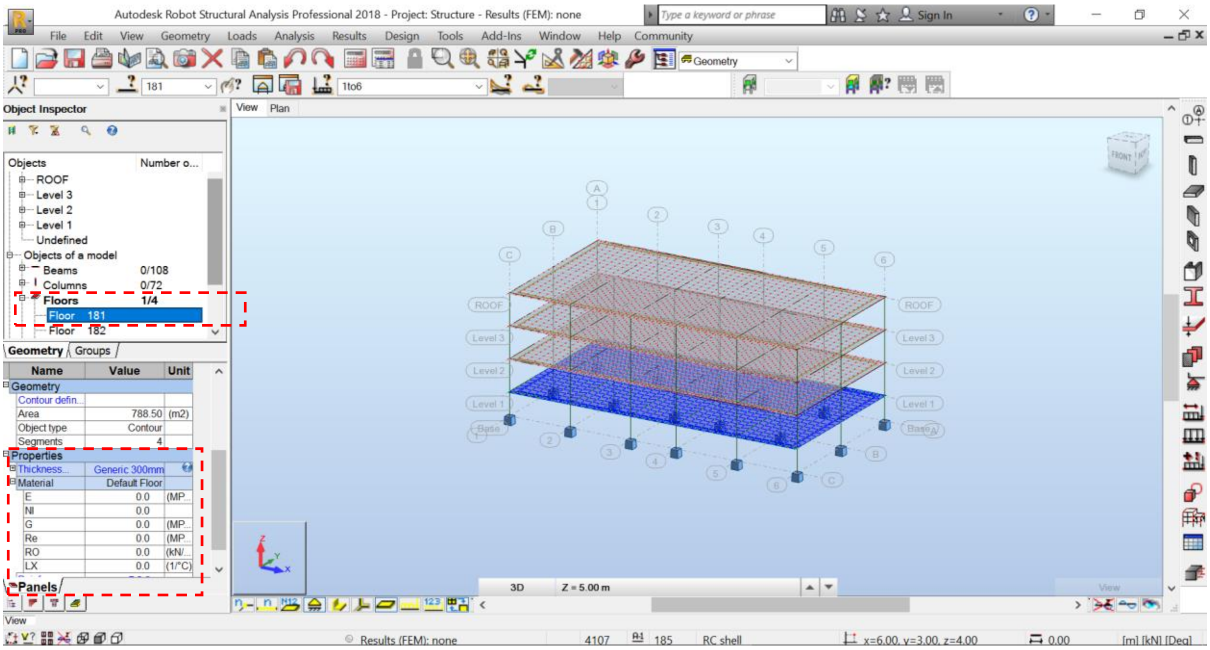


6 There is error in this case because of material properties problem.





If you select one of the floors, then you will see that the material used is “Default Floor” and it has zero values for its material properties.



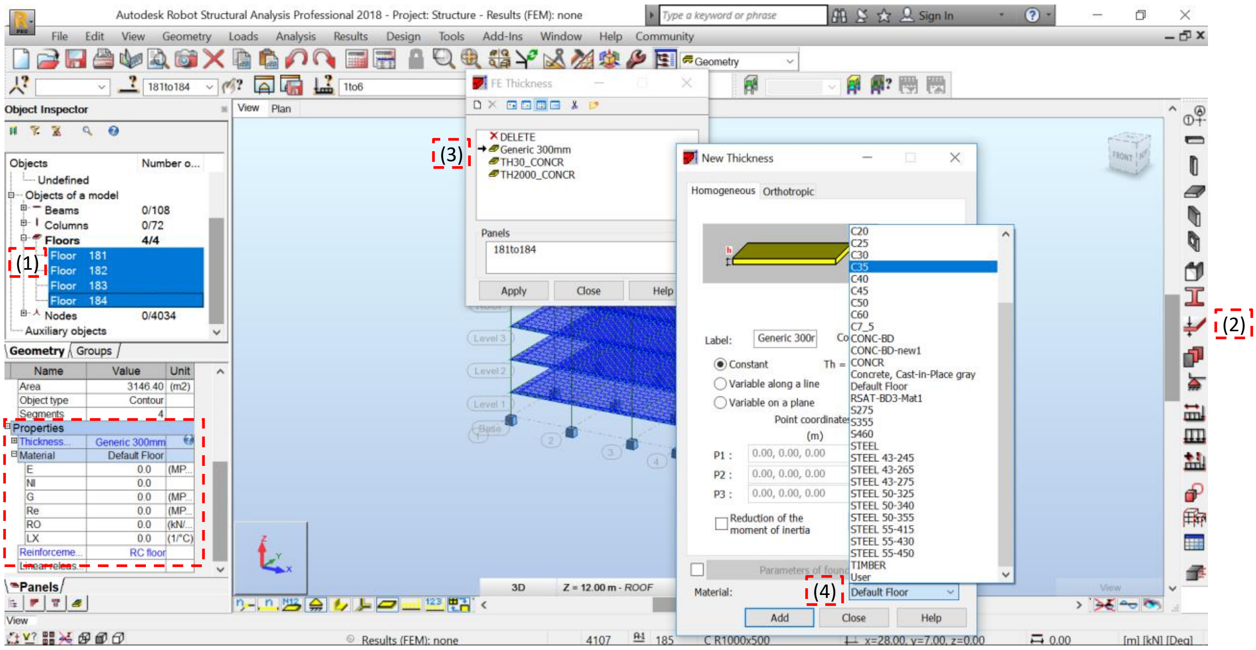
7 You can resolve the problem in RSA or in Revit. In RSA,

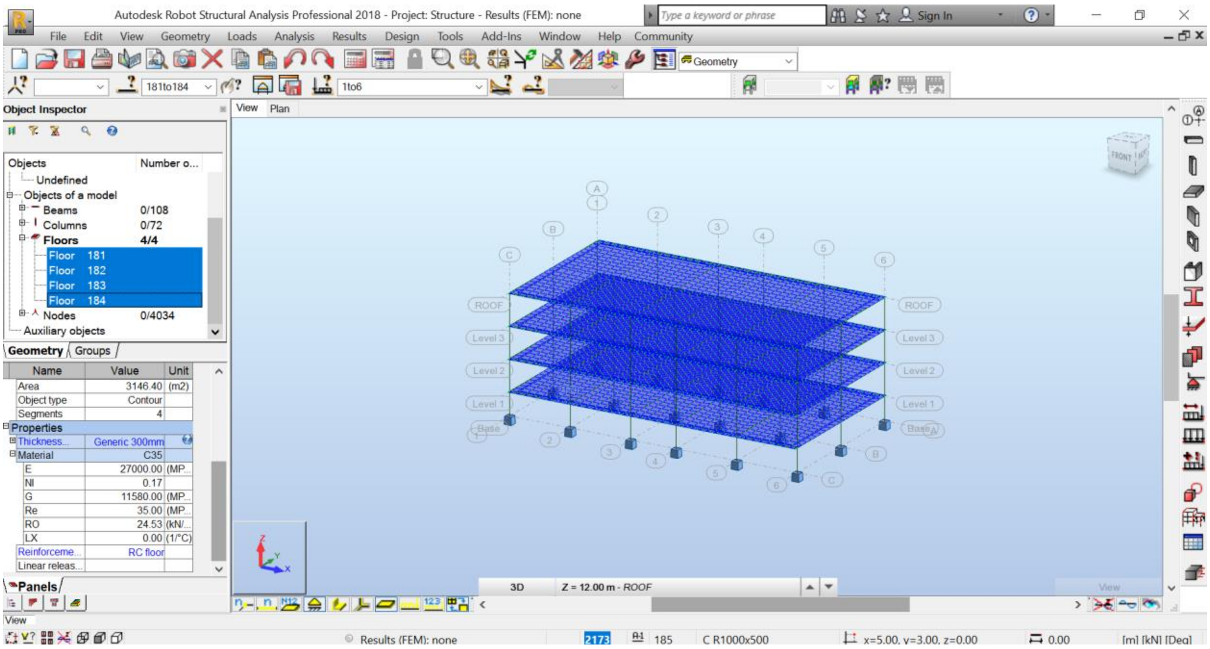
(1) Select all the floors.

(2) Click “Thickness”. 

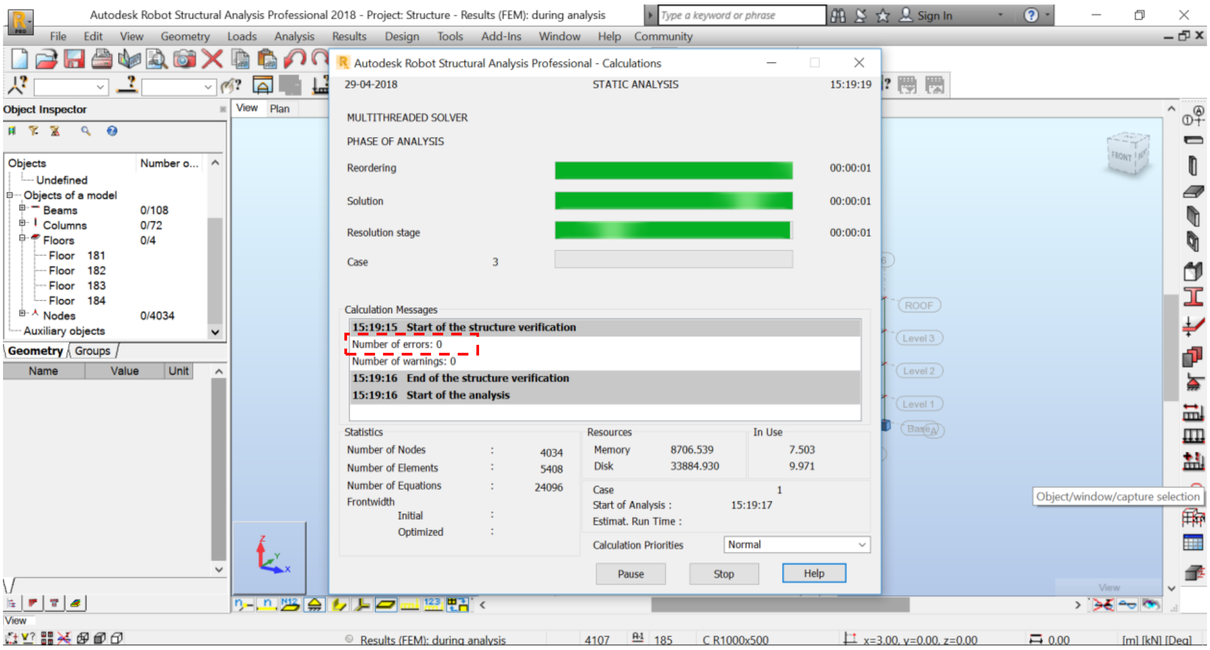
(3) Double click on “Generic 300mm”.

(4) Change the material from “Default Floor” to “C35” and click “Add” to modify it. Then all the floors with this thickness type “Generic 300mm” will be changed.

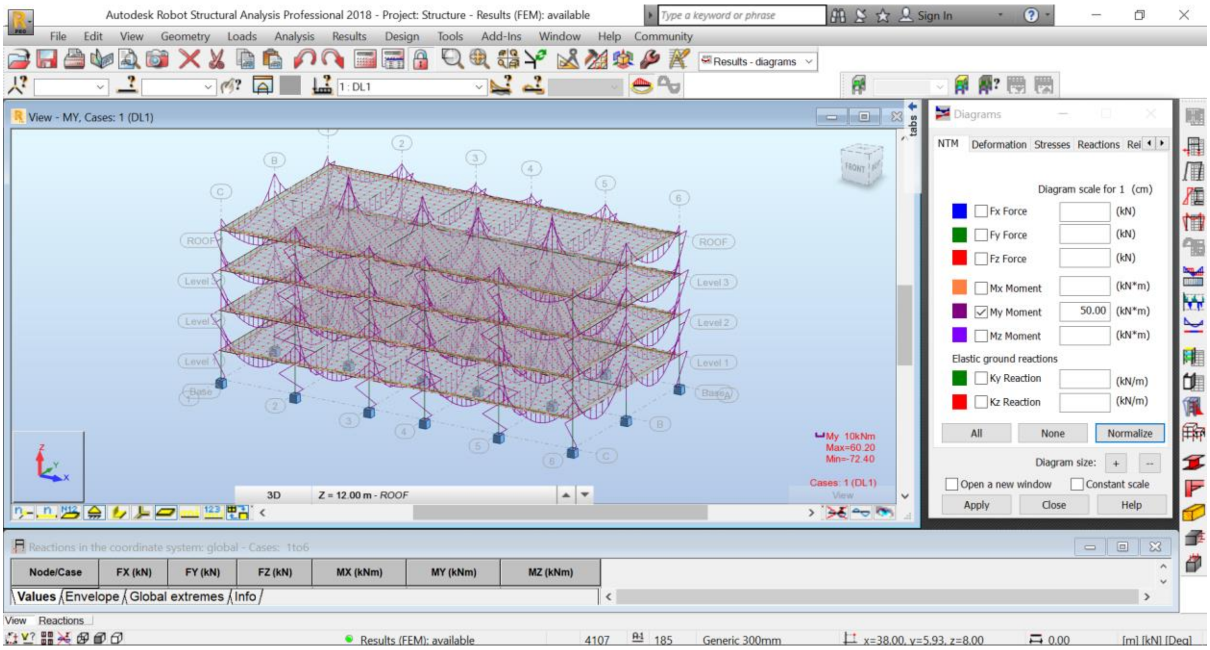




8 Carry out Calculation again to analysis the structure with the new material properties of the floors. This time there is no error.

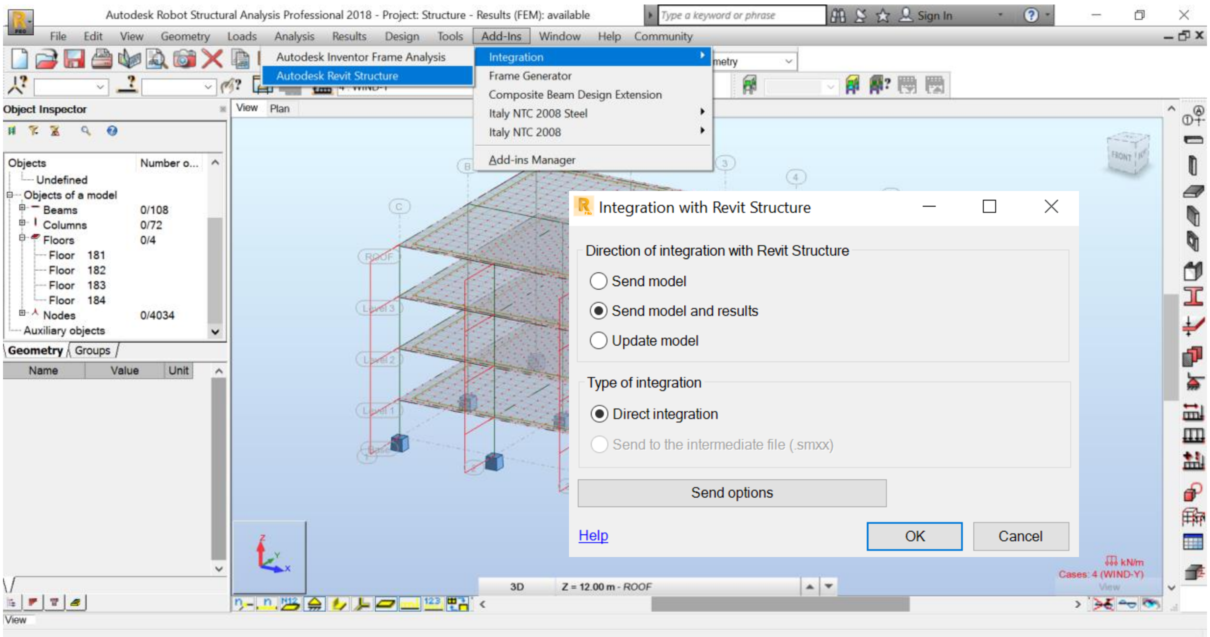


You can examine the analysis result in RSA or send it back to Revit for viewing and documentations.

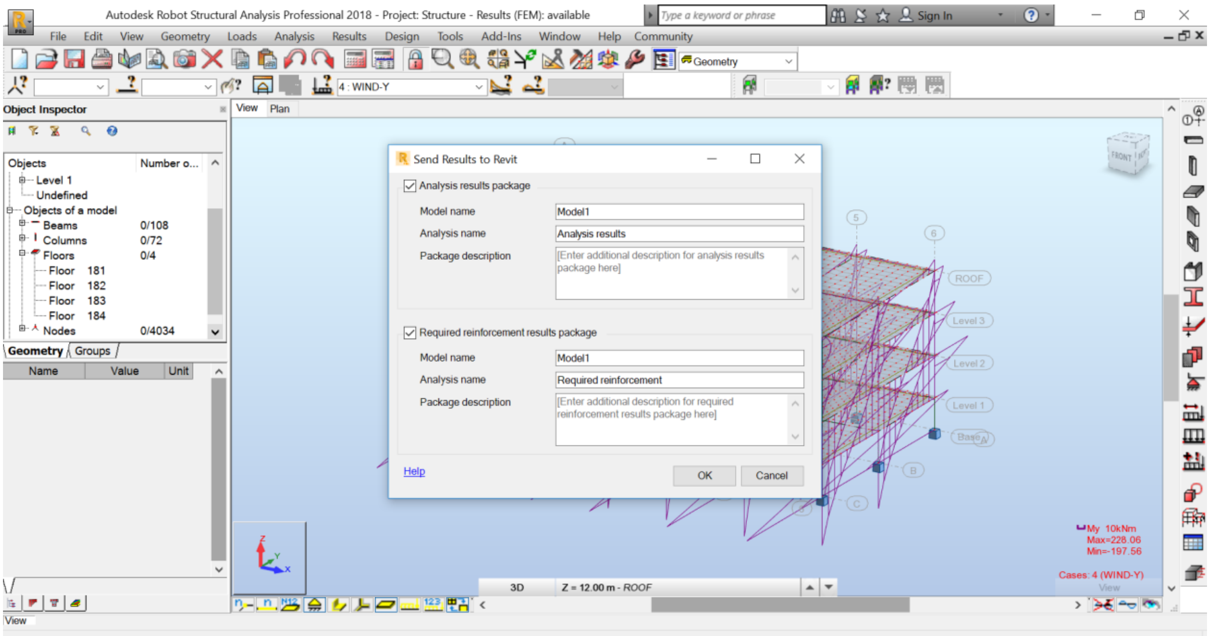


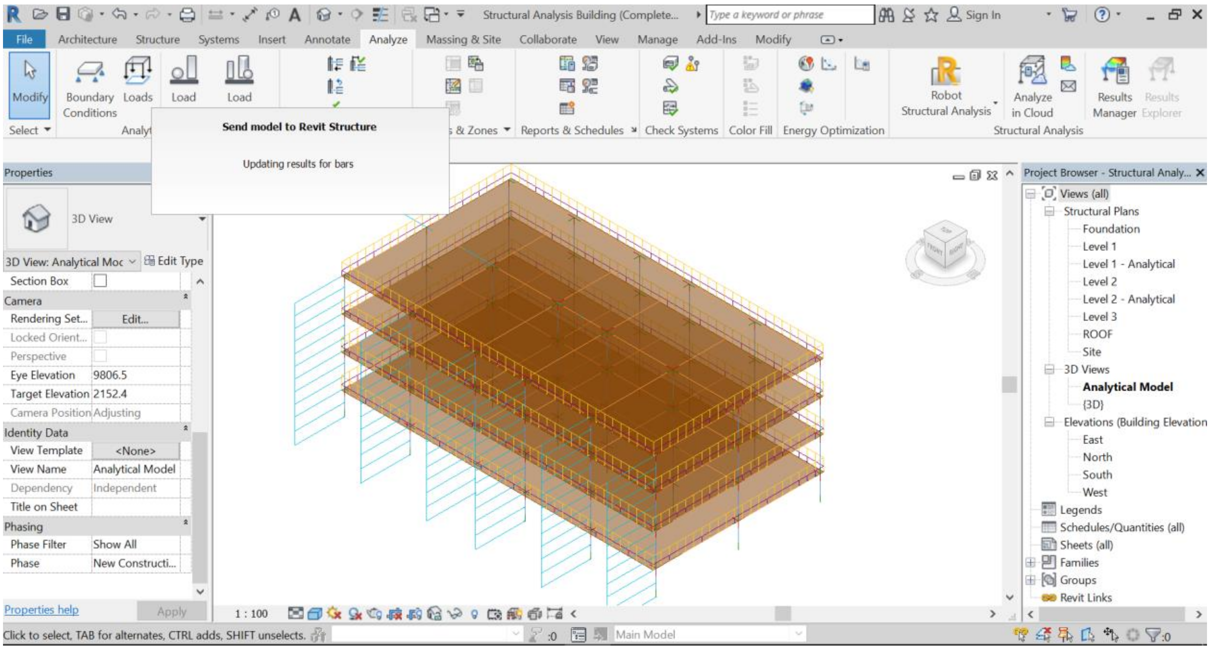
16.2.2 Sending RSA Analysis Results to Revit

1 Click Add-Ins ➢ Integration ➢ Autodesk Revit Structure. Select “Send model and result” and “Direct integration” and click “OK”.

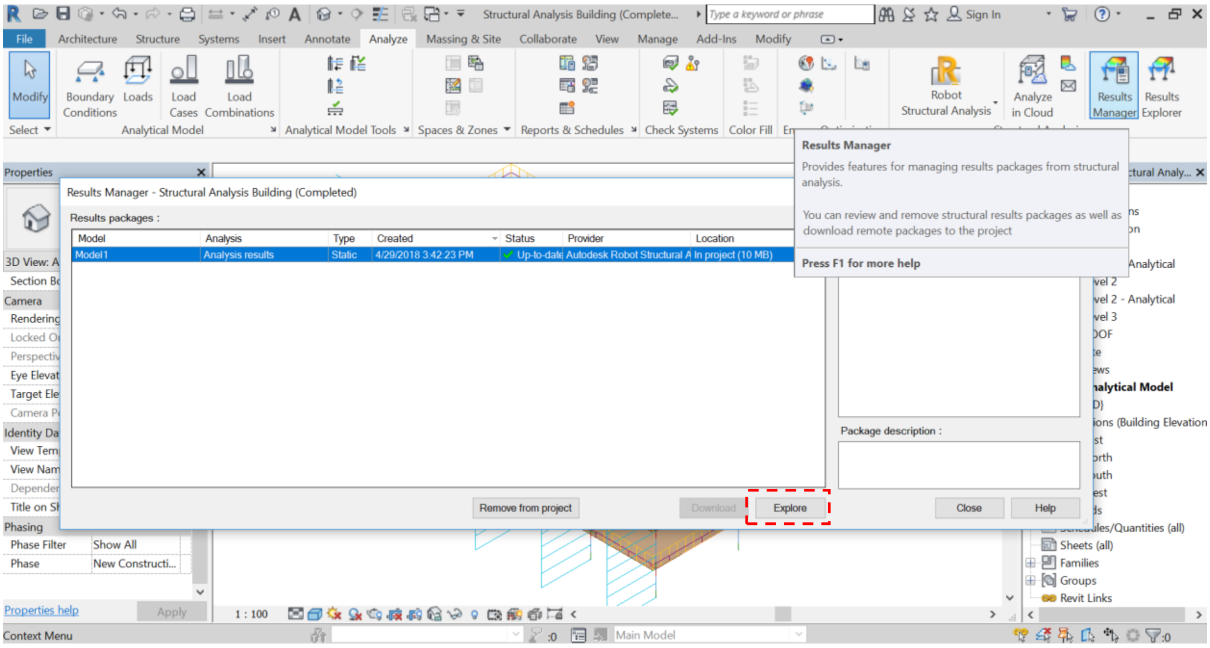


2 You can send the Analysis results and/or the Required reinforcement results.

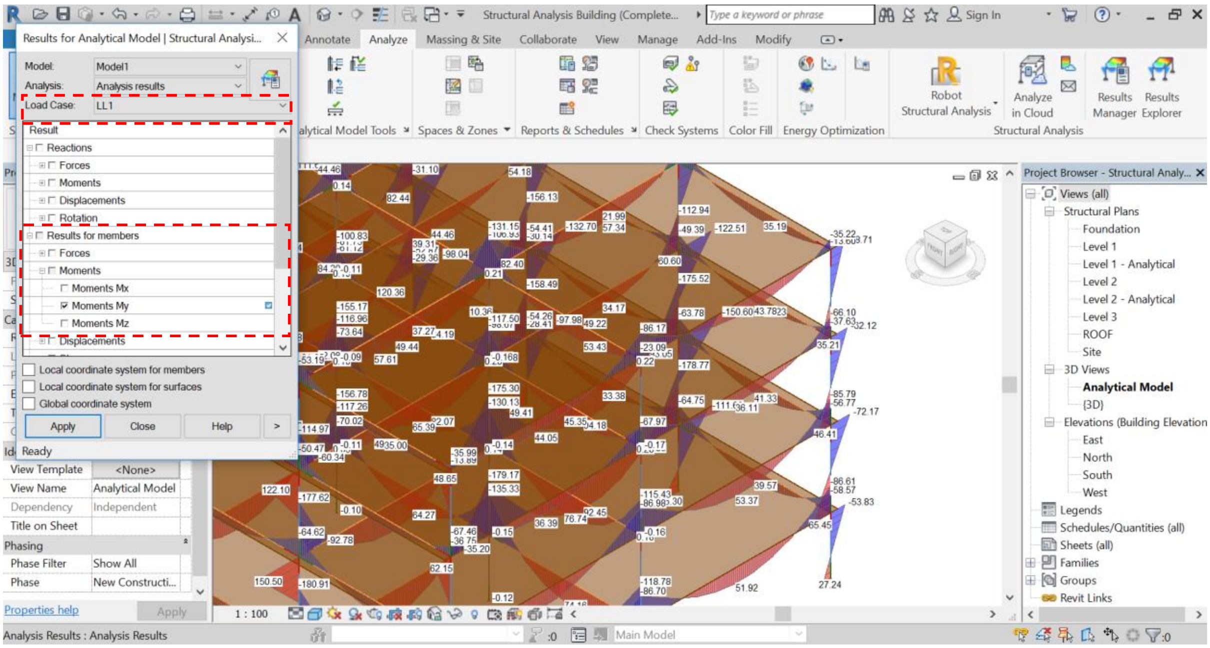




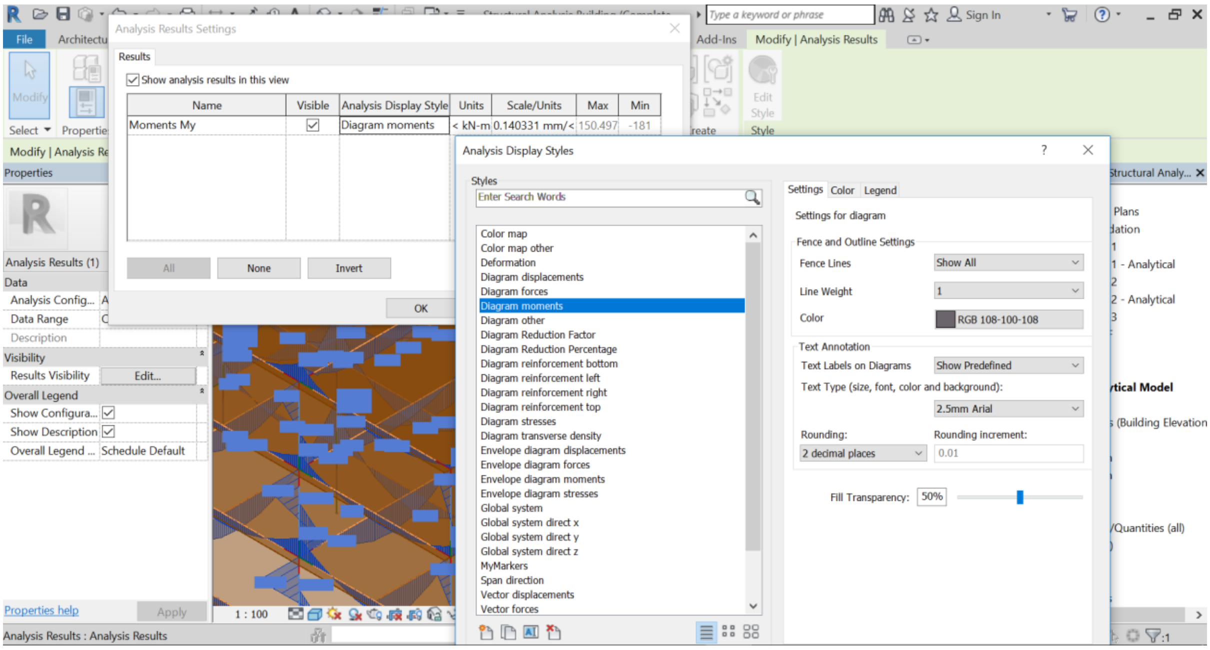
3 After sending the analysis results from RSA, go to Revit and click Results Manager from the Analyze tab. You will see the latest analysis results and you can click explore to view the results.

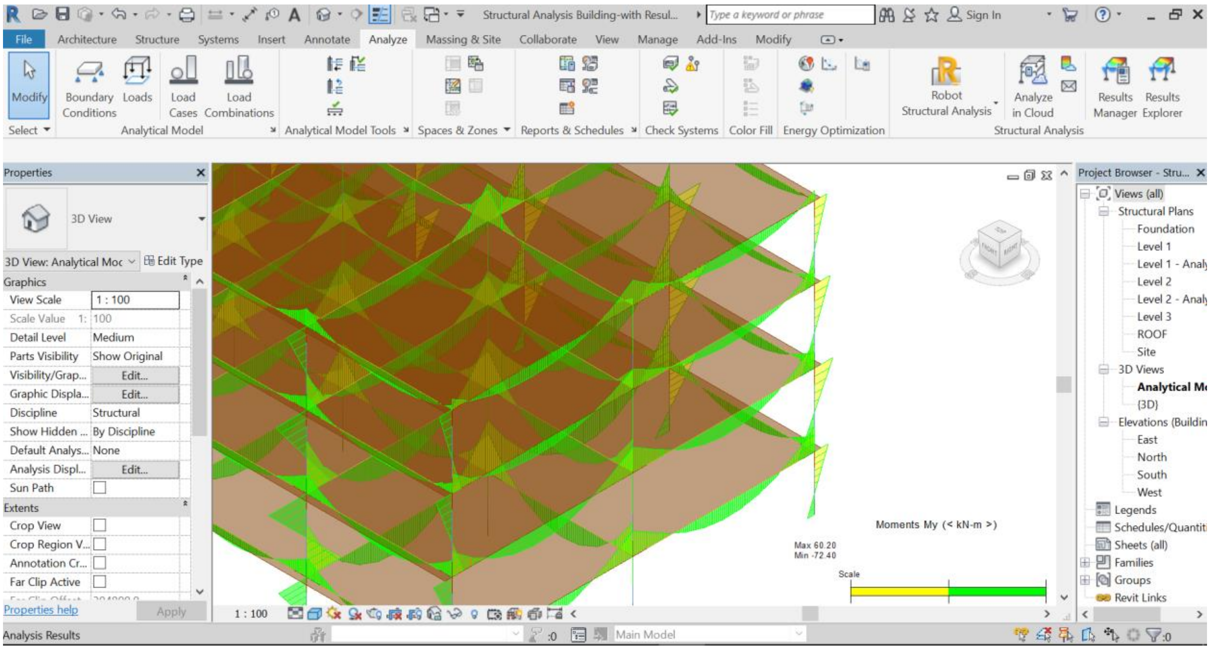


4 Select Load Case "LL1" and Results for member ➢ Moment My



5 If you select the moment diagram, you can edit its visibility setting such as Scale and Color.





16.3 Revit and other Structural Analysis Programs

16.3.1 Revit and CSI Software

Although RSA is the most compatible, effective and efficient way to link your Revit model, there are other available linkages between Revit and other structural analysis software such ETABS and S-FRAME.

For Revit and ETABS linkage, there is a CSI developed plugin called CsiXRevit that provides a bi-directional link between ETABS (also SAP2000 and/or SAFE) and Revit (2016 and above). Four (4) options are available to interchange data:

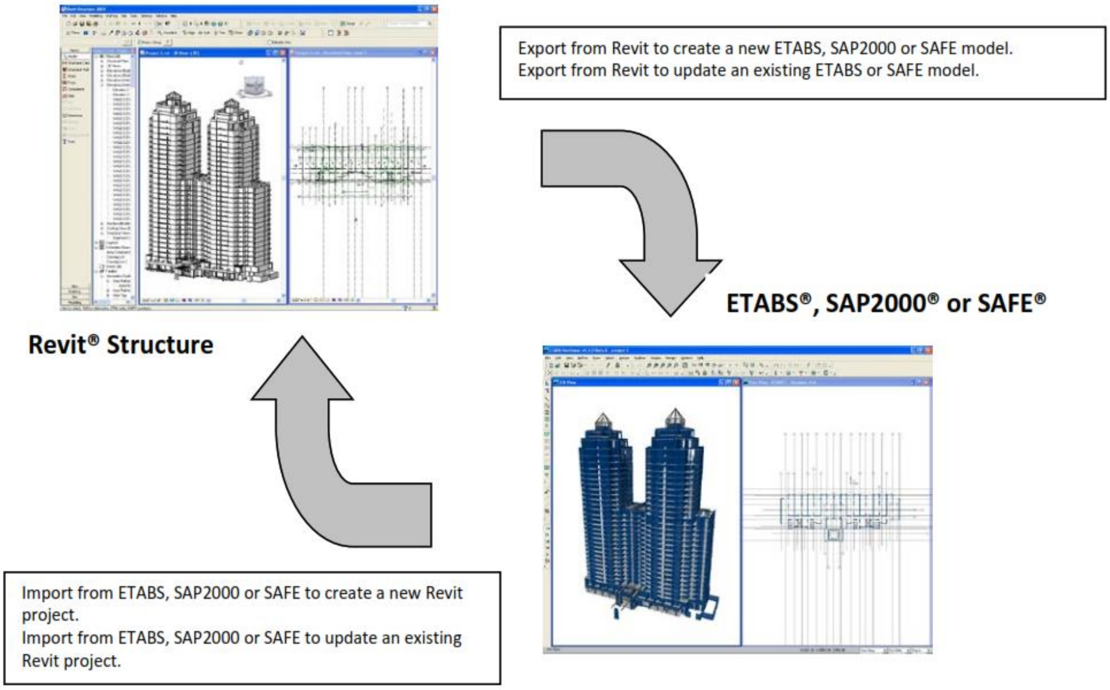
- from Revit Structure to create a new CSI model
- from Revit Structure to update an existing CSI model
- from CSI software to create a new Revit Structure project
- from CSI software to update an existing Revit Structure project

These options provide for true round-tripping of data between CSI software and Autodesk Revit® Structure.

The content below is extracted from the CSiXRevit Data Exchange Documentation (for Revit 2018).

(Source: https://www.csiamerica.com/sites/default/files/CSiXRevit_2018_Manual.pdf)

The flow of information is represented in the schematic below:



The table below provides an overview of the data imported in ETABS when exporting from Revit to create a new ETABS model:

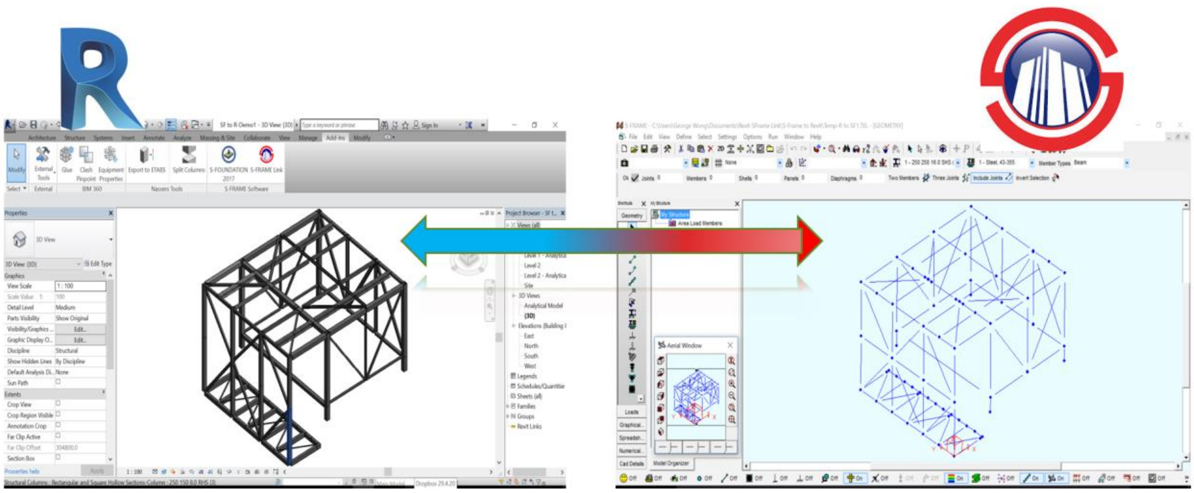
Action	Project Element	Supported	Notes
Creation of…			
	Grid Lines		
	Story Levels		
	Materials		
	Structural Columns and Structural Framing		Transfers geometry, offsets, end releases, and beam and brace alignment points into ETABS. Imports columns based on splice information in ETABS.
	Steel Column and Framing Family Types		Creates and maps equivalent ETABS frame sections. Makes Auto-Select Lists in ETABS for all families used in the Revit project.
	Concrete Column and Framing Family Types		Creates and maps equivalent ETABS frame sections.
	Walls		Slanted or warped walls not imported.
	Wall Family Types		Creates and maps equivalent ETABS wall sections.
	Wall Openings		
	Floors		Sloped floors with more than four outer corners are projected on a horizontal plane.
	Floor Family Types		Creates and maps equivalent ETABS slab and deck sections.
	Floor Openings		
	Footings		Creates fixed joint restraints in ETABS wherever a footing occurs in Revit.
	Point / Line Loads		
	Area Loads		Non-uniform area loads not imported.
	Load Cases		Creates both an ETABS load pattern and load case for each Revit load case.
	Load Combos		

You can see that the link is not fully compatible, care must be taken in using it. Critical items should be checked after export / import. These checking should include but not limited to

- Critical loads values
- Material and Section properties
- Local axes direction of critical members
- Connectivity of secondary members
- Loads transfer from slabs to supporting beams/walls

16.3.2 Revit and S-FRAME Analysis Software

The software S-FRAME Analysis is another popular structural analysis software in North America. It has a free bi-direction link provided for transferring models with Revit. Similar to ETABS' CSiXRevit link, the S-FRAME and Revit link is not fully compatible. However most of the information needed for structural analysis can be transferred from and to Revit seamlessly.



CHAPTER 17 - DRAWINGS PRODUCTION in REVIT

One of the objectives of using Building Information Modelling (BIM) is to share information seamlessly across all processes of the building life-cycle. Information shared from design to drawings is one of these key information transfer processes. In Revit, all 3D elements modelled are parametric and drawings of the building model composed of these elements can be produced by putting different views into sheets. As such, any change in detail of the Revit model will be reflected in the drawings automatically and this is one of the ultimate goals of using Revit to avoid loss of information when date is transferred from design output to drawings production.

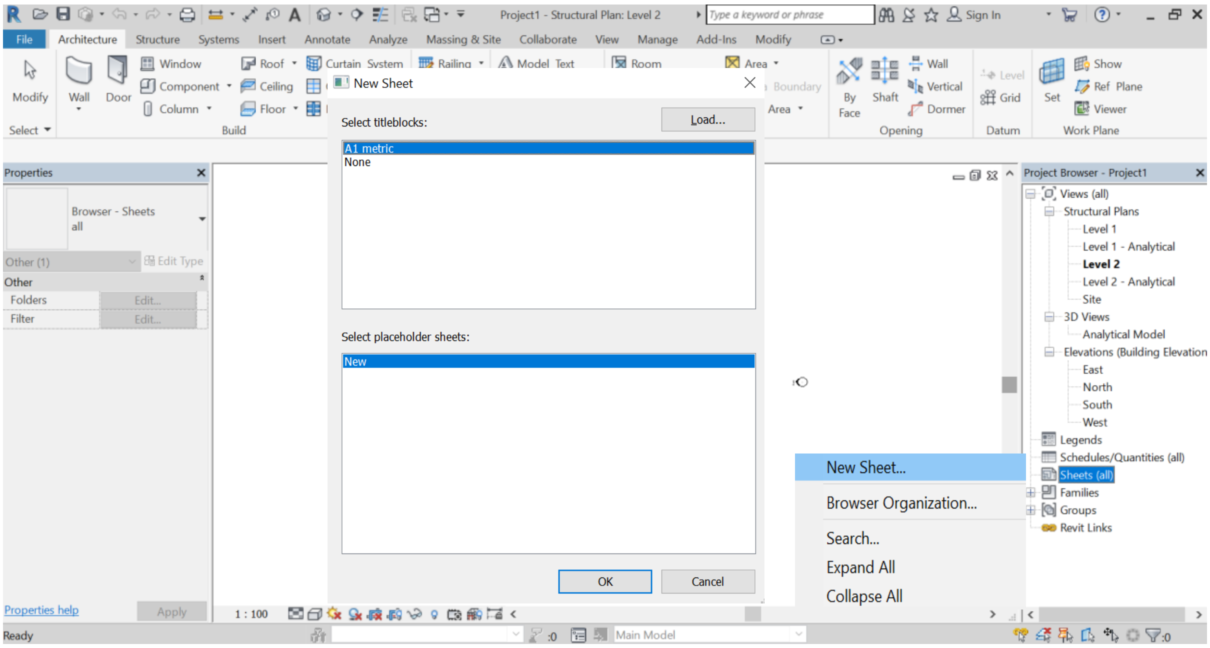
In this section, you will learn

- How to create drawing sheet and customize a title block in Revit.
- How to create General Arrangement (Framing Plans) from Revit model.
- How to create Sections Drawings from Revit model.
- How to create R.C. Details Drawings from Revit model.

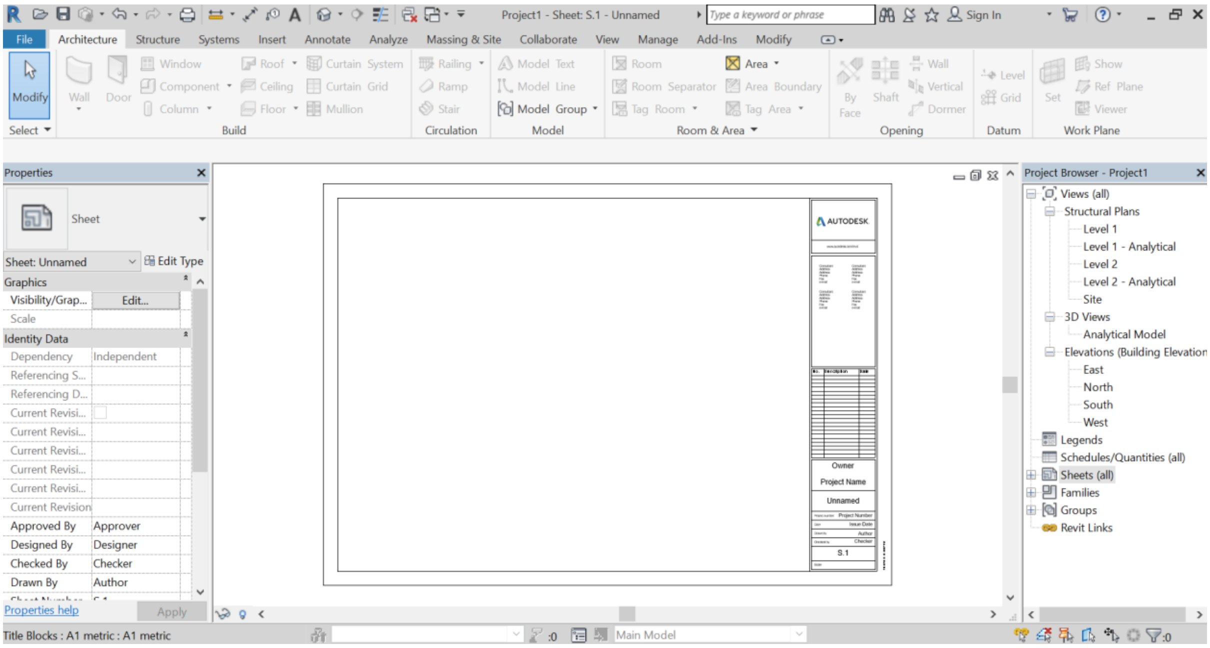
17.1 Creating Drawing Sheets in Revit

The simplest way to create a drawing sheet in Revit is to use its available template and modify it to suite your use.

1 Select “Sheets (all)” from the Project Browser. Then right click and click “New Sheet…”.

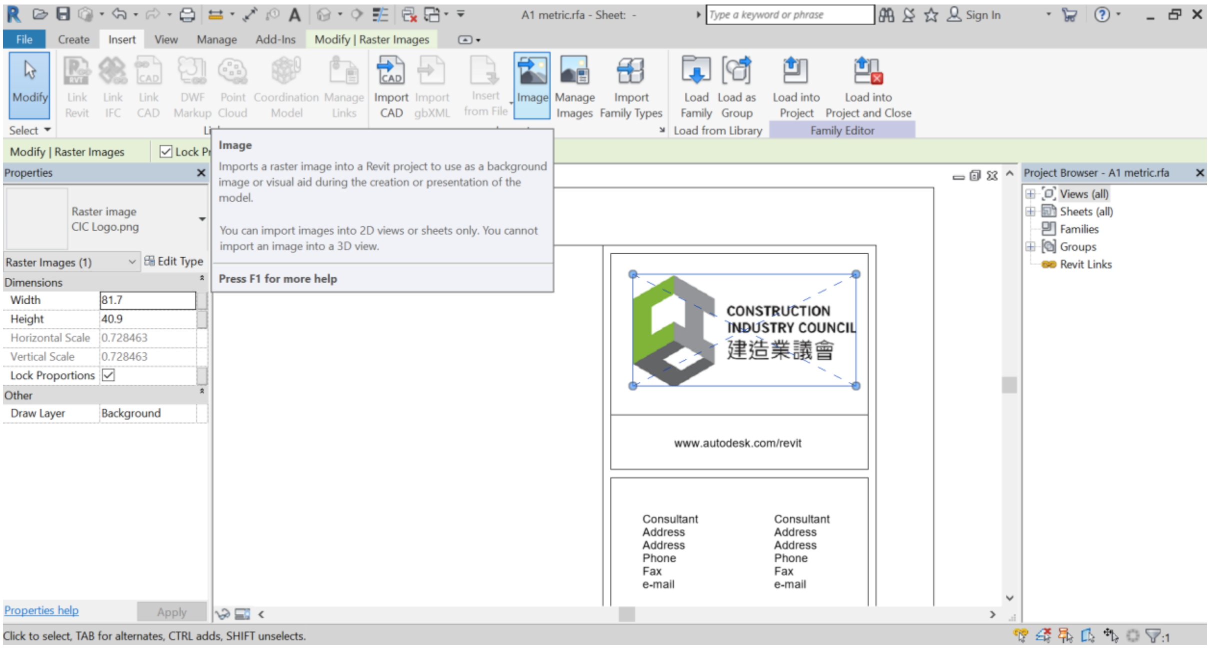


2 Select A1 metric and click “OK”.

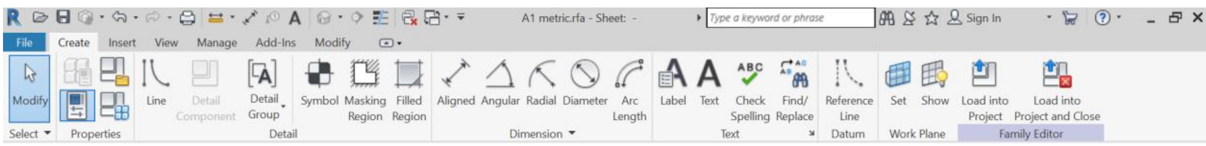


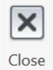
3 Double click on the sheet to edit it. Note that now you are in Family Editor and any change made will affect all sheets of the same type created before and in the future.

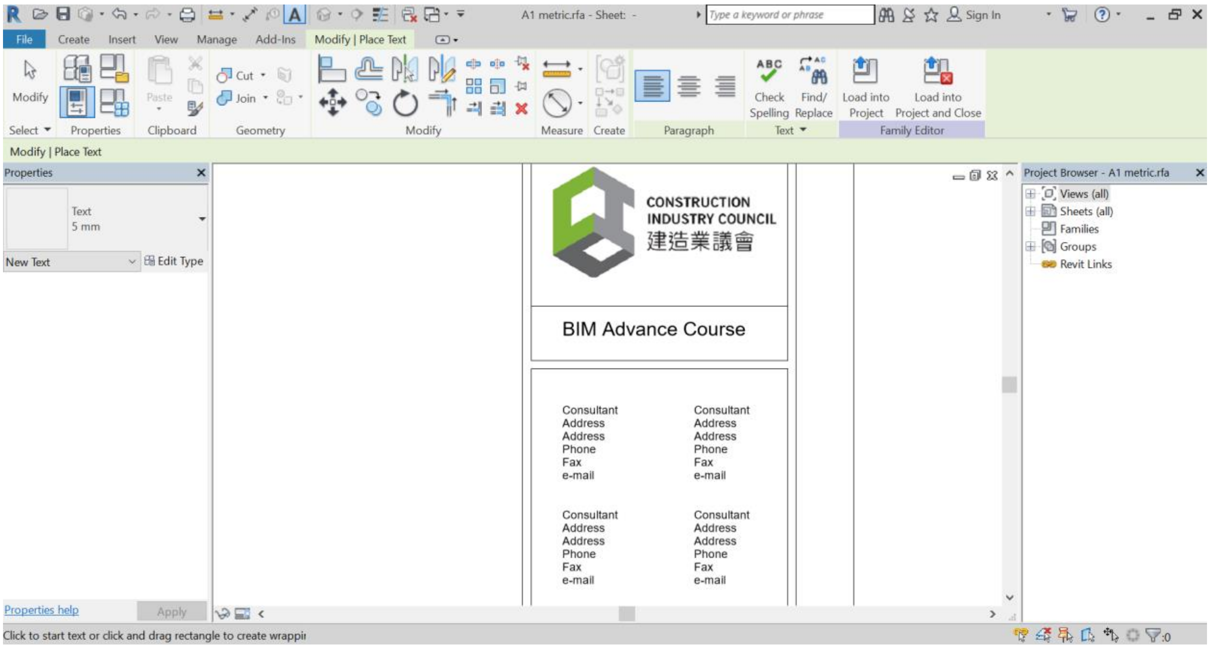
4 Now delete the Autodesk logo and insert a CIC logo (can be found in your course folder).



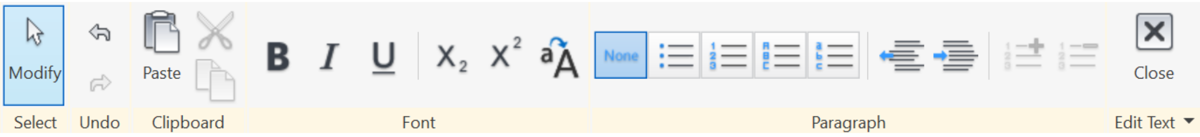
5 From the Create tab, you can create many components such as Line, Text and Label to customize your title block.



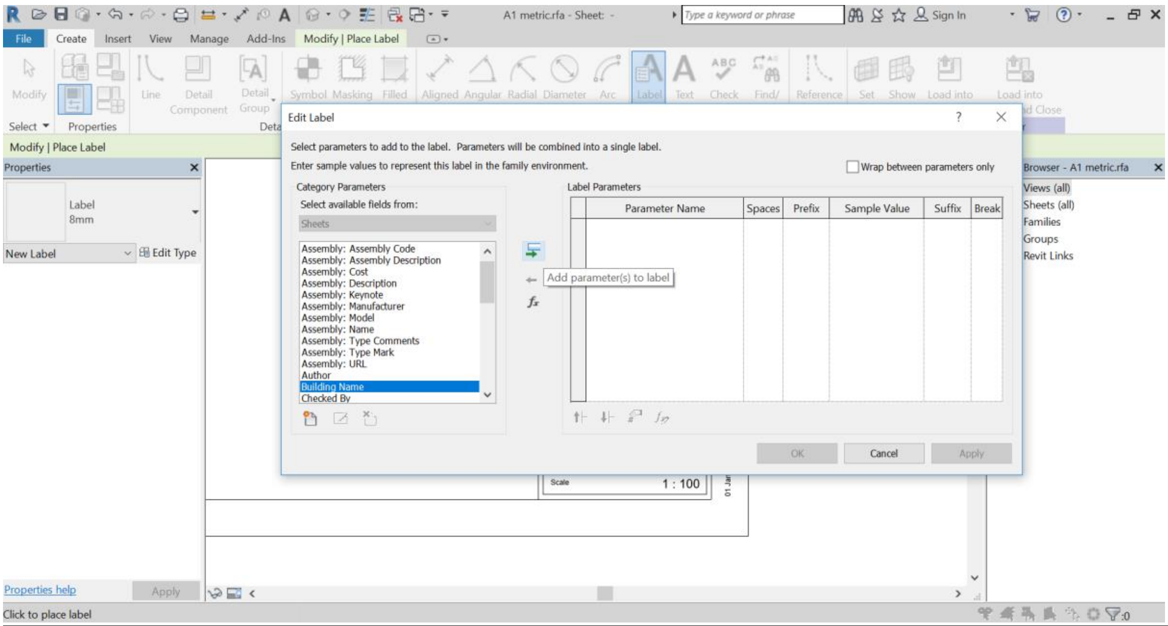
6 Select and delete the text "www.autodesk.com/revit". Then click Create tab ➢ Text panel ➢ Text, and choose "5mm" from type selector. Pick a location inside the box under the CIC logo and type in "BIM Advance Course". Click "Close"  to add the text.



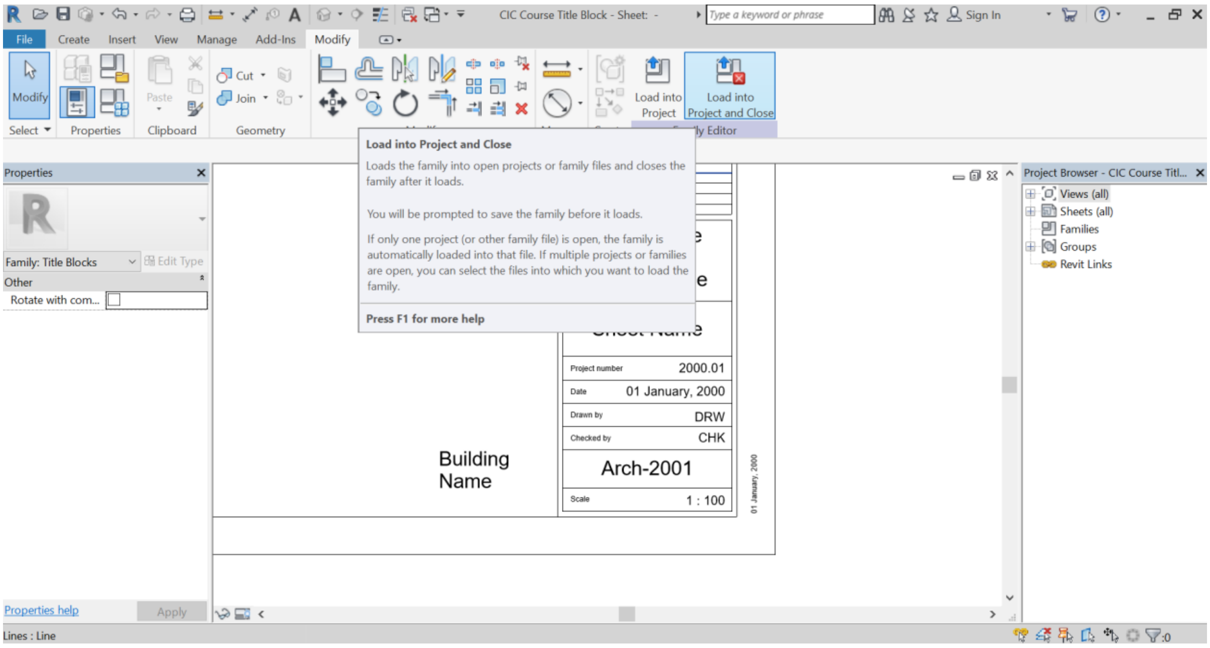
(Notice that there are many tools available to help you to format the text.)



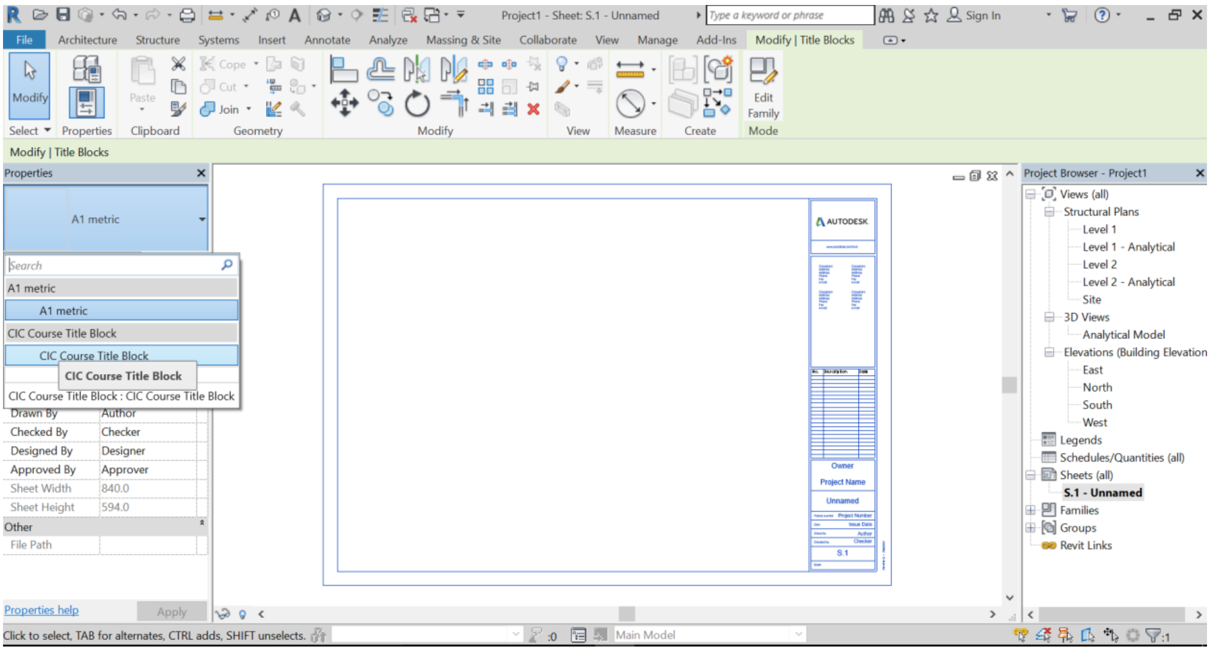
7 You can also include information specific to the project as "Label" in the sheet. The advantage of this is that you only need to change the information once and Revit will automatically update all drawings that contain such information. To insert a label into a sheet, click Create tab ➢ Text panel ➢ Label. Then select "Building name" and click "Add parameter(s) to label". Click "OK".



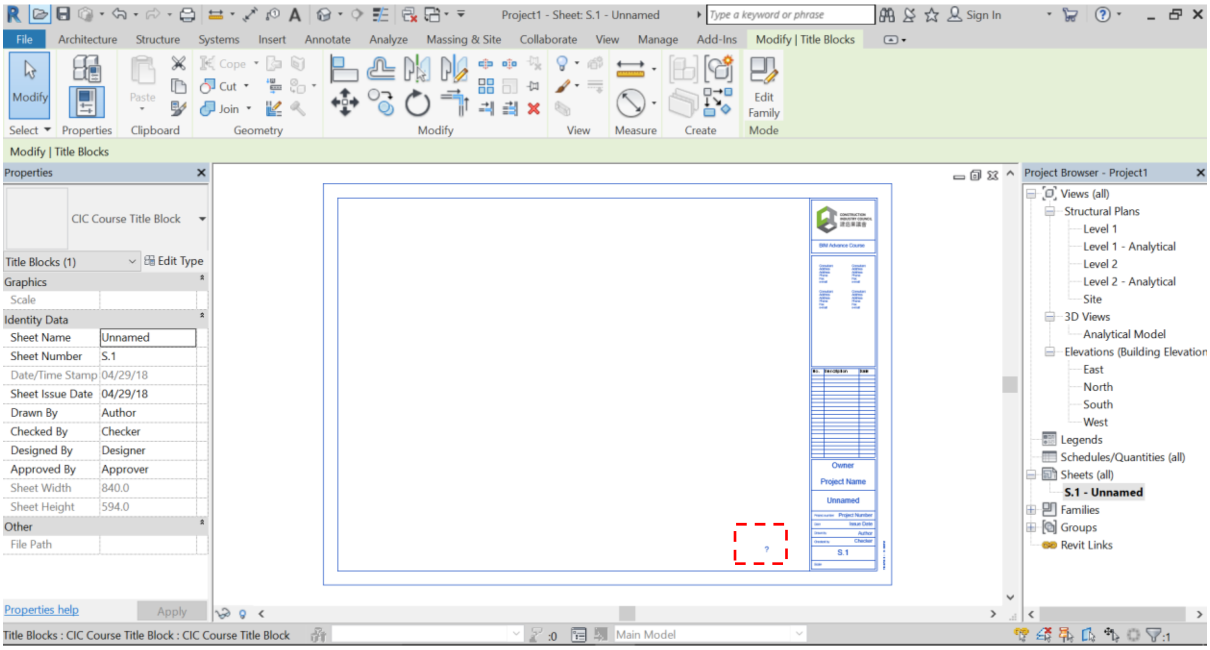
8 Now save this title block as a new family. Click File ➢ Save as ➢ Family, and name it as "CIC Course Title Block". Next click "Load into Project and Close".



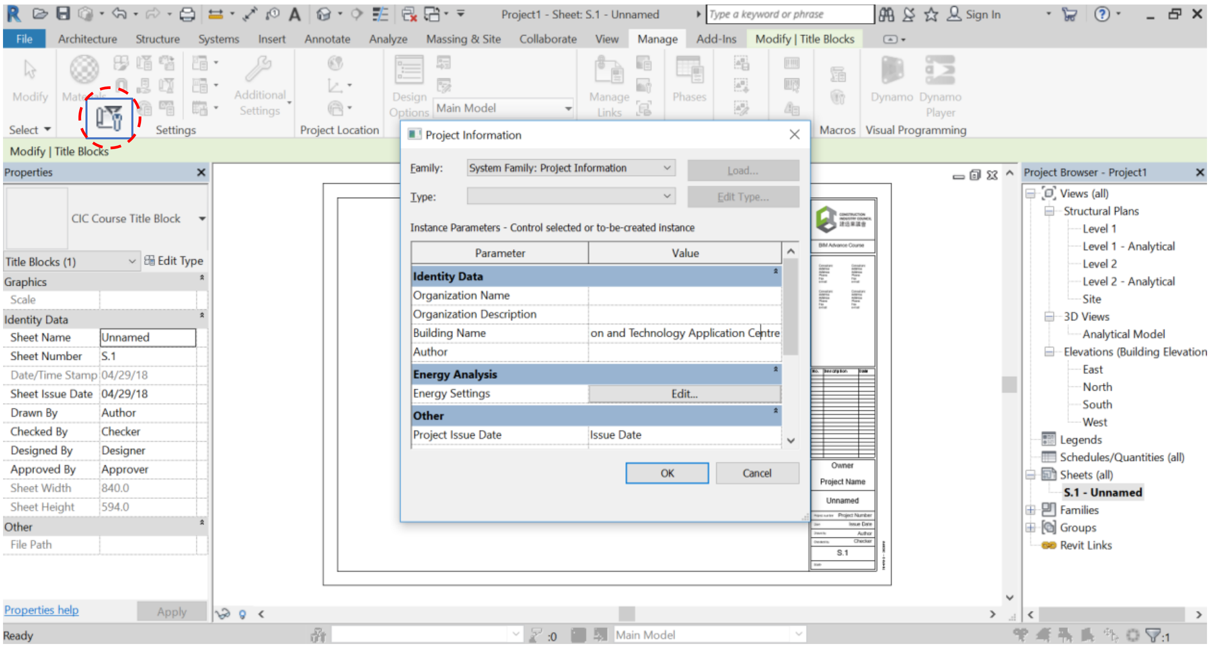
9 Go to the sheet view again and select the title block. Now you can choose the CIC BIM Course Title Block from the type selector.

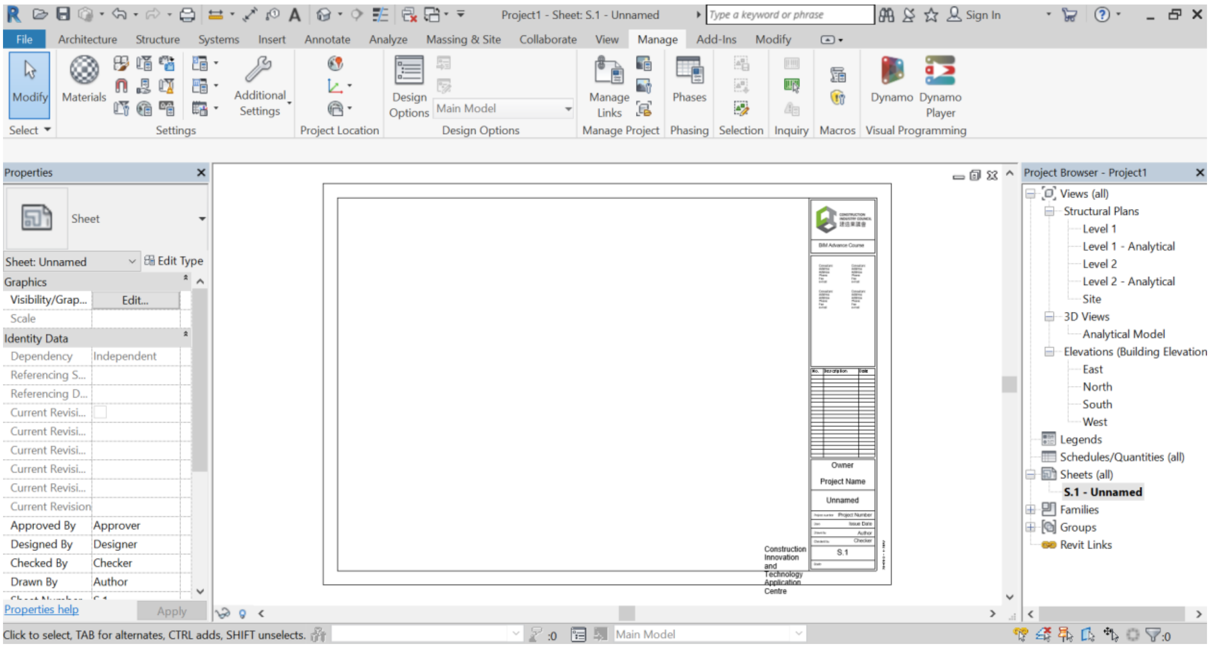


10 Select the CIC Course Title Block and you have your customized title block to prepare your drawings. You will notice a "?" mark when you select the title block. This is because the label information, in this case "Building name", has not been input for this project.



11 To input project information, click Manage tab ➢ Settings panel ➢ Project Information. Type in “Construction Innovation and Technology Application Centre” for the Building Name.





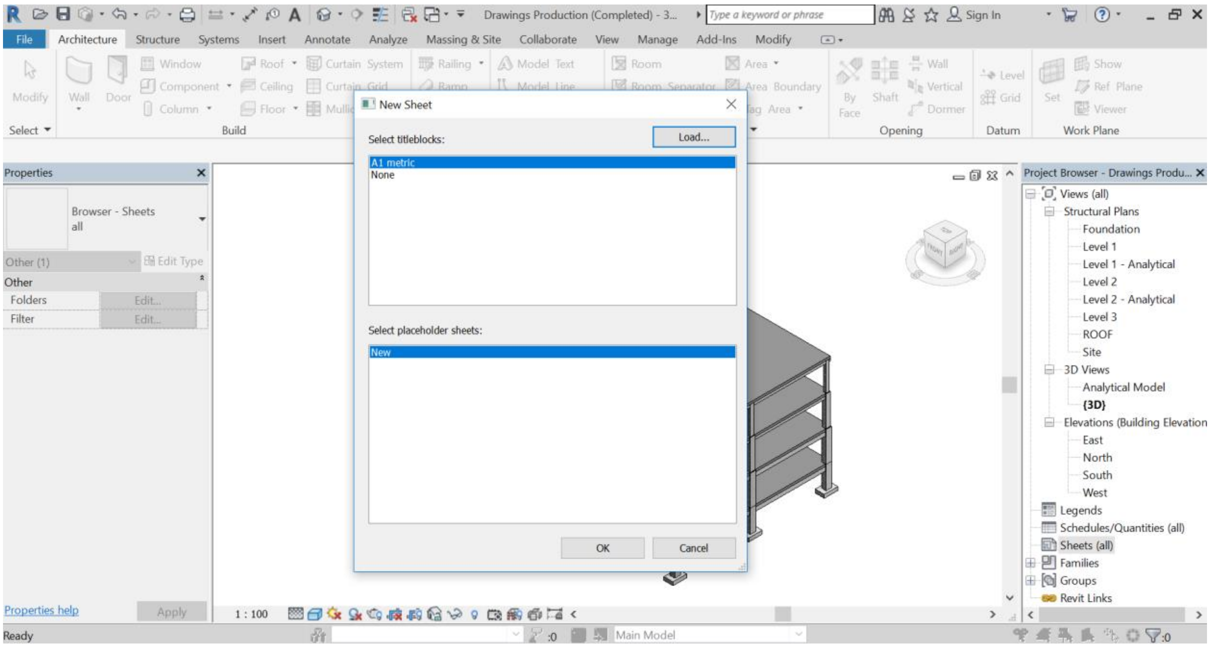
You can align the label to a better position by double clicking on the title block to open the family editor.

17.2 Creating General Arrangement / Framing Plans in Revit

To add General Arrangement / Framing Plans in Revit,

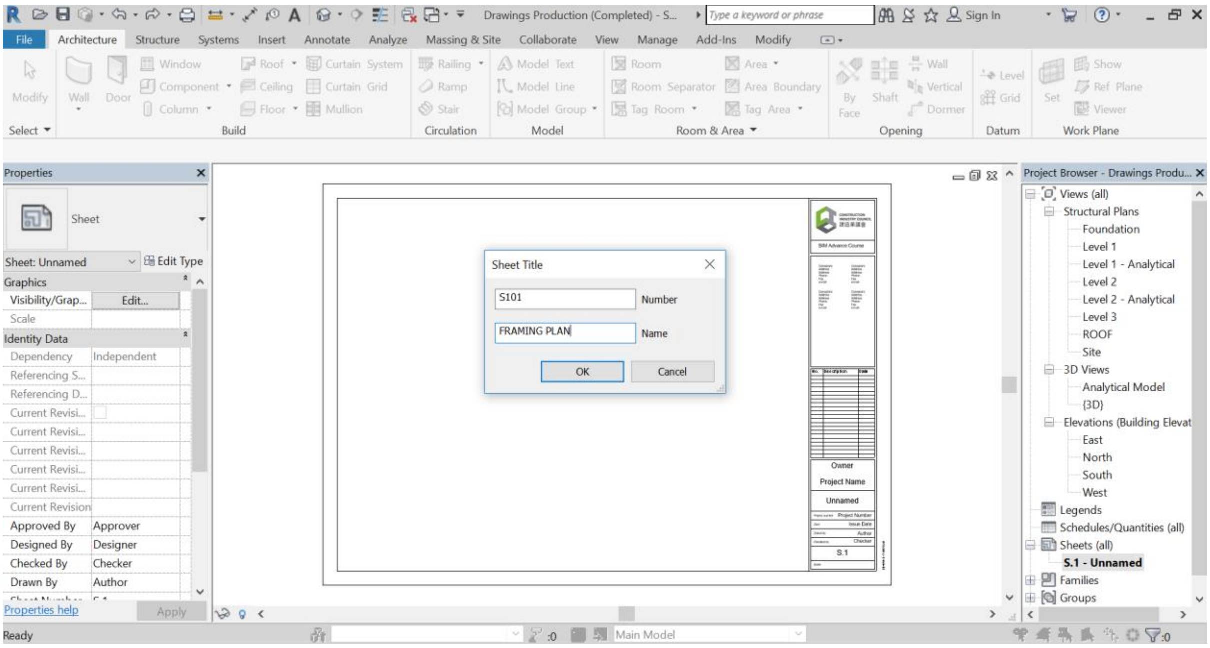
1 Open from the course folder the “Drawings Production.rvt” file.

2 From the Project Browser, select “Sheets (all)” and right click to add new sheet.

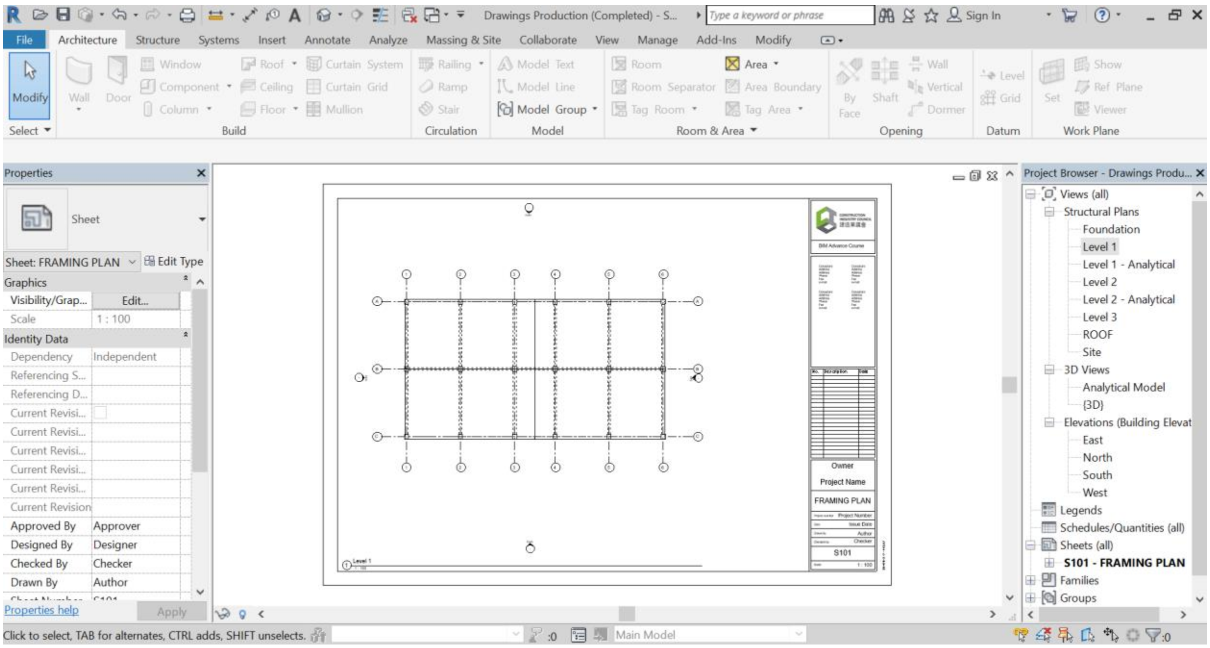


3 Click “Load” and find the “CIC Course Title Block” family (which you should had saved in the course folder) to load it into the project.

4 Select the newly created sheet (S.1 - Unnamed) and right click to rename it. Change the (Sheet) Number to “S101” and rename it to “FRAMING PLAN”. Click “OK”.

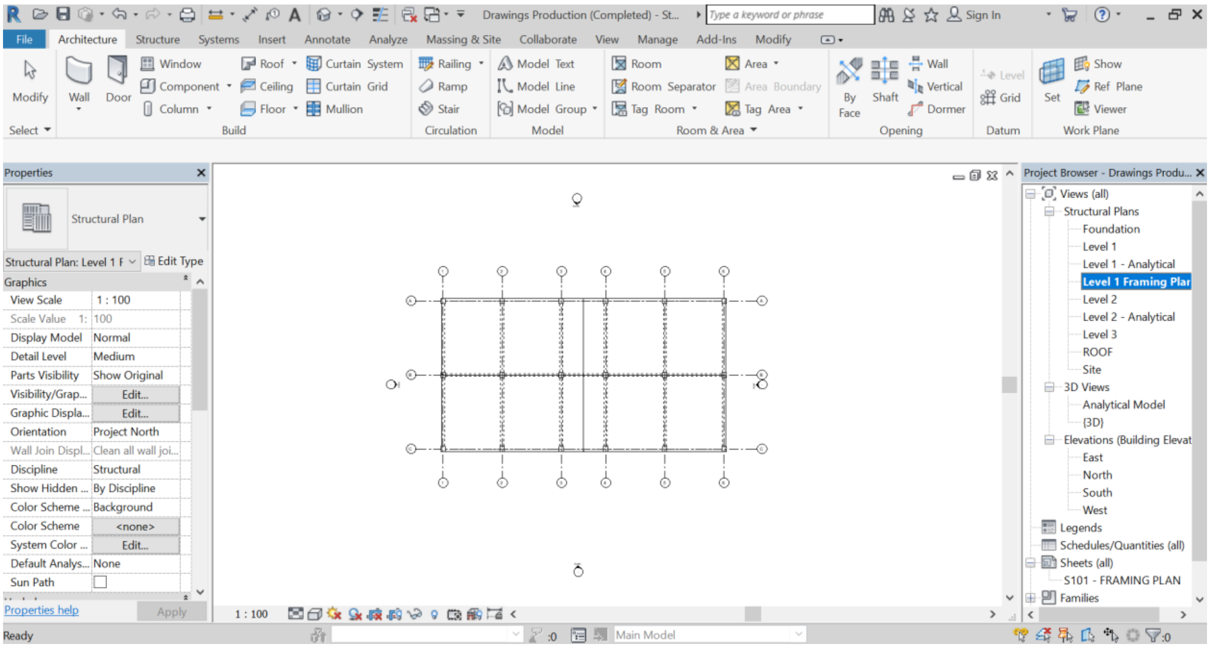


5 Now click on “Level 1” (Structural Plan) and press down the left mouse key then drag it into the sheet area. You have just created a framing plan drawing.



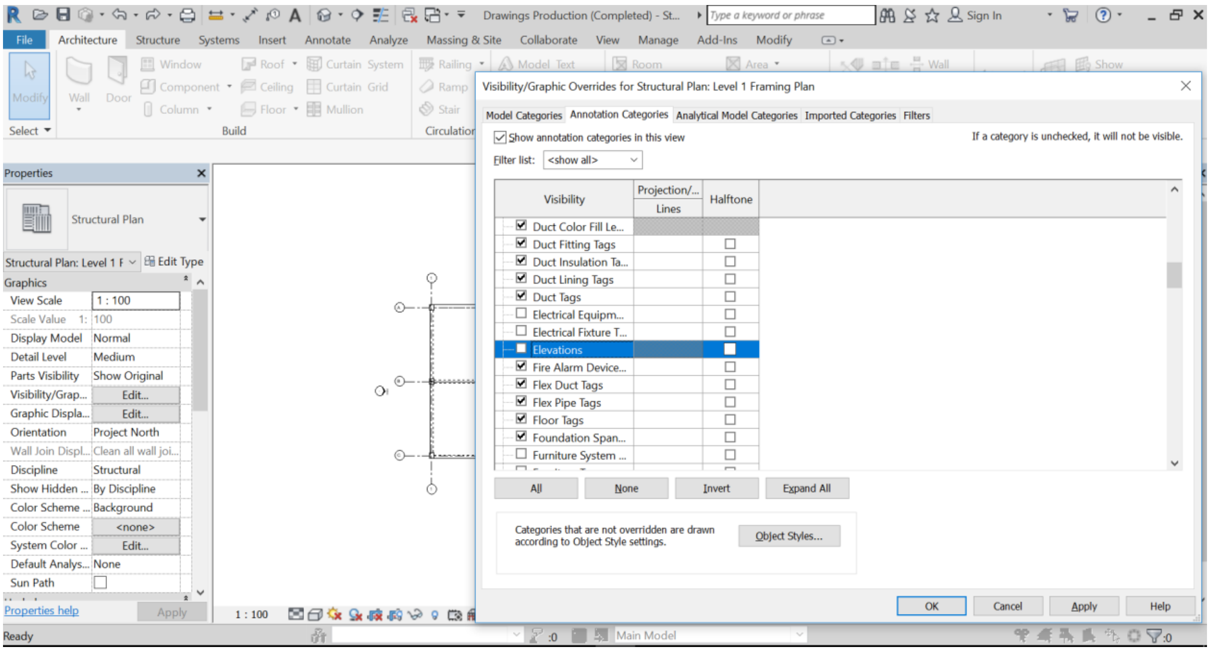
6 The framing plan drawing just created, however, need to be refined for submission to authority. This will involve some visibility settings to control how the framing plan looks like. Since one view can only be place into one sheet, therefore a more proper way to do this is to use a duplicated view instead of the original Level 1 structural plan.

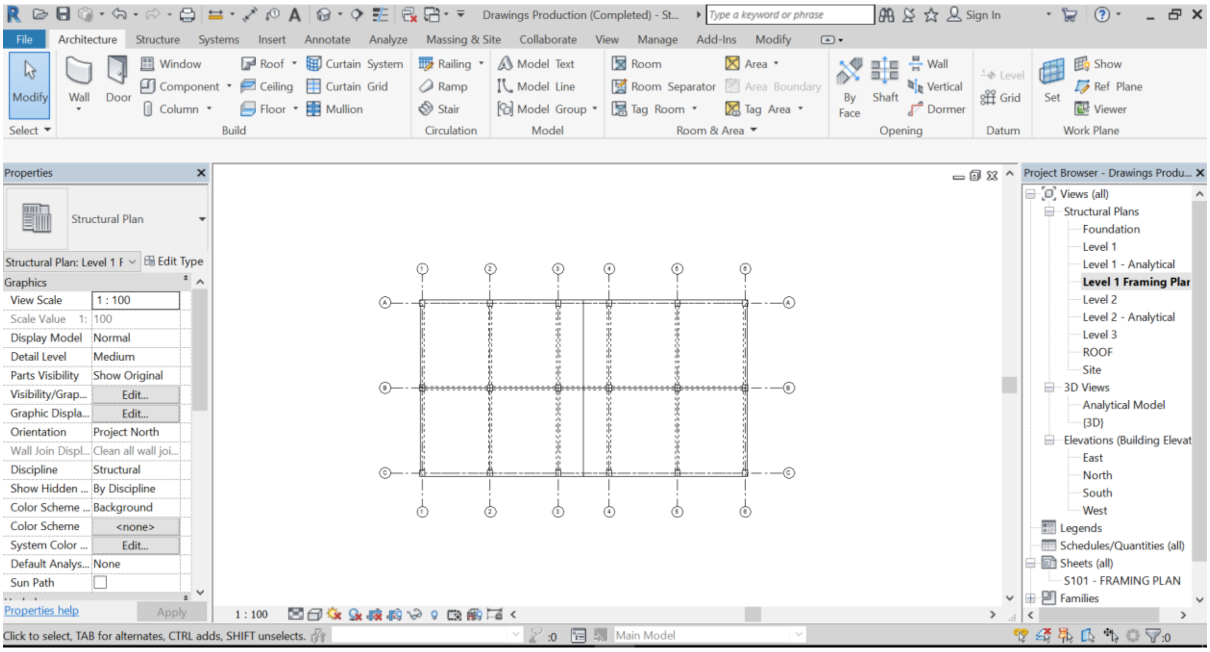
7 Delete the level 1 view from the S101 drawing. Then click on Level 1 (Structural plan) and right click to Duplicate View ➢ Duplicate with Detailing. Rename this duplicated view to “Level 1 Framing Plan”.



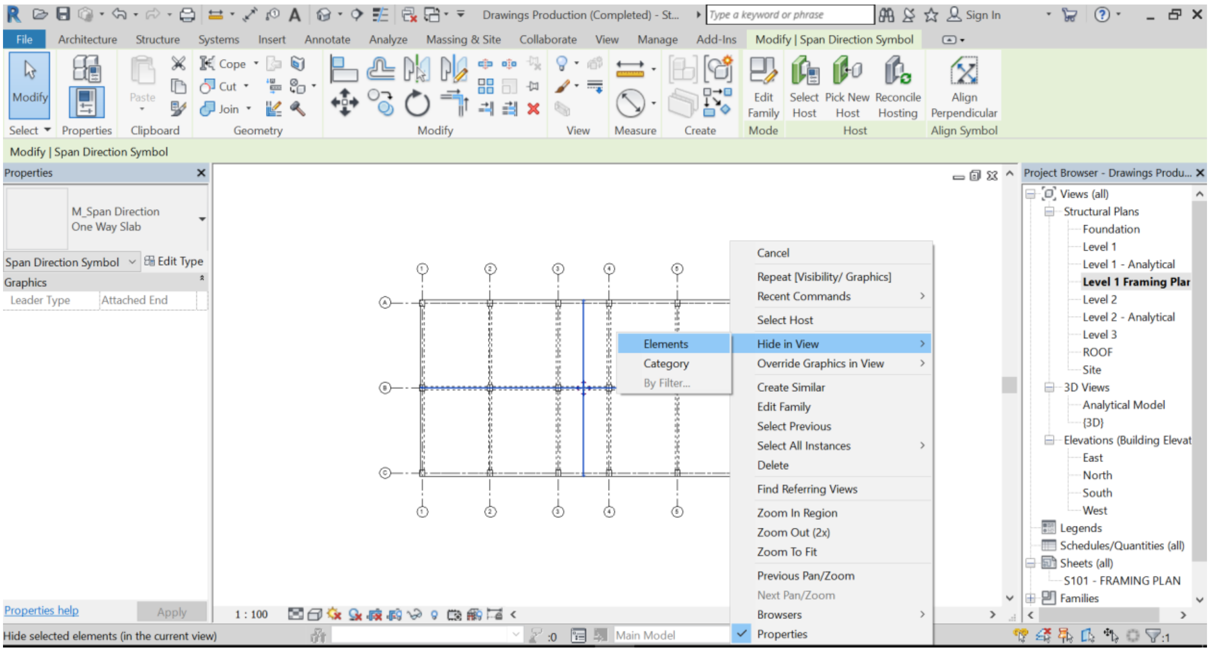
8 Using this "Level 1 Framing Plan" view, you can hide details that you do not want to be shown. There are two ways to hide details. First way is by controlling an element's visibility.

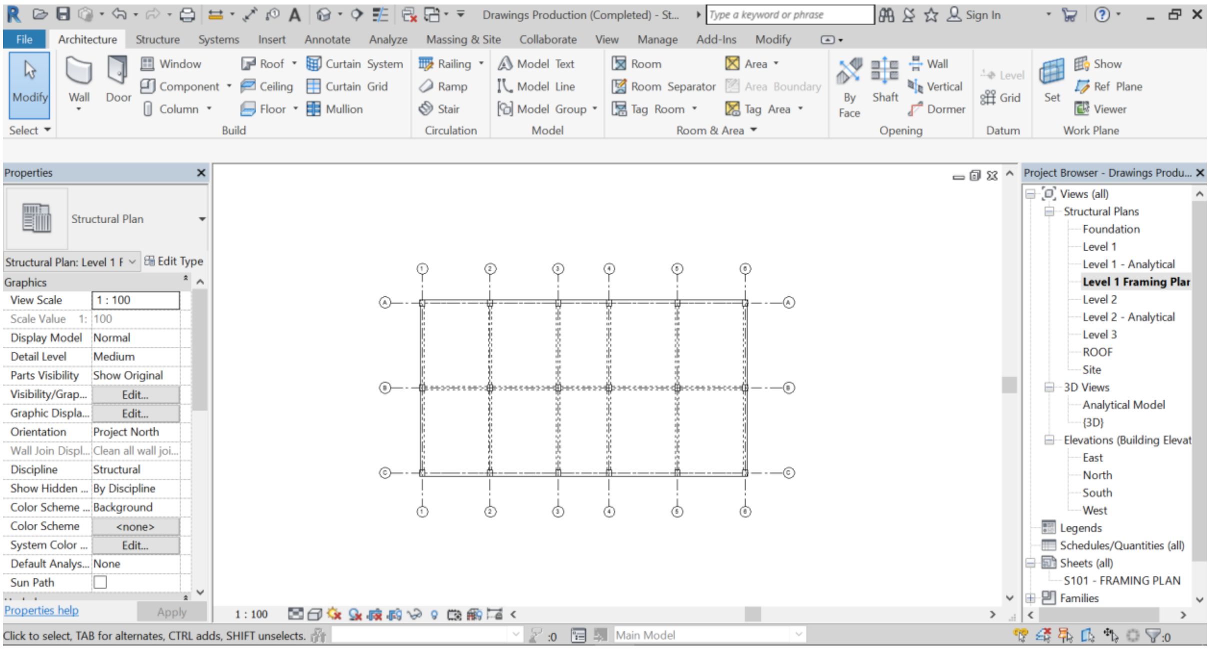
Type "vv" for Visibility/Graphics control. In the "Annotation Categories", unselect "Elevations". Then you will not see any elevation symbol in this view.





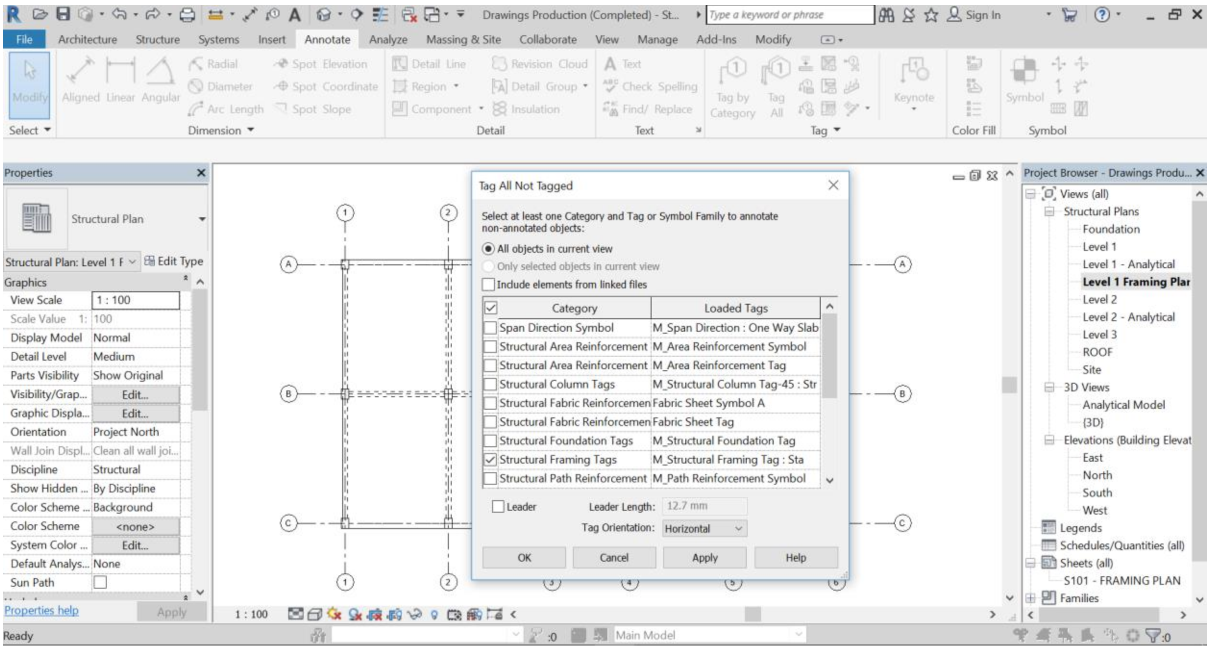
The other way to hide an element in the view is to select that element then right click and "Hide in View" ➢ Elements.

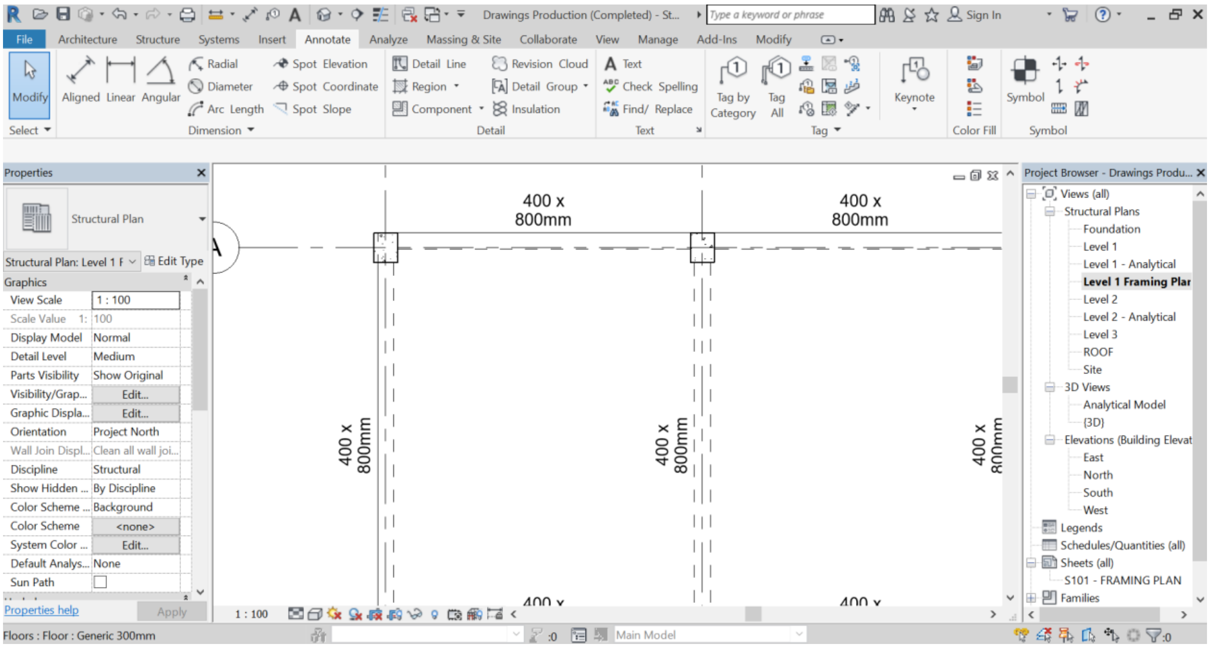




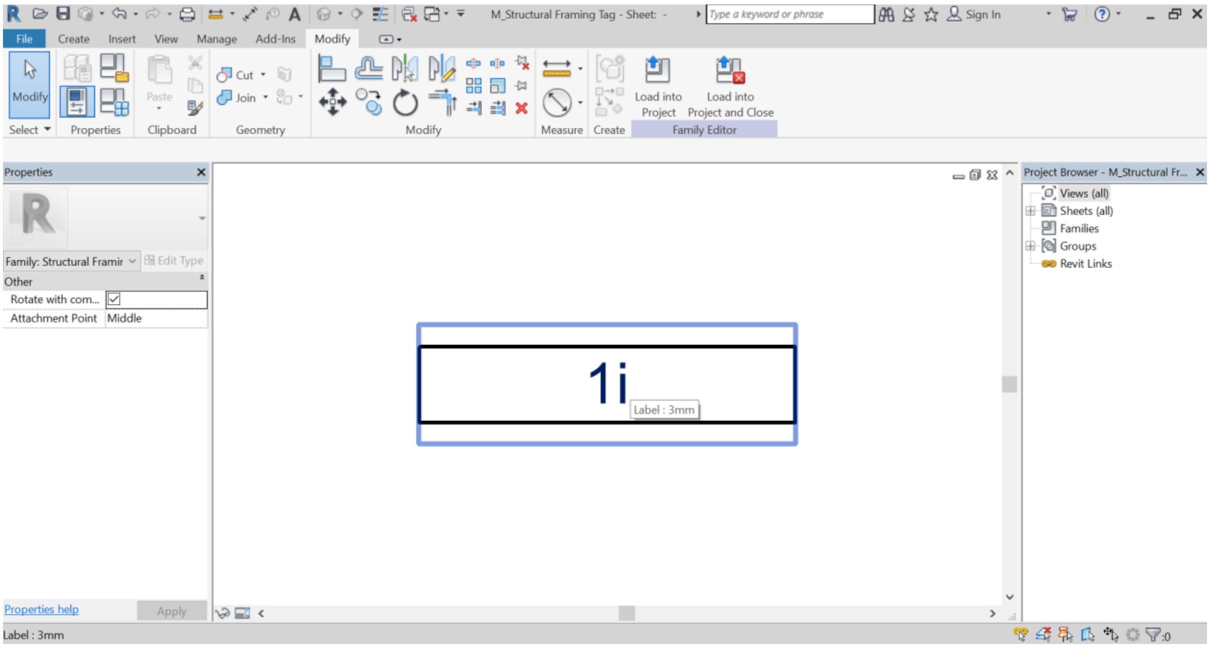
9 Next, to add beam sizes on the framing plan.

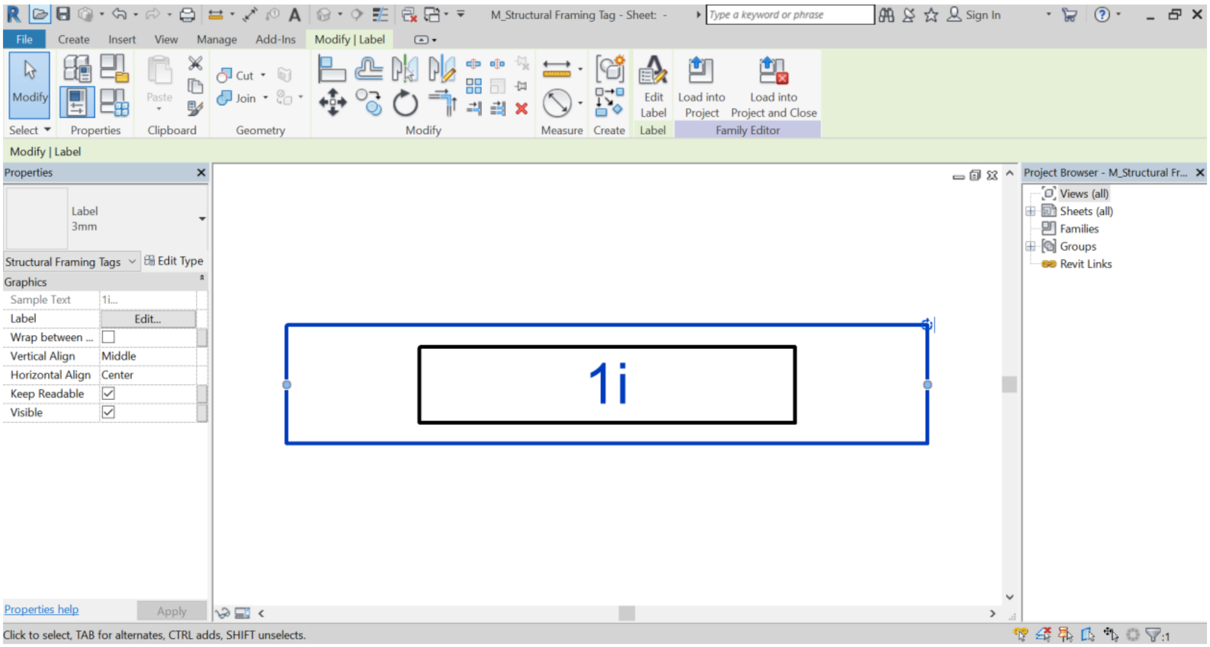
Click Annotate tab ➢ Tag panel ➢ Tag All. Select "All objects in current view" and "Structural Framing Tags" and click "OK".



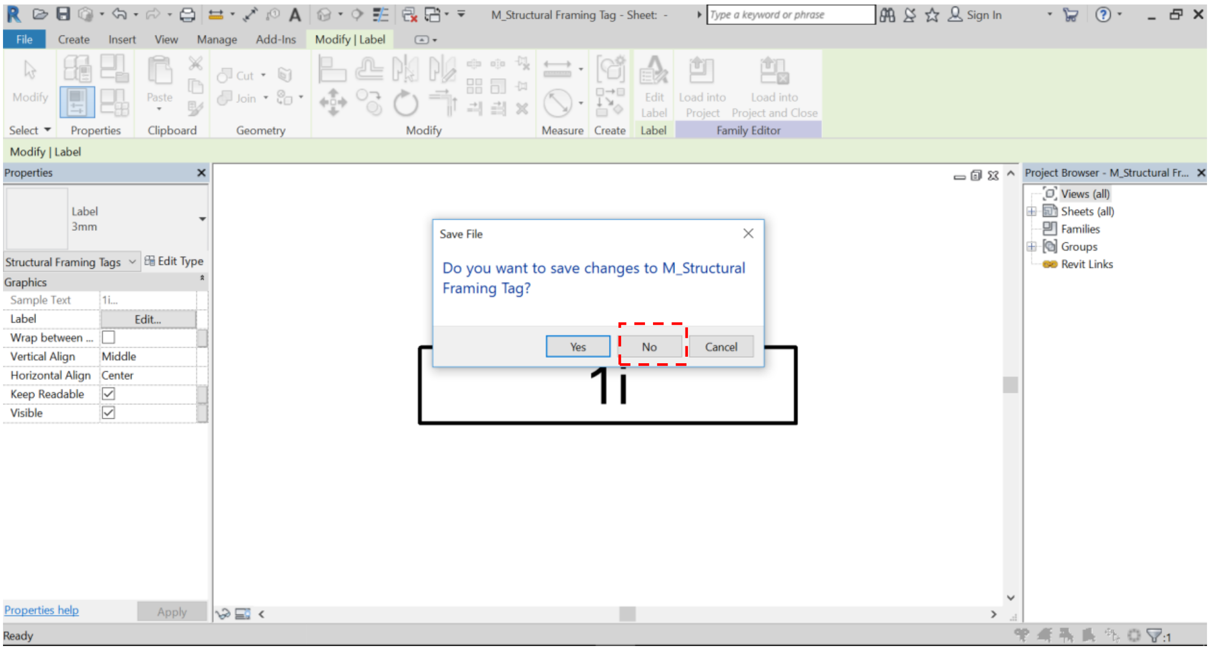


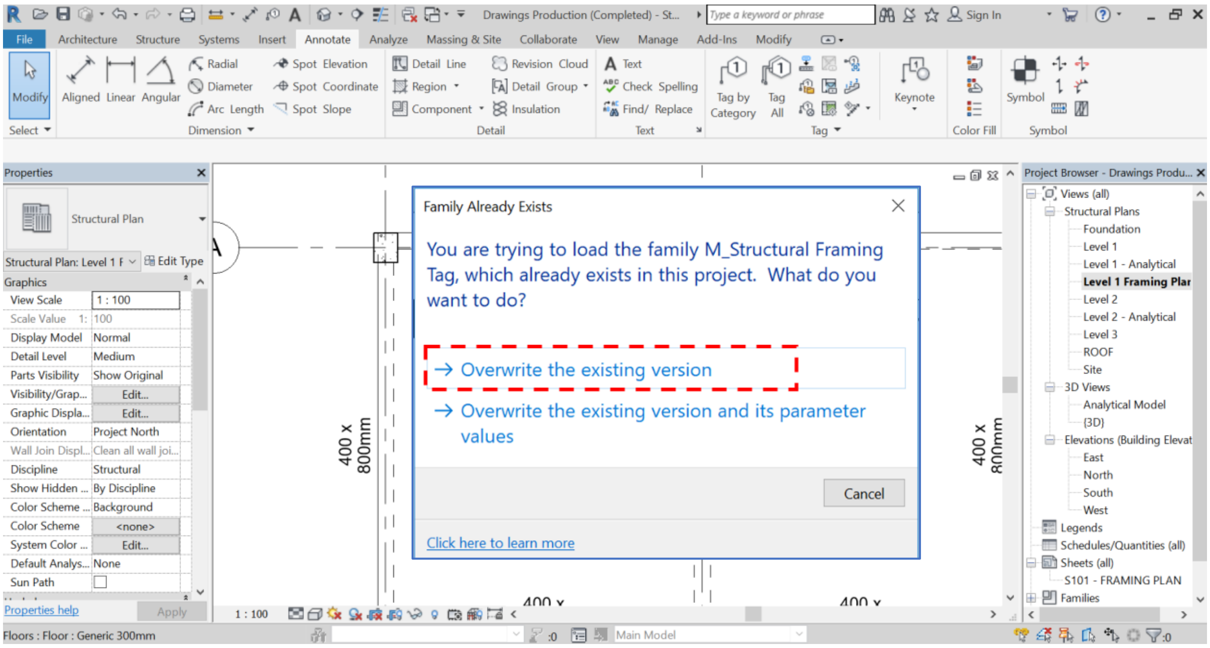
10 The text does not display nicely. You can double click on it and edit the family. Select the Label and drag the boundary longer then "Load into Project and Close".

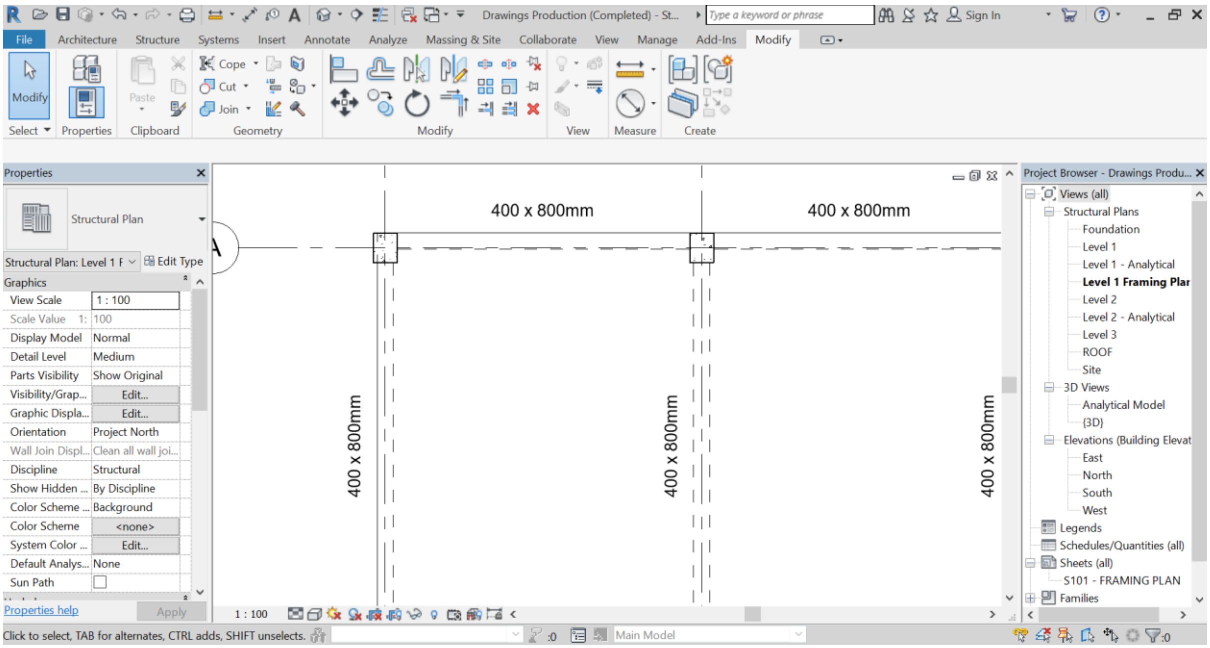




No need to save change to “M_Structural Framing Tag.rfa”. Then “Overwrite the existing version”.



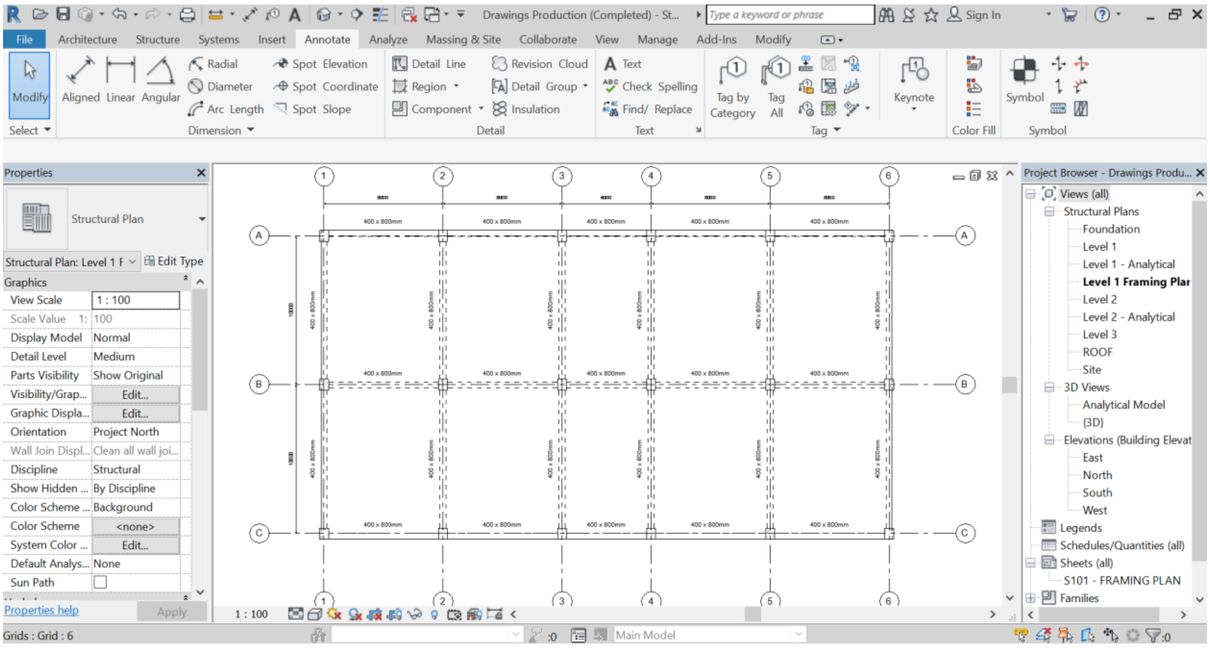




11 Lastly, add some dimensions to the plan.

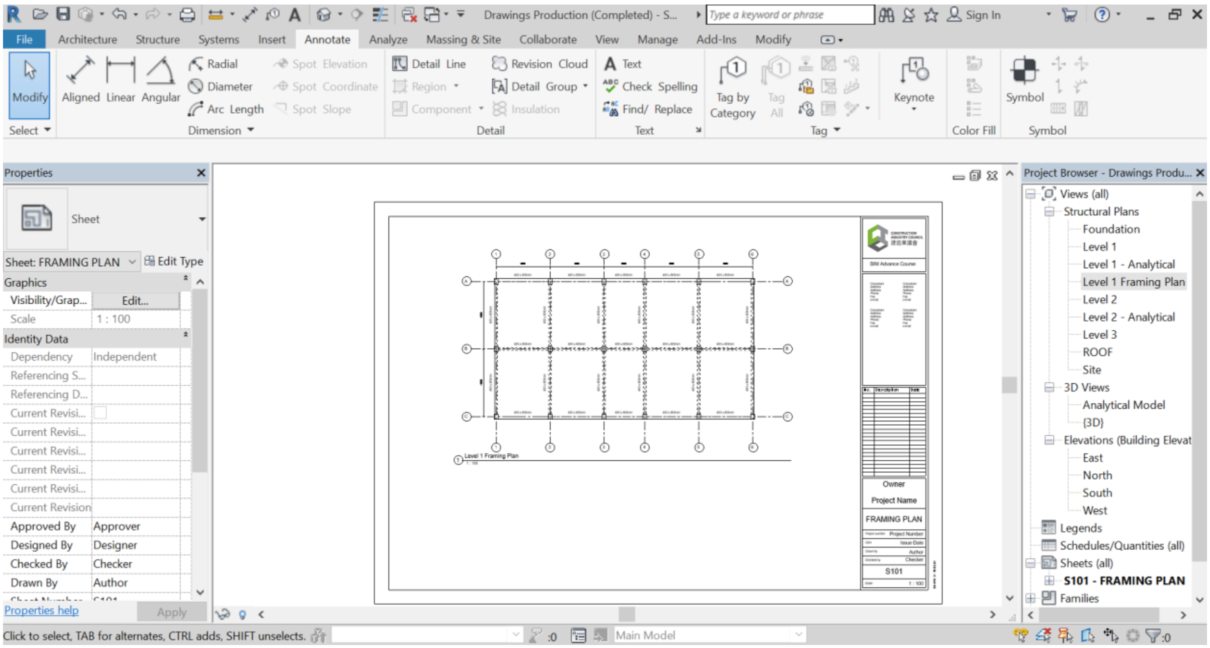
Click Annotate tab ➢ Dimension panel ➢ Aligned. Then pick Grid-1, then Grid-2 to Grid-6 and click anywhere right to Grid-6. Press “ese”.

Repeat Aligned Dimension and click on Grid-A to Grid-B.



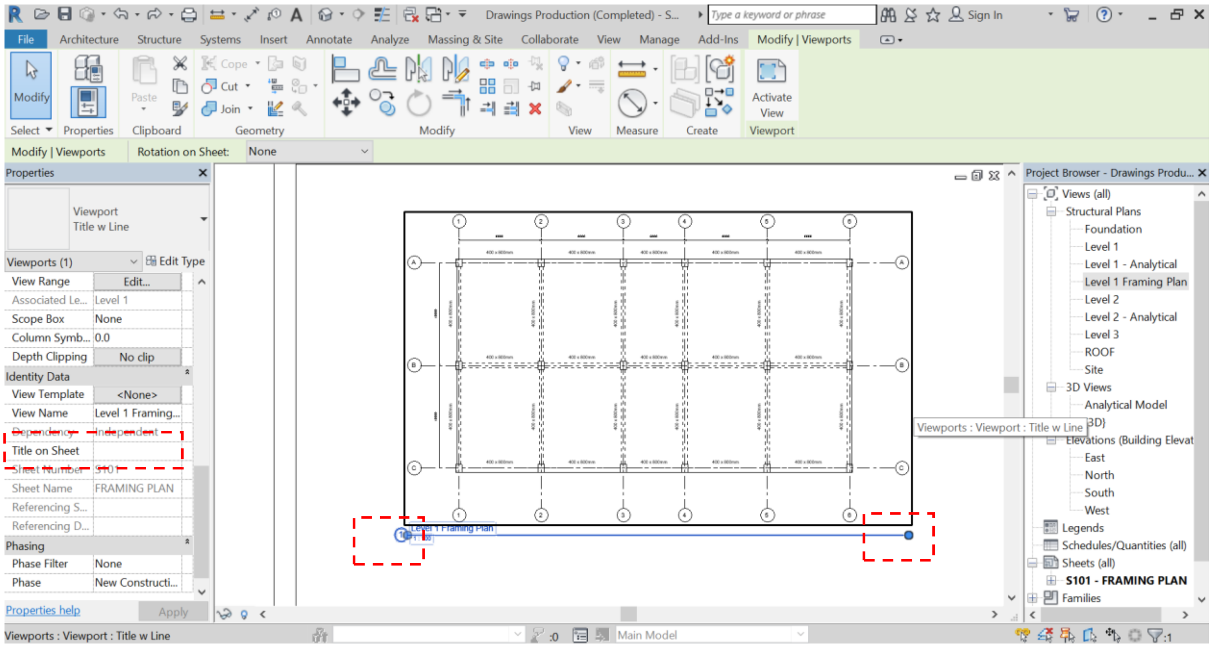
12 Now the “Level 1 Framing Plan” view looks more like a framing plan for submission.

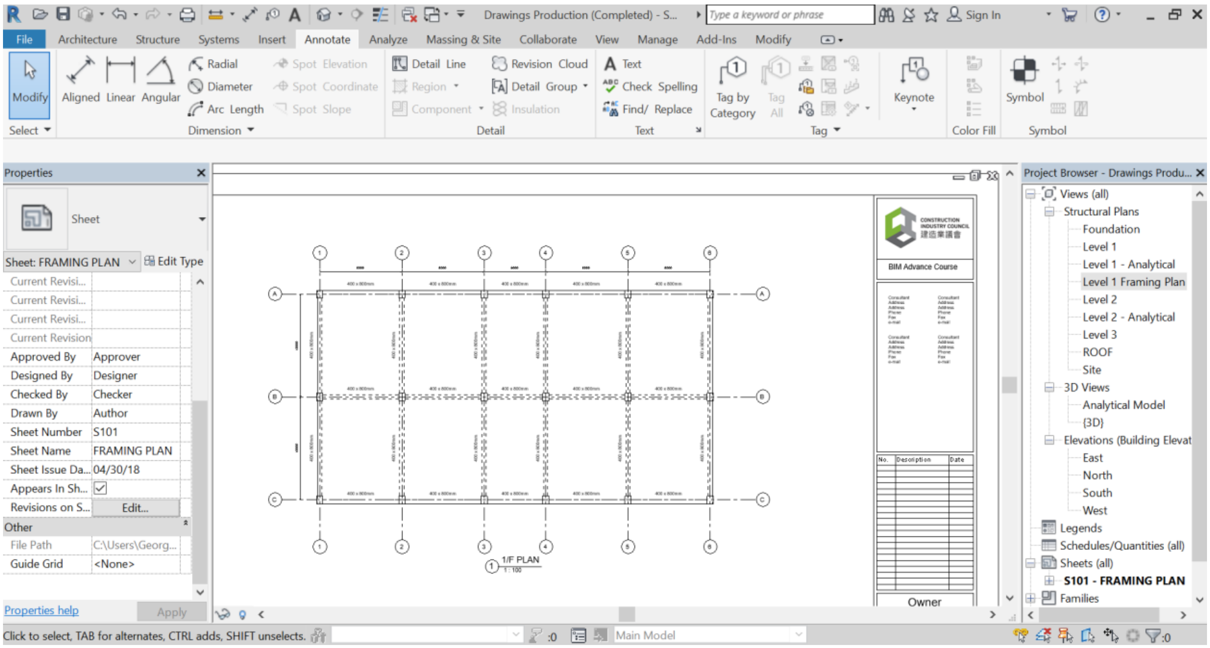
You can insert this view into Sheet “S101”. First go to Sheet “S101” (i.e. double click on S101 in the Project Browser). Then drag the “Level 1 Framing Plan” view into the sheet area.



With some final refinements on the viewport and the drawing will be ready for submission.

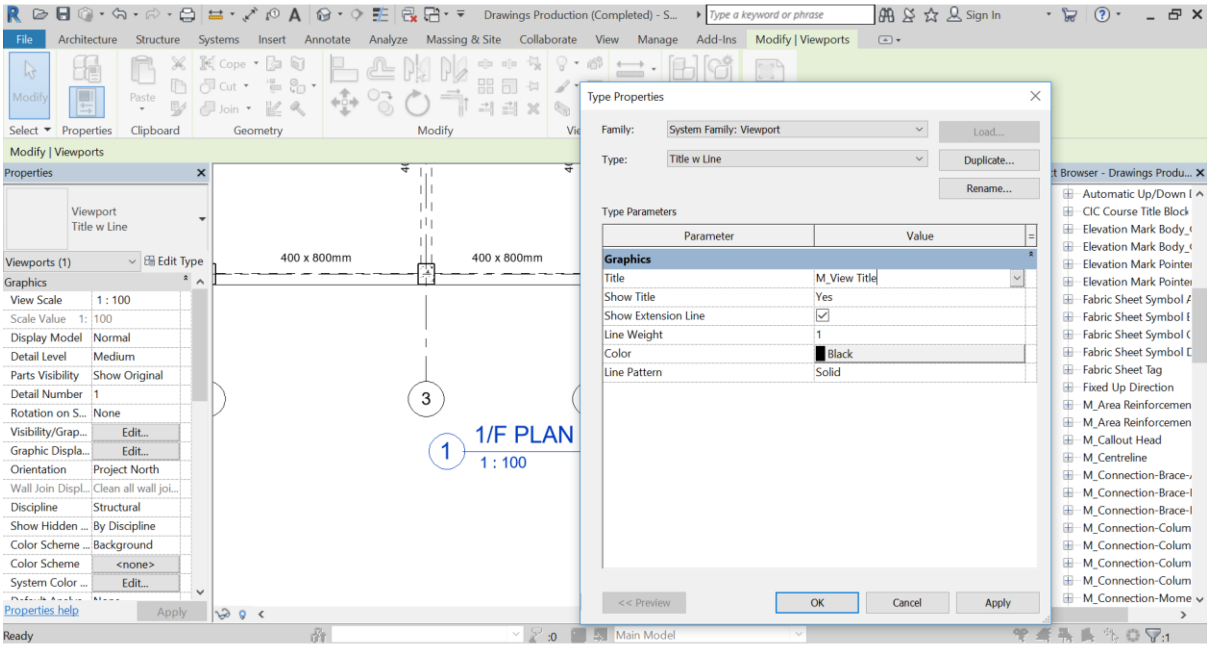
13 Select the viewport. (1) Then drag the title line to appropriate length and location. (2) Type in “1/F PLAN” in “Title on Sheet” at the Properties Palette.



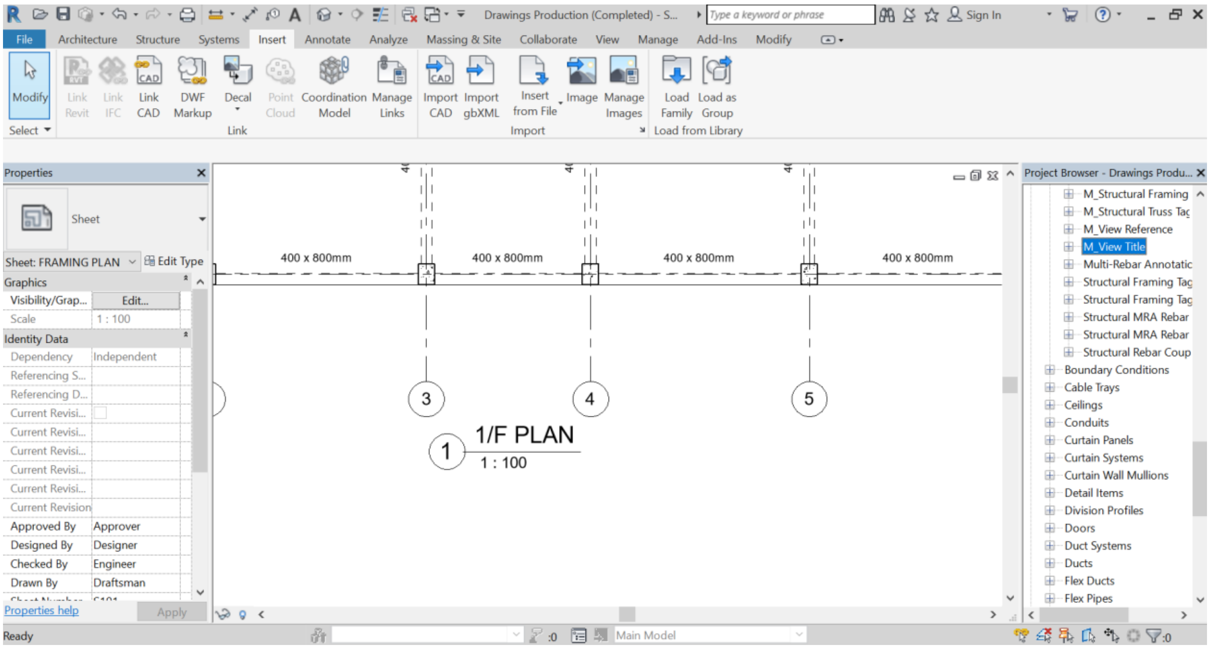


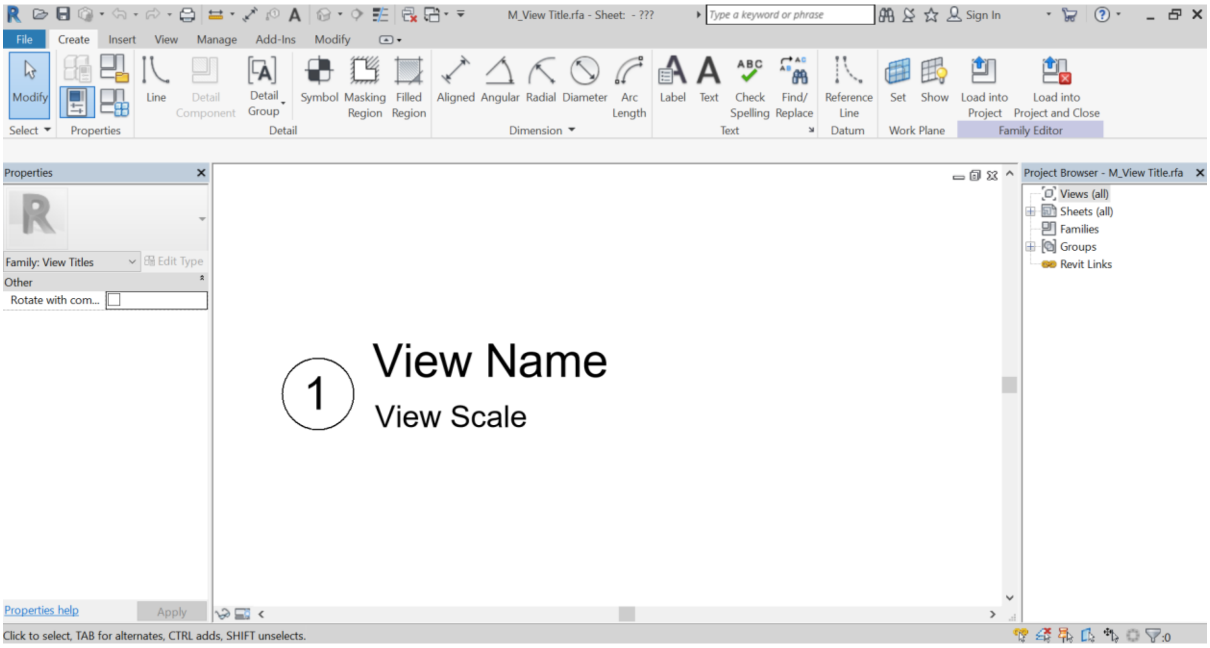
14 The title of the 1/F PLAN still does not look professional because of the bubble and detail number “1”. You need to edit the “View Title” family to remove that.

First, notice that when you select the title and click “Edit Type”, you will see that the family used for the title style is “M_View Title”.

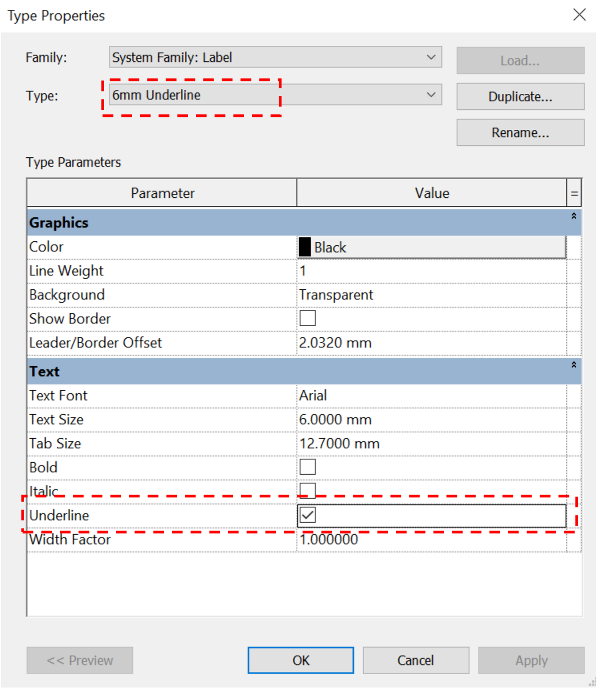


Now from the Project Browser, find "Families" ➢ "Annotation Symbols" ➢ "M_View Title". Right click and click edit.



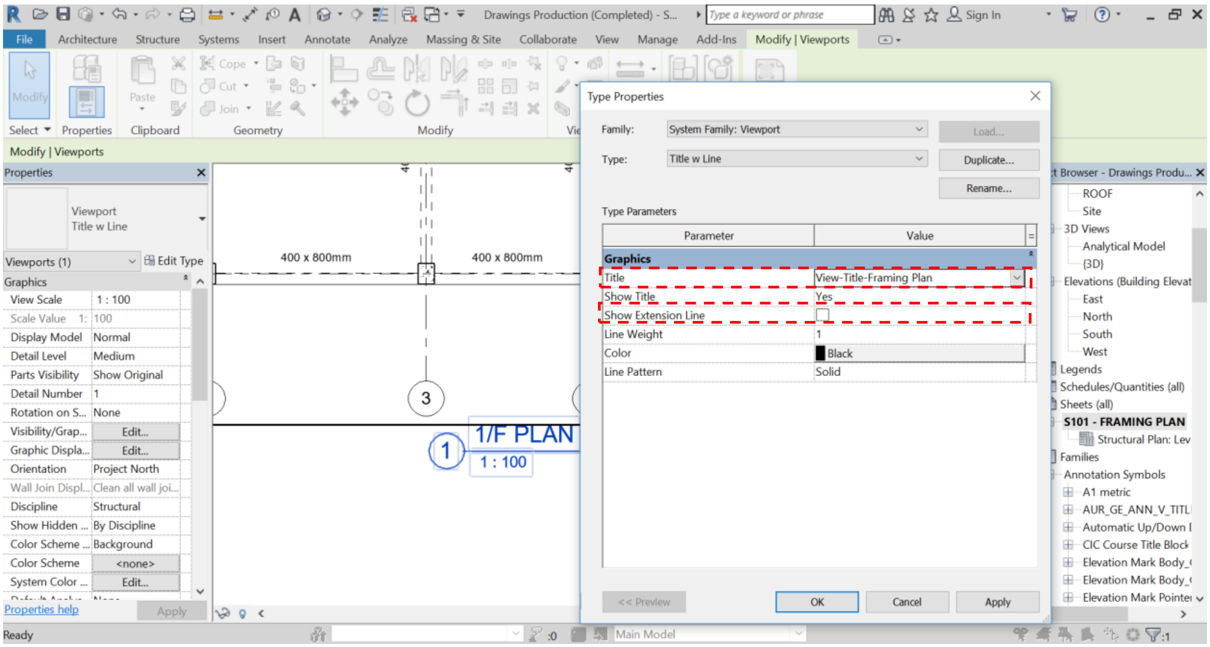


Now delete the bubble and the label “1”. Then select the “View Name” label and click “Edit Type” in the properties palette. Duplicate the type and rename it to “6mm Underline”. Remember to check the “Underline” box.

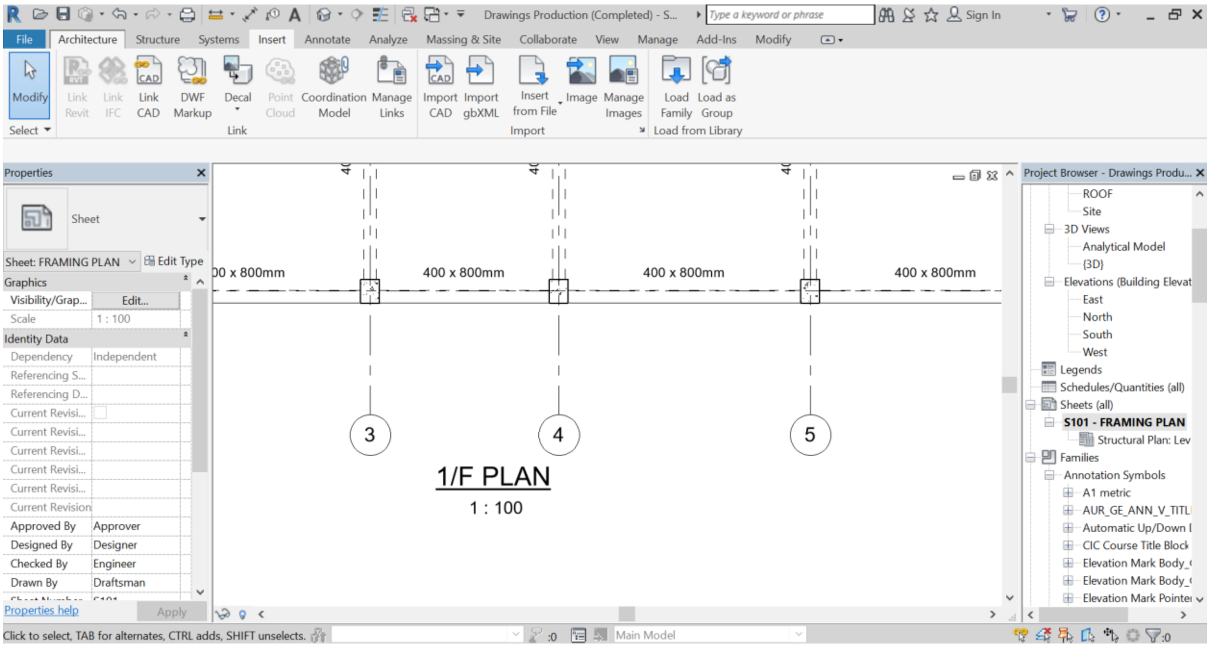


Click “OK”. Then save this edited family as “View-Title-Framing Plan” in the course folder. Now click “Load into Project and Close”.

Now when you select the title and click “Edit Type”, you can change the “Title” style to the one you have just created which is “View-Title-Framing Plan”. Remember to uncheck the “Show Extension Line” too.

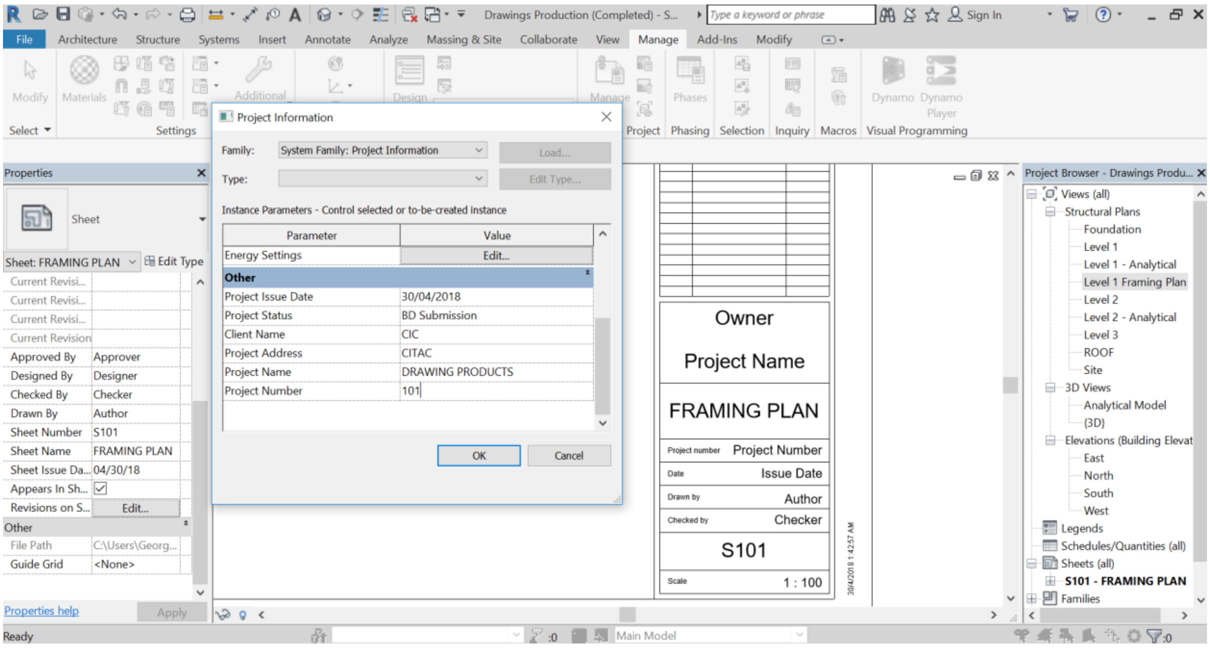


Finally, the title style looks good for submission.

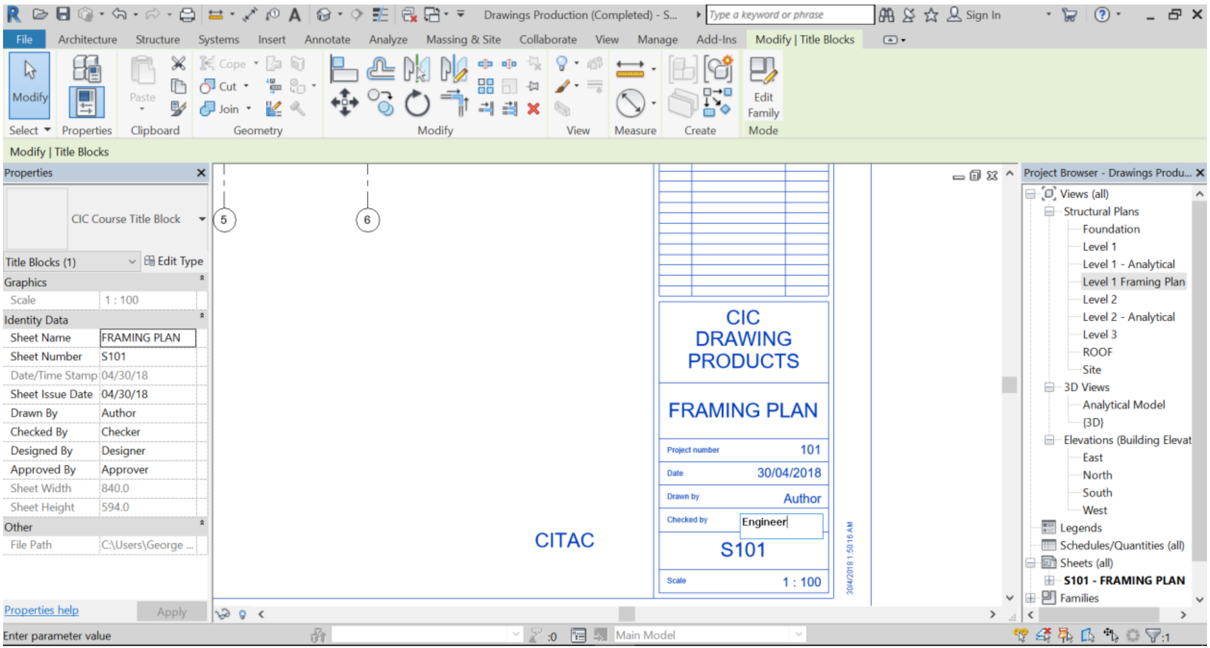


15 Input the project information to complete the drawing.

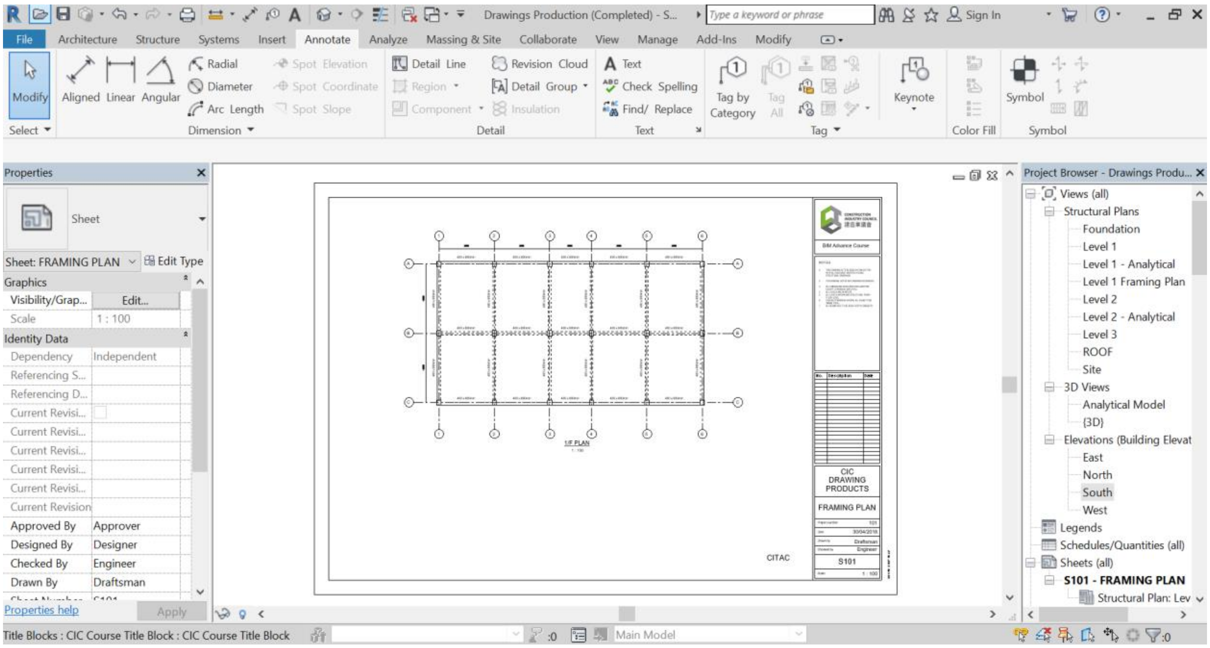
Click Manage tab ➢ Settings panel ➢ Project Information. Type in the information as shown below.



16 You can also edit the parameter by clicking on the label in the sheet view.



Now with some general notes, the sheet (drawing) is ready to be printed for submission.

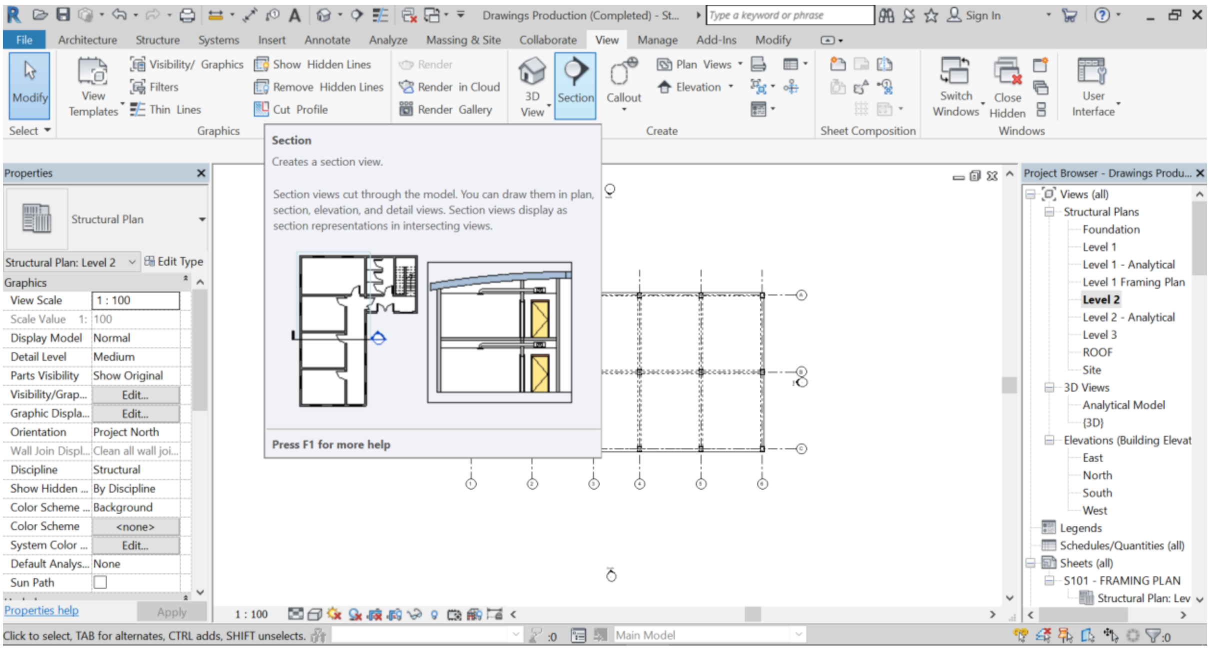


17.3 Creating Sections Drawings in Revit

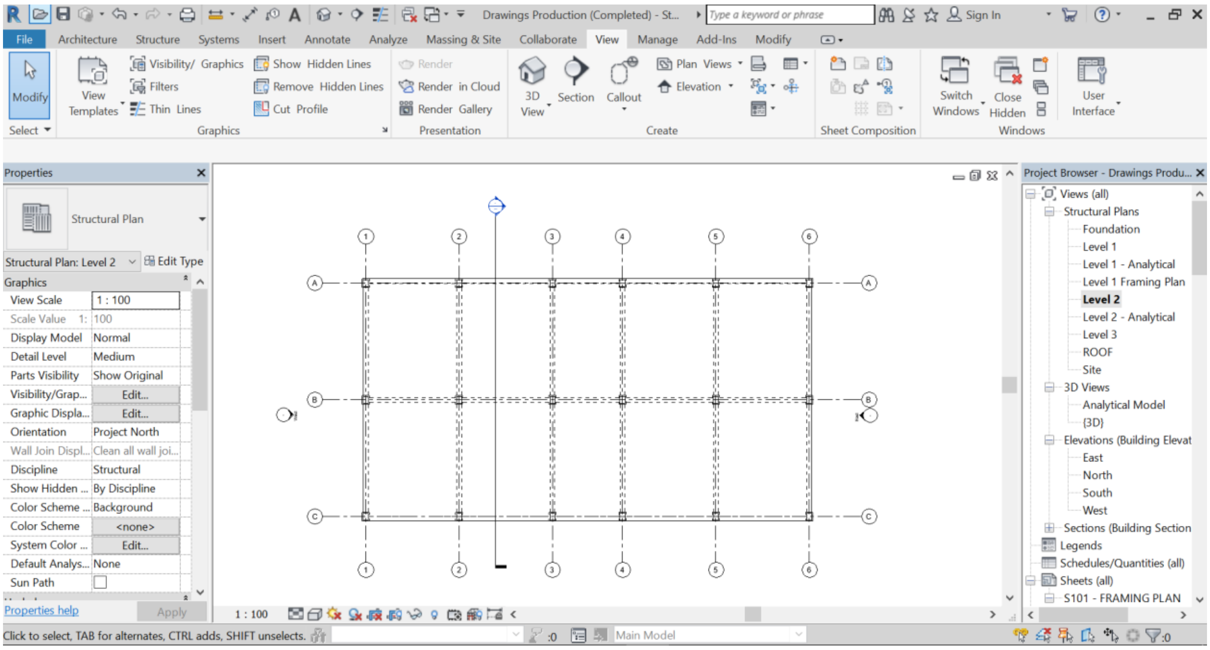
The create a section of your building,

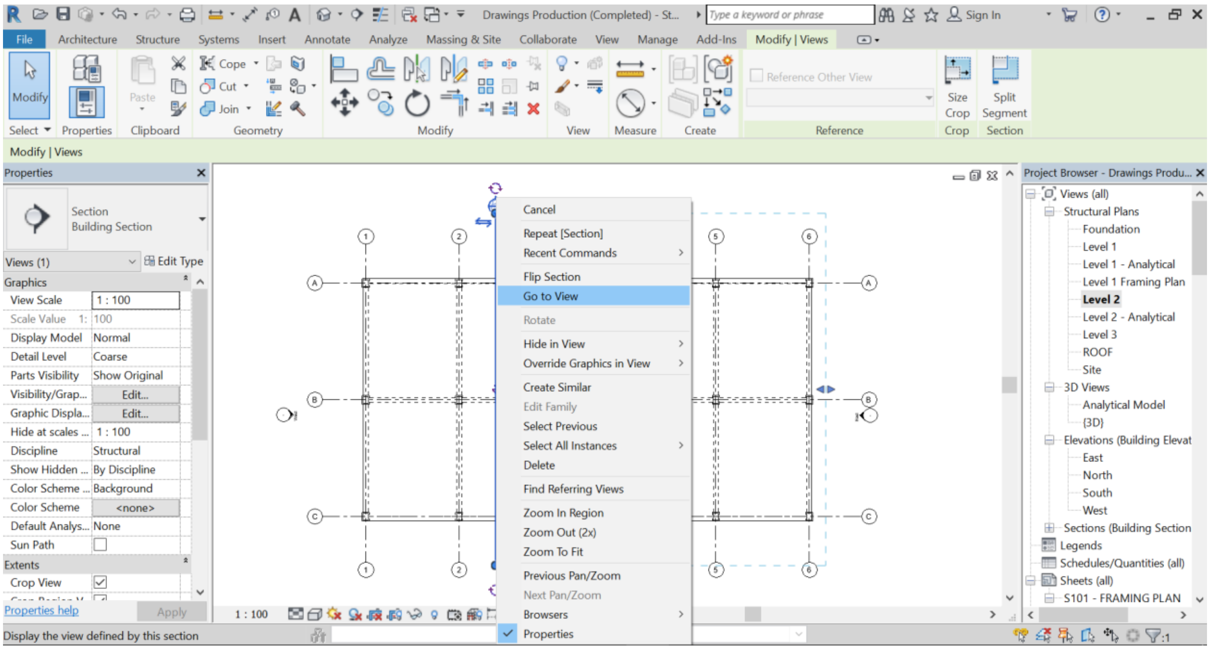
1 Go to Level 2 (Structural Plan).

2 Click View tab ➢ Create panel ➢ Section.

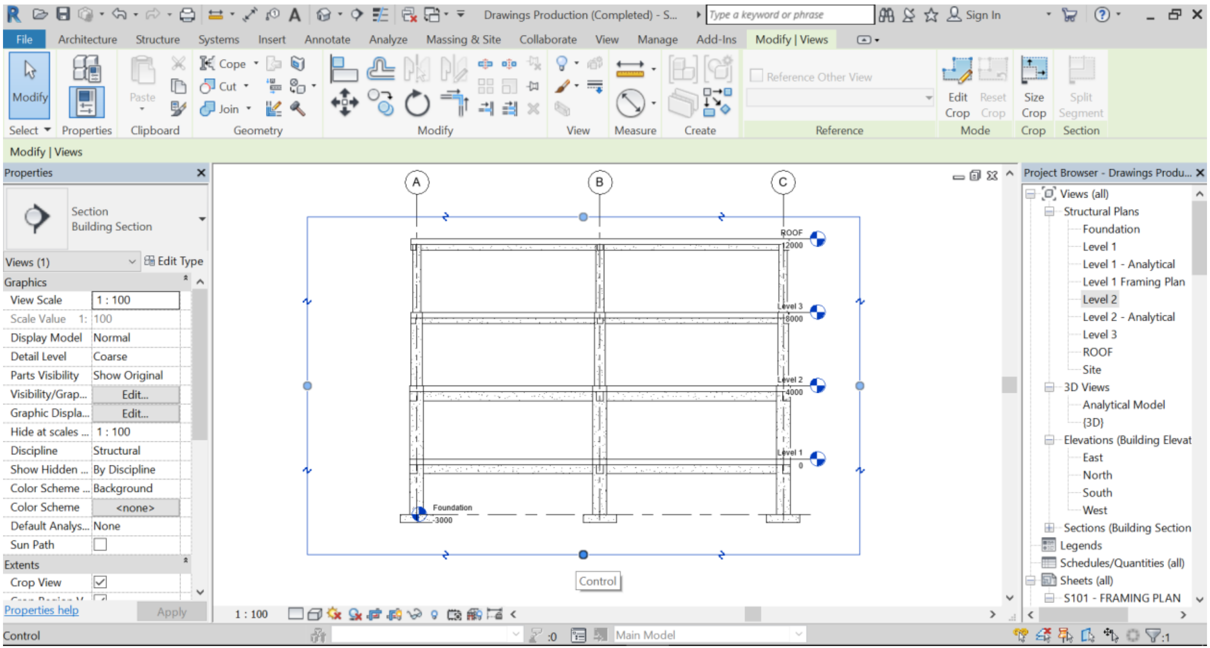


3 Cut a section between Grid-2 and Grid-3 as shown below. Then select the Section line and click "Go to View".

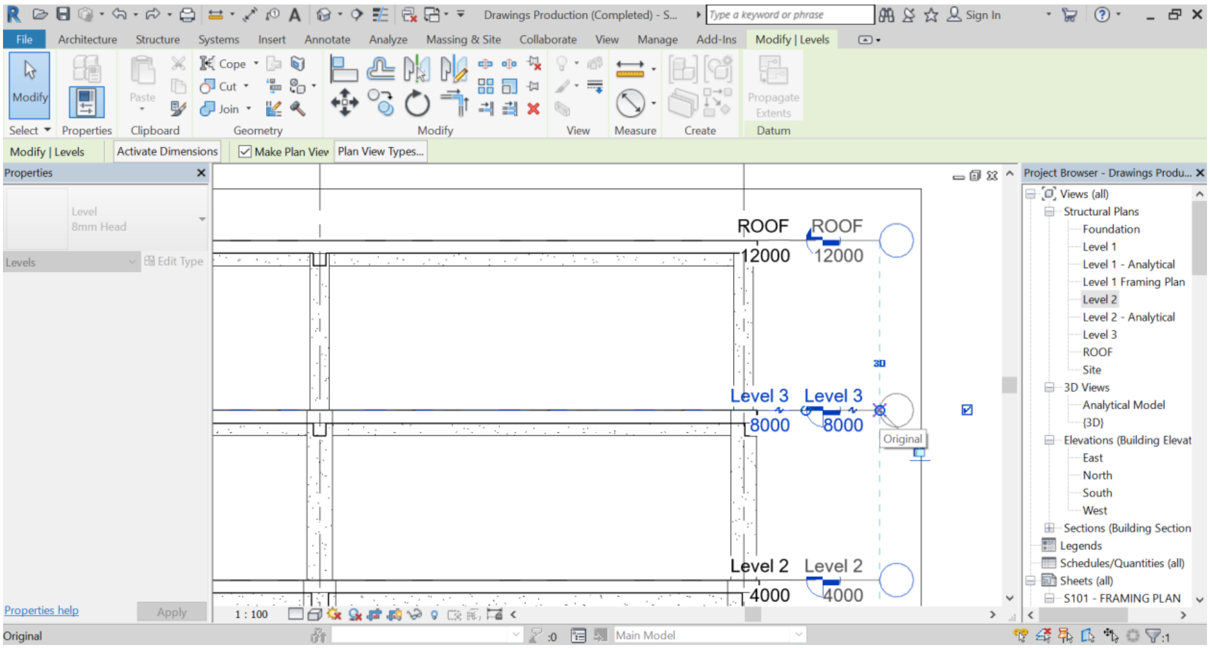




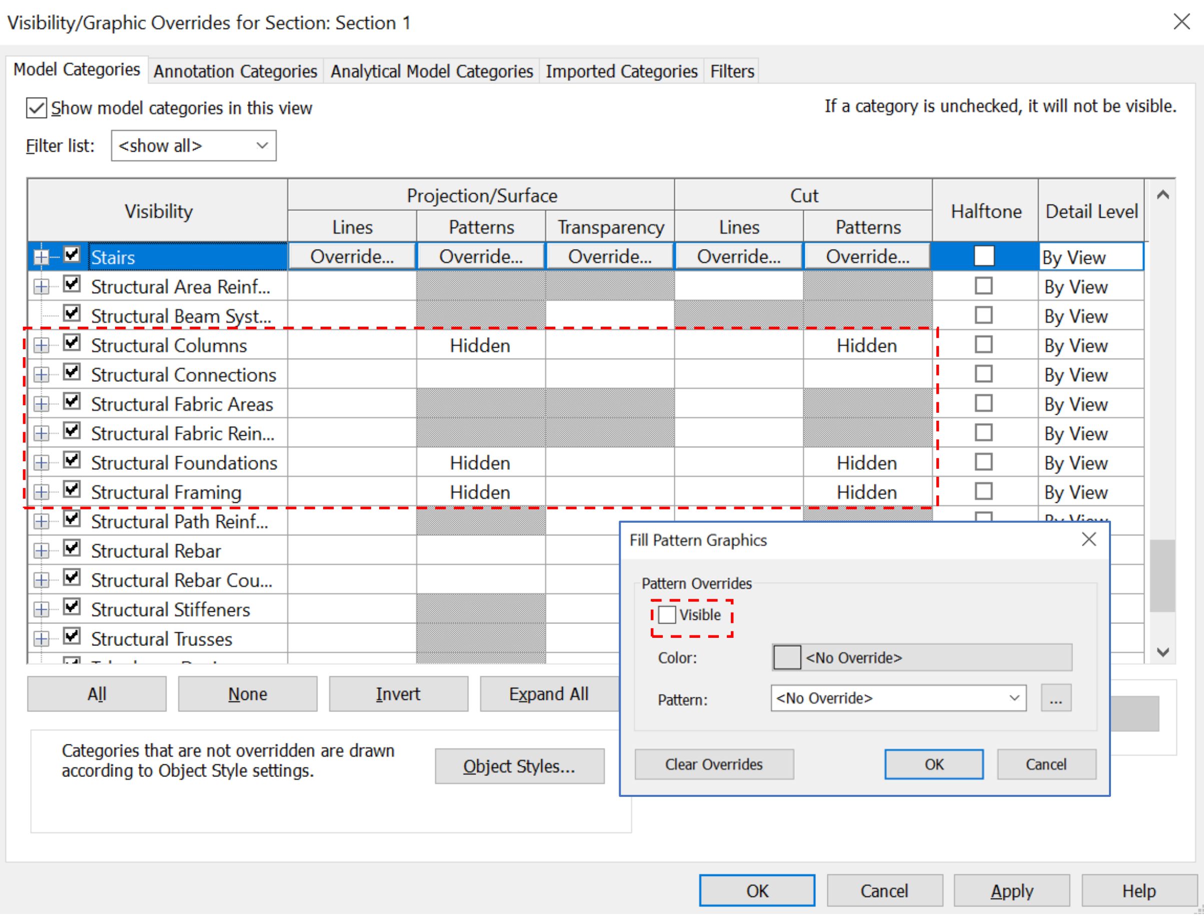
4 Select the View box and drag the control point downward to view the footings as well.

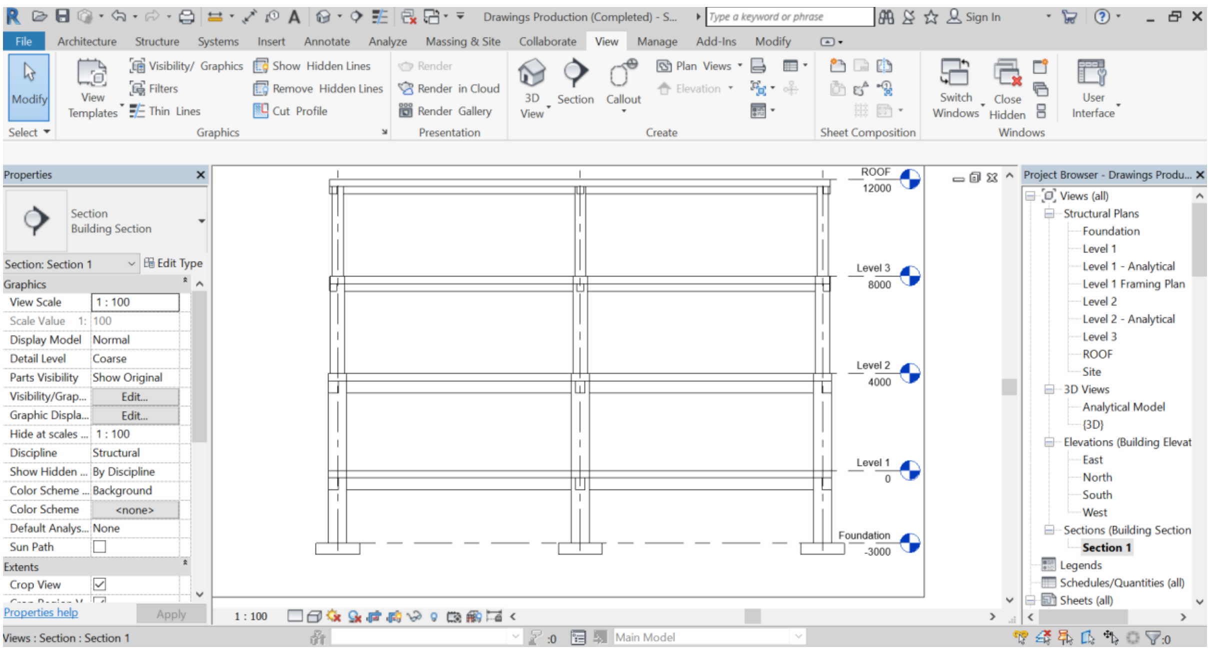


5 Select the level line and drag it further away from the building. Note that all level lines will be moved.



6 To hide the pattern on beams, columns and footings, type “vv” for Visibility/Graphics Control and edit the “Structural Columns”, “Structural Foundations” and “Structural Framing” settings as below.

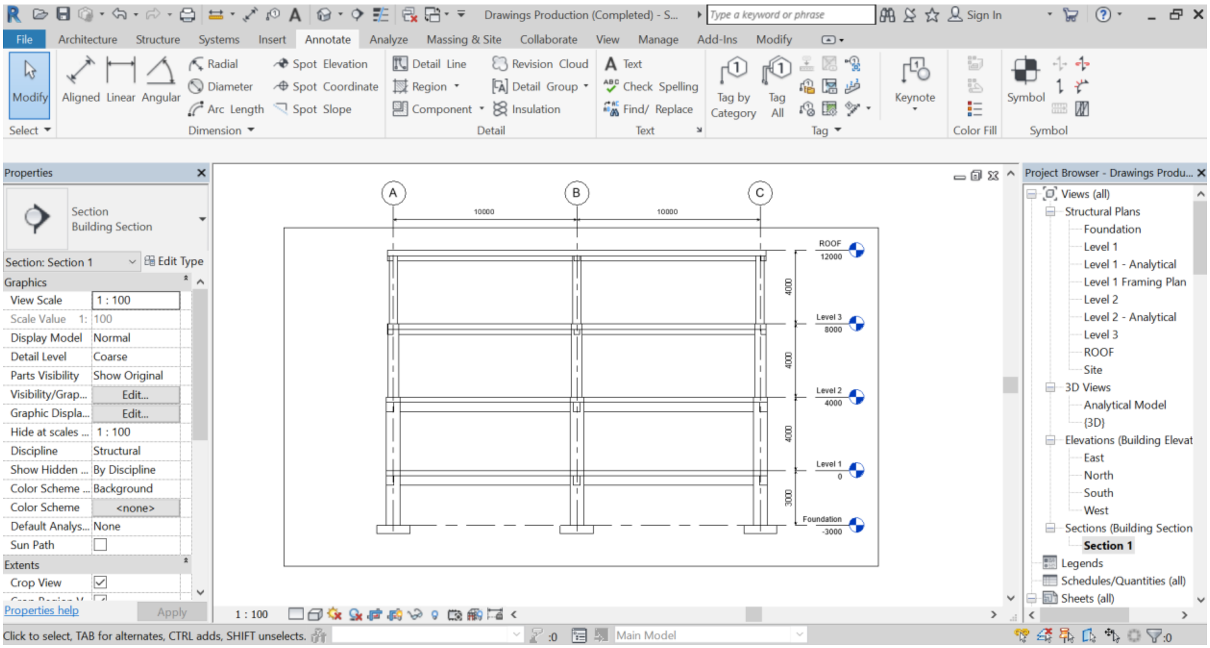




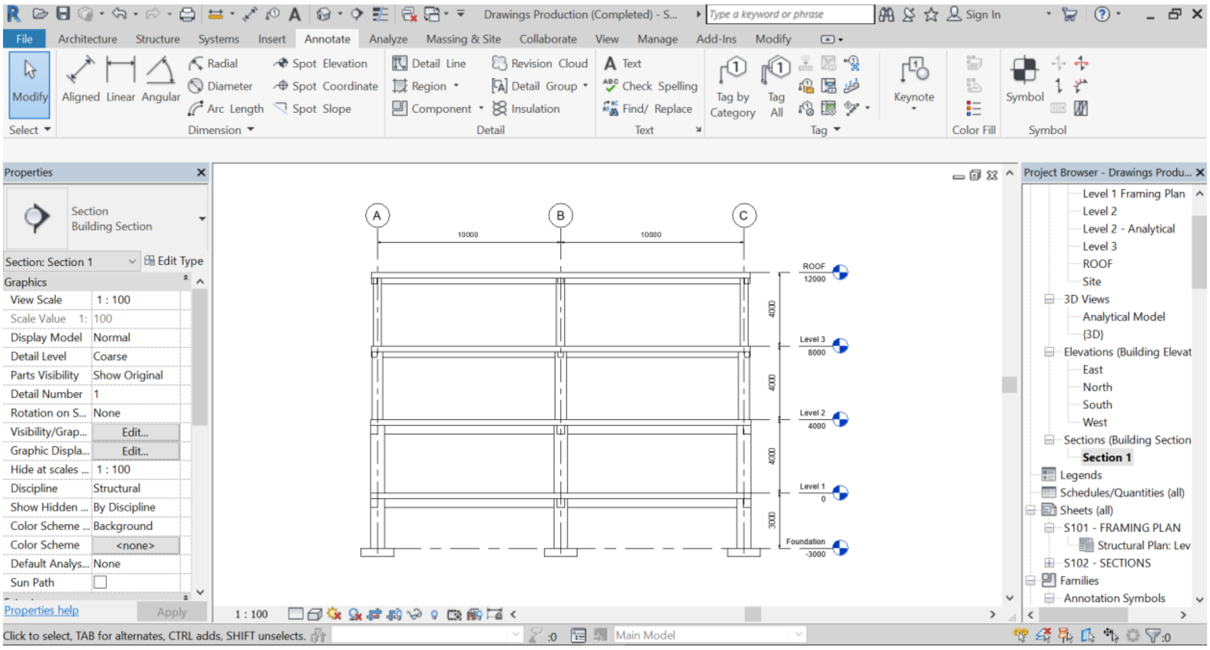
7 Now add some dimensions to the section.

Click Annotate tab ➢ Dimension panel ➢ Aligned. Then pick Grid-A, then Grid-B and Grid-C and click anywhere right to Grid-C. Press “ese”.

Also add some dimension for the levels.



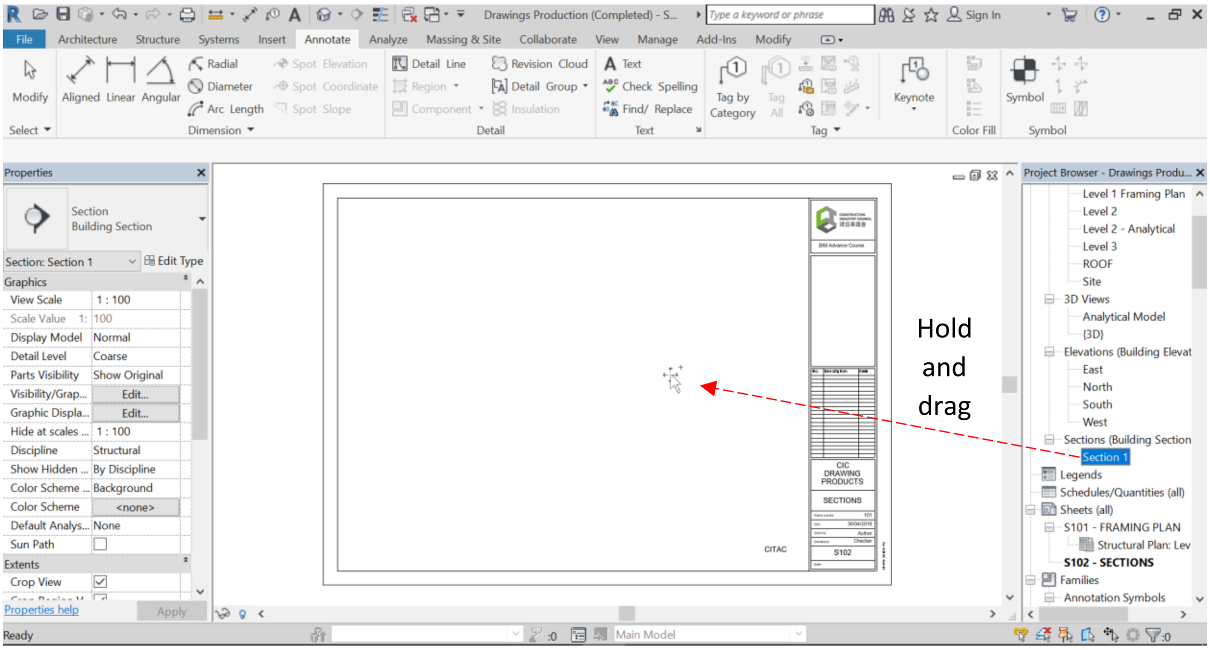
8 Hide crop view .

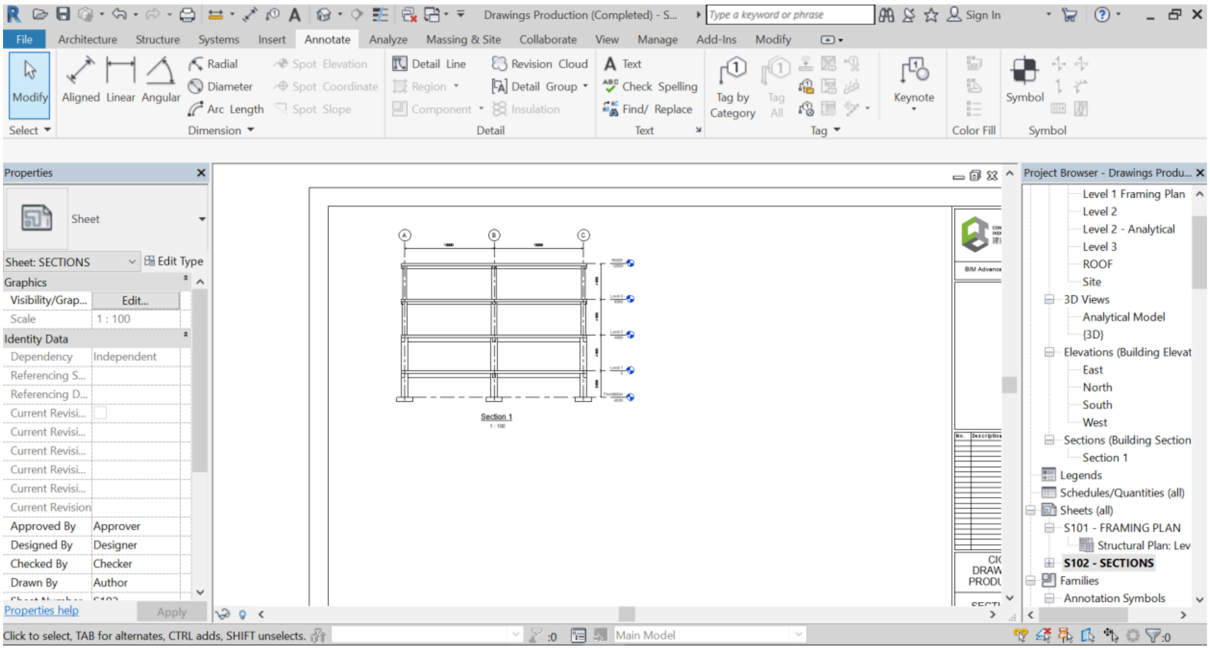


9 Next add a new sheet to place the section prepared above. Select "Sheets (all)" from the Project Browser and then right click to add a New Sheet with the same CIC Course Title Block.

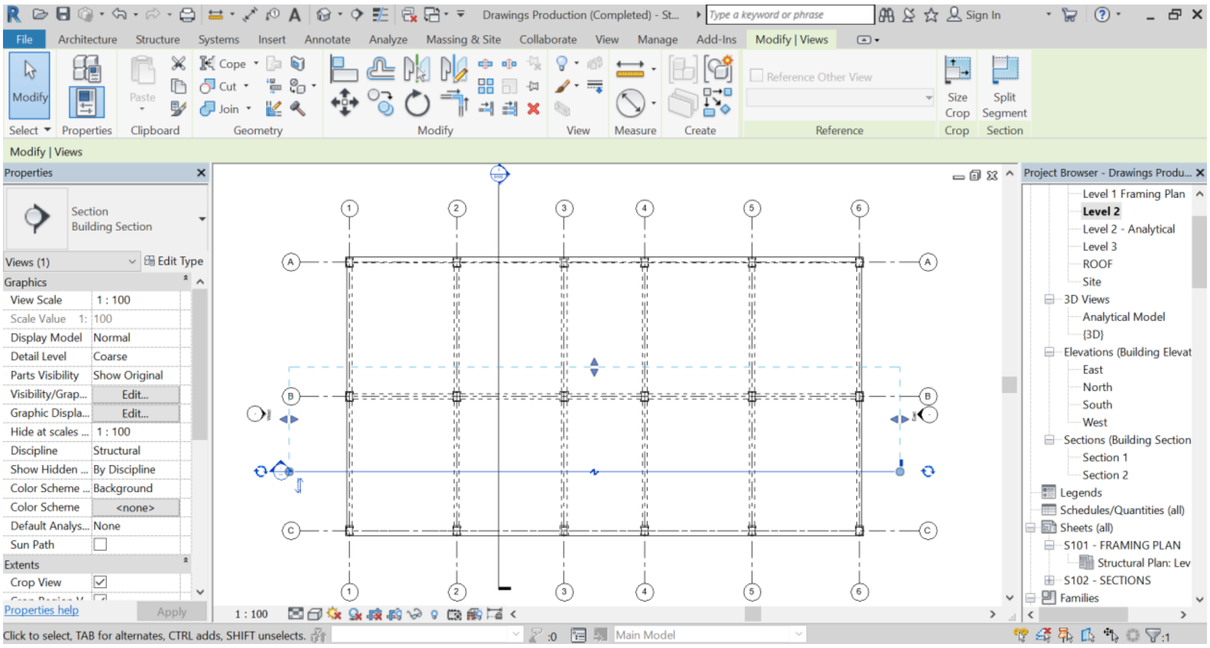
11 Select the new sheet (S102) and right click to rename. Rename it to "SECTIONS". Click "OK".

12 While still in sheet S102, find and click Section 1 from the Project Browser and drag it into the sheet. Then move the title to the centre of the section.

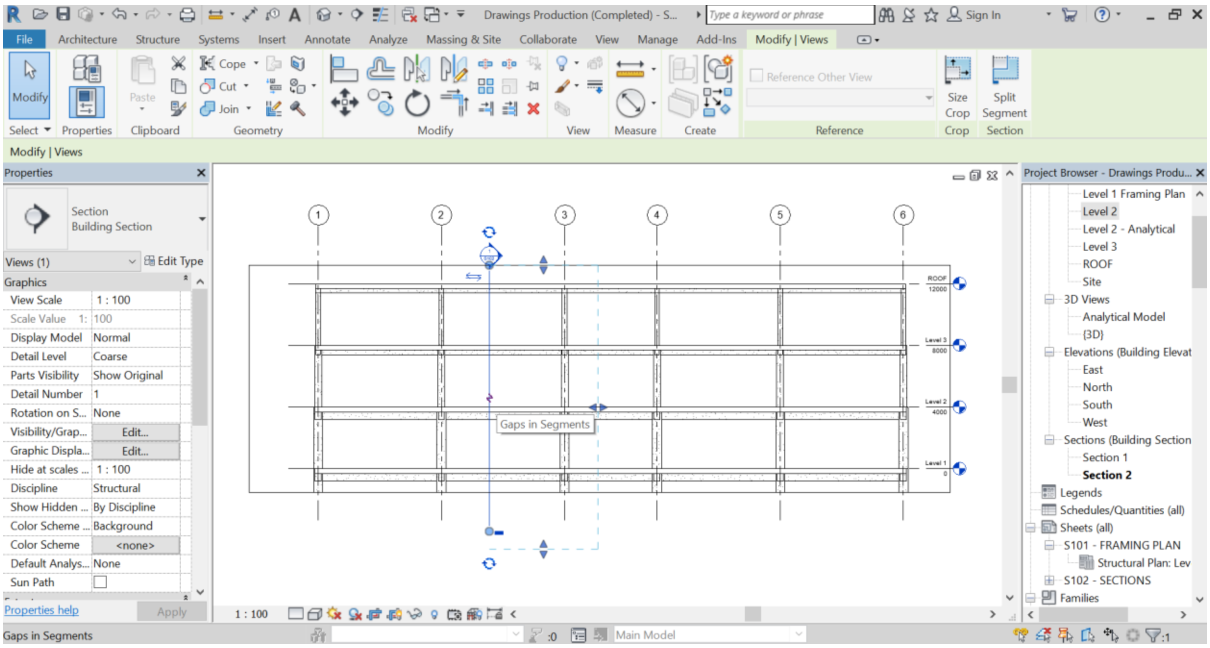


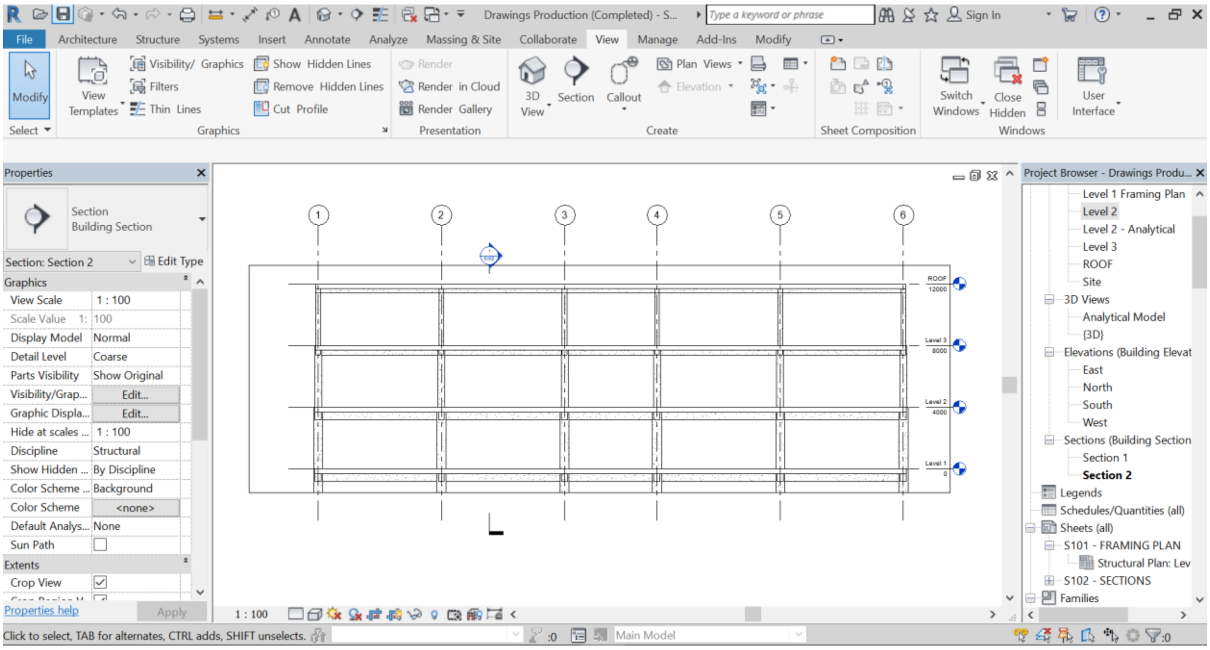


13 Go to Level 2 (Structural Plan), then click View tab ➢ Create panel ➢ Section. Add another section between Grid-B and Grid-C as shown below.

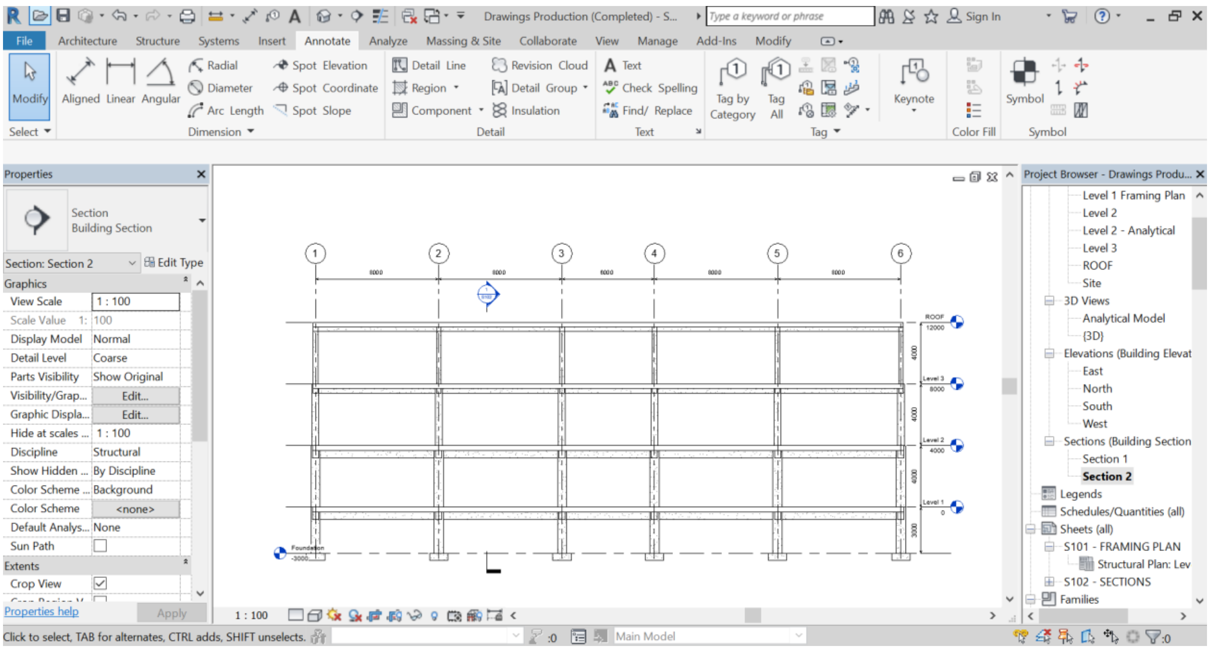


14 Go to the newly formed section view (Section 2) and adjust the cut line of Section 1 as shown below.

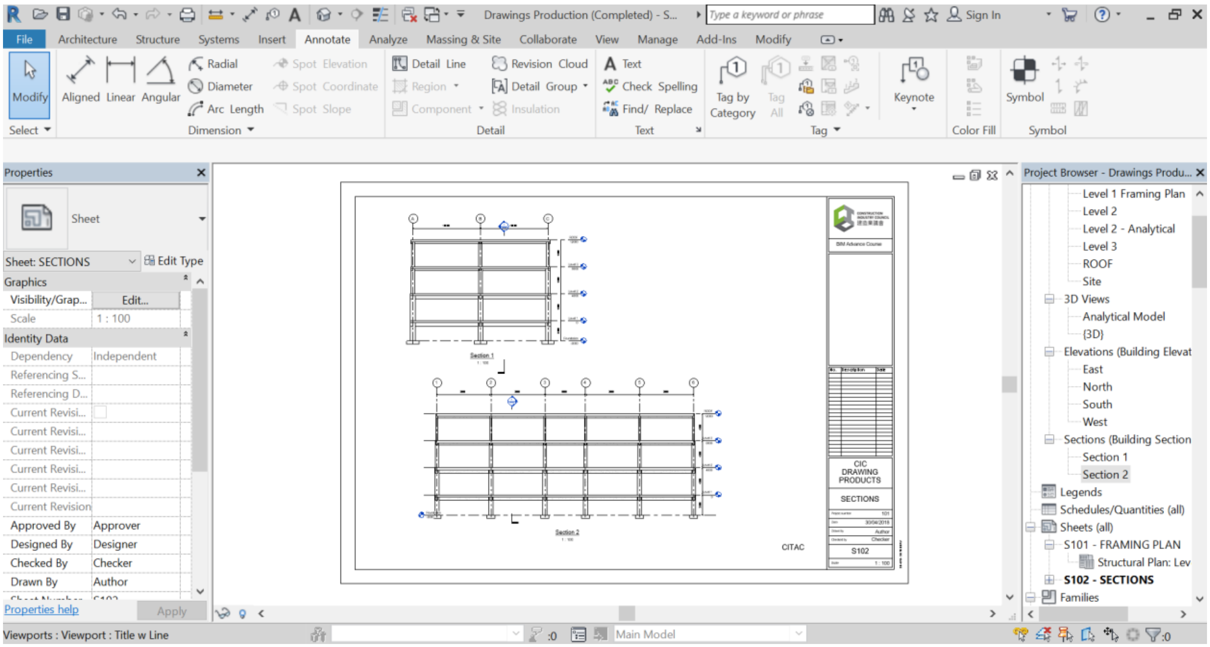




15 Add dimensions and hide crop region as in Steps 7 and 8 above.



16 Go to Sheet "S102" (double click on it from the Project Browser). Then click on Section 2 and drag it into the sheet S102.



17.4 Creating Column R.C. Detail Drawings in Revit

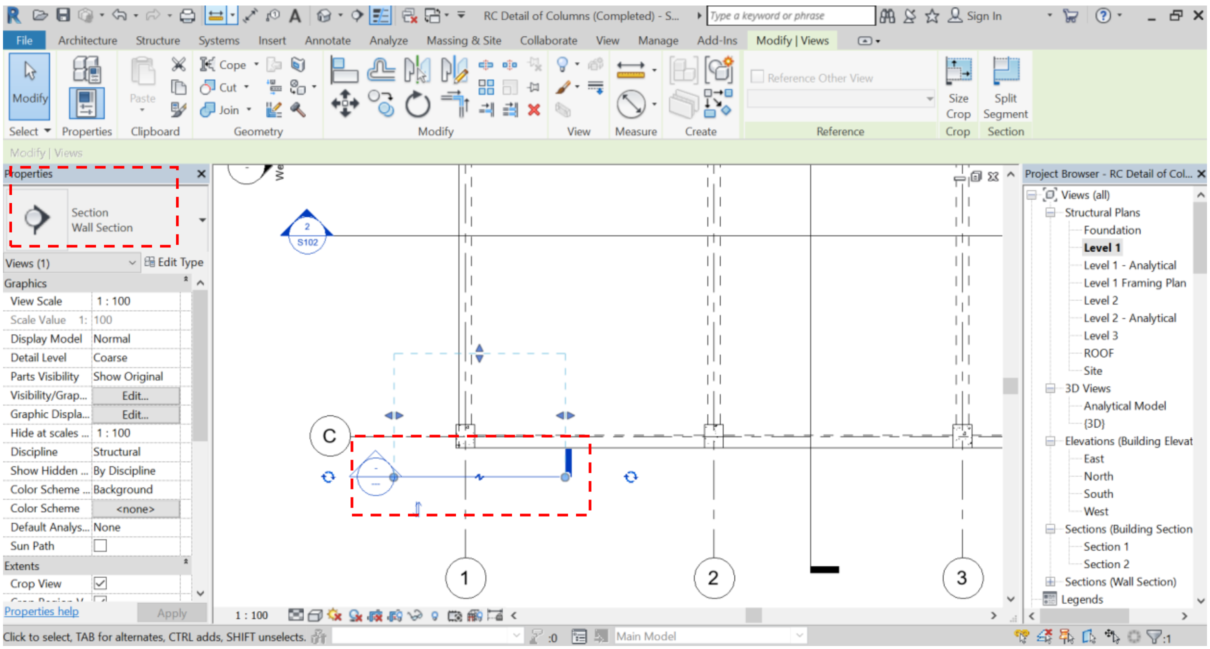
You have learnt how to model 3D rebars in Revit earlier in this course. In this part, you will practice this skill again and learn how to create R.C. detail drawings for beams and columns.

To create R.C. detail drawings for columns,

1 Open from the course folder the "RC Detail of Columns.rvt" file.

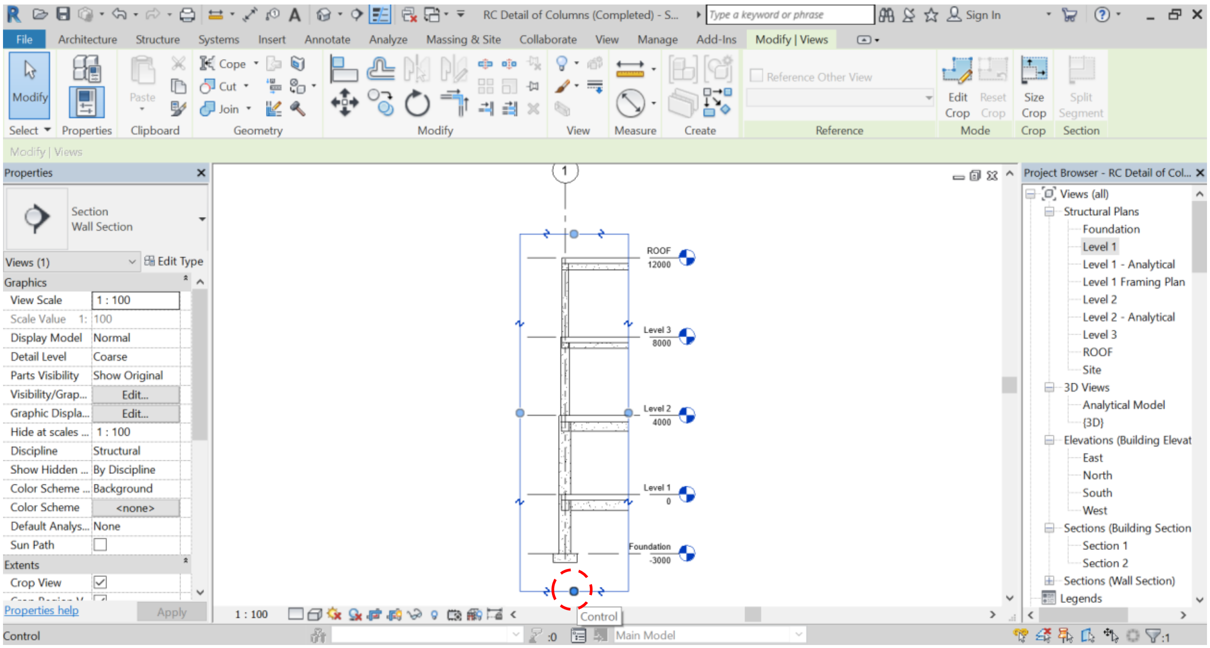
2 Go to Level 1 (Structural Plans).

3 Click View tab ➢ Create panel ➢ Section (choose Wall Section from Type Selector). Then create a wall section as shown below.



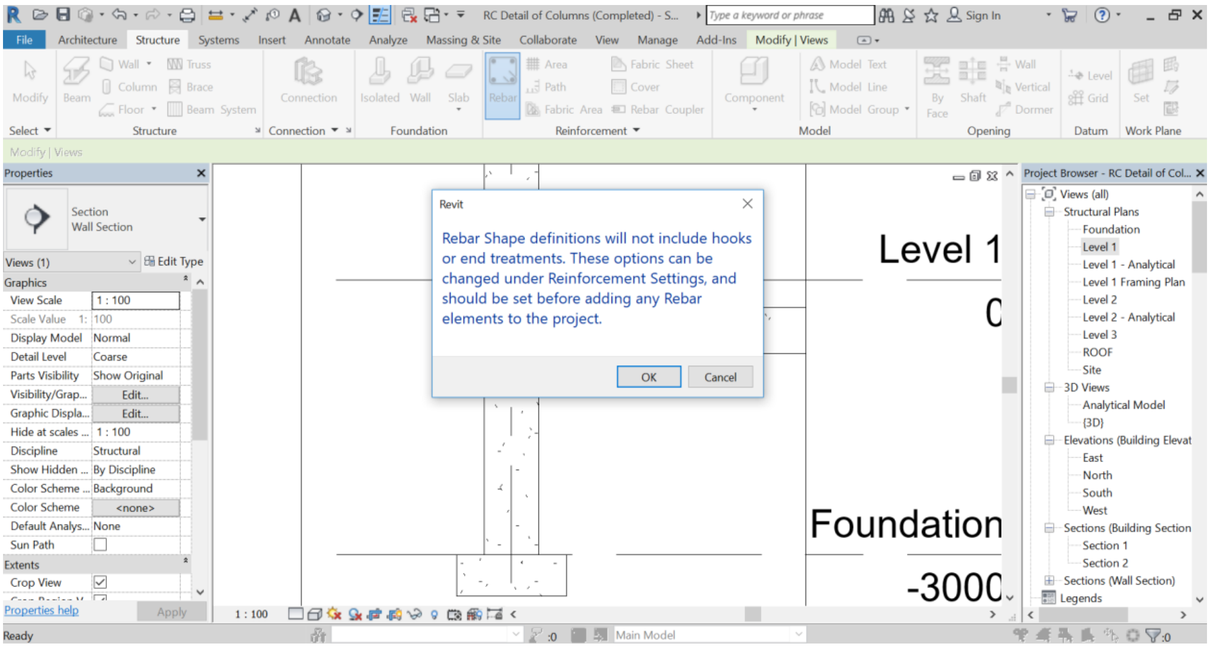
4 Double click on the wall section to go this wall section view. Alternatively, you can select the wall section line then right click and select "Go to View".

At this wall section view, drag the bottom control point until you can see the footing.

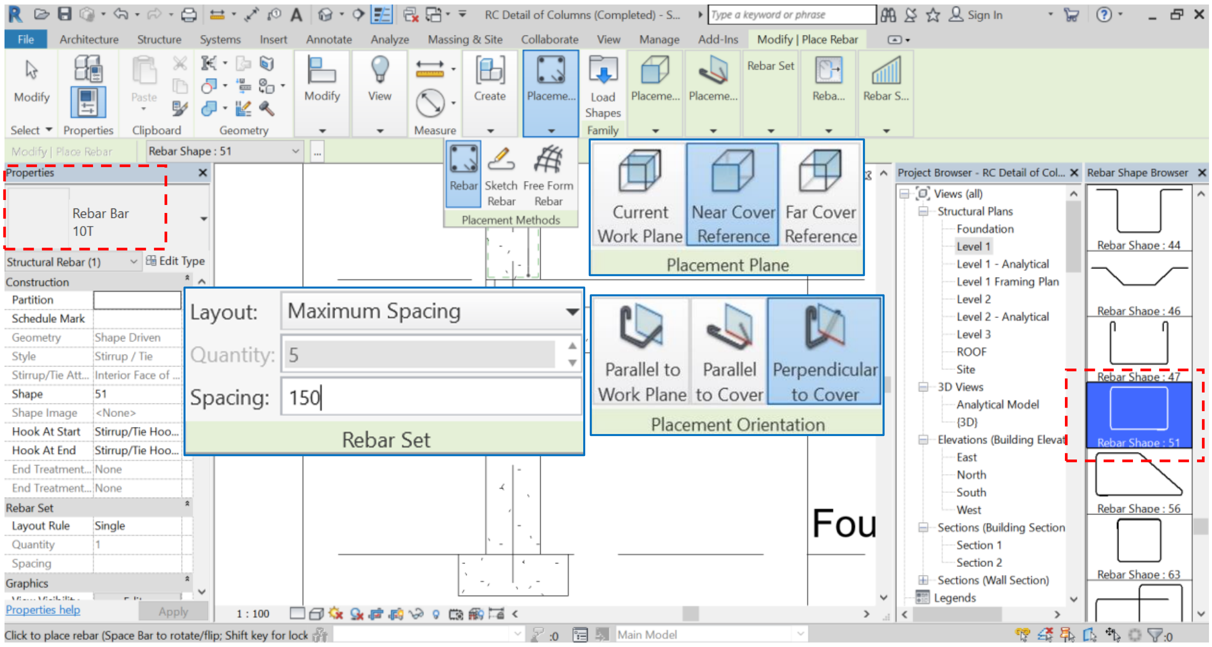


5 Now, zoom to the column from Foundation to Level 1 and start to add 3D rebars.

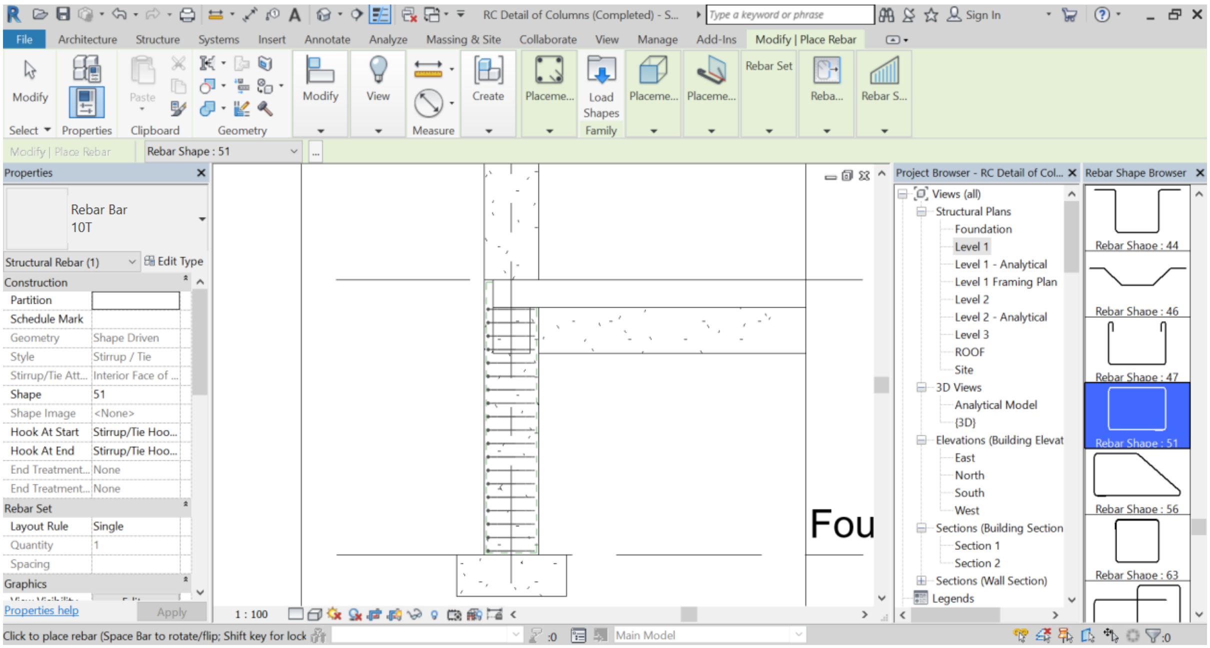
Click Structure tab ➢ Reinforcement panel ➢ Rebar. Click "OK" to the message.



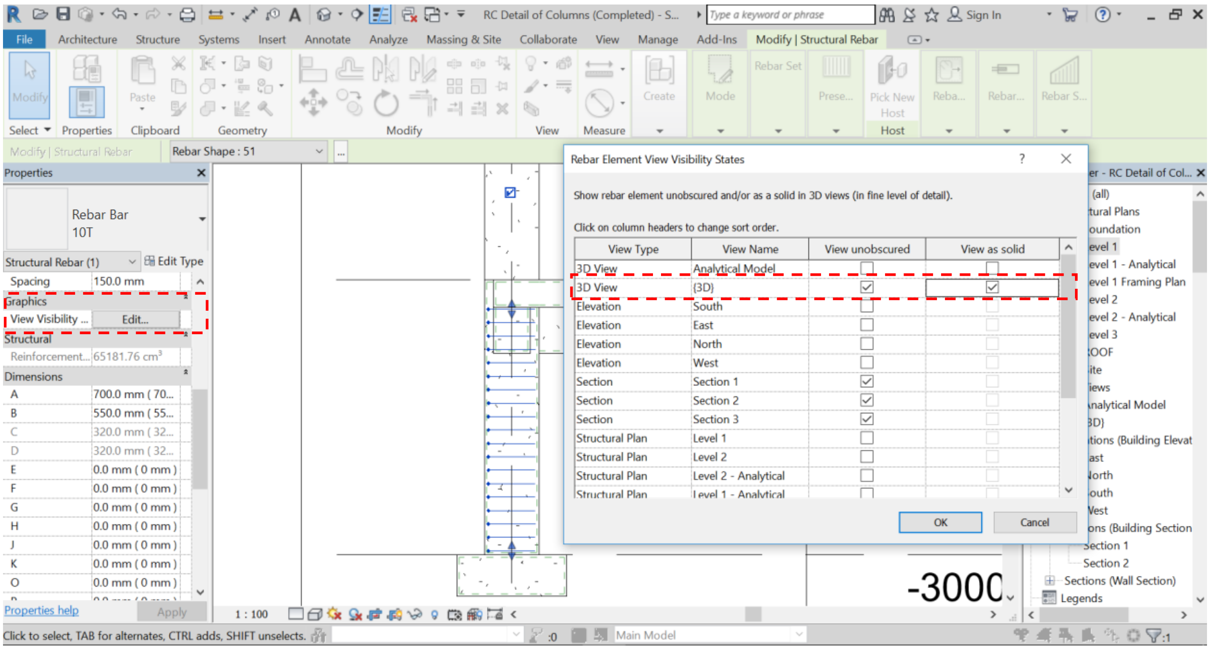
6 Select Rebar Shape: 51 with Rebar Bar 10T and set "Placement Methods", "Placement Plane", "Placement Orientation" and "Rebar Set" according to the setting shown below.

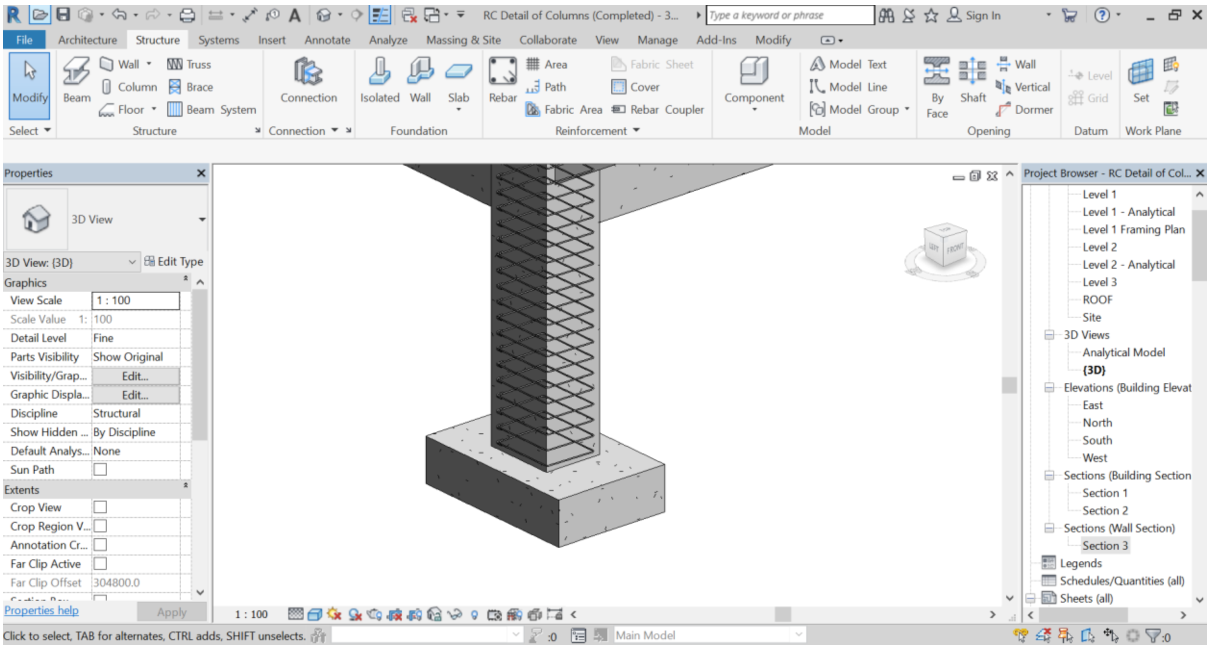


7 Hover near to the left edge of the column, Revit will show you a temporary set of rebars. Click to place the rebars as indicated.

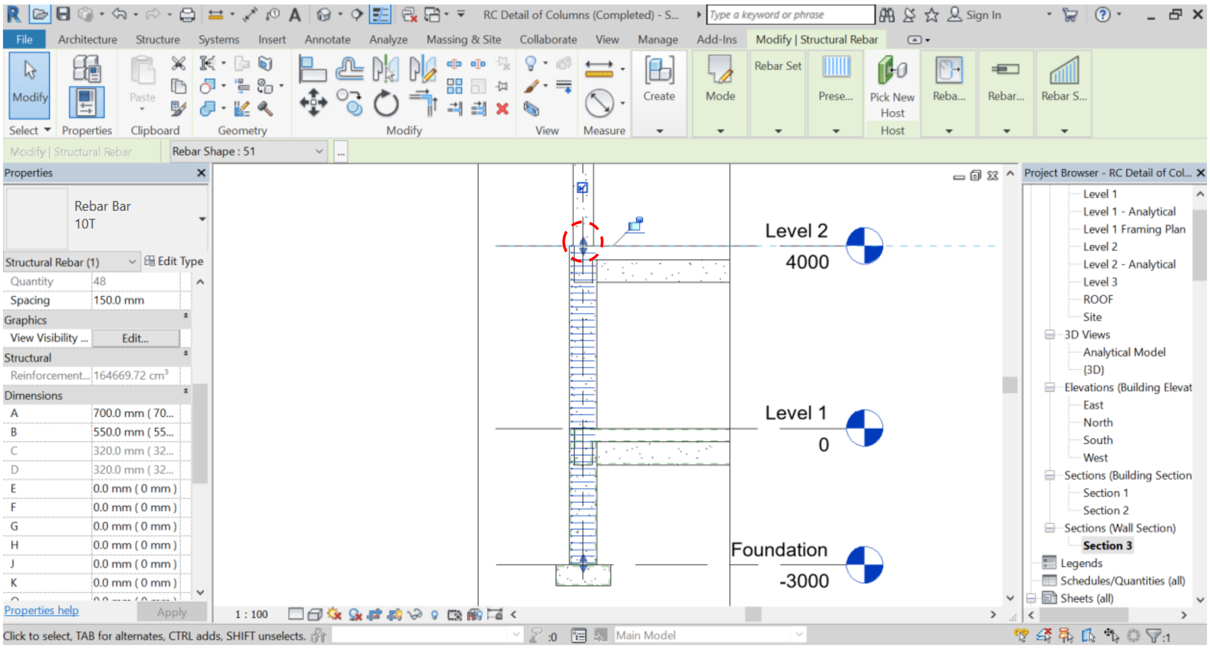


8 Select the rebar set and edit its View Visibility from the Properties Palette. Set “View unobscured” and “View as solid” for 3D View (View Name {3D}). Then go to 3D view to see the rebar set just created.

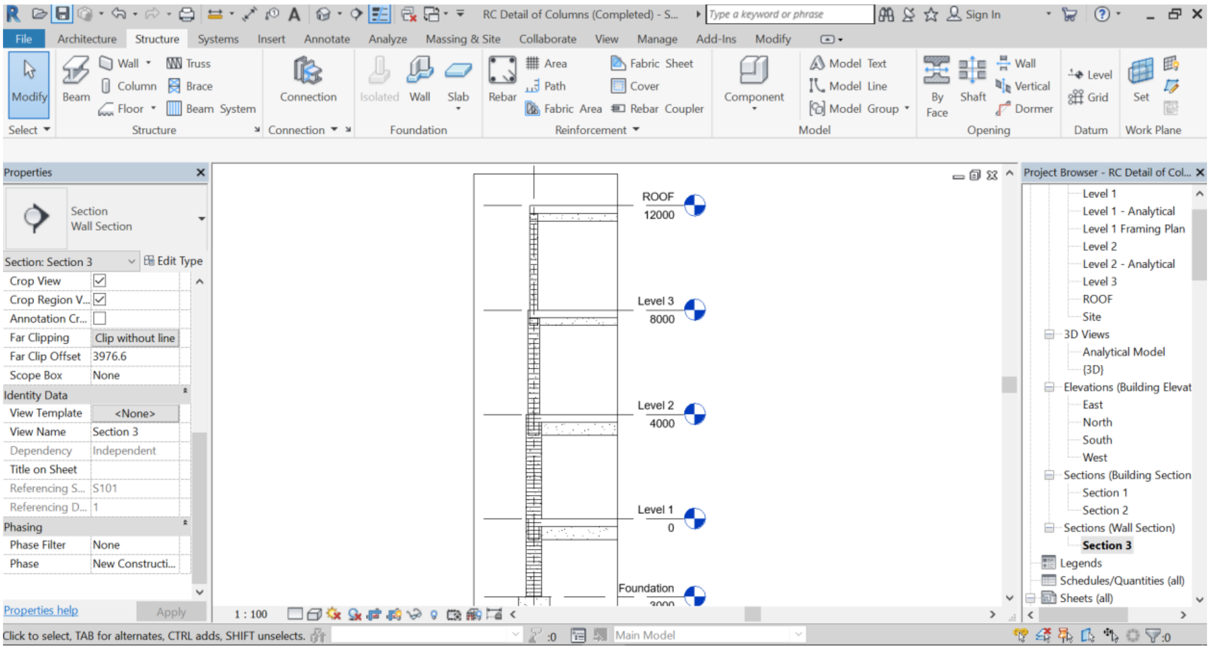




9 Go back to the wall section view (Section 3 in the Project Browser). Then select the rebar set and sketch it up to Level 2.

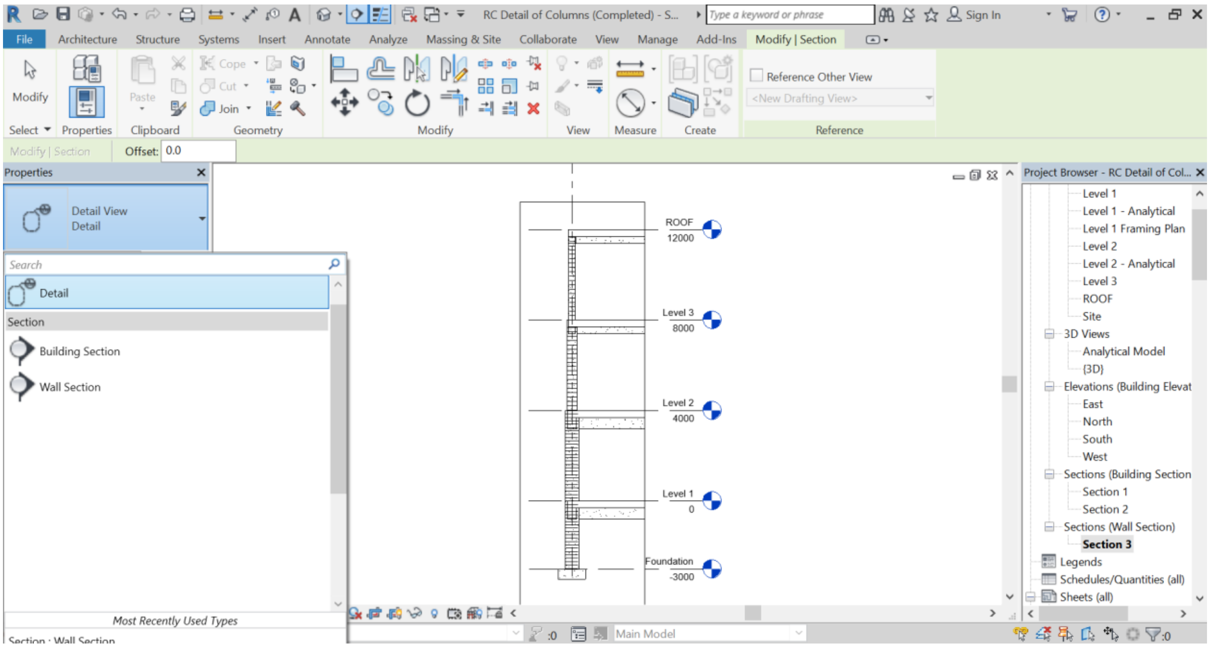


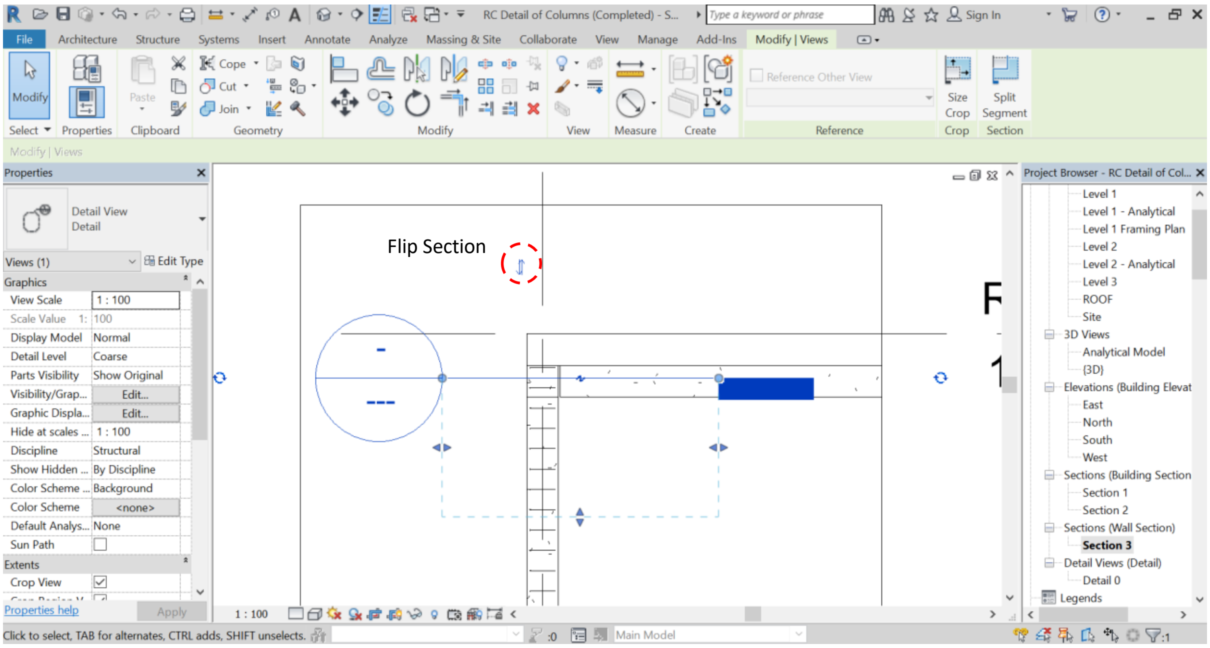
10 Create similar rebar set with maximum spacing set to 200mm for the same column from Level 2 to ROOF.



11 Now, create a detail view to add main rebars for the column.

Click View tab ➢ Create panel ➢ Section and choose Detail View in the Type Selector. Then create a Detail View near ROOF level as shown below. (Click the Flip Section button if your detail view is opposite to what shown below.)

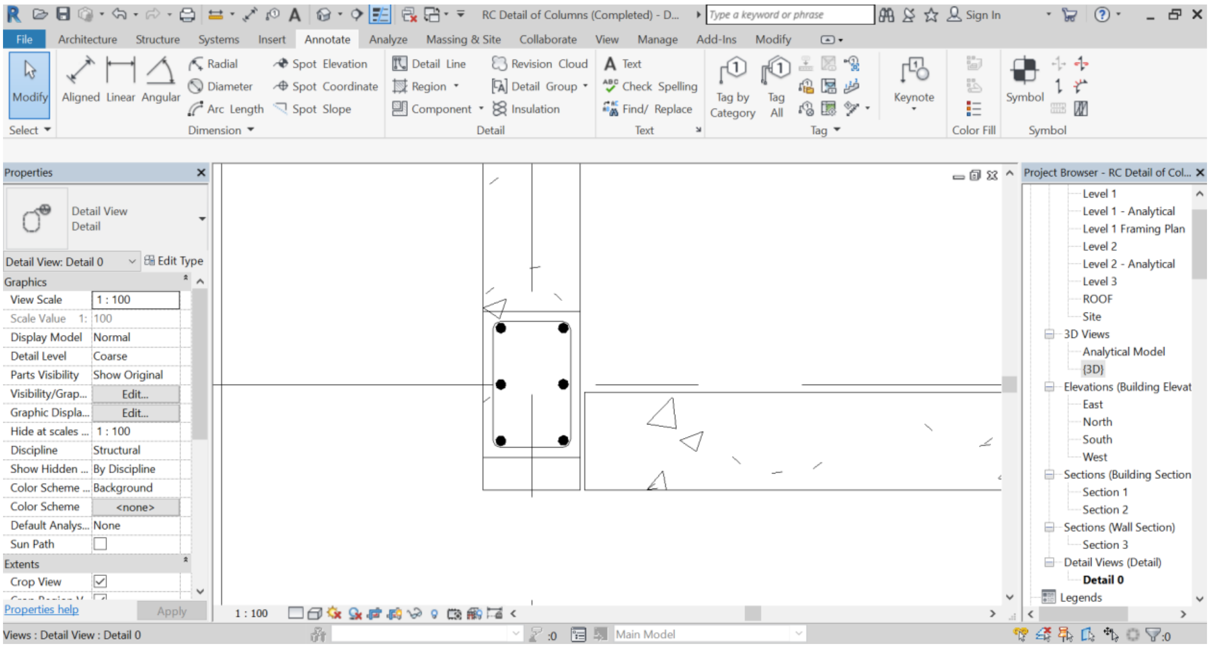




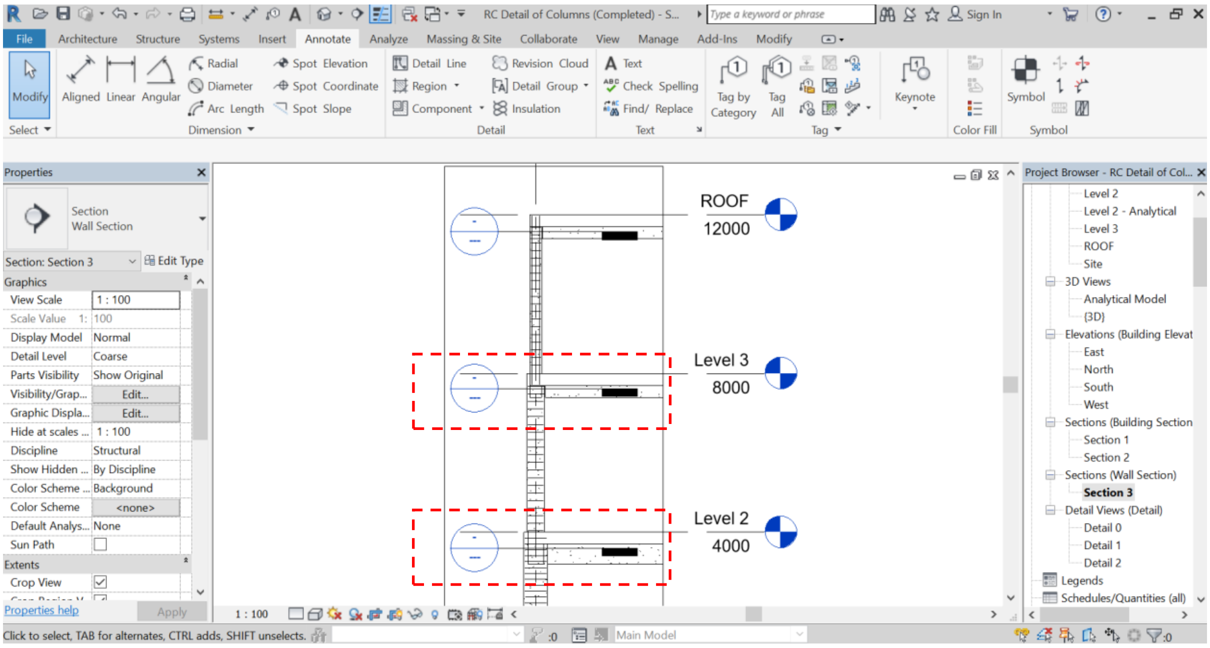
12 Right click on the Detail View and select “Go to View” to go to this detail view. Now, zoom to the column and start placing rebars.

Click Structure tab ➢ Reinforcement panel ➢ Rebar. Select Rebar Shape: 00 with Rebar Bar 32T. Set Placement Plane to “Current Work Plane”, Placement Method to “Perpendicular to Cover” and Rebar Set to “Fixed Number and Quantity = 3”.

Hover over to the left and right edges of the column and place the rebar sets.

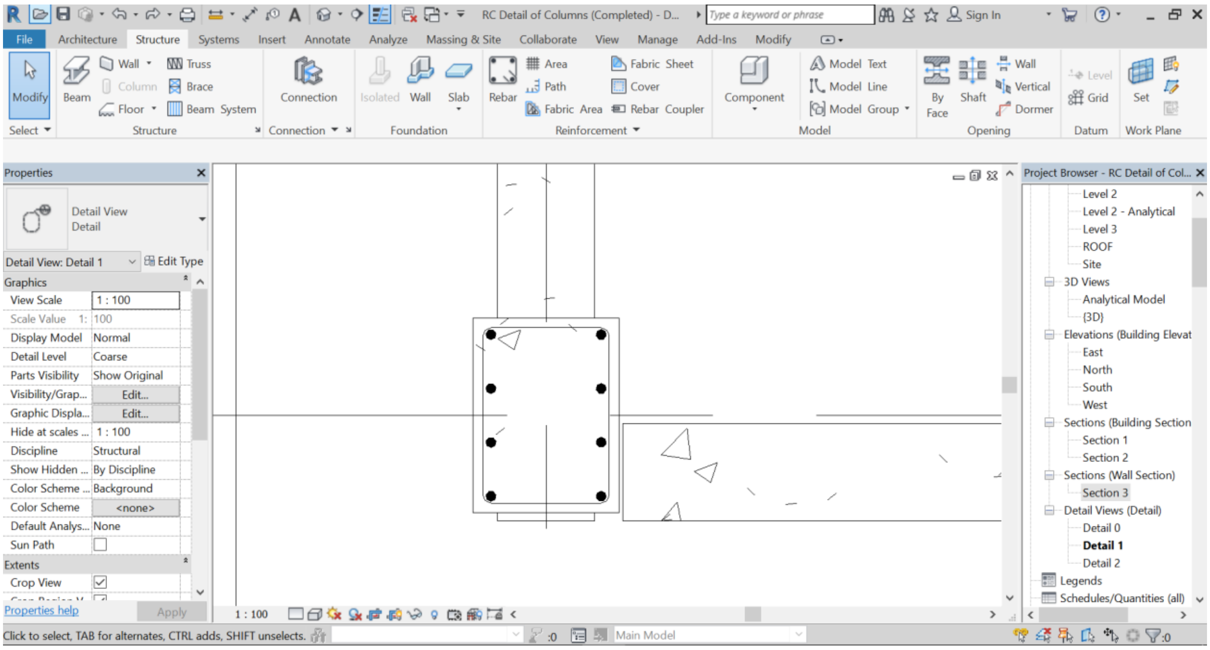


13 Go to Wall Section view (Section 3) and create two more Detail Views near Level 3 and Level 2 by copying the existing detail view.



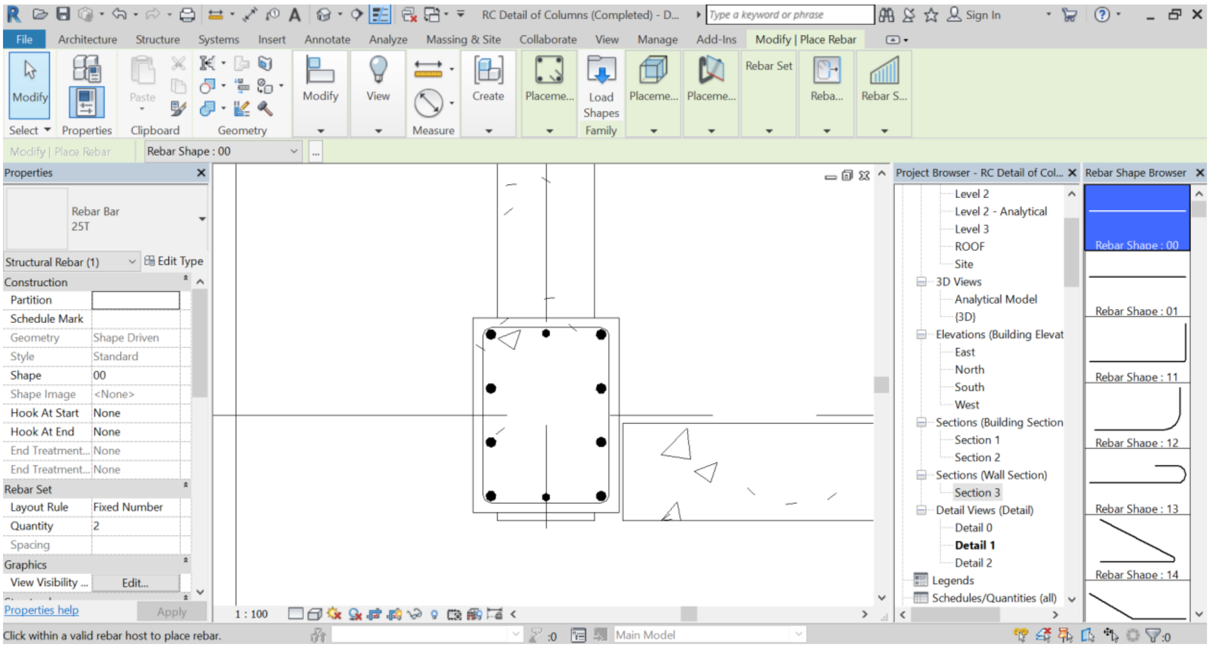
14 Go to the detail view near Level 3 (double click on it). Place rebars along the left and right edges of the column with the following settings.

Rebar Shape: 00 with Rebar Bar 32T. Set Placement Plane to “Current Work Plane”, Placement Method to “Perpendicular to Cover” and Rebar Set to “Fixed Number and Quantity = 4”.

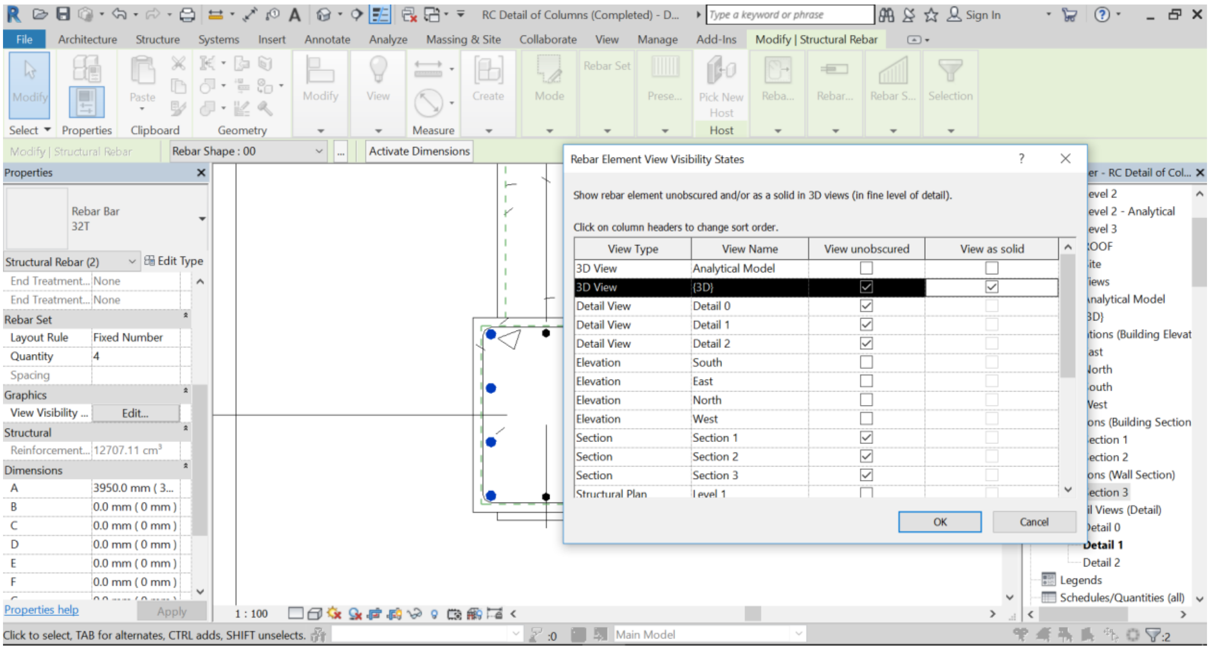


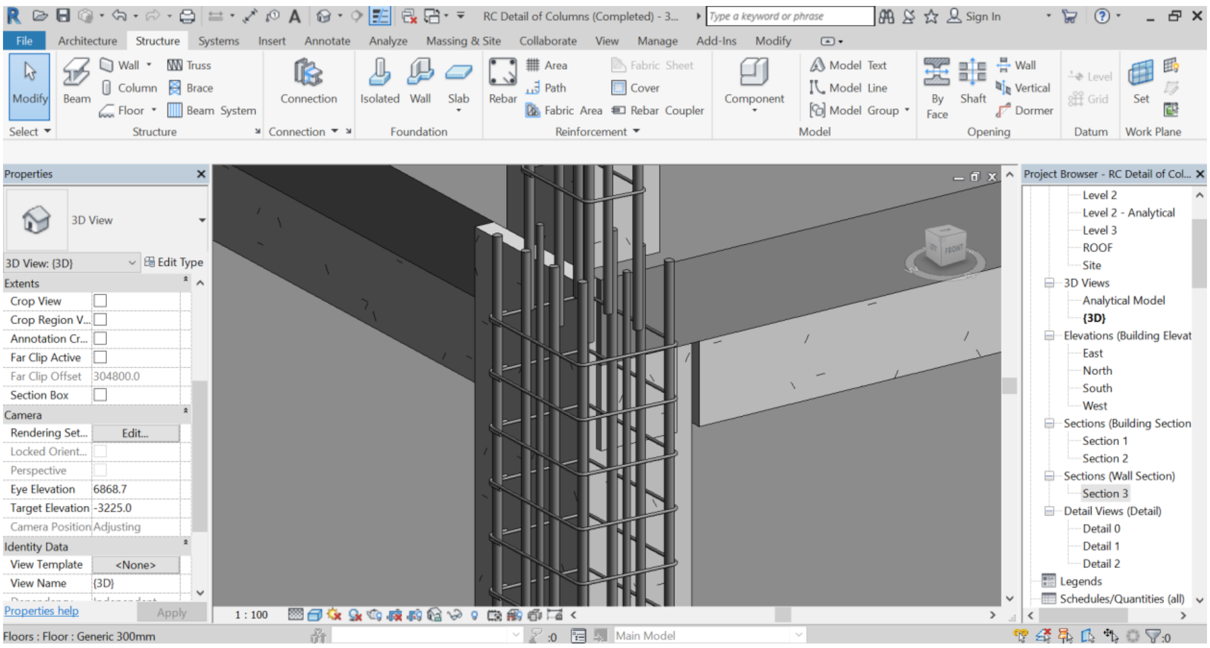
15 Place another set at the middle face of the column using the following setting:

Rebar Shape: 00 with Rebar Bar 25T. Set Placement Plane to “Current Work Plane”, Placement Method to “Perpendicular to Cover” and Rebar Set to “Fixed Number and Quantity = 2”.

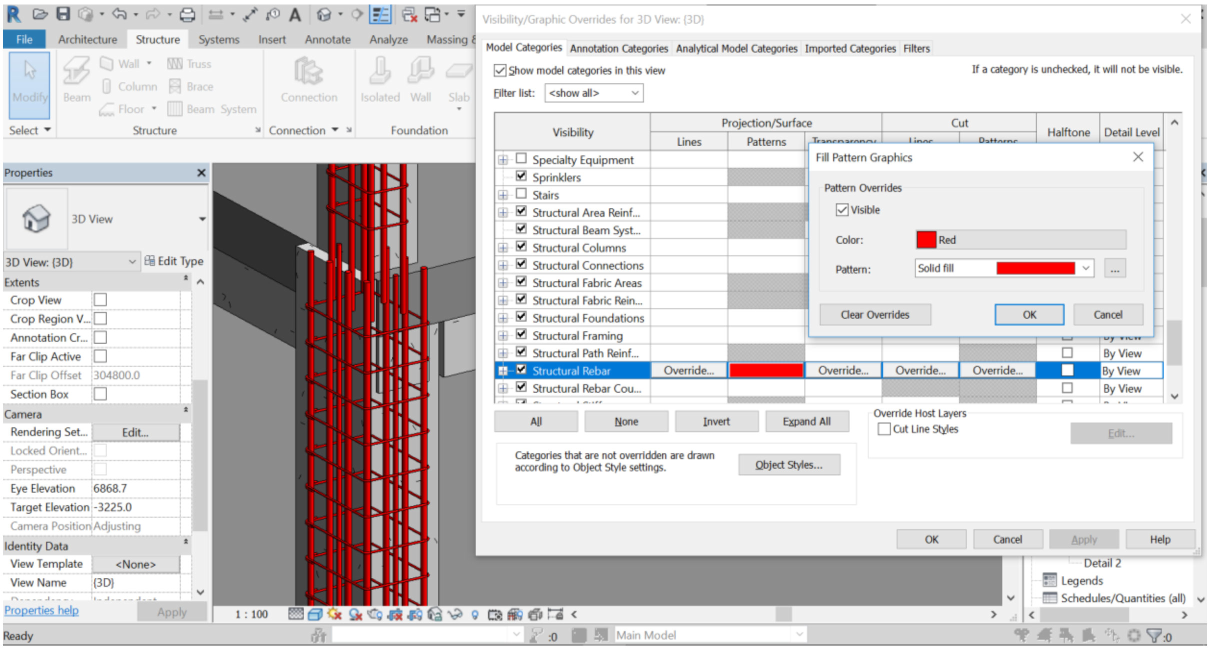


Remember to set the View Visibility if you want to view these rebars unobscured in 3D view.

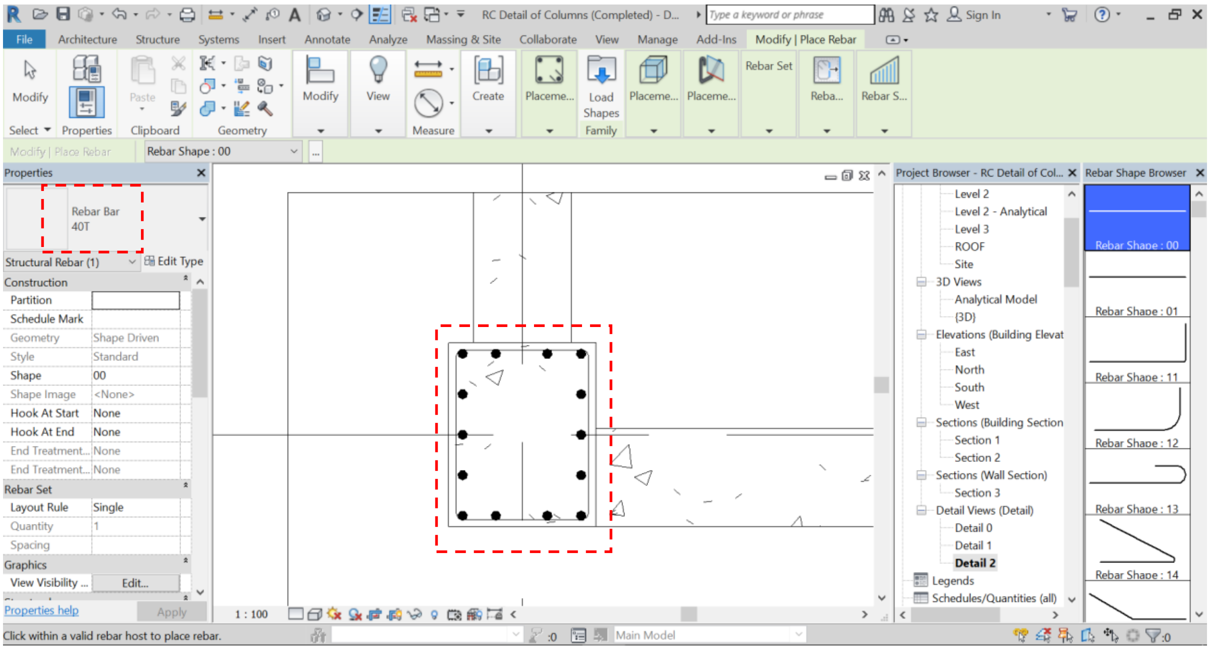




You can change the colour of the rebars for easy viewing. Type “vv” for Visibility/Graphics control and edit the setting for “Structural rebar” as below.



16 Use the steps mentioned above, place the following rebars for the column in Detail 2.



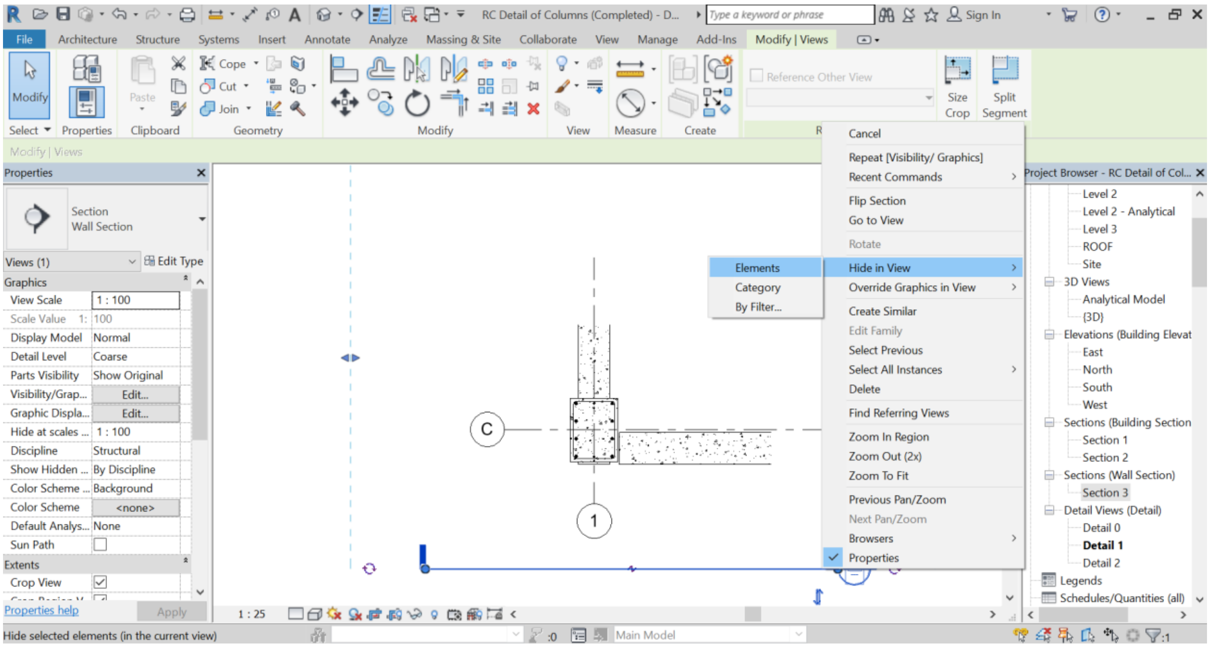
17 Now, you can tidy up the detail views for R.C. detail drawing of the column.

(1) Change the scale of the view to 1:25.

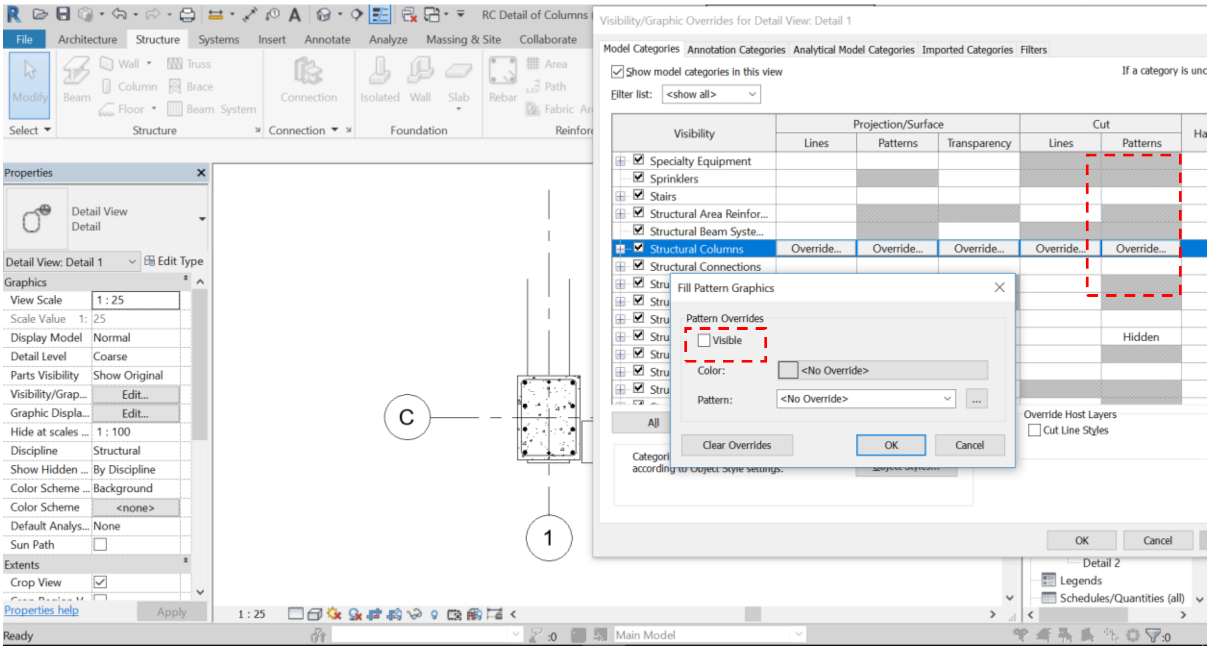
(2) Hide the crop region by clicking the button . (Buttons near the bottom of the drawing area.)

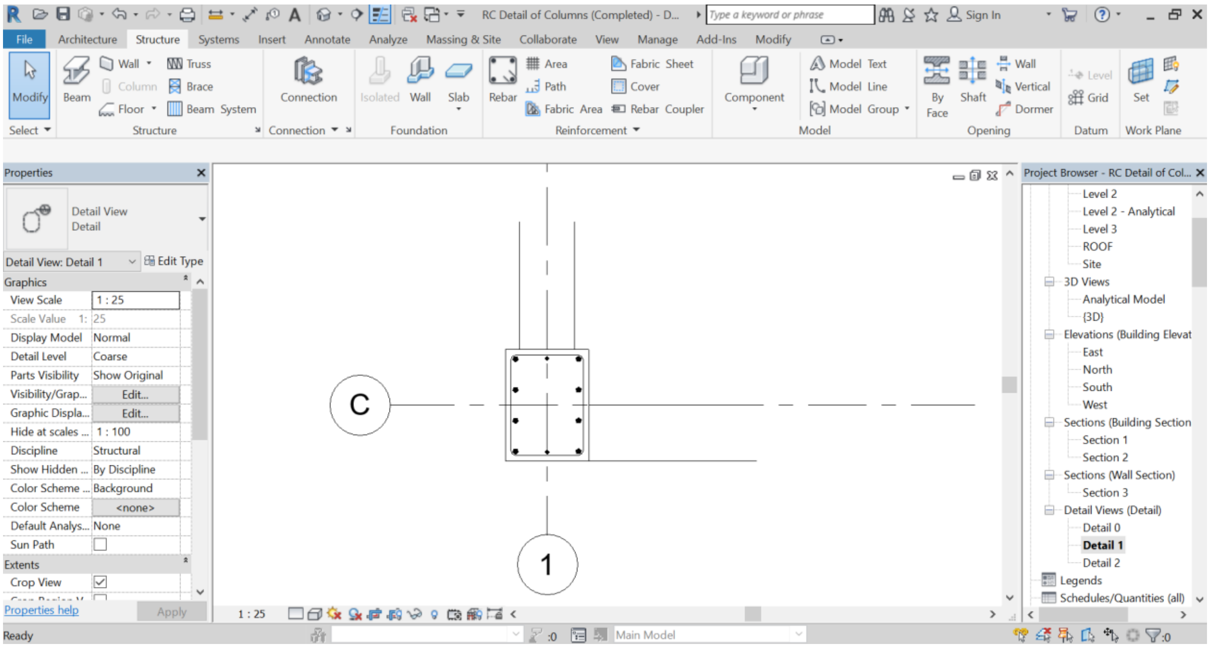
(3) Move the Grid Bubbles closer and hide the bubbles at the opposite ends.

(4) Select the section view, right click and hide it in view.

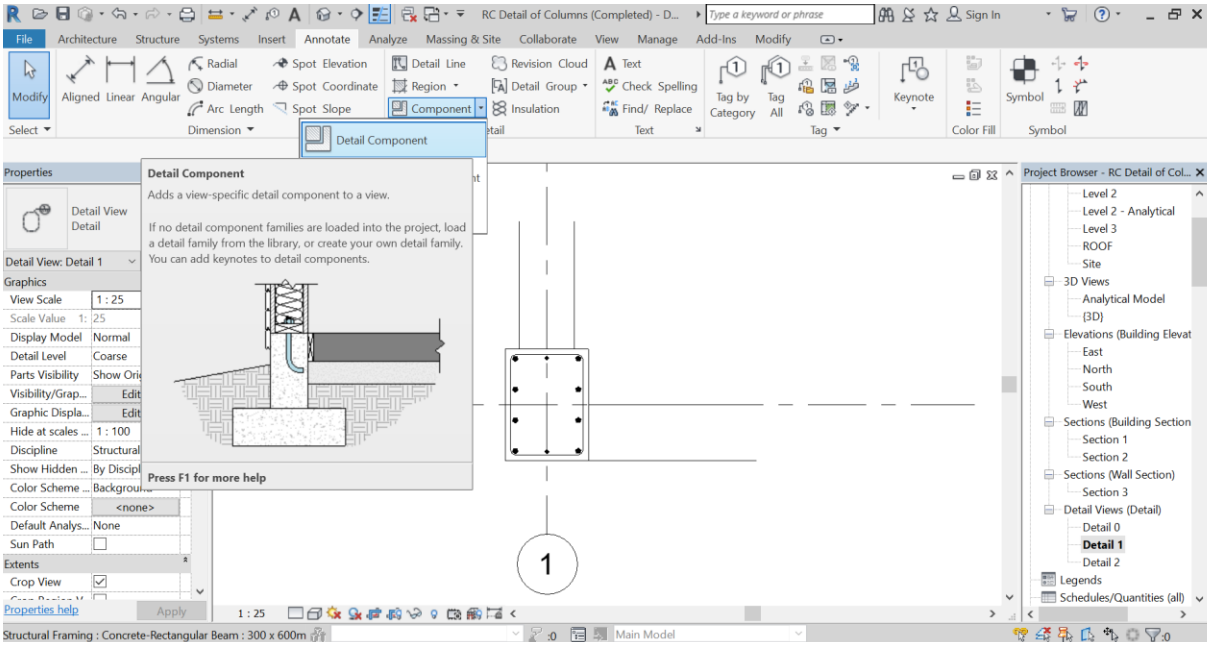


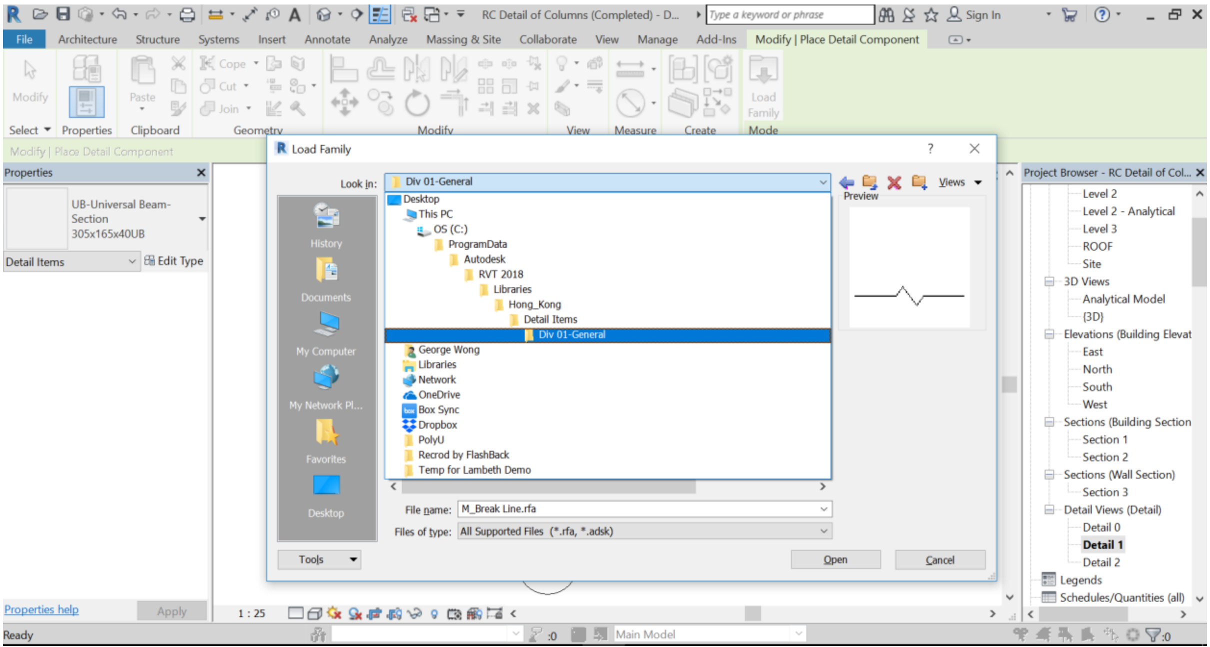
(5) Hide the pattern of concrete for beams and columns. (Go to Visibility/Graphics control and set turn the Cut Pattern not visible for Structural Framing and Structural Columns.





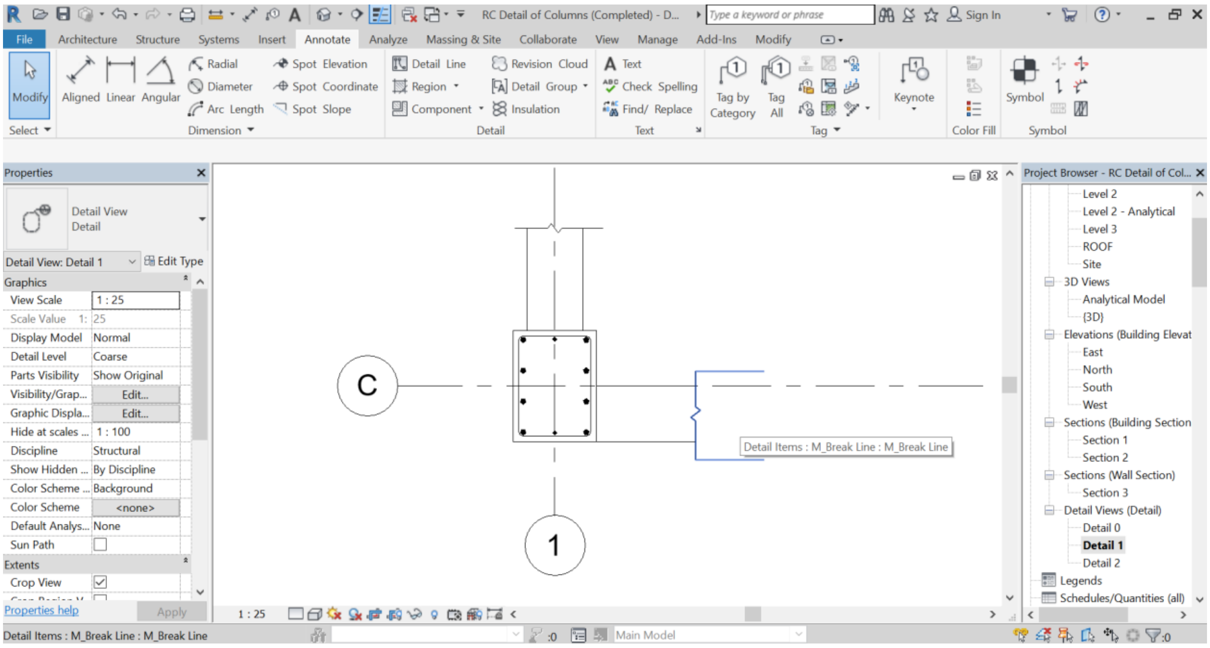
(6) Finally add some break lines for the beams. Break lines are Detail Components and can be found in Annotate tab ➢ Detail panel ➢ Component. Click "Load Family" and find the break line detail item "M_Break Line" from the path shown below. (C:\ProgramData\Autodesk\RVT2018\Libraries\Hong_Kong\Detail Items\Div 01-General)



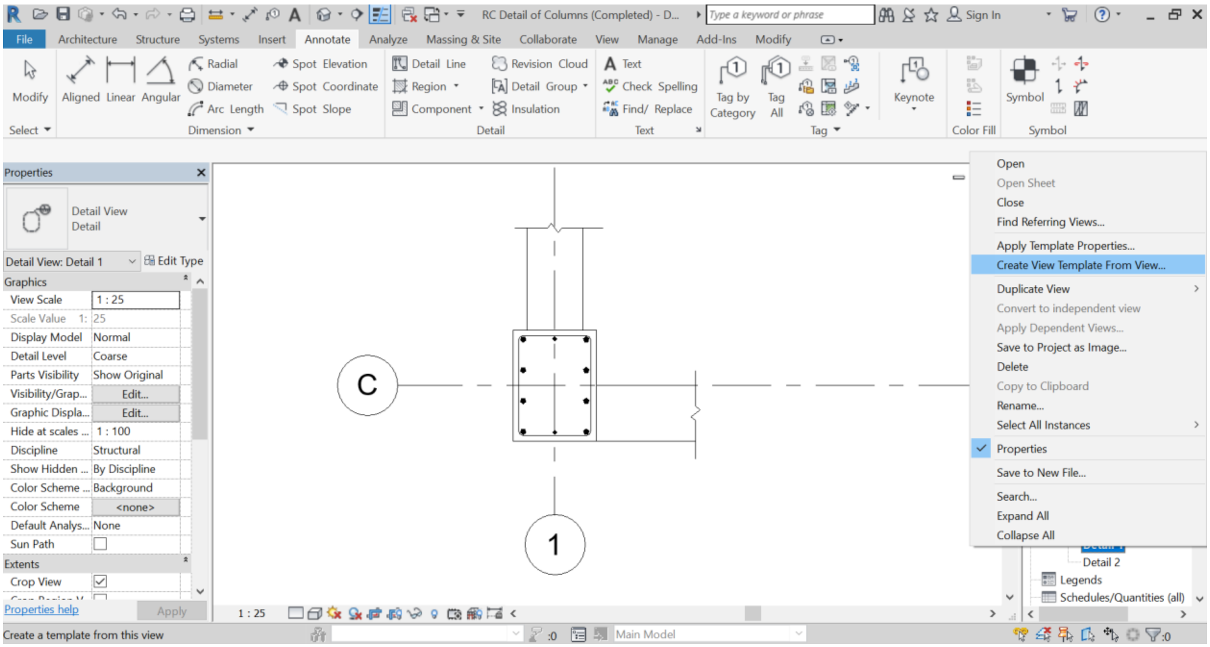


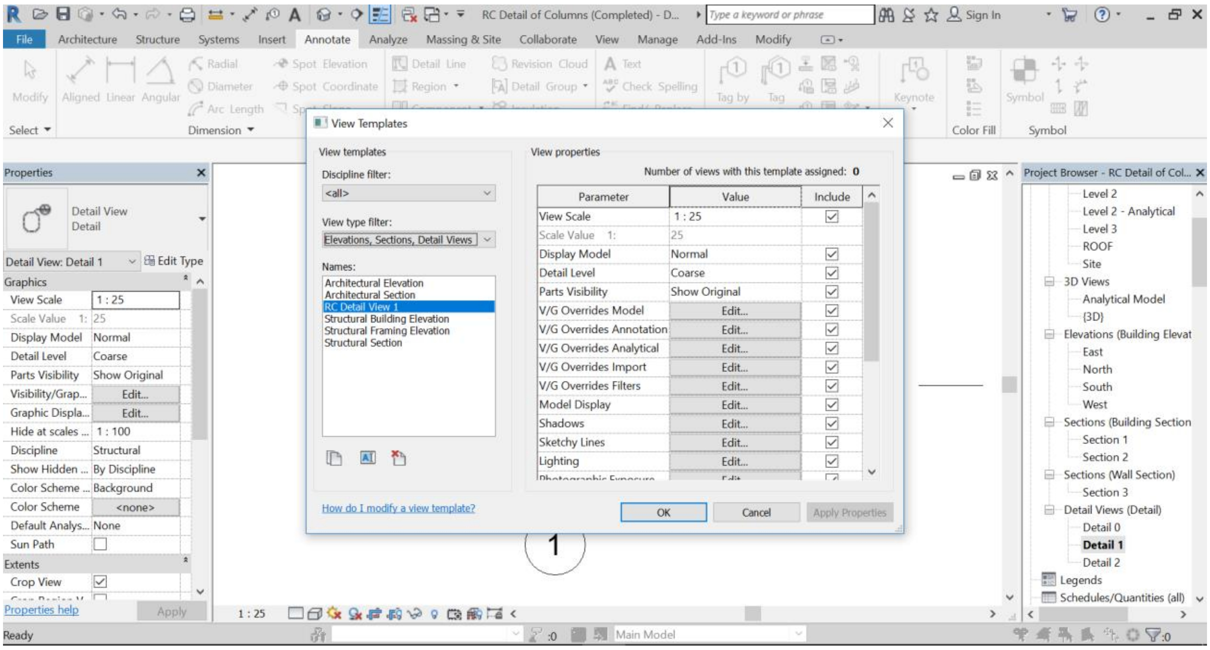
Place the break lines as shown below. (Note: you can press "spacebar" to rotate the break line and sketch it by dragging the arrows.)

Repeat adding break lines for detail views "Detail 0" and "Detail 2".



(7) Now, save the view setting as a View Template and apply it to other detail views. To do this, select Detail 1 from the Project Browser and right click to "Create View Template From View". Name it as "RC Detail View 1" and click "OK".

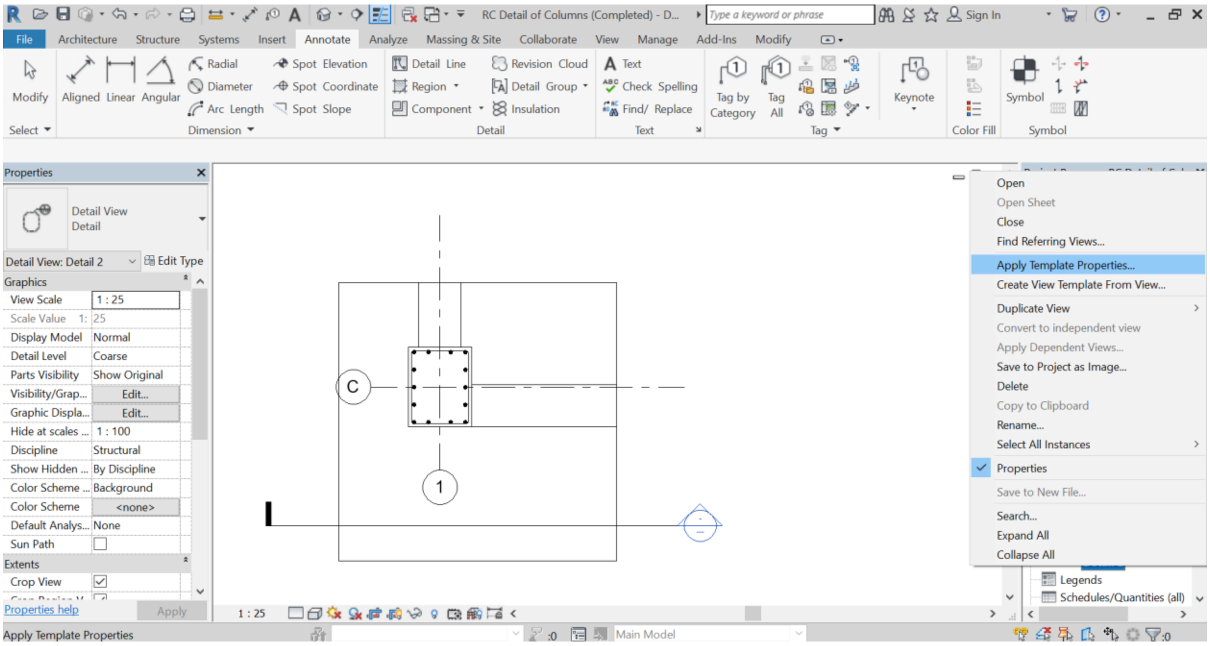


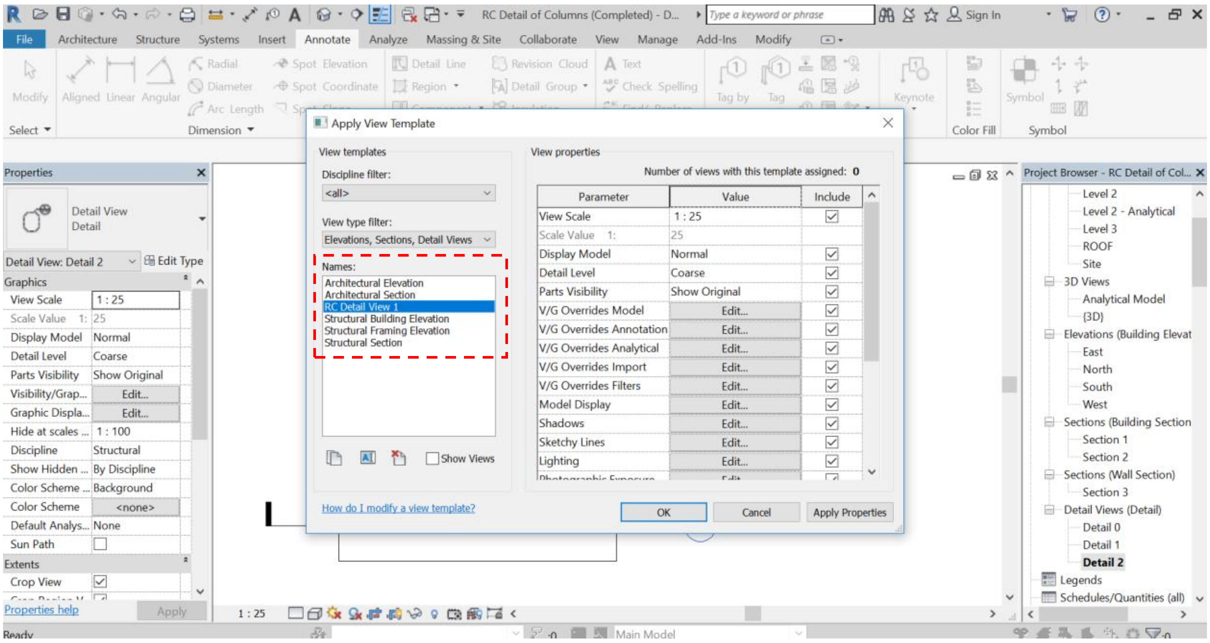


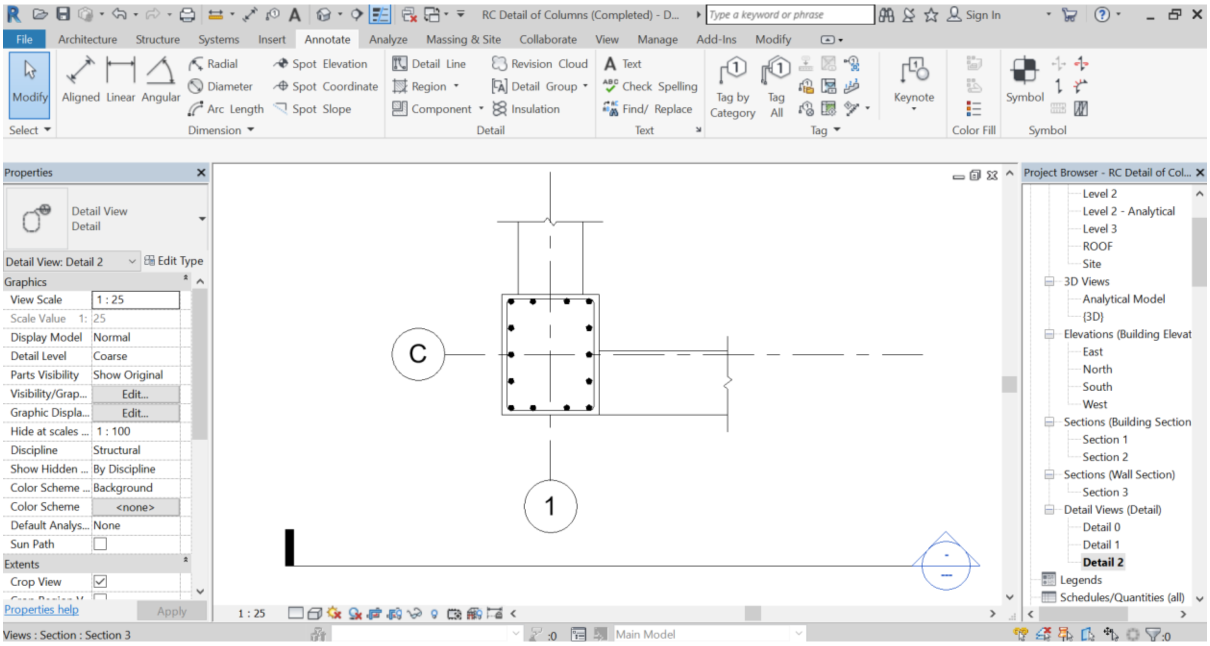
(8) To use the view template "RC Detail View 1" for other detail views. Select Detail 2 from the Project Browser and right click to "Apply Template Properties".

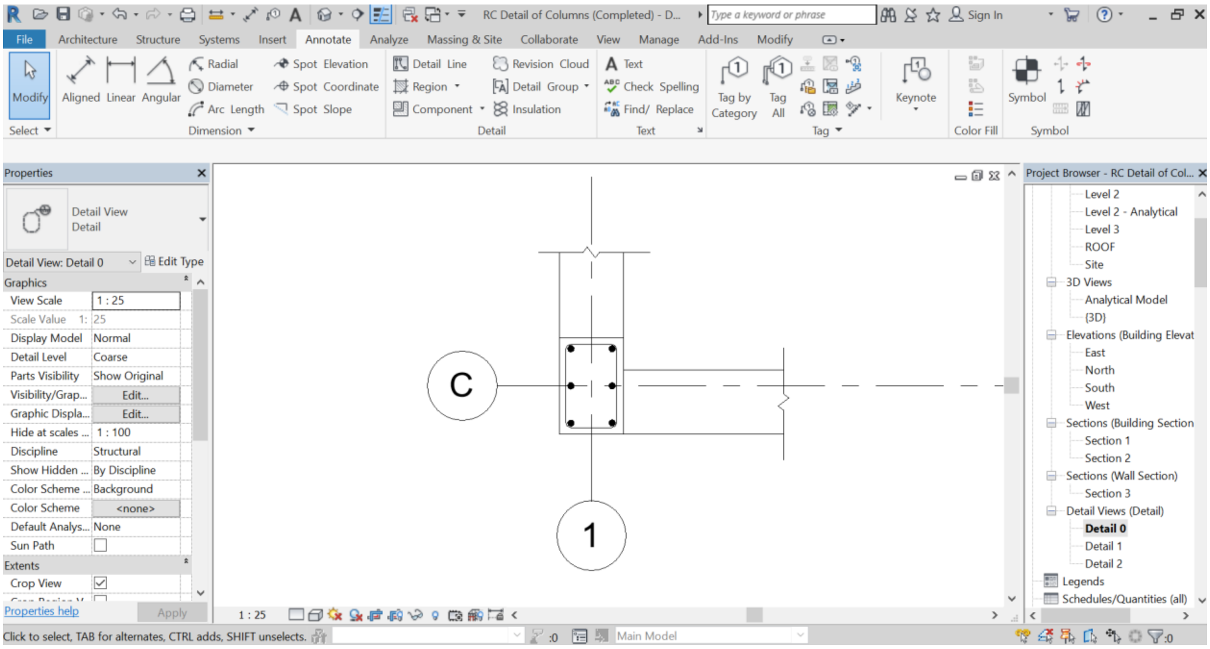
Then find and select the view template named "RC Detail View 1" and click "OK".

Repeat this for detail view "Detail 0".

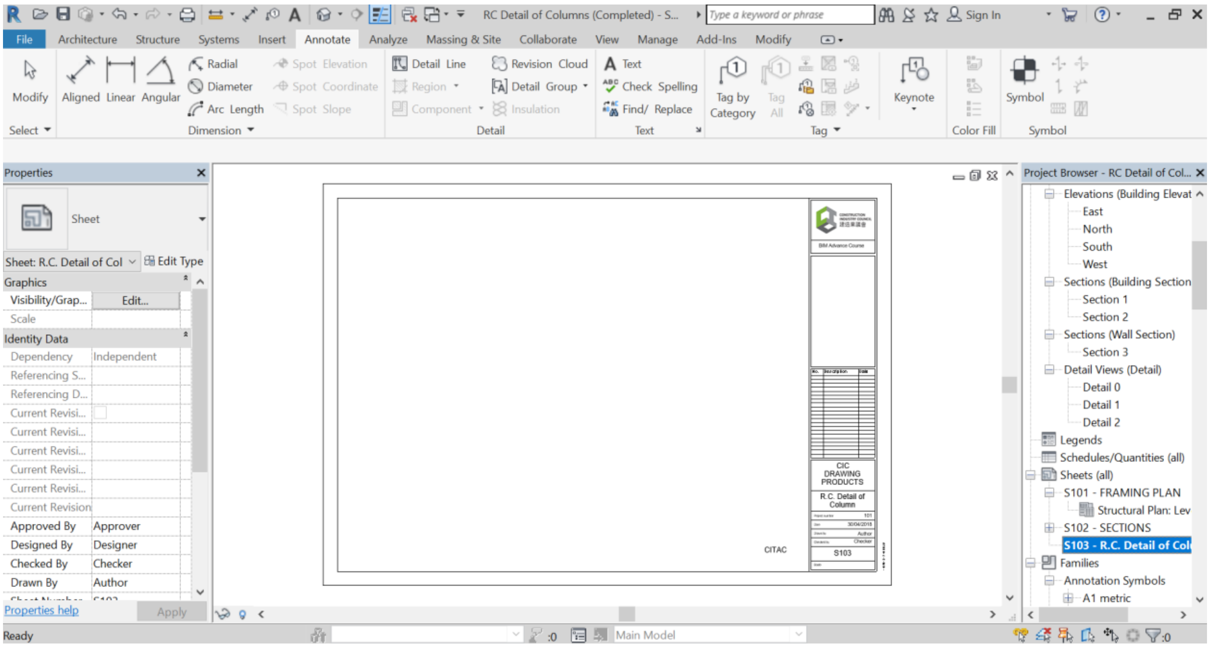

Open
Open Sheet
Close
Find Referring Views...
Apply Template Properties...
Create View Template From View...
Duplicate View
Convert to independent view
Apply Dependent Views...
Save to Project as Image...
Delete
Copy to Clipboard
Rename...
Select All Instances
Properties
Save to New File...
Search...
Expand All
Collapse All


Apply View Template
View templates
Discipline filter:
View type filter:
Elevations, Sections, Detail Views
Names:
Architectural Elevation
Architectural Section
RC Detail View 1
Structural Building Elevation
Structural Framing Elevation
Structural Section
View properties
Number of views with this template assigned: 0
OK
Cancel
Apply Properties

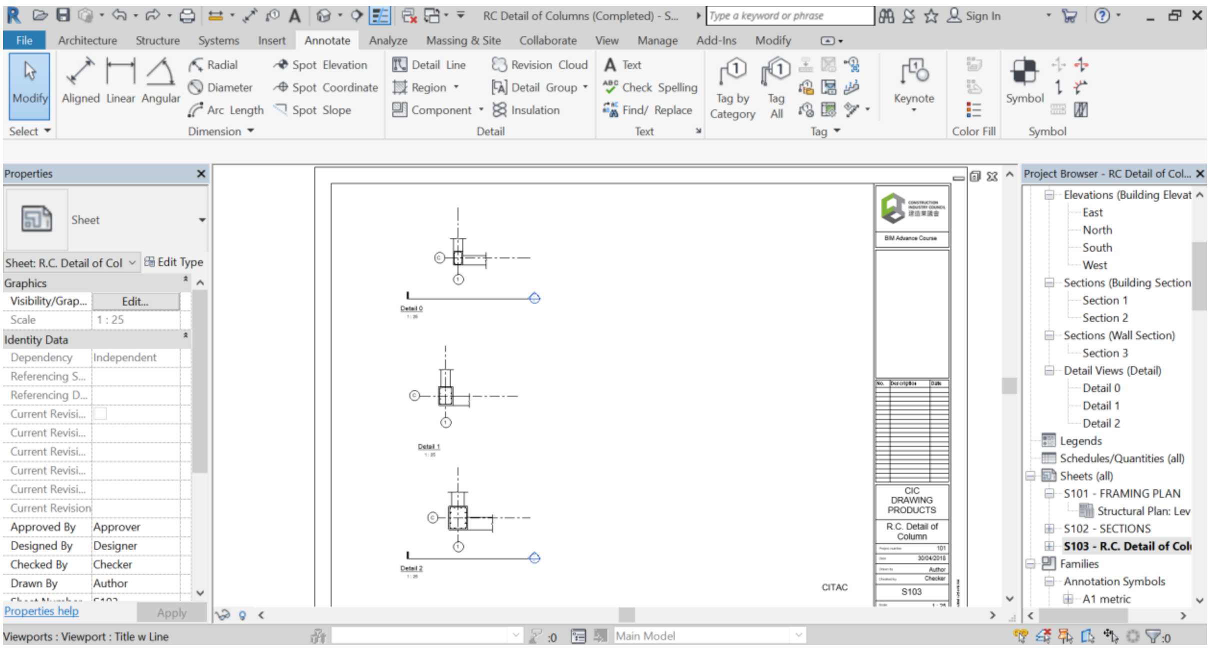




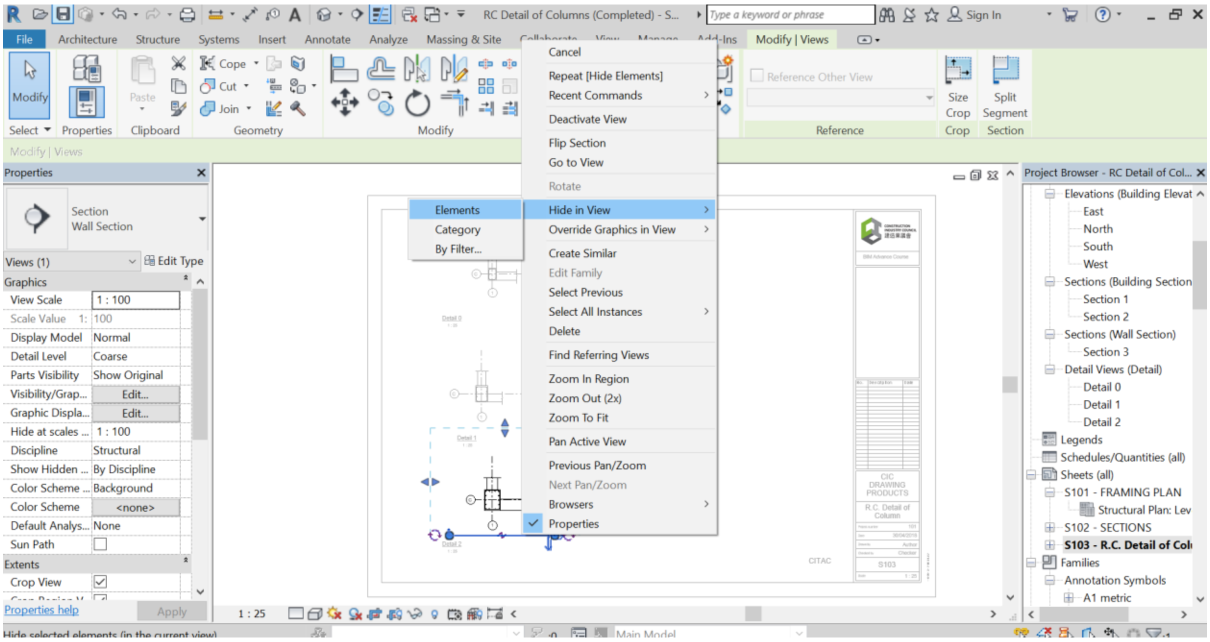
18 To create the drawing R.C. Detail of Column, add a new sheet (right click on “Sheets (all)” and select “New Sheet”). Use the CIC Course Title Block then rename this sheet to “R.C. Detail of Column” (right click and rename).



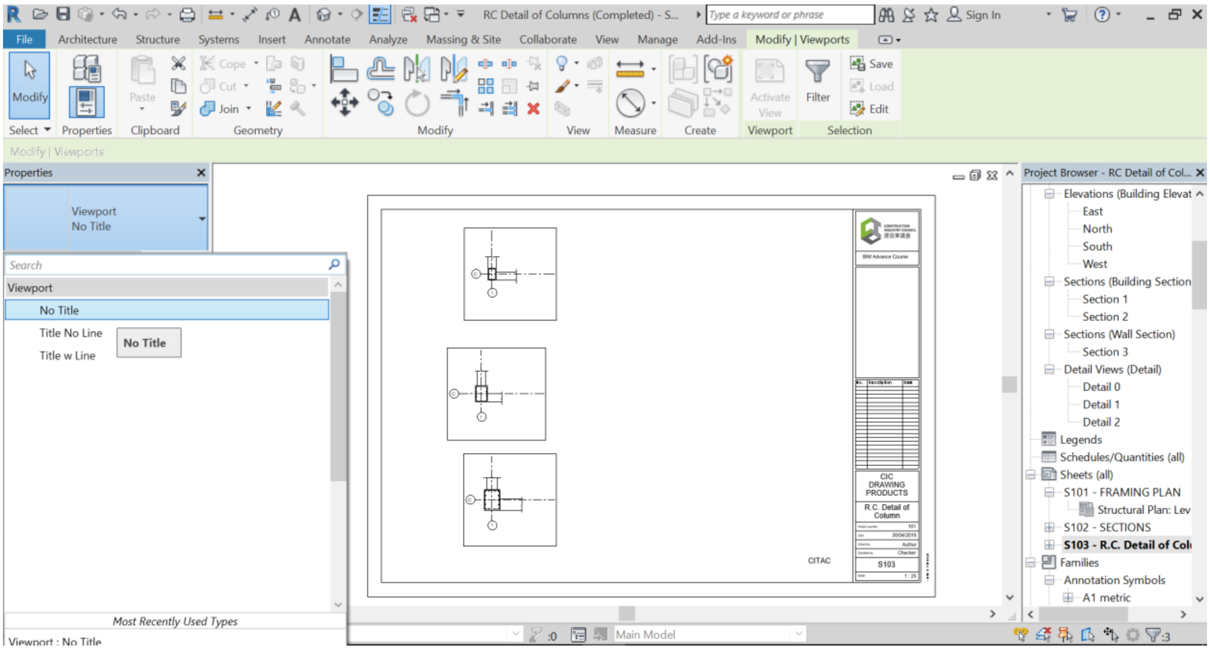
19 While still activated this sheet, select Detail View "Detail 0" and drag it onto the sheet "S103 – R.C. Details of Column". Continue to drag "Detail 1" and "Detail 2" onto the sheet.

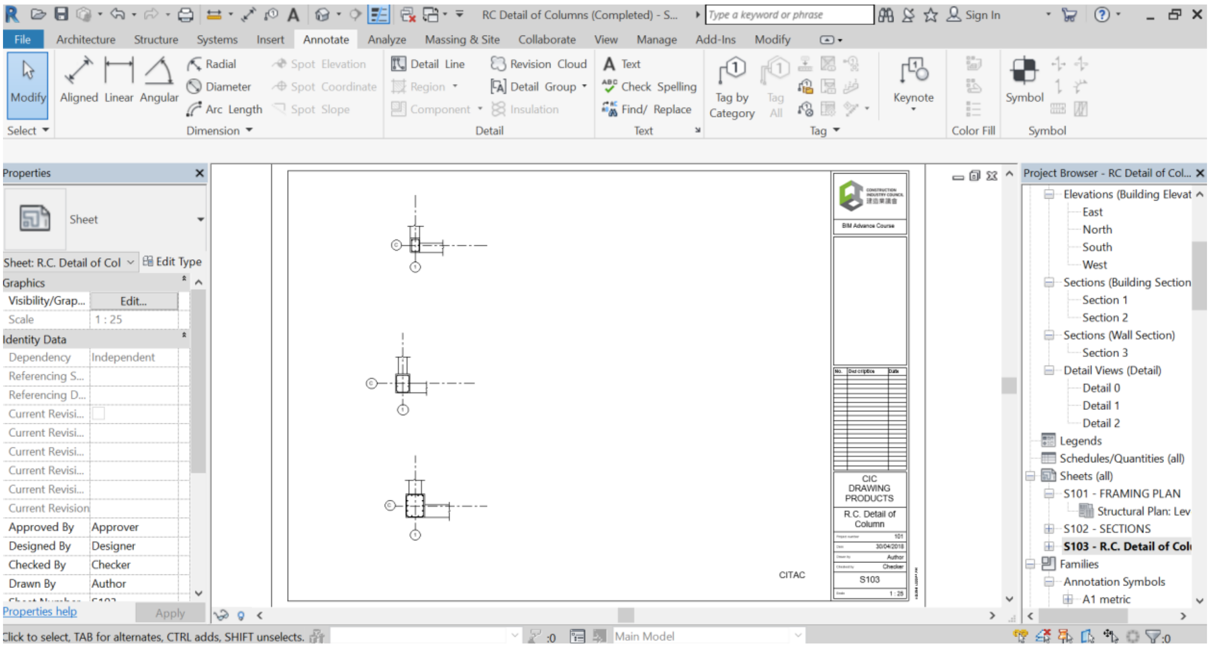


20 Double click the details and select the section line then hide it in view.

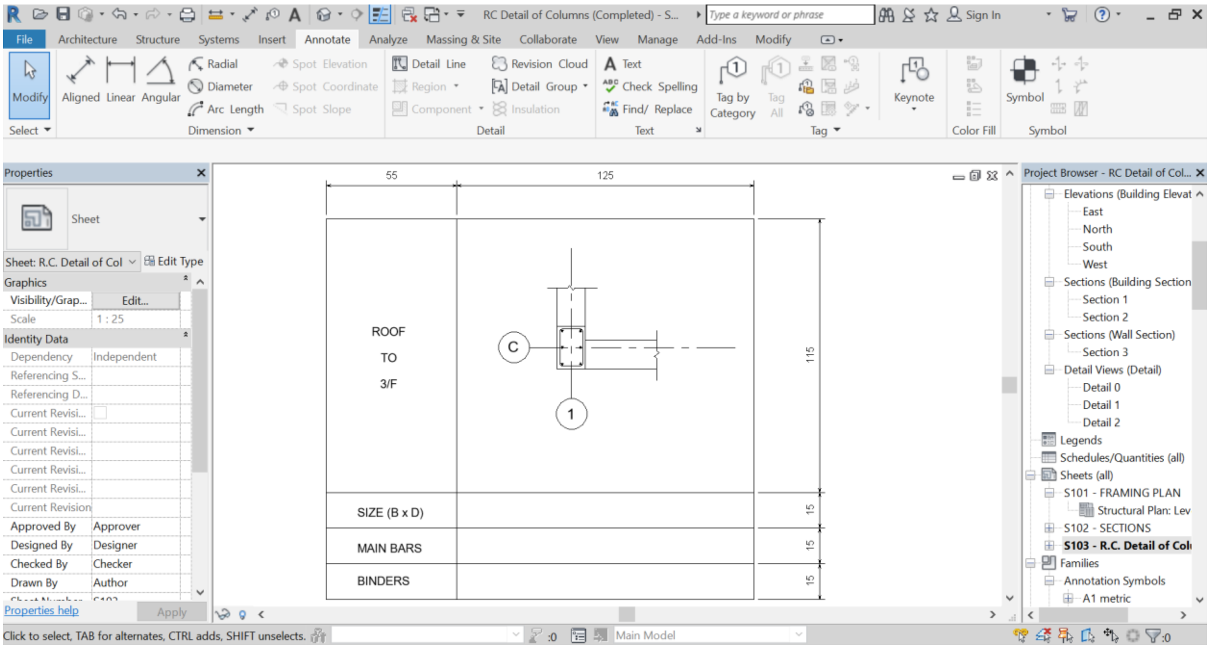


21 Select all viewports and change the type to “No Title”.

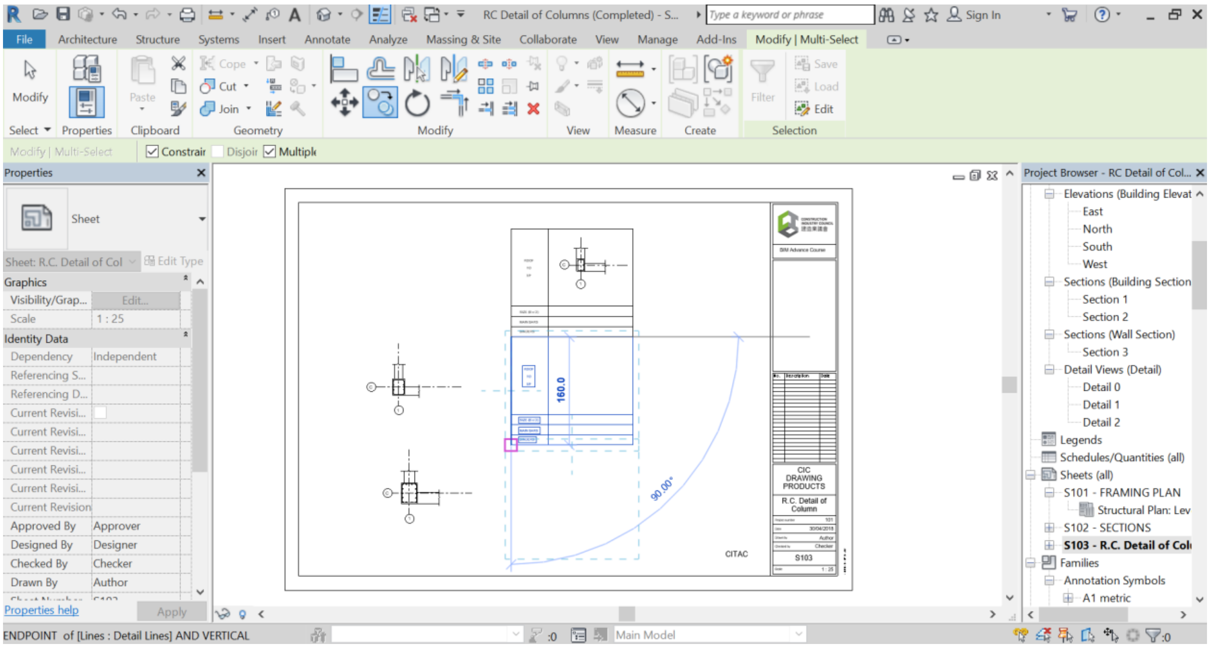




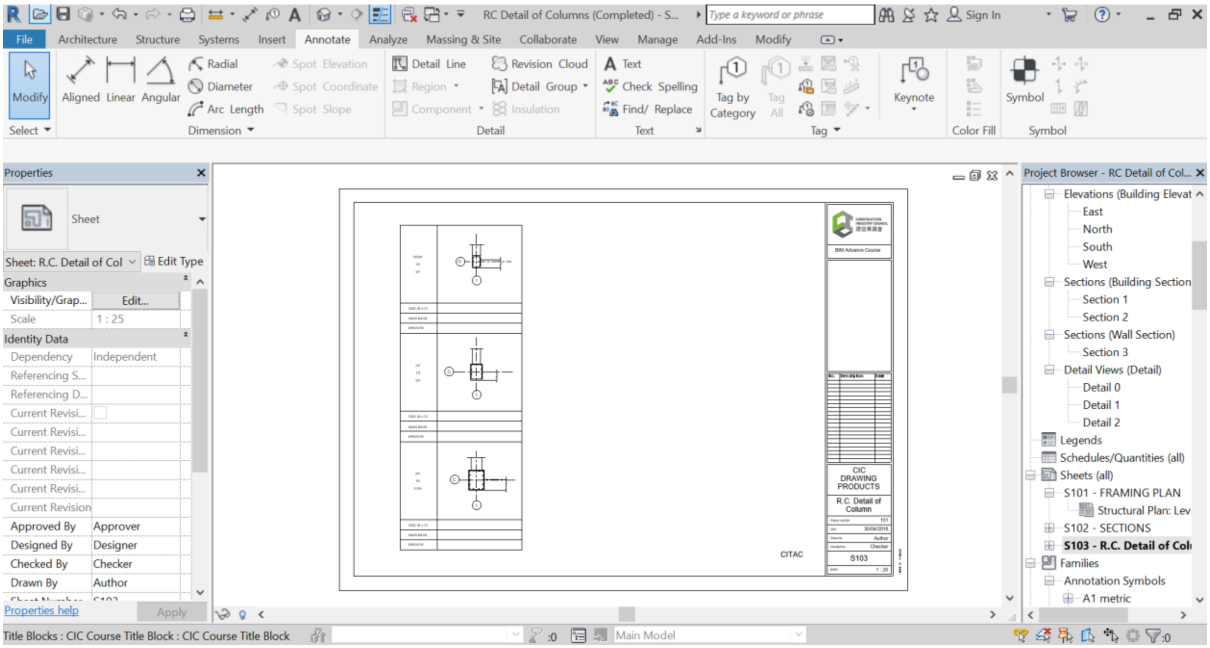
22 Next, create a table to place the details. You can draw the table using “Detail Line” and “Text” from the Annotate tab. Use the dimensions shown below as reference to draw the table.

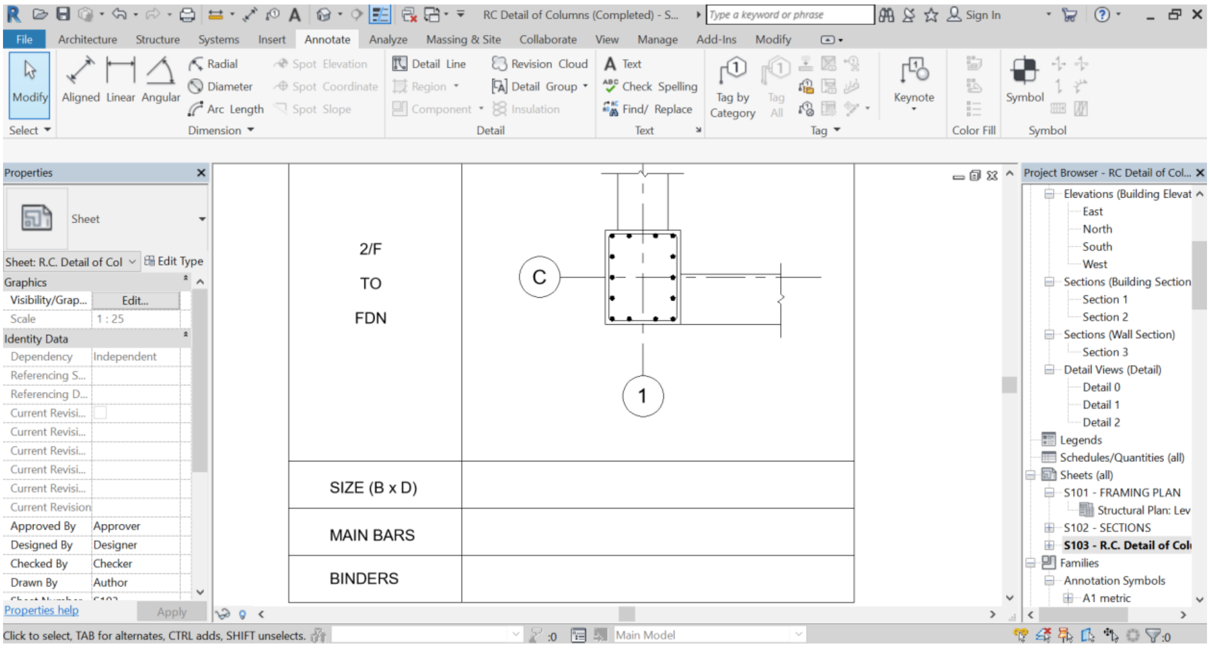


23 Copy the table for one level to form a complete table for the R.C. details.



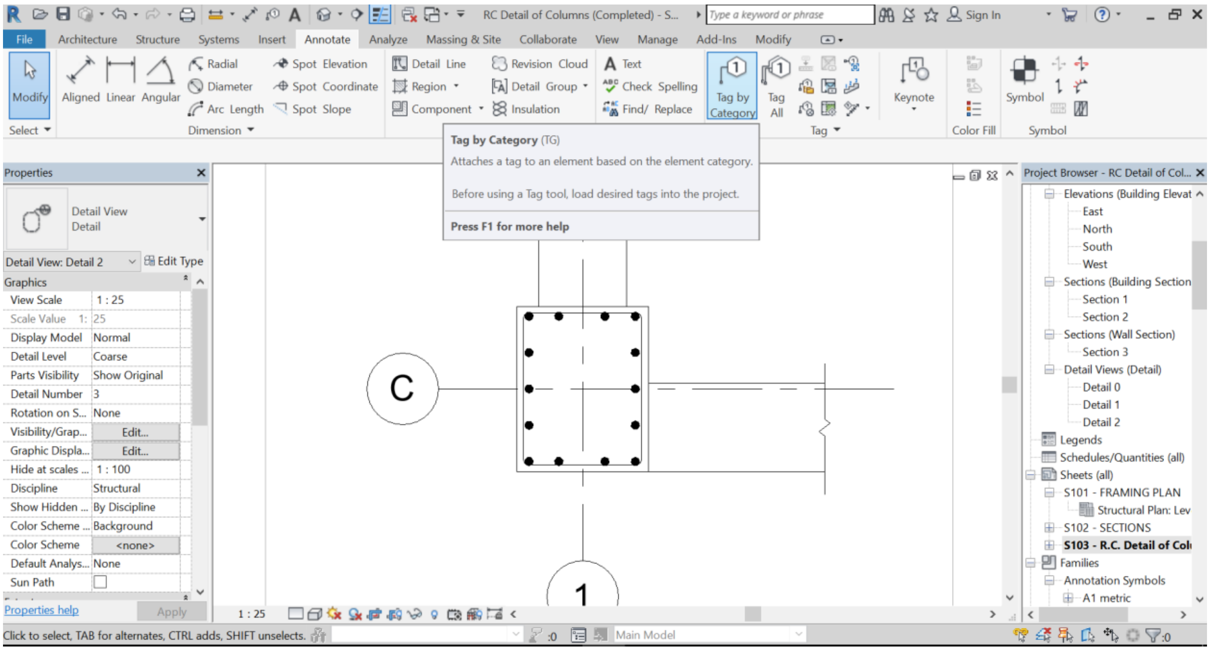
24 Revise the text according to the details and the R.C. Detail of Column is almost ready.

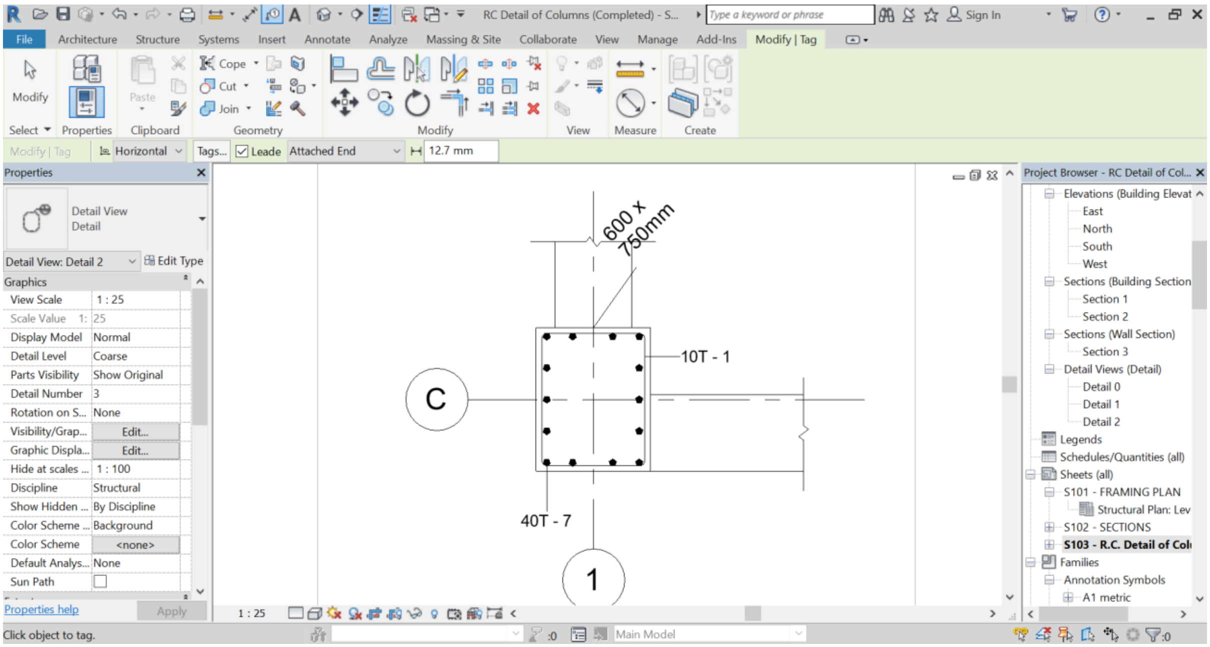




25 Double click on the viewport of the 2/F to FDN detail to annotate it with some tags.

Click Annotate tab ➢ Tag panel ➢ Tag by Category. Then tag the column, one main bar and the link. You will need to modify these tags for better presentation.

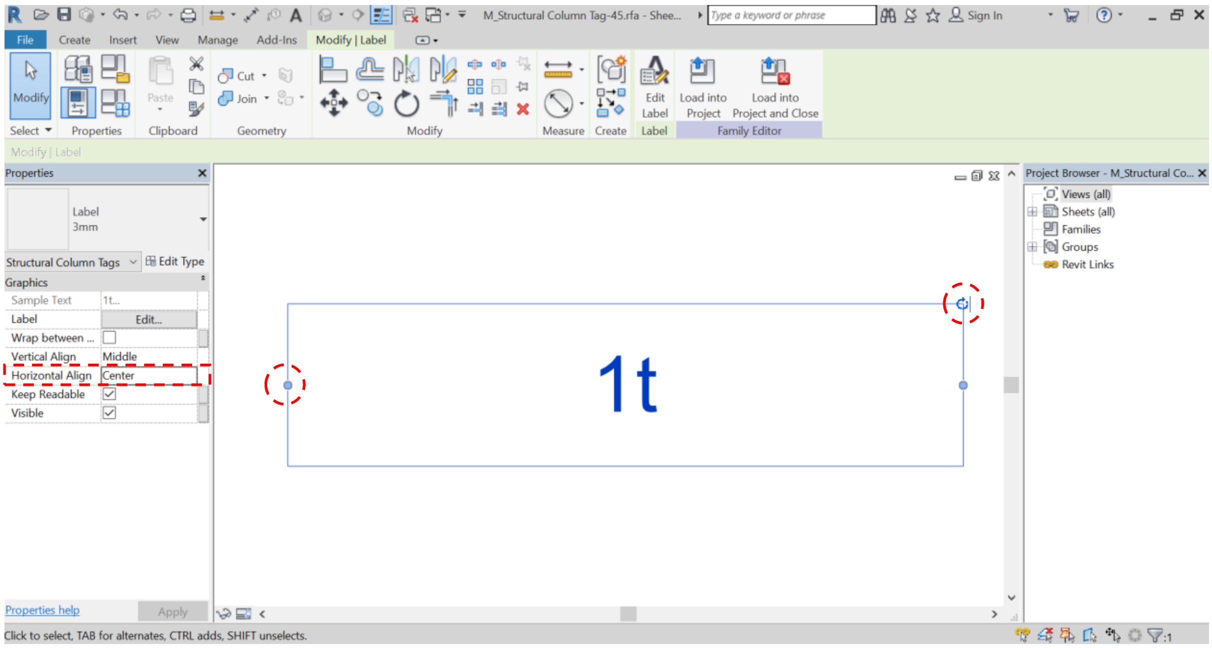




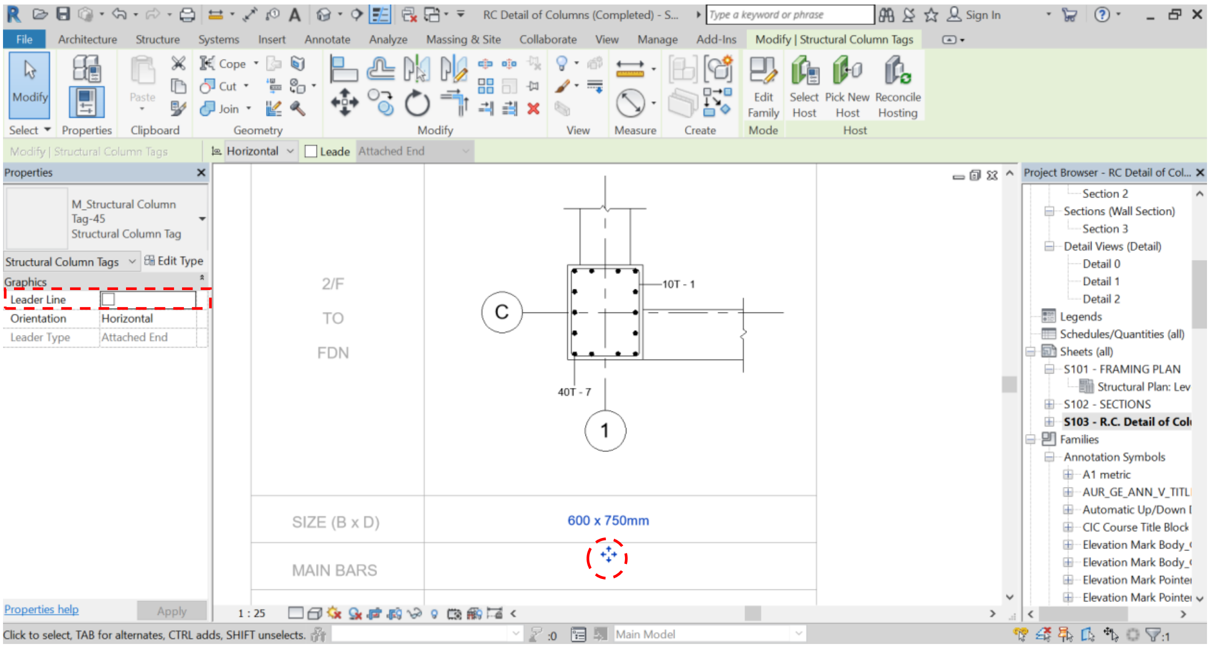
26 Double click on the column tag then edit the following items:

- Rotate the tag to align horizontally.
- Change the “Horizontal Align” to “Center”.
- Drag the tag boundary longer.

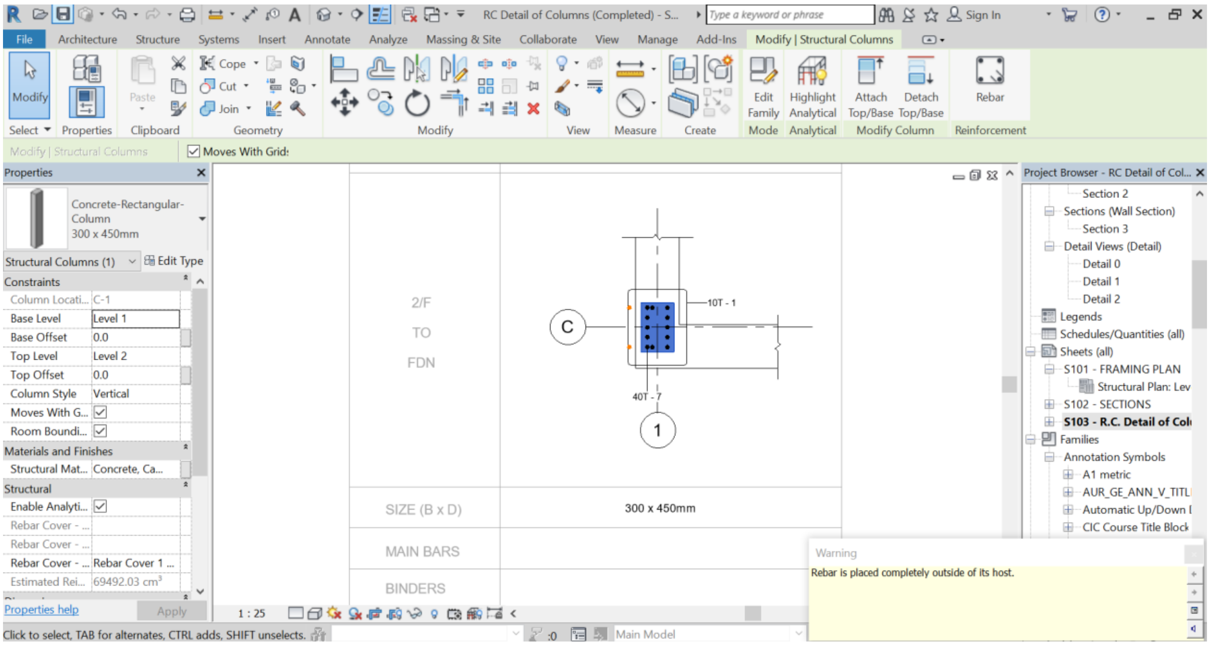
Then “Load into Project and Close”. Do not save the family and “Overwrite the existing version”.



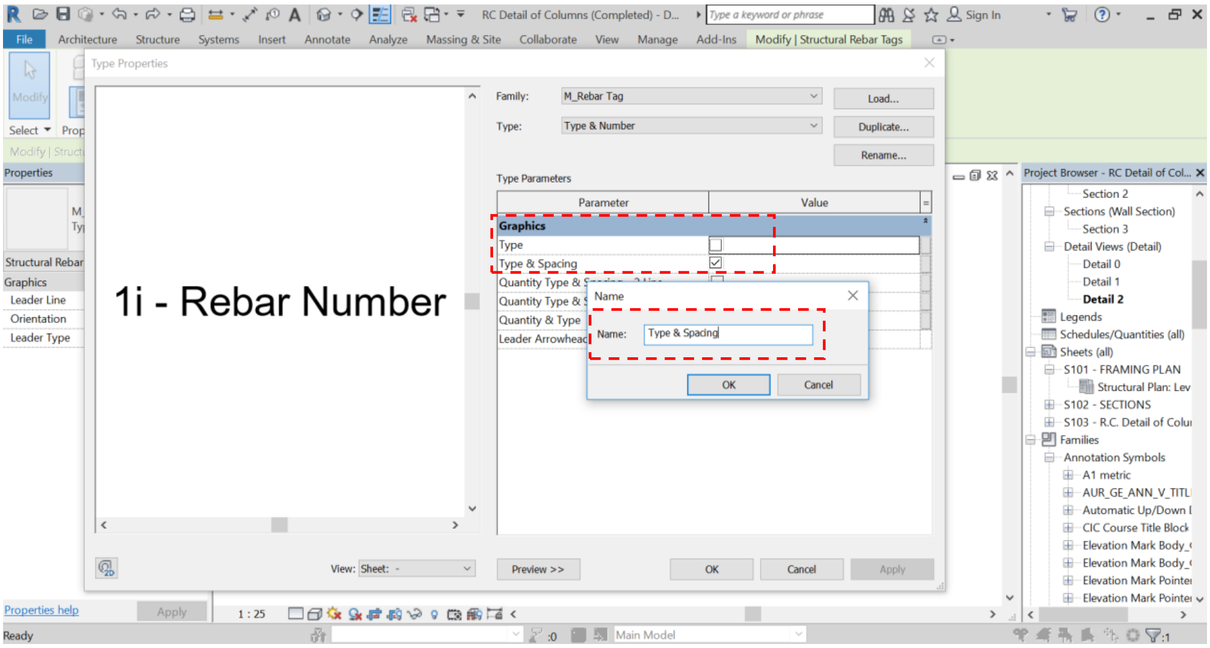
27 Activate the detail in the drawing (Sheet S103 - R.C. Detail of Column). Move the column tag to the box for “Size” and remove “Leader Line” from the tag.



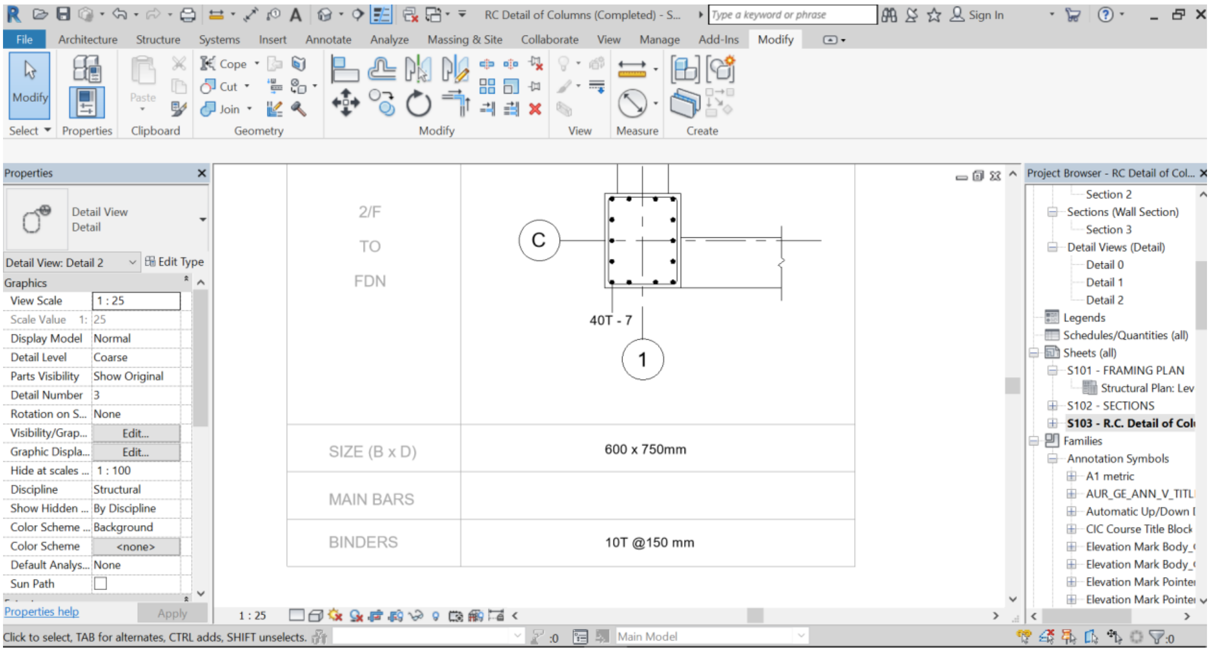
Now, if you change the column size, the drawing will be updated automatically.



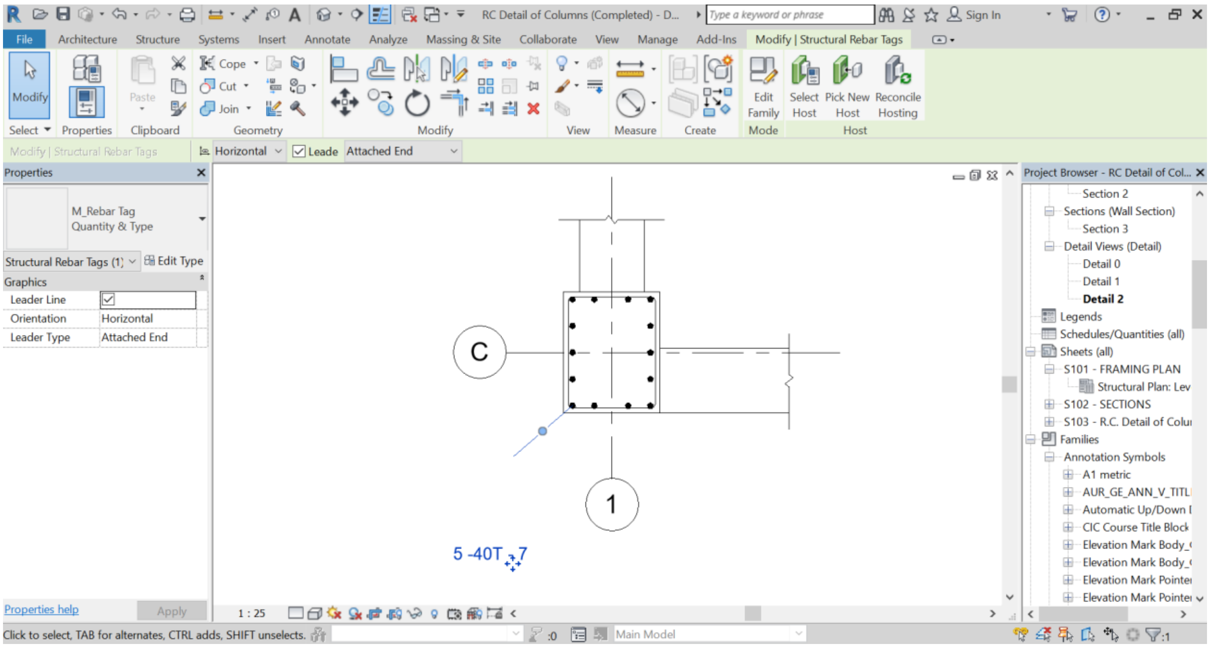
28 Next, change the type of links/binders tag. Select the binder tag and click “Edit Type”. Duplicate and rename to “Type & Spacing”. Select “Type & Spacing” instead of “Type”.



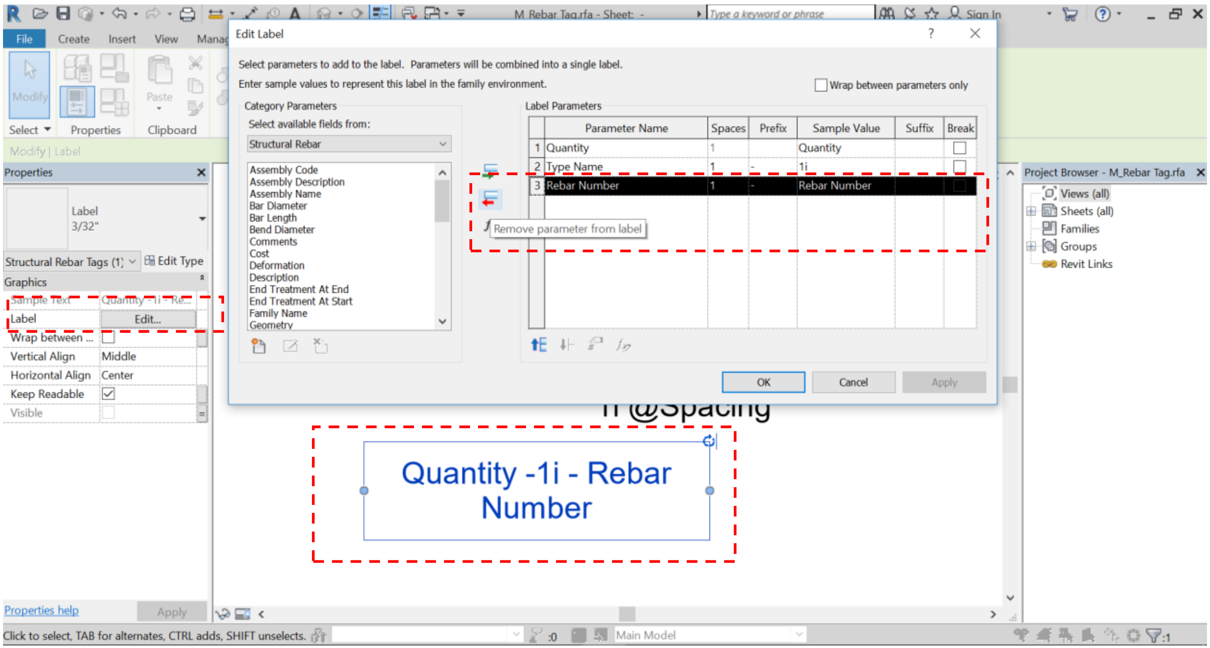
29 Go to the sheet view and double click on the detail then move the tag to appropriate location.



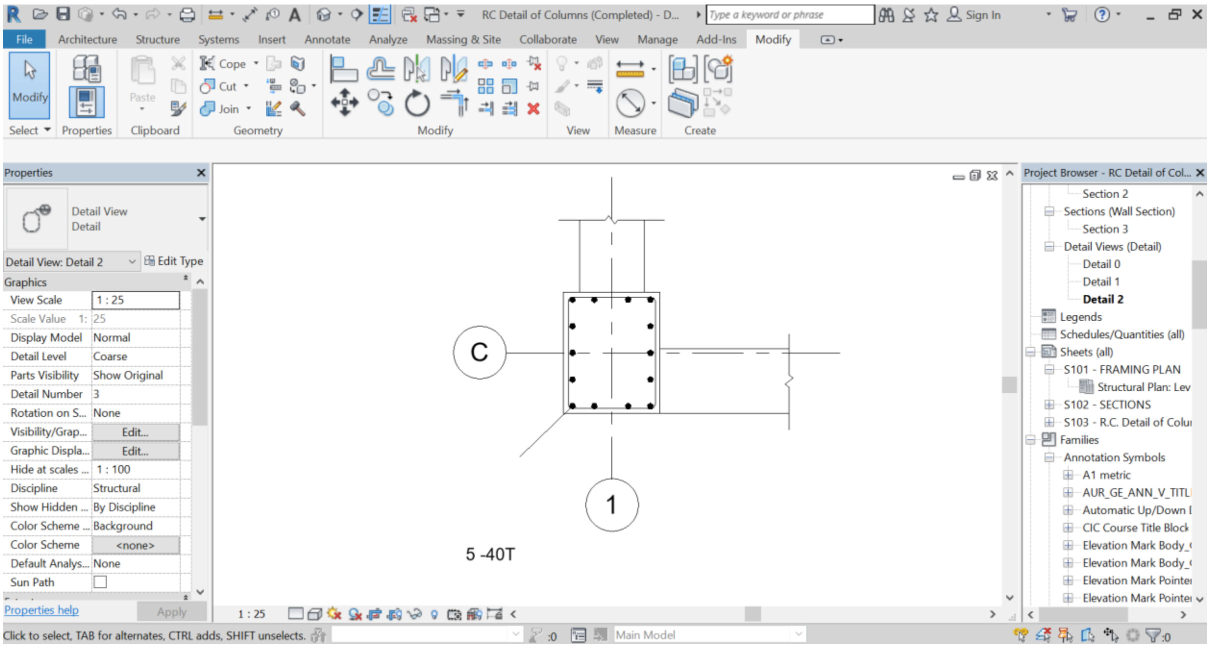
29 Finally, change the type of tag for the main bar to "Quantity Type & Number" and duplicate it with name "Quantity & Type". To get rid of the rebar number, you have to double click on the tag and edit its family.



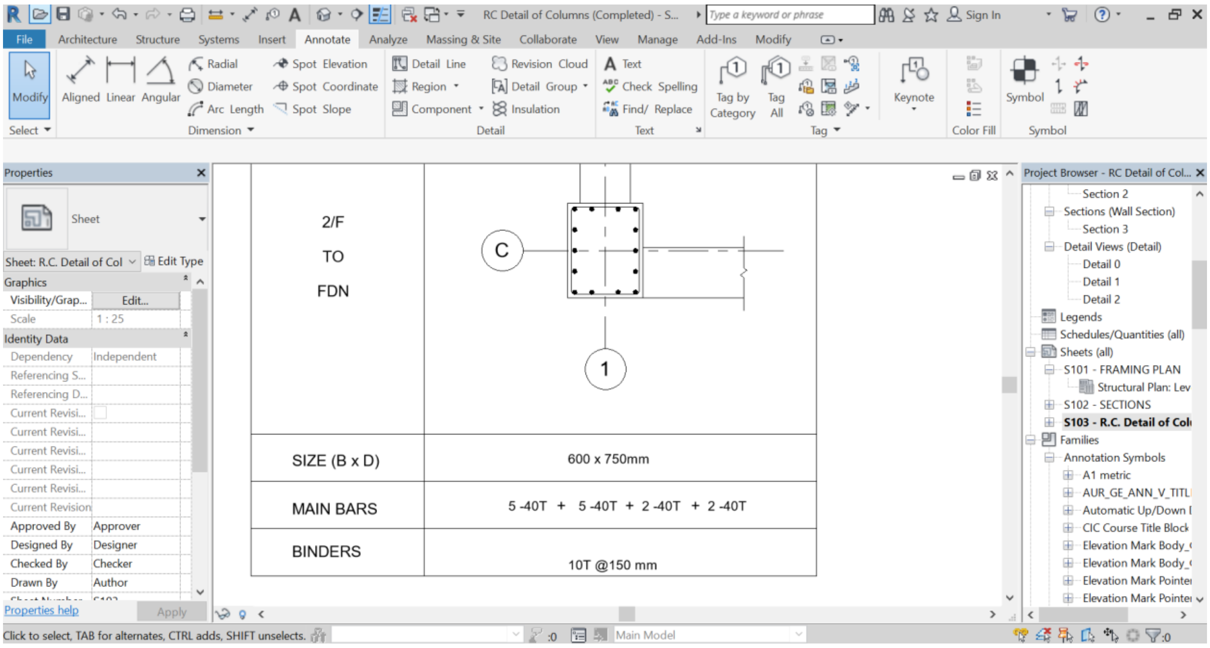
30 Find the corresponding label and “Edit” it. Remove the parameter “Rebar Number” from the label.



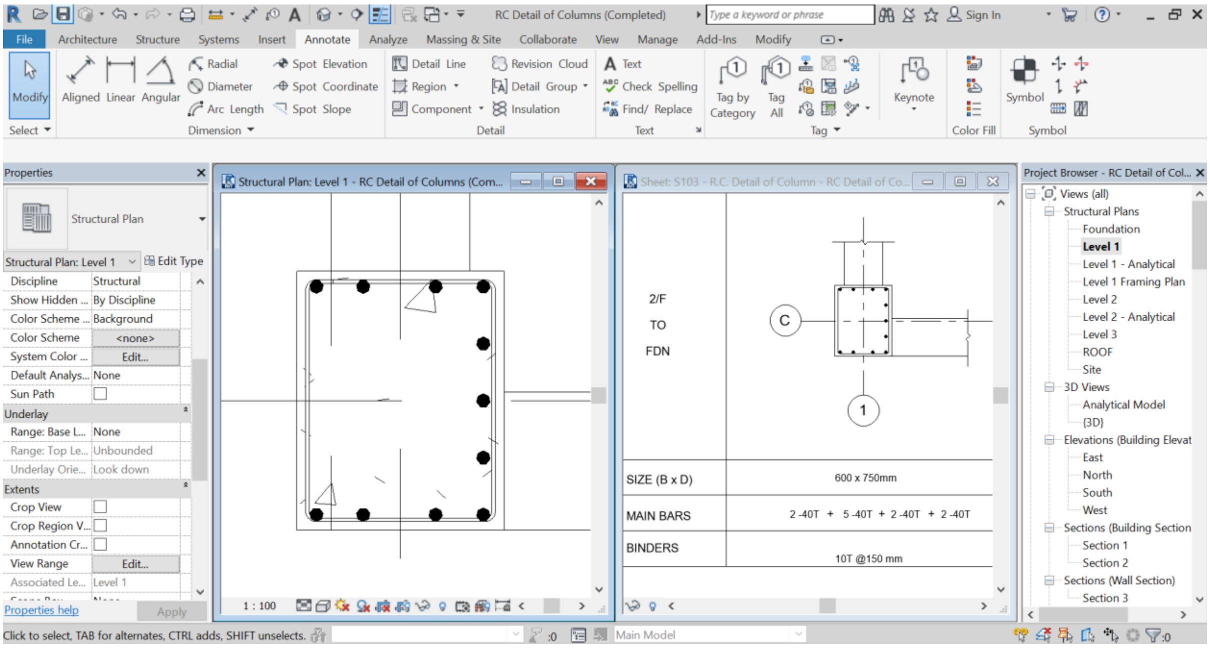
Then “Load into Project and Close”. Do not save the family and “Overwrite the existing version”.



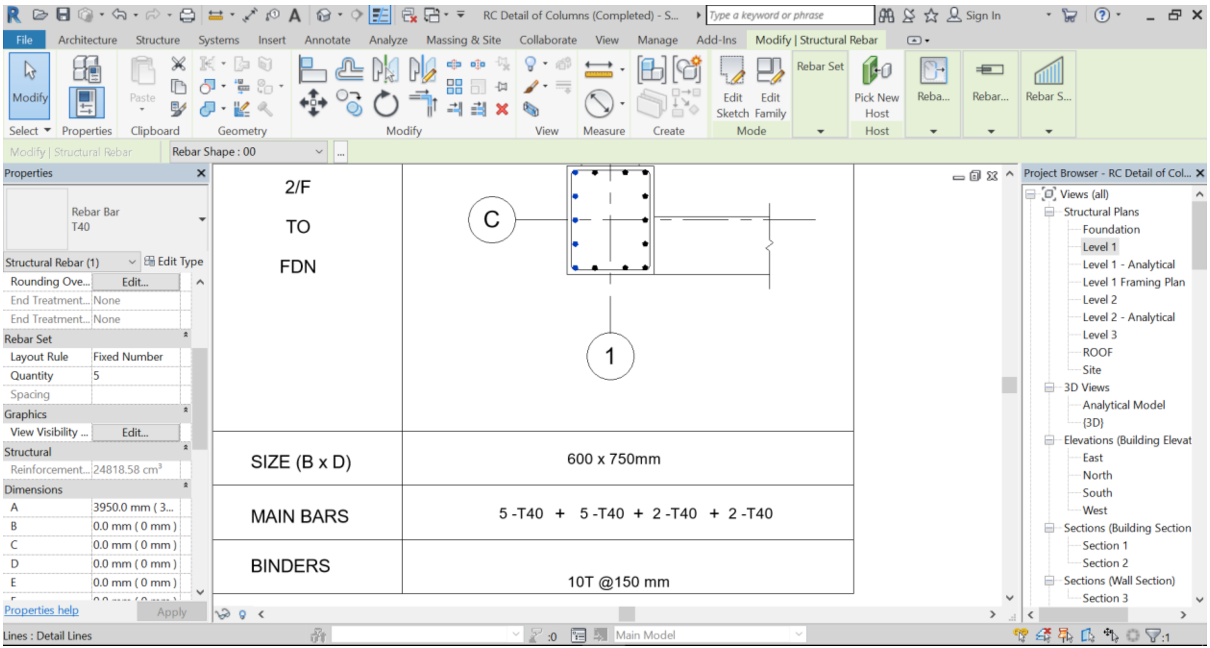
31 Tag other main bars and move these tags to appropriate location in the drawing.



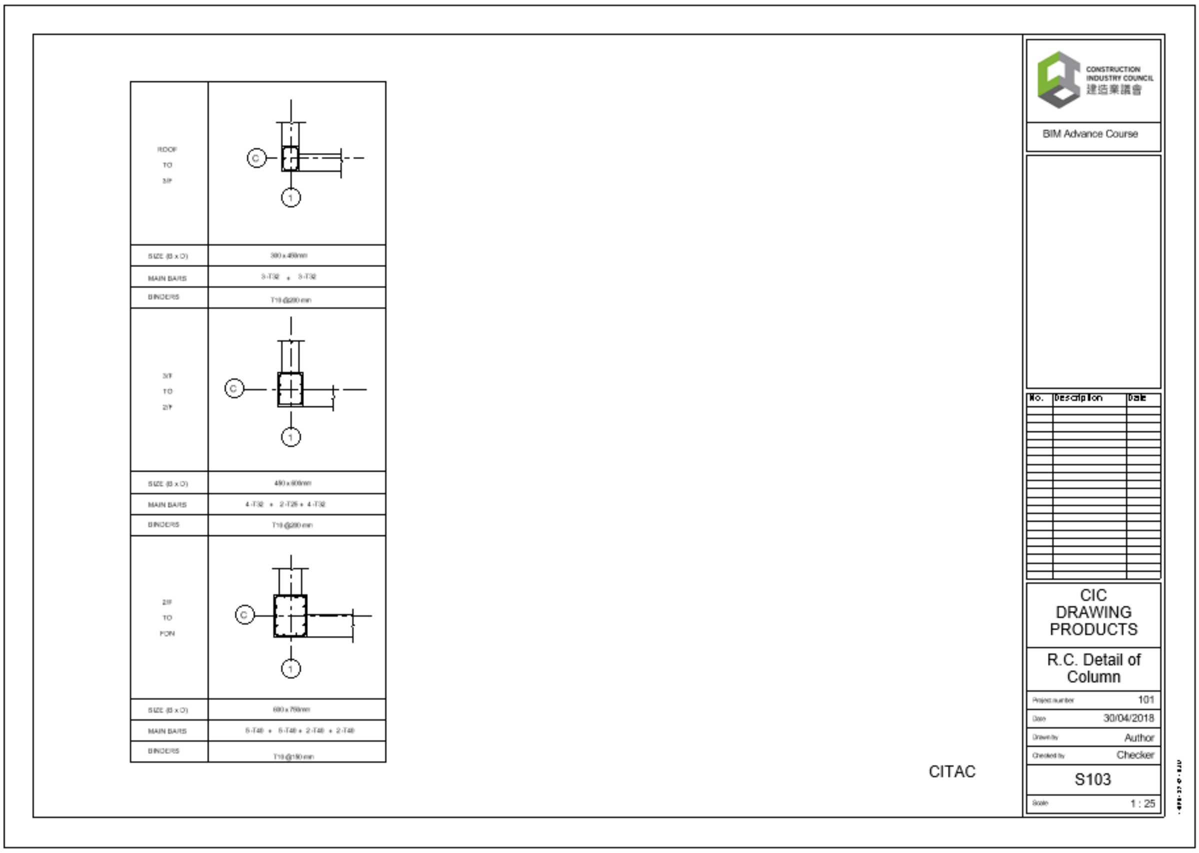
Now, when you change the rebar set in your model (say in Level 1 plan), the drawing R.C. Detail of Column will automatically be updated.

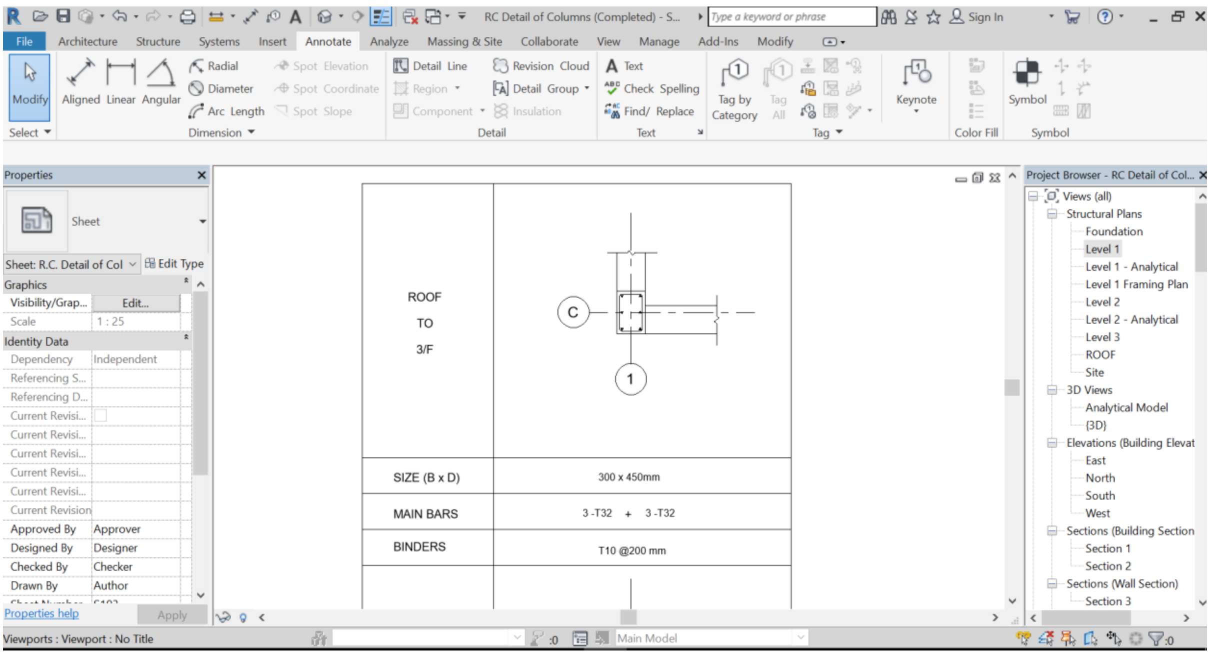


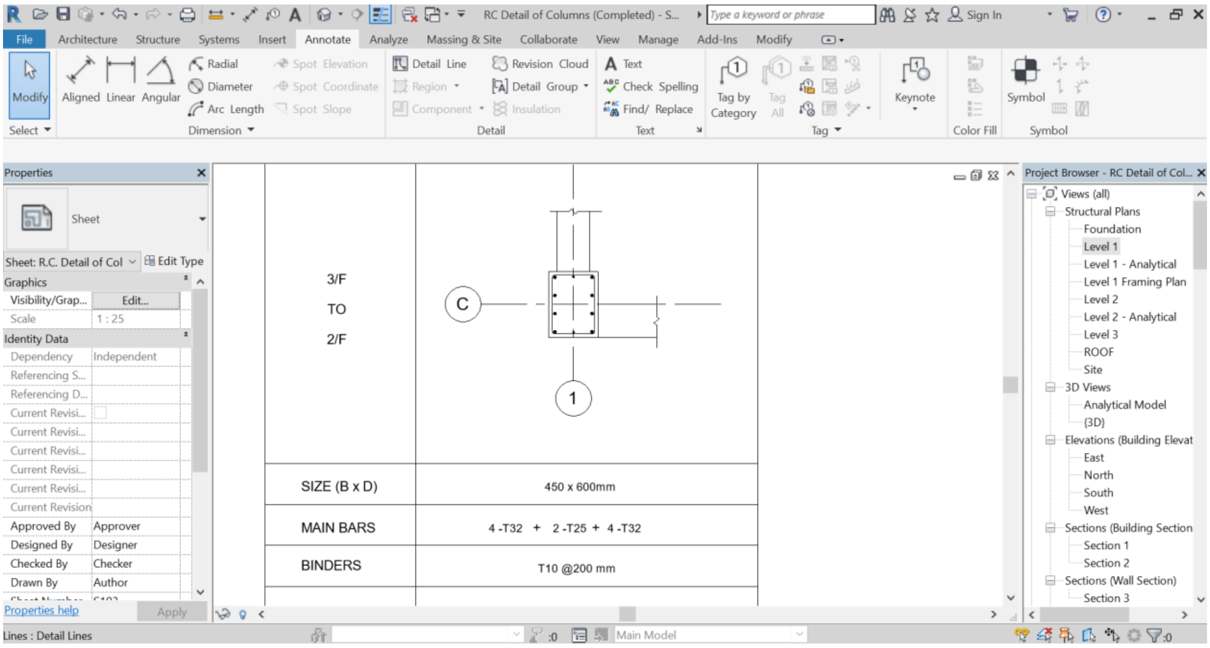
32 One minor detail still needed to be changed. In Revit, the default naming of rebar is, for example, 40T instead of T40 that is commonly used in Hong Kong. To change this, select the 40T rebar in any view then click "Edit Type". Rename it to "T40" and click "OK". Now the tag looks more familiar.

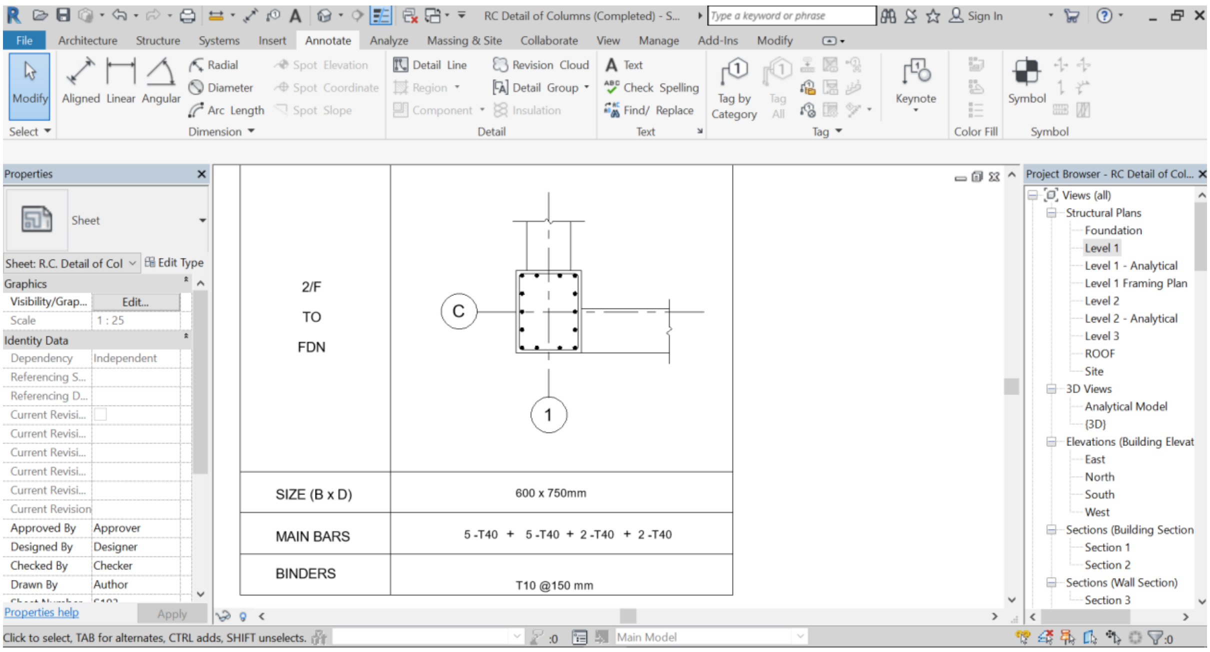


33 Revise and refine the other details with reference to the above steps. Then you should be able to get a R.C. Detail of Column drawing like the one below.


ROOF
TO
3/F
SIZE (B x D)
MAIN BARS
BINDERS
3/F
TO
2/F
2/F
TO
FDN
BIM Advance Course
CIC
DRAWING
PRODUCTS
R.C. Detail of
Column
101
30/04/2018
Author
Checker
S103
1 : 25
CITAC


RC Detail of Columns (Completed) - S...
File
Architecture
Structure
Systems
Insert
Annotate
Analyze
Massing & Site
Collaborate
View
Manage
Add-Ins
Modify
Properties
Sheet
Sheet: R.C. Detail of Col
Edit Type
Graphics
Visibility/Grap...
Edit...
Scale
1 : 25
Identity Data
Dependency
Independent
Approved By
Approver
Designed By
Designer
Checked By
Checker
Drawn By
Author
Properties help
Apply
ROOF
TO
3/F
SIZE (B x D)
300 x 450mm
MAIN BARS
3 -T32 + 3 -T32
BINDERS
T10 @200 mm
Project Browser - RC Detail of Col...
Views (all)
Structural Plans
Foundation
Level 1
Level 1 - Analytical
Level 1 Framing Plan
Level 2
Level 2 - Analytical
Level 3
ROOF
Site
3D Views
Analytical Model
{3D}
Elevations (Building Elevat
East
North
South
West
Sections (Building Section
Section 1
Section 2
Sections (Wall Section)
Section 3
Viewports : Viewport : No Title
Main Model





17.5 Creating Beam R.C. Detail Drawings in Revit

The steps to create R.C. Detail drawings for beams are similar to that for columns. First, you create some section views, then place rebars and finally create new sheets, annotate and tidy up the drawings.

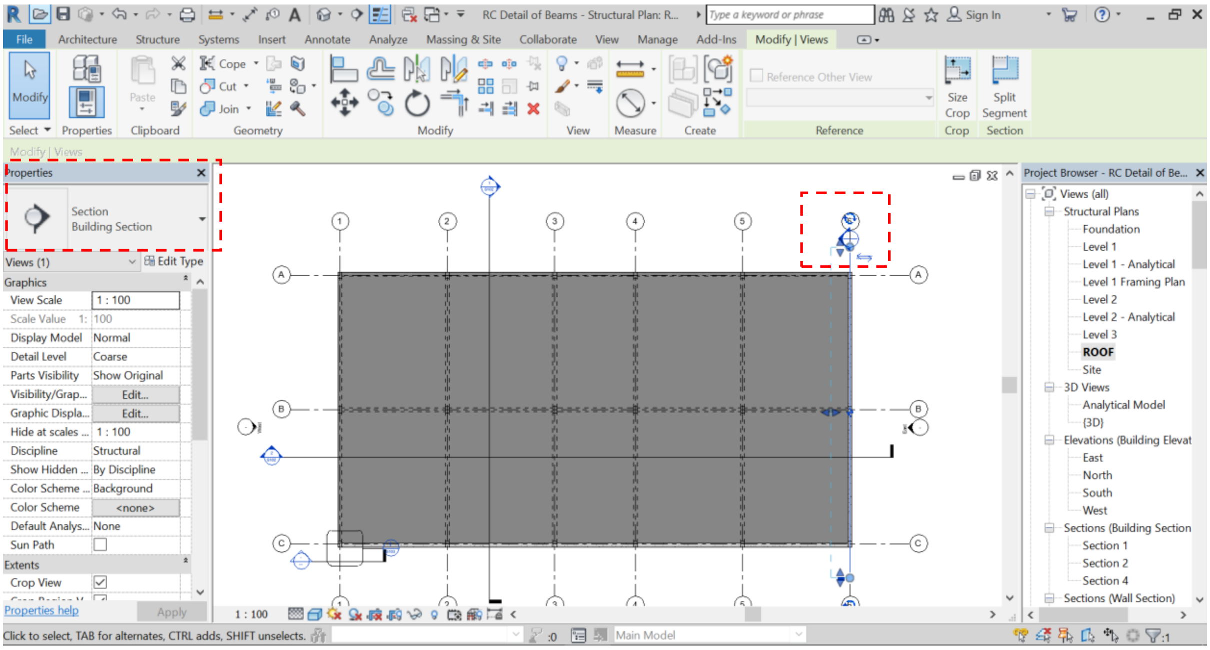
To create a R.C. Detail of Beam drawing,

1 From the course folder, open the file "RC Detail of Beams.rvt".

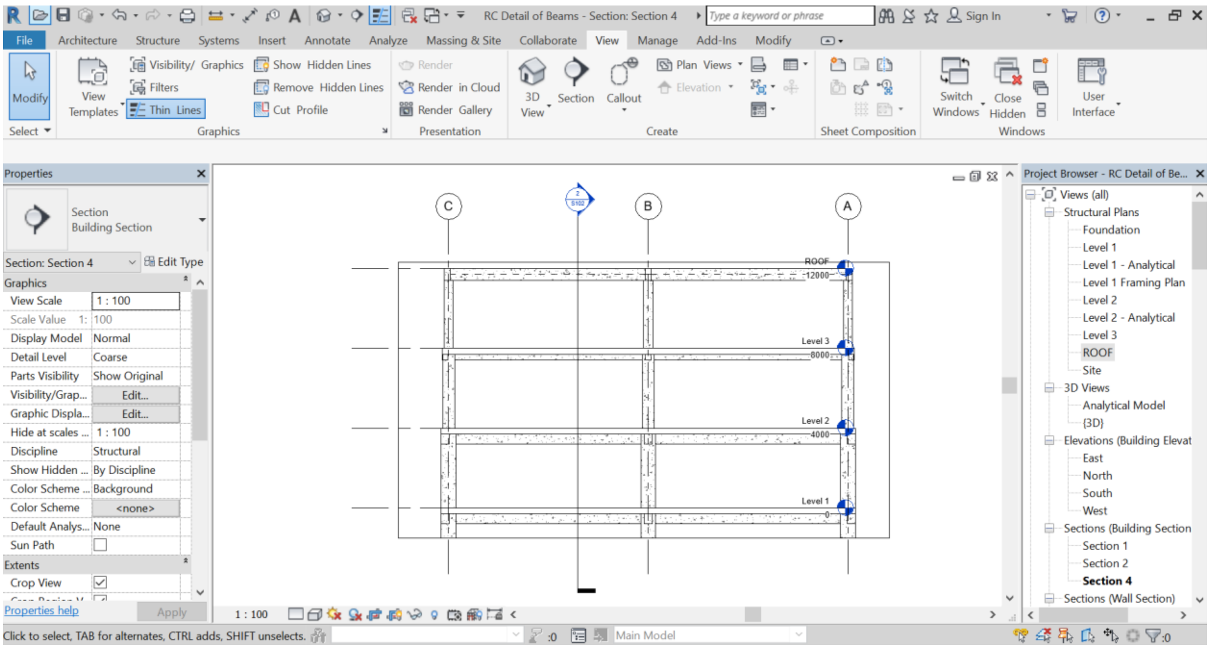
2 Go to ROOF (Structural Plan).

3 Click View tab ➢ Create panel ➢ Section. Make sure that "Building Section" is selected.

4 Create a section alone Grid-6 and flip its direction as shown below.



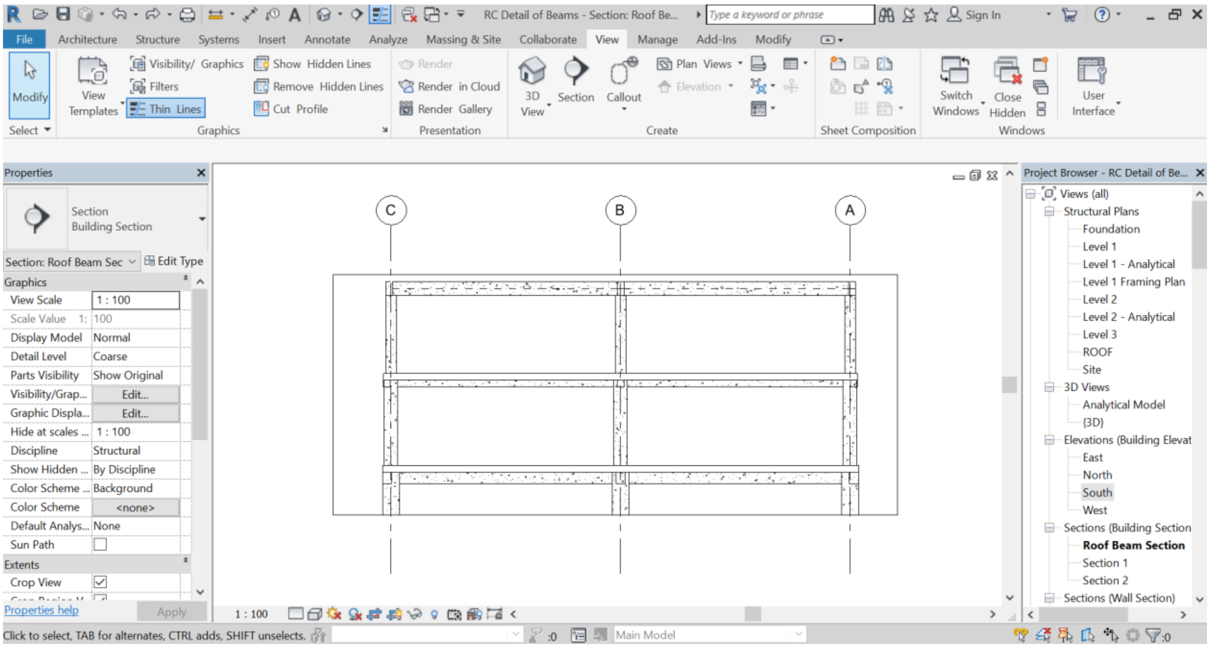
5 Go to this section by double clicking it or select it then right click and select “Go to View”.



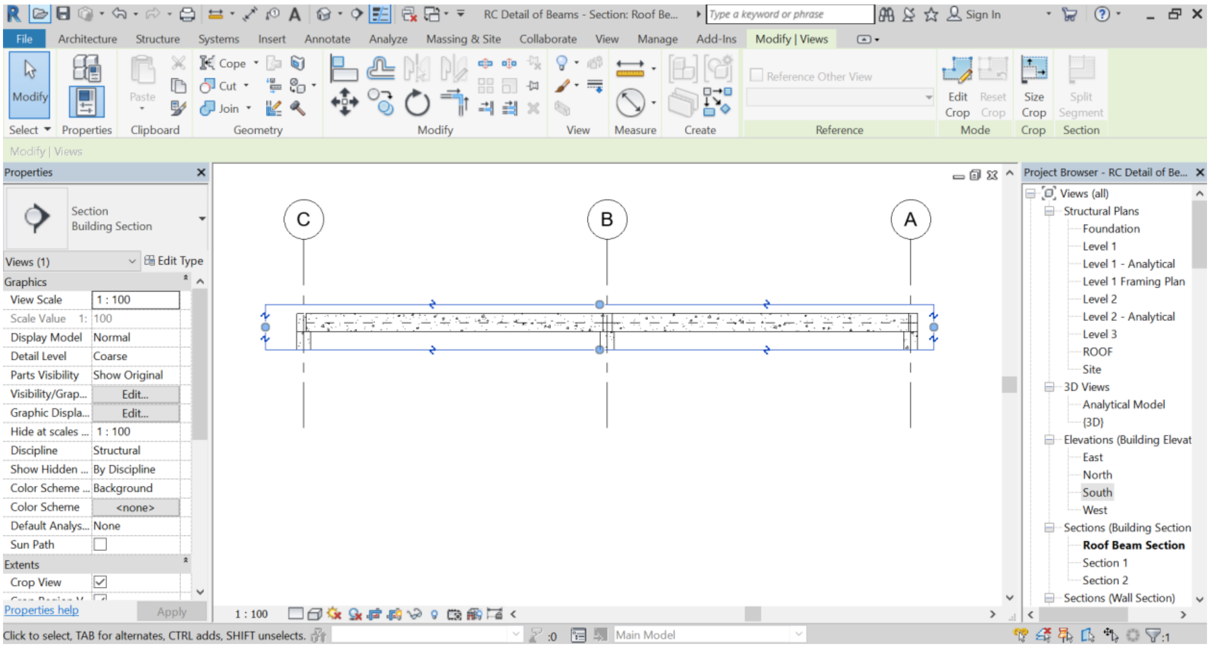
6 Clean up the section view with the following steps:

(1) Rename the section view to “Roof Beam Section”. (Right click near the view name in the Project Browser, then select rename.)

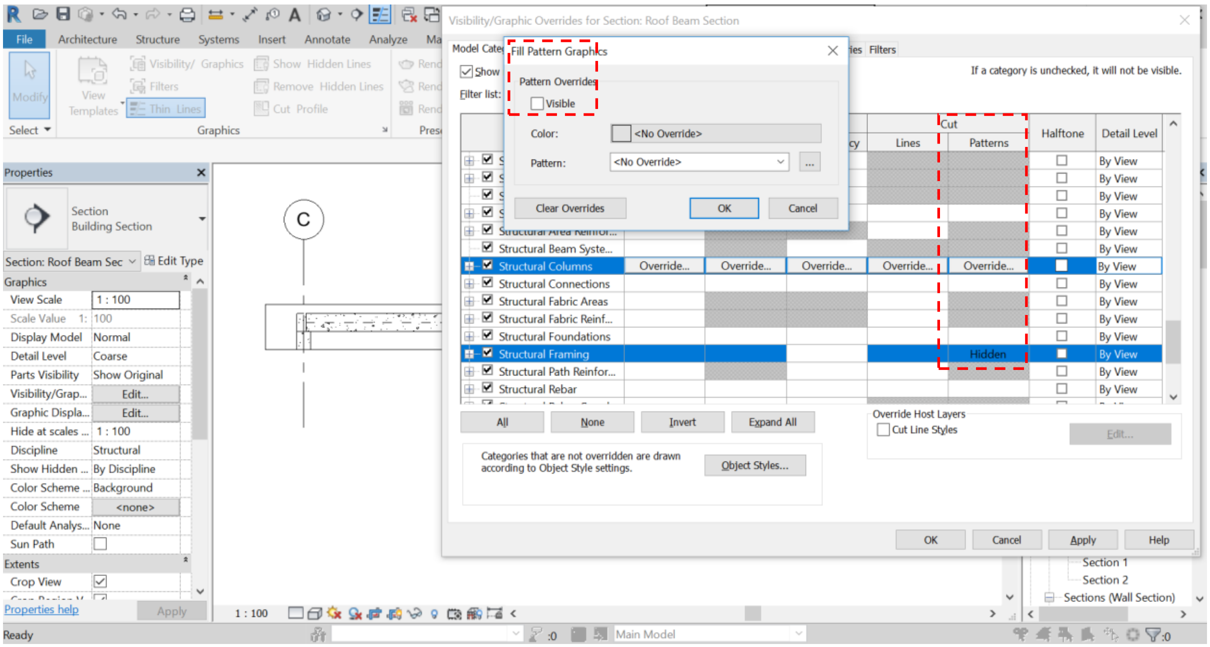
(2) Hide elements that are not necessary in this view. Select levels and section line, then right click and select “Hide in View ➢ Elements”.

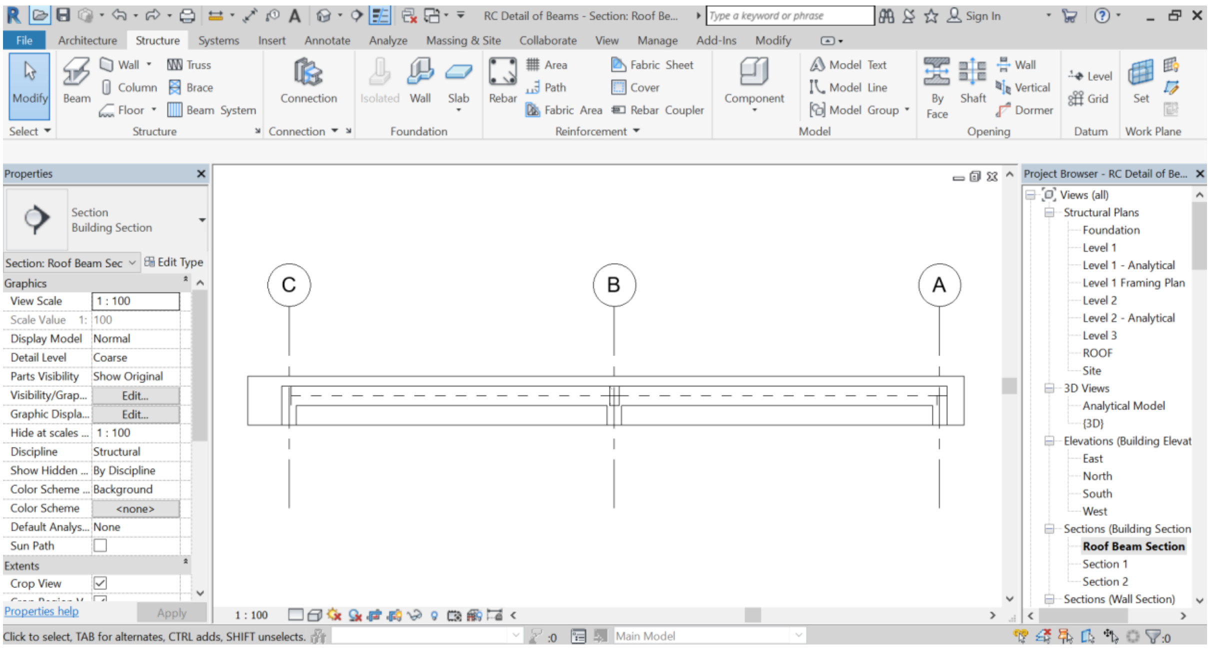


(3) Select the View crop region and drag the crop region until it shows only the Roof part as shown below.

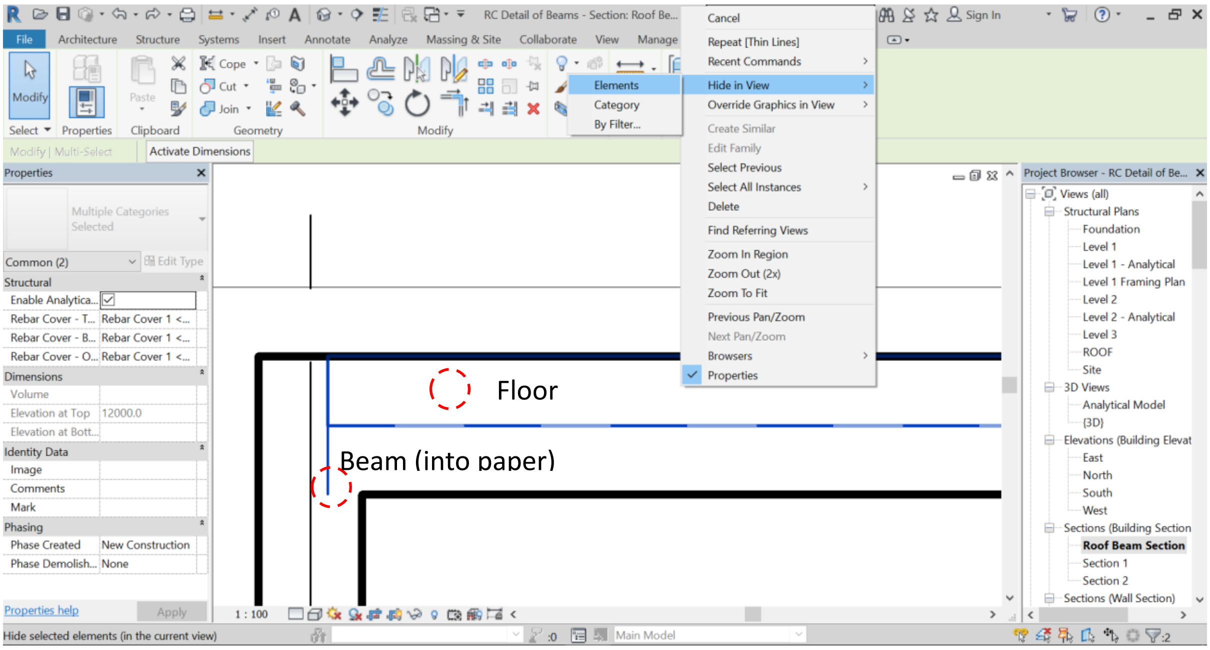


(4) Clear the hatch pattern of the beams and columns. Type "vv" for Visibility/Graphics control, then edit that for Structural Columns and Structural Framing. Make the pattern in Cut view not visible for these elements.

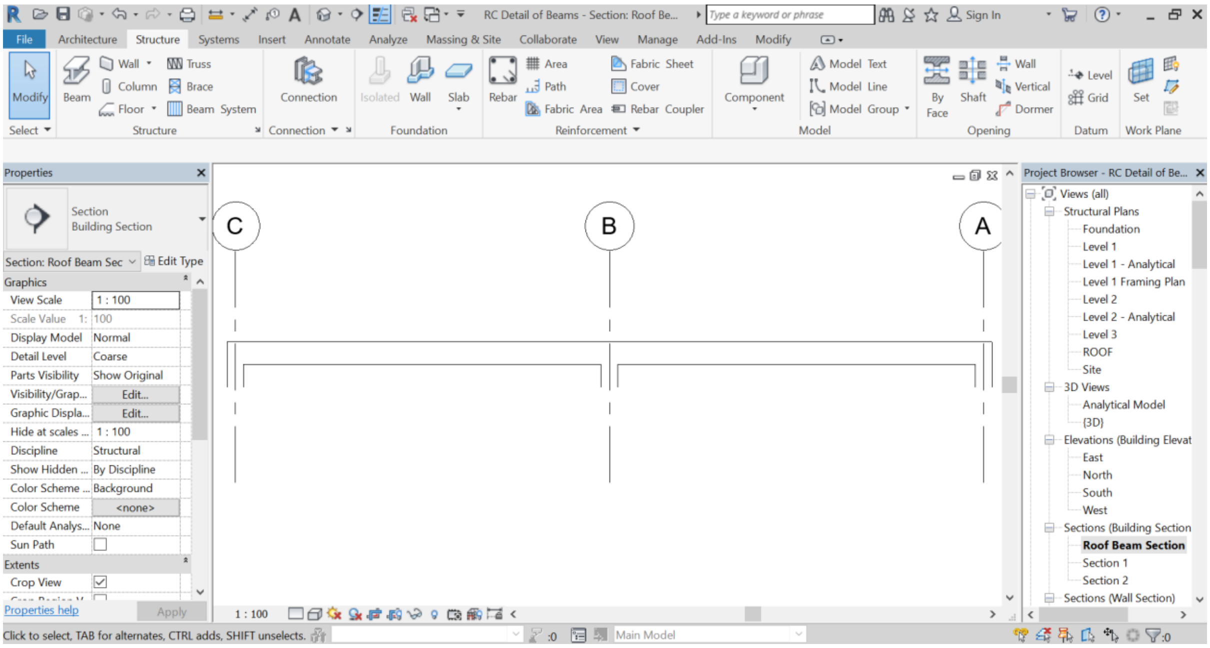




(5) There are still some lines showing elements that are not to be shown in a R.C. Detail drawing. Carefully select the floor and the beams that go into the paper and hide them in view. (Select then right click and select "Hide in View ➢ Elements". (Use the "tab" key to help toggle between elements to assist your selection.)



Then hide the view crop region (click ) and give a clean outline of the roof beams from the model. This outline of the beams will automatically update if you change the size or location of the beams.

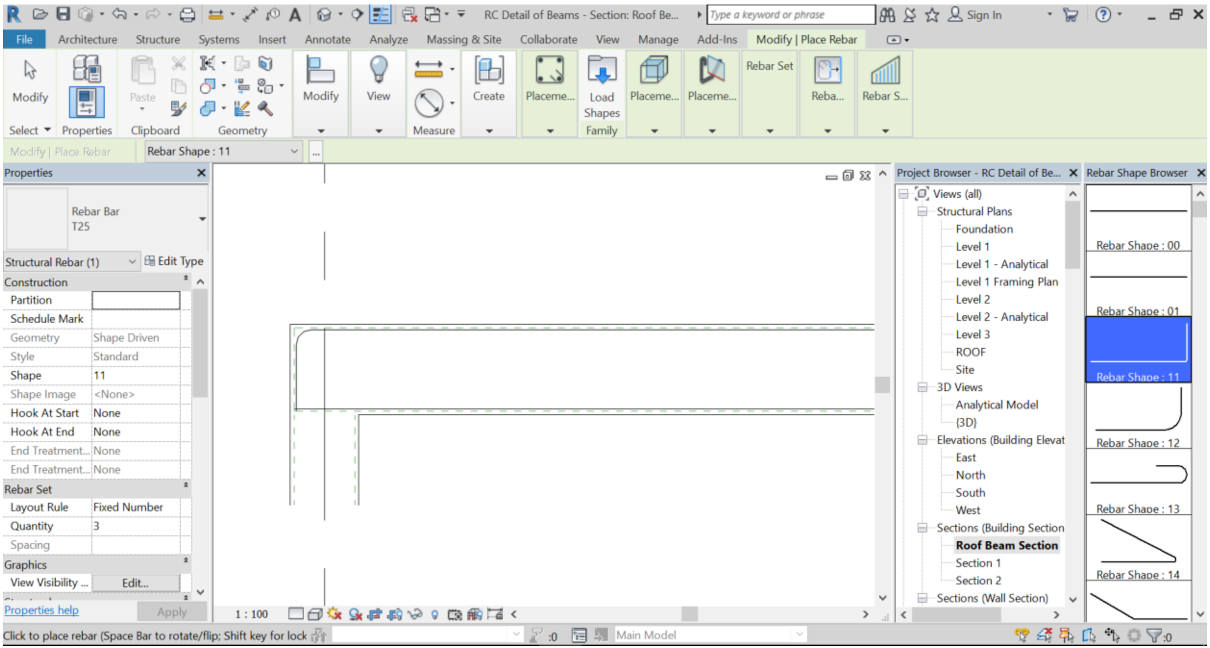


7 The next step is to add reinforcement or rebars to the beams.

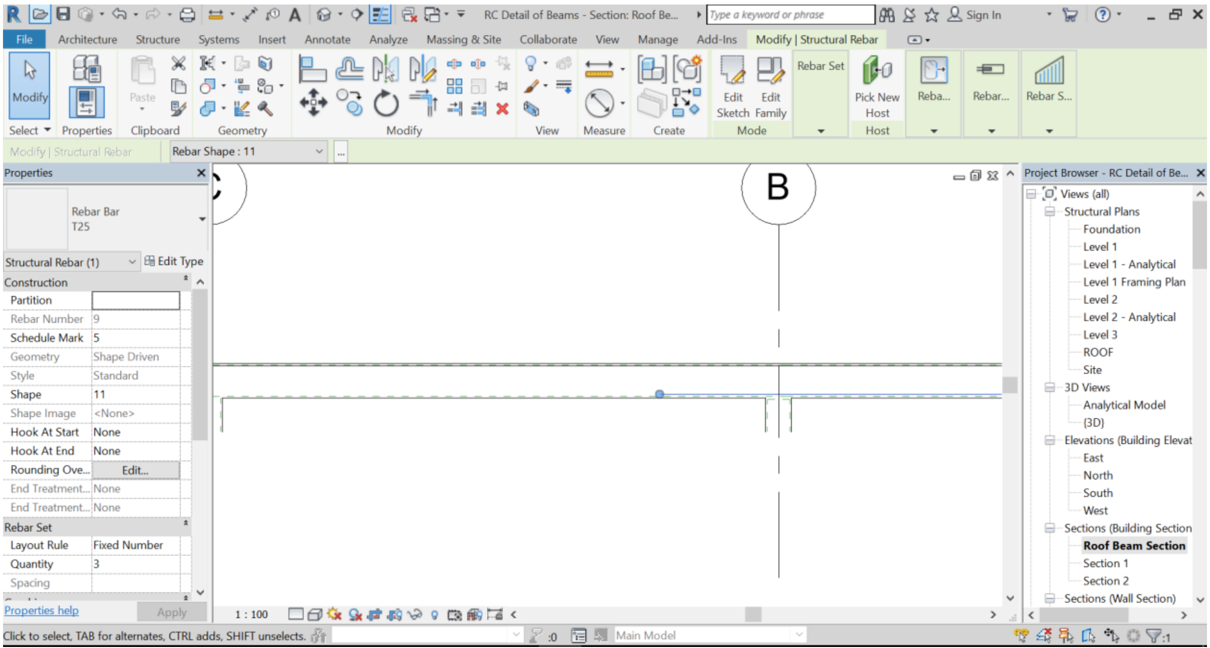
Click Structure tab ➢ Reinforcement panel ➢ Rebar.

Select Rebar Shape: 11 with Rebar Bar T25. Set Placement Plane to “Current Work Plane”, Placement Method to “Parallel to Work Plane” and Rebar Set to “Fixed Number and Quantity = 3”.

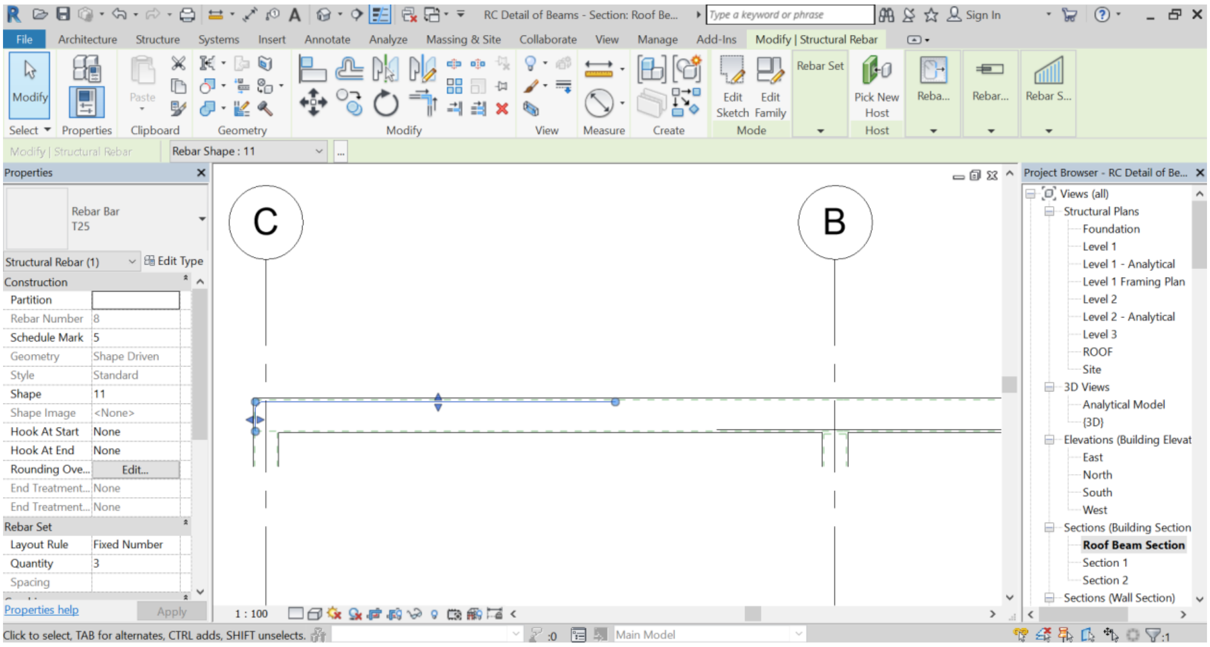
Hover over to top and bottom of the beam near Grid-C and click to place the rebar sets. (Press "esc" a couple times to end the command.



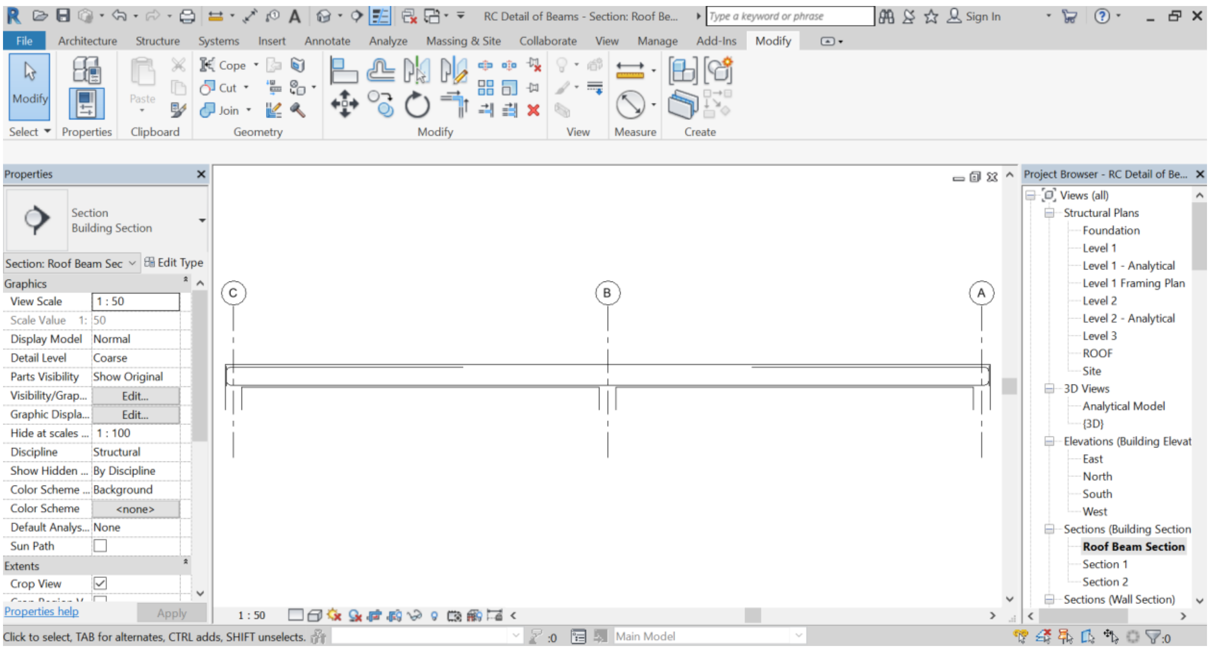
8 Select the bottom rebar set and drag its left end to near Grid-B as shown below.



9 Select the top rebar set and drag its right end to the middle of Grid-C and Grid-B as shown below.



10 Copy these two rebar sets by mirroring them about Grid-B. (The Mirror – Pick Axis command is under Modify tab. (Also change the Scale of the view to 1:50.)

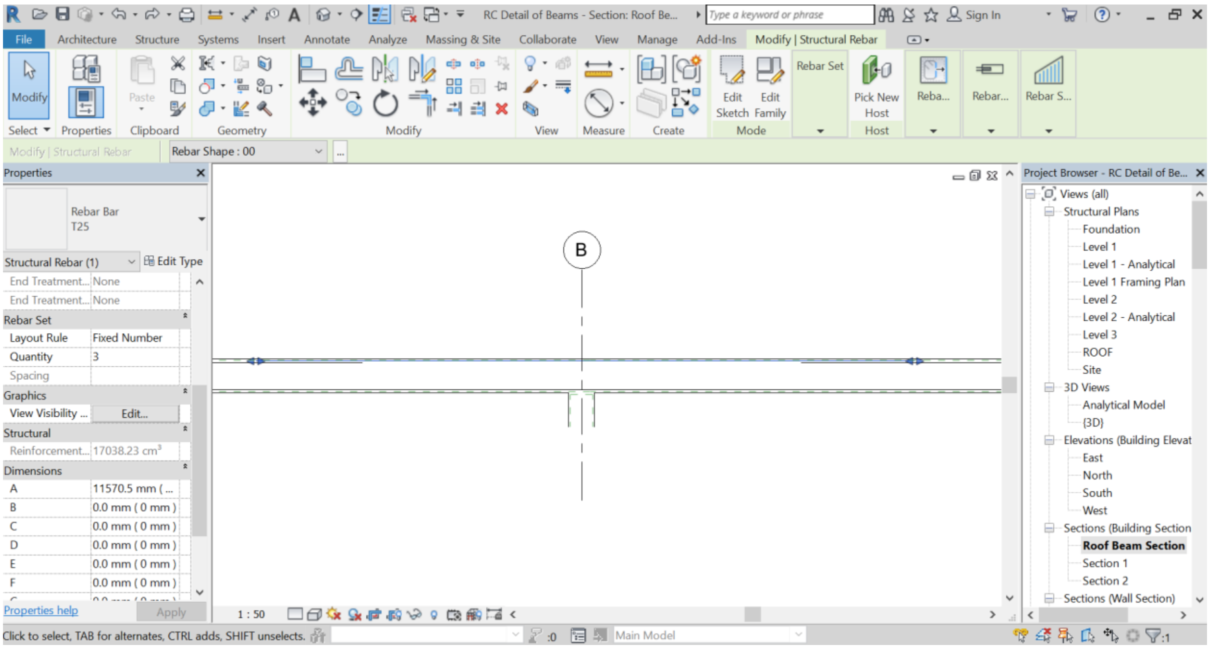


11 Next, add rebar set to the top of the beams at Grid-B.

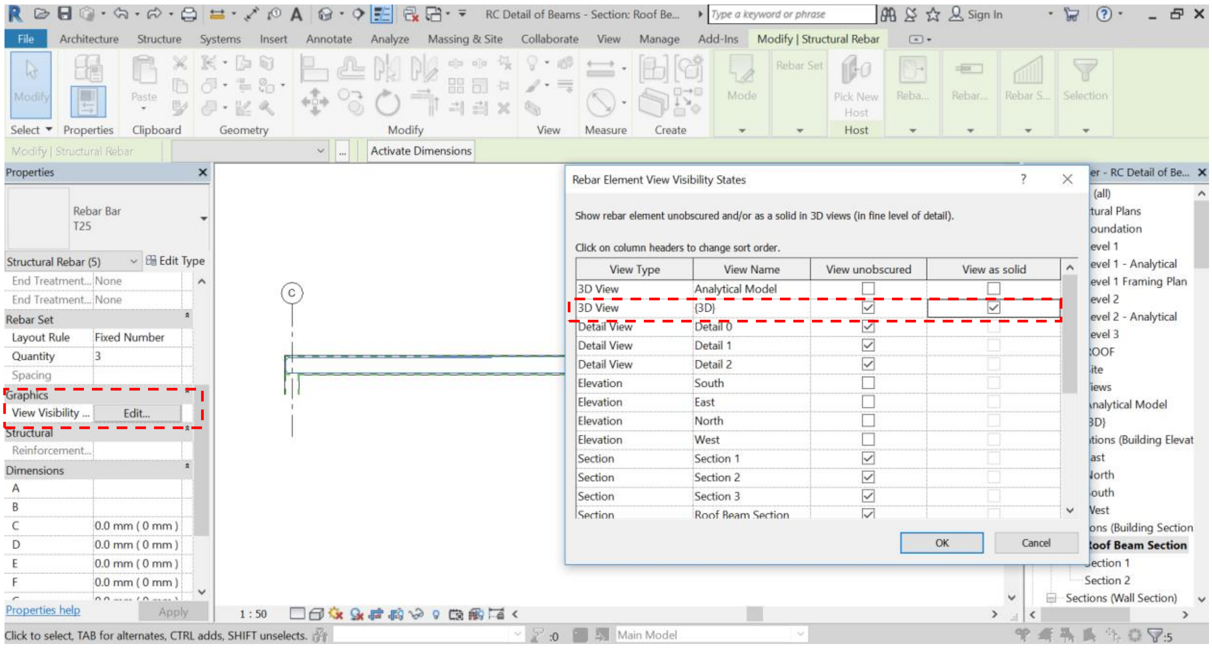
Click Structure tab ➢ Reinforcement panel ➢ Rebar.

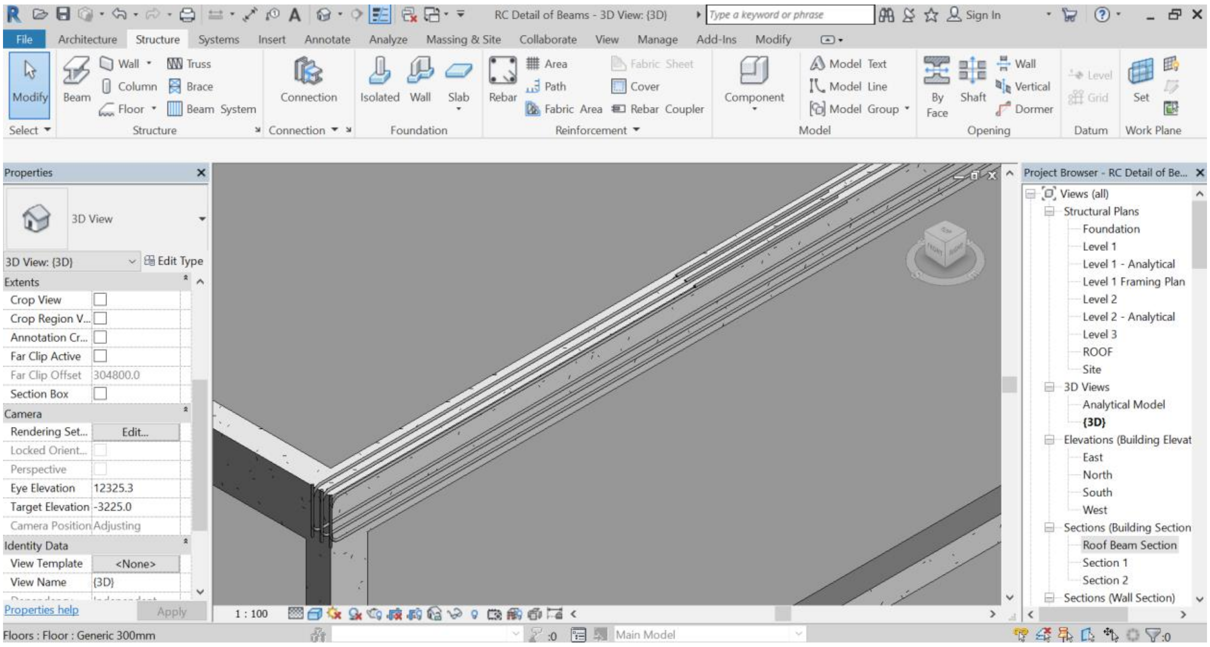
Select Rebar Shape: 00 with Rebar Bar T32. Set Placement Plane to “Current Work Plane”, Placement Method to “Parallel to Work Plane” and Rebar Set to “Fixed Number and Quantity = 3”.

Then select this rebar set and drag its ends near to the middle of Grid-C and Grid-B as shown below.



12 Now, select all the rebar sets and change its view visibility for 3D View {3D} to both "View unobscured" and "View as solid". Then go to 3D view to view the rebar sets you added.





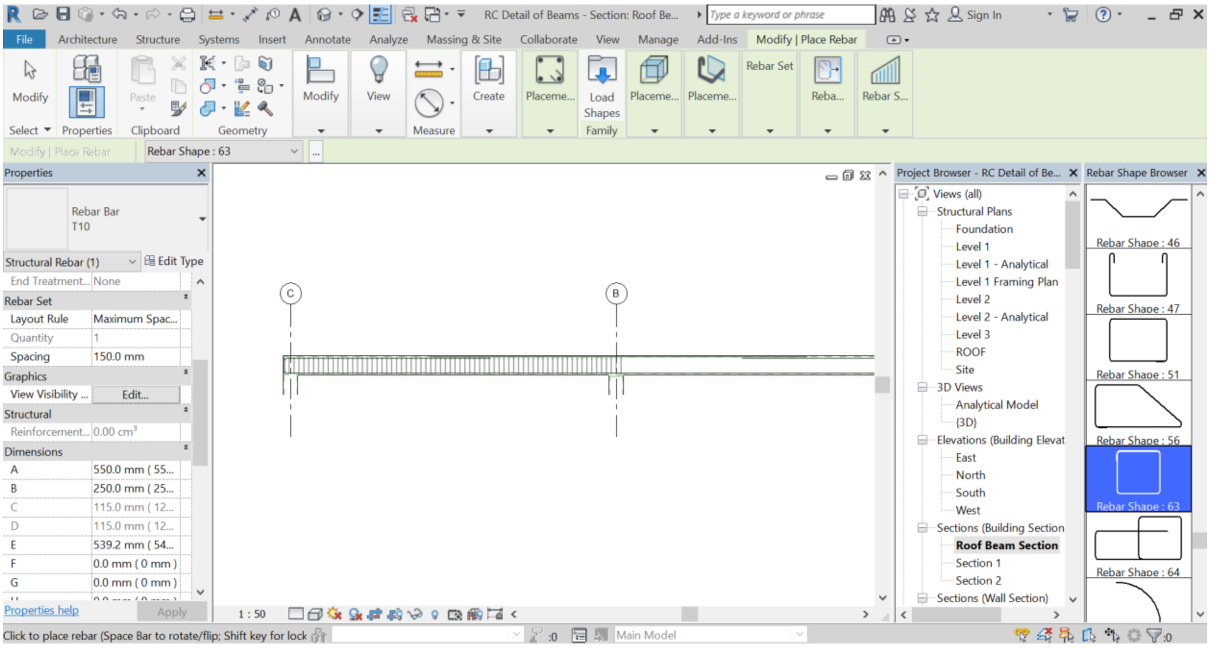
13 Next, add some transverse reinforcement or links to finish placing rebars to the beams.

Go to the section view "Roof Beam Section", then

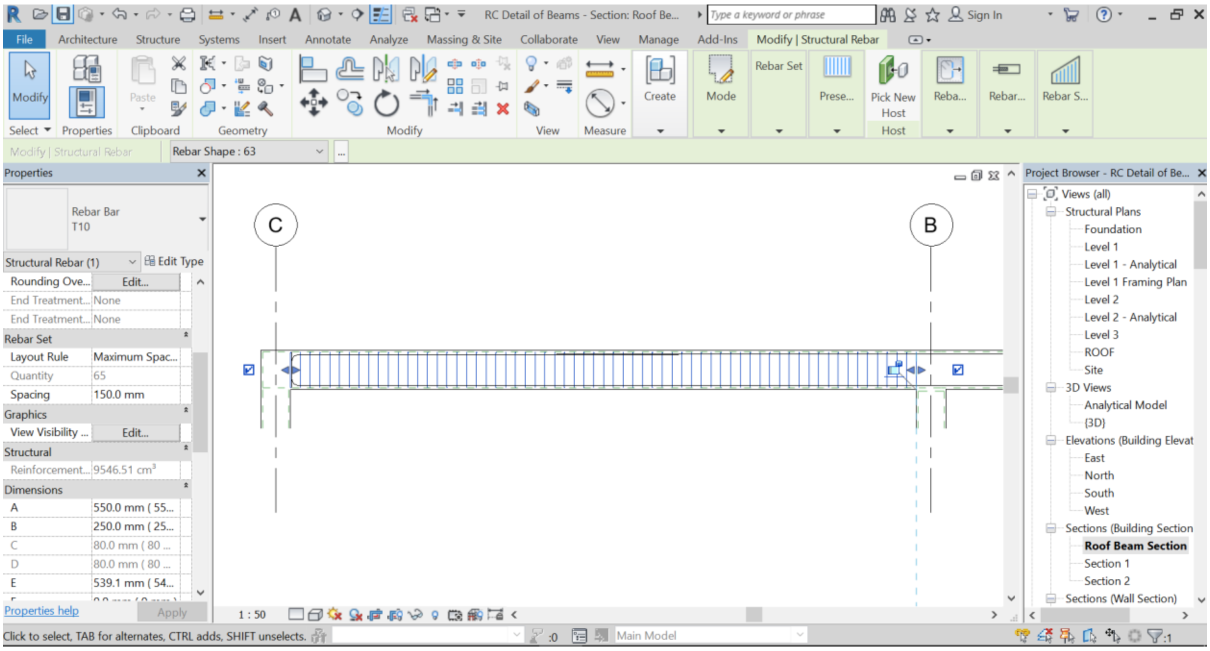
Click Structure tab ➢ Reinforcement panel ➢ Rebar.

Select Rebar Shape: 63 with Rebar Bar T10. Set Placement Plane to "Current Work Plane", Placement Method to "Perpendicular to Cover" and Rebar Set to "Maximum Spacing and Spacing = 200mm".

Hover over the left beam and click to place the links.



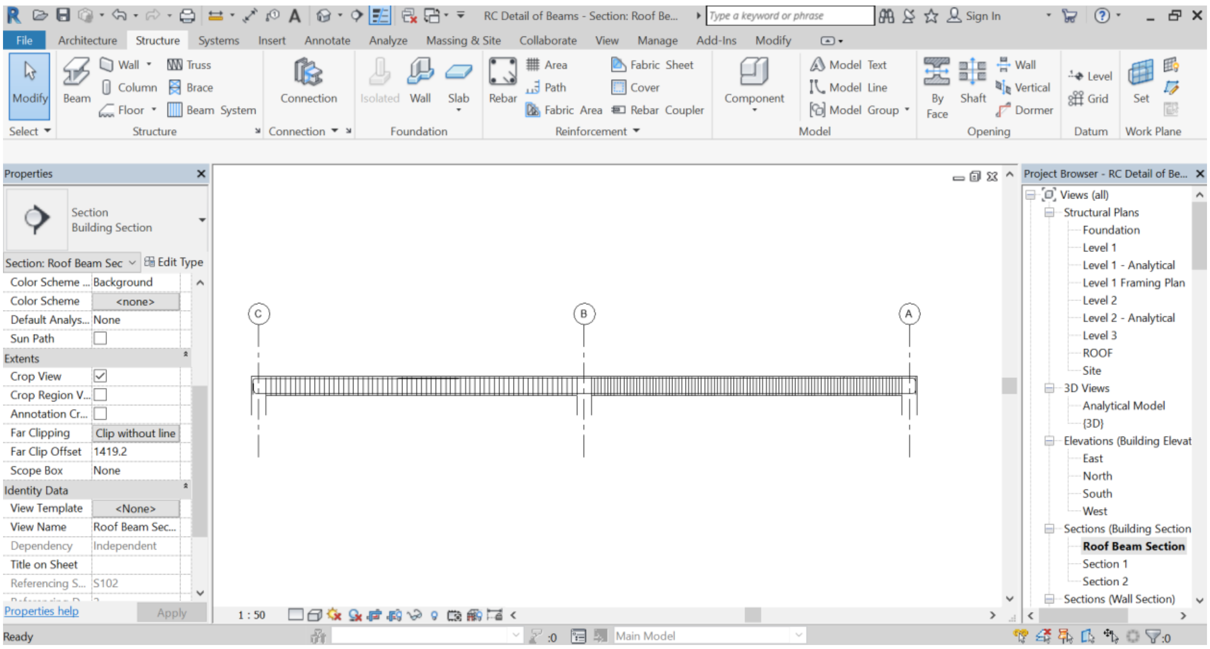
14 Select the links and drag its ends to just outside the column face.

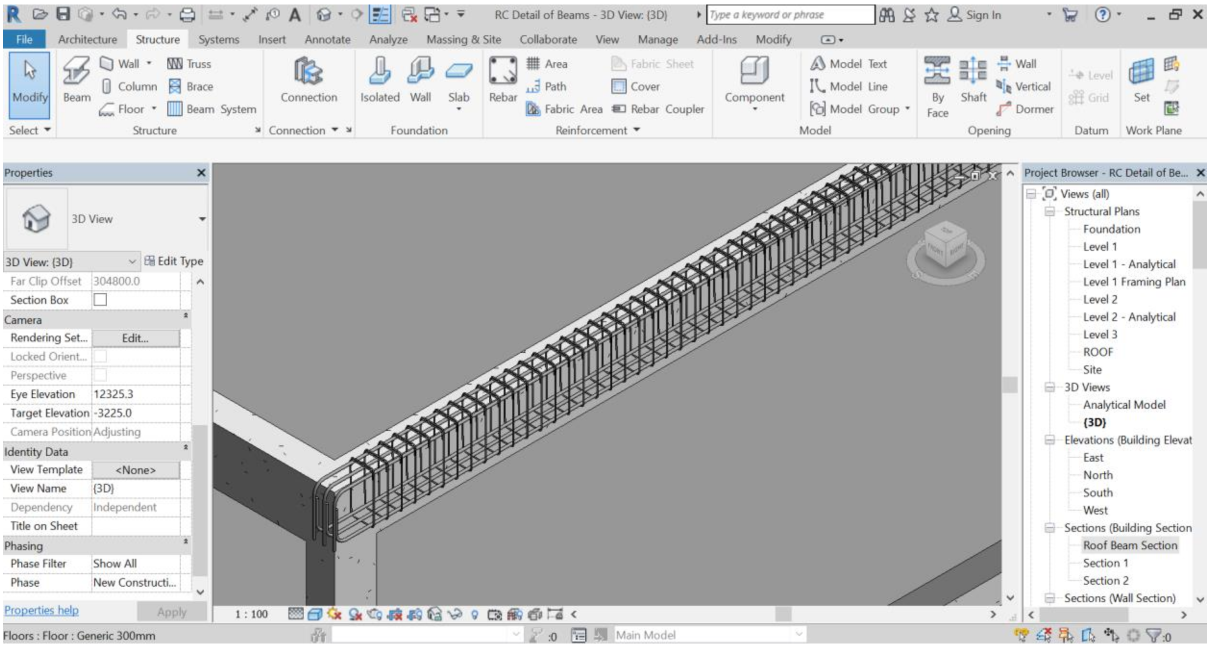


15 Add links to the right beam with the follow settings.

Select Rebar Shape: 63 with Rebar Bar T10. Set Placement Plane to “Current Work Plane”, Placement Method to “Perpendicular to Cover” and Rebar Set to “Maximum Spacing and Spacing = 100mm”.

Hover over the right beam and click to place the links. Drag the links to outside the column face as before.



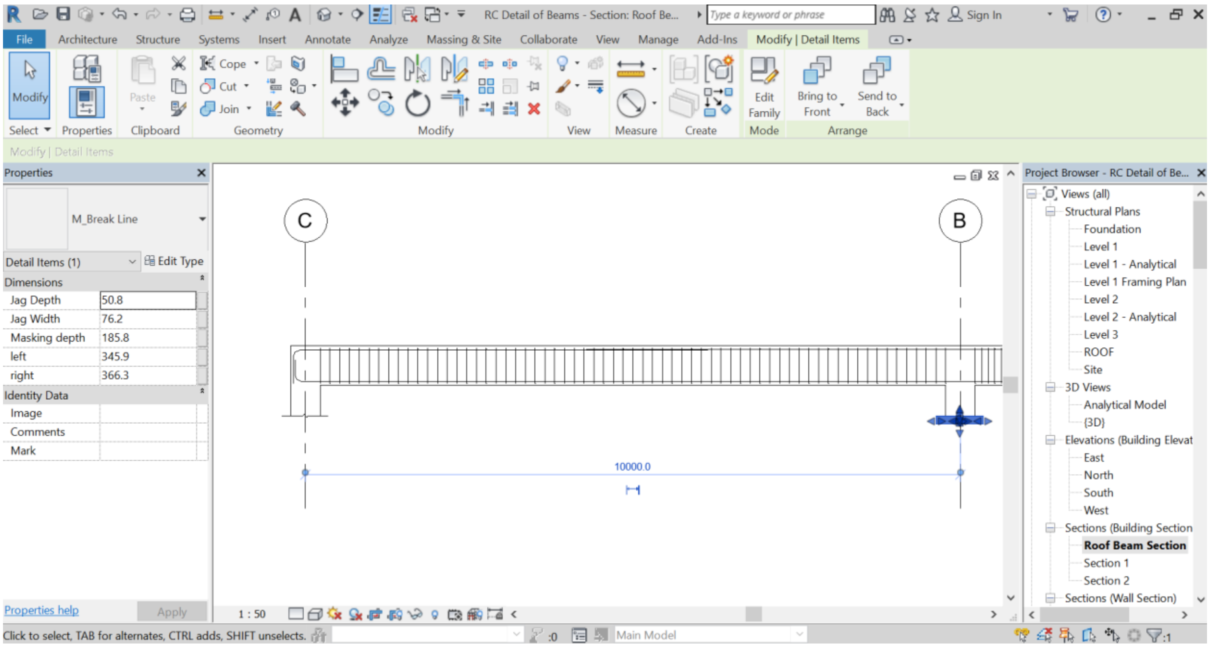


16 After placing all the reinforcement, you can begin to cut some sections at the beams and start annotating all these detail sections.

First, tidy up and annotate the "Roof Beam Section".

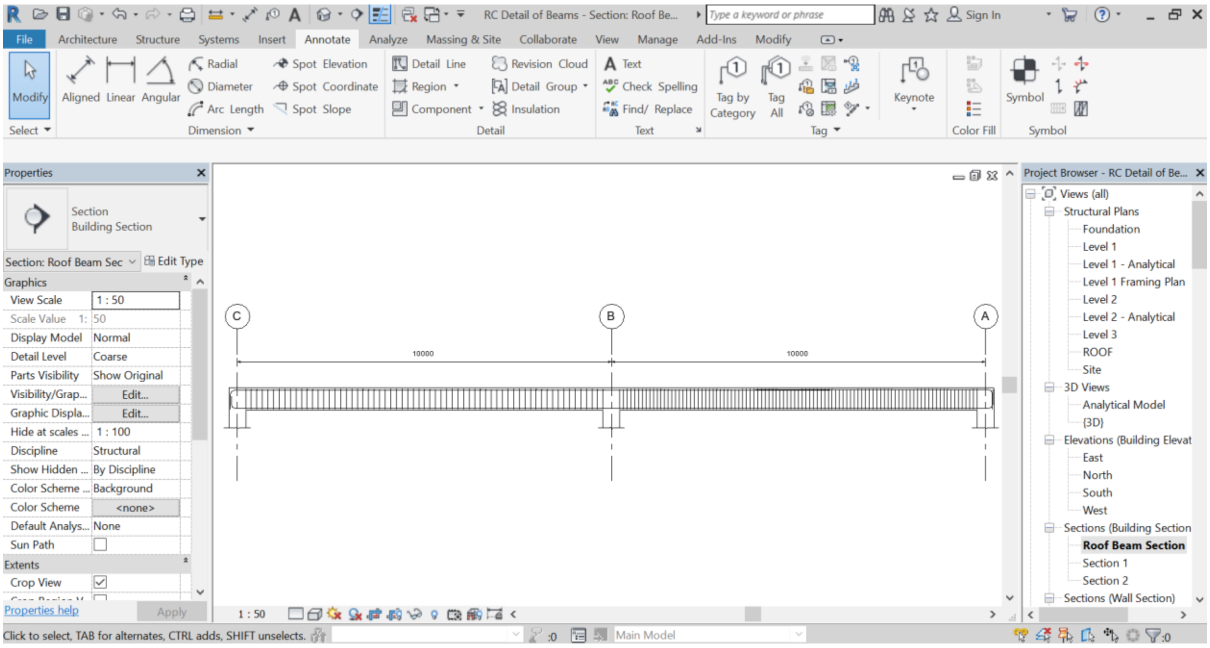
(1) Add break lines to the columns. Break lines are Detail Components and can be found in Annotate tab ➢ Detail panel ➢ Component. Click "Load Family" and find the break line detail item "M_Break Line" from the path shown below. (C:\ProgramData\Autodesk\RVT2018\Libraries\Hong_Kong\Detail Items\Div 01-General)

(Press "spacebar" to change the orientation of the break line if necessary.)



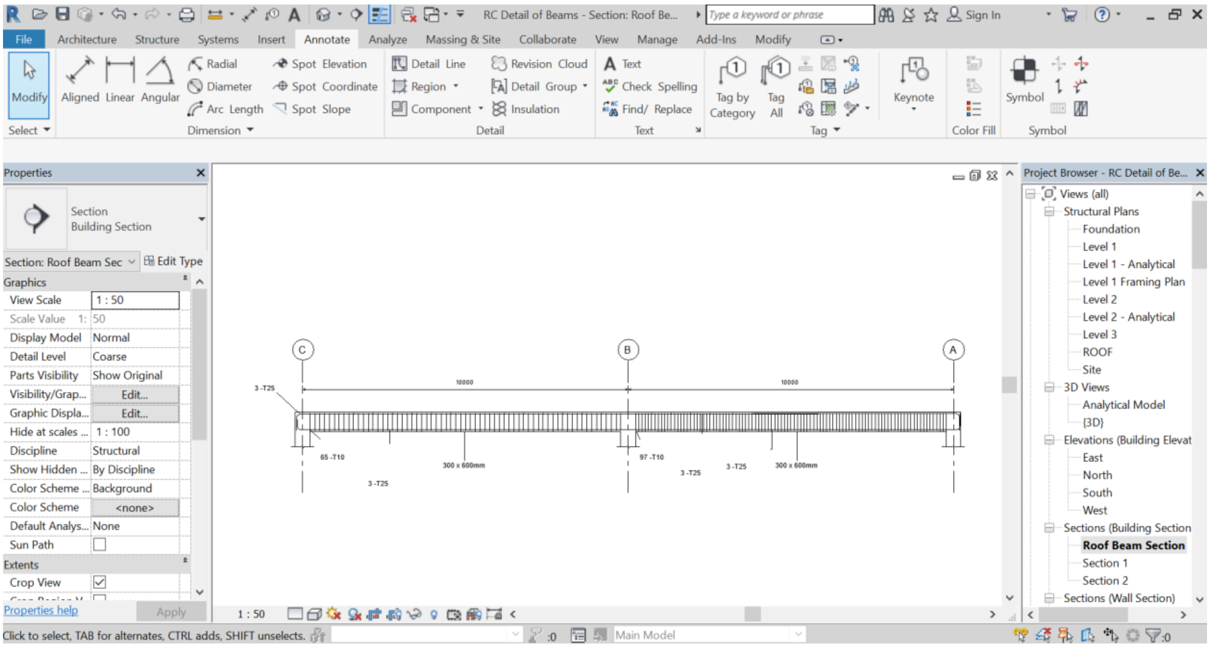
(2) Add dimensions between the Grids.

Click Annotate tab ➢ Dimension panel ➢ Aligned. Then pick Grid-C, then Grid-B and Grid-A and click anywhere right to Grid-A. Press "ese".



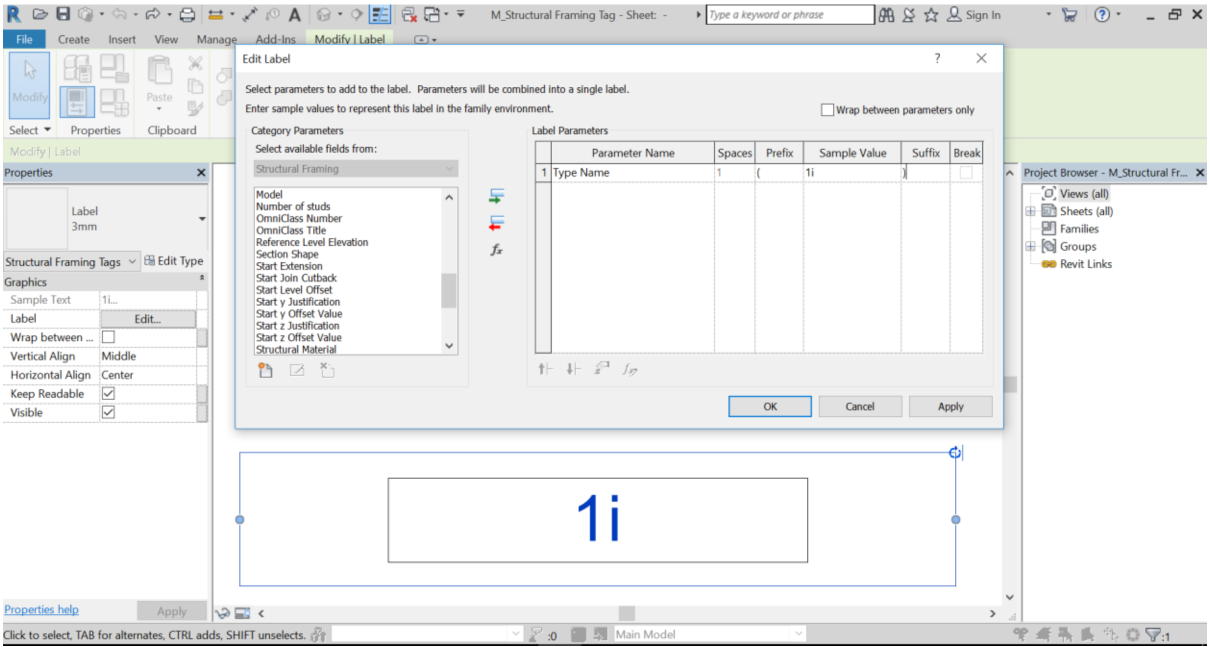
(3) Now, add tags to annotate the beams and their reinforcement.

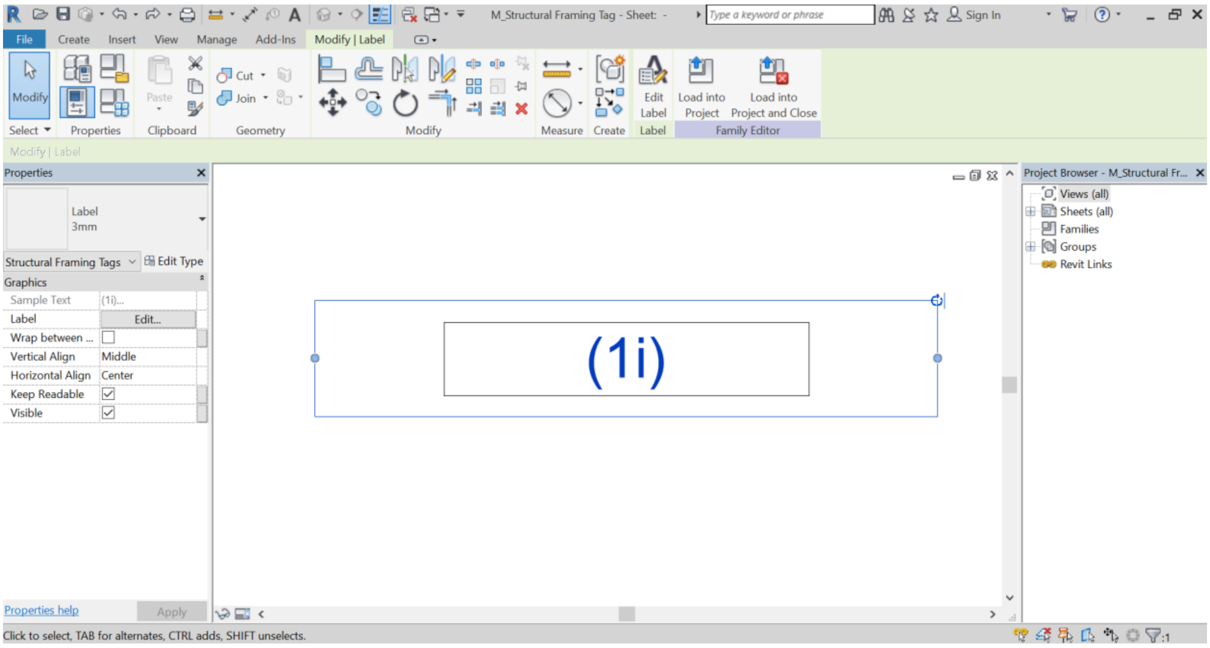
Click Annotate tab ➢ Tag panel ➢ Tag by Category. Then tag the beams, the top and bottom rebars and the links. You will need to modify these tags for better presentation.



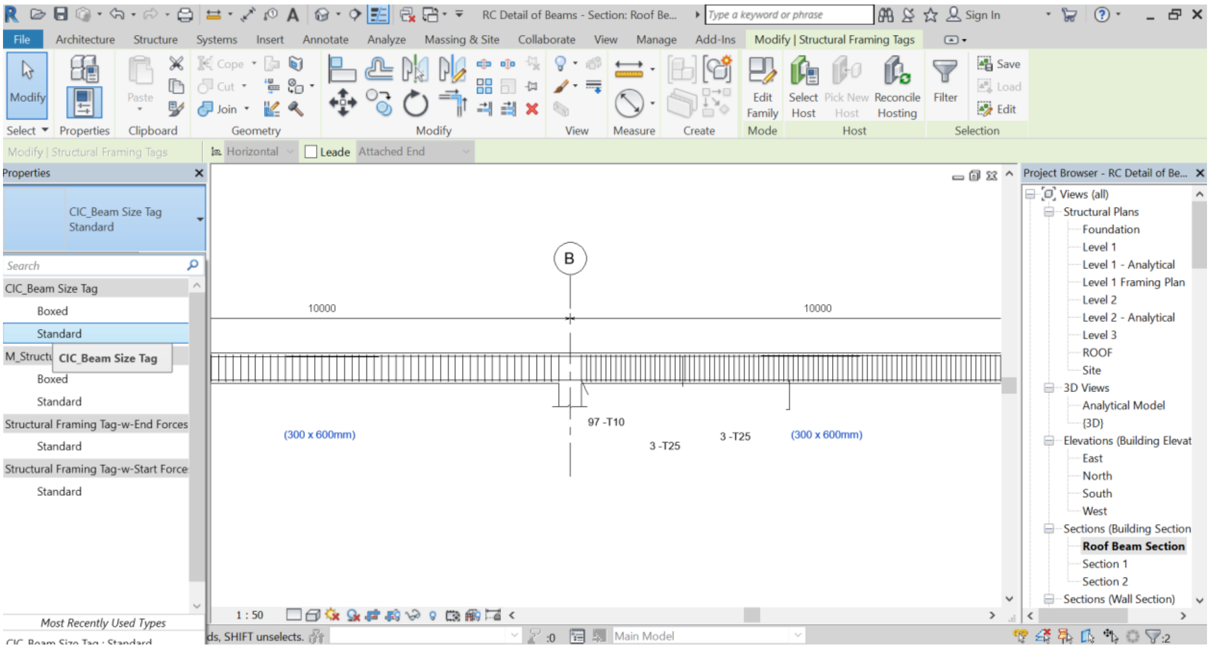
(4) Now, double click on the beam tag "300 x 600 mm" to customize it.

In the Family Editor, select the label and click "Edit". For the Type Name parameter, add "(" as prefix and ")" as suffix. Click "OK" then save as a new family named "CIC_Beam Size Tag".





Now, select the two beam tags and change their type to "CIC_Beam Size Tag" (Standard) and "Leader Line" as well.

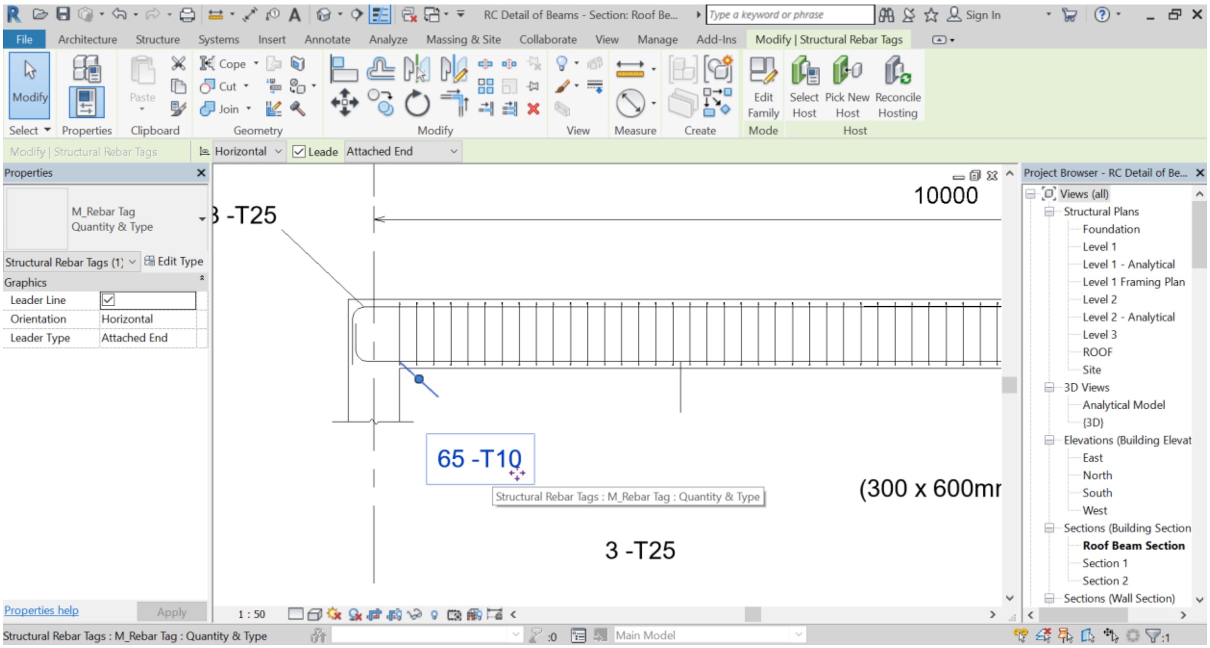


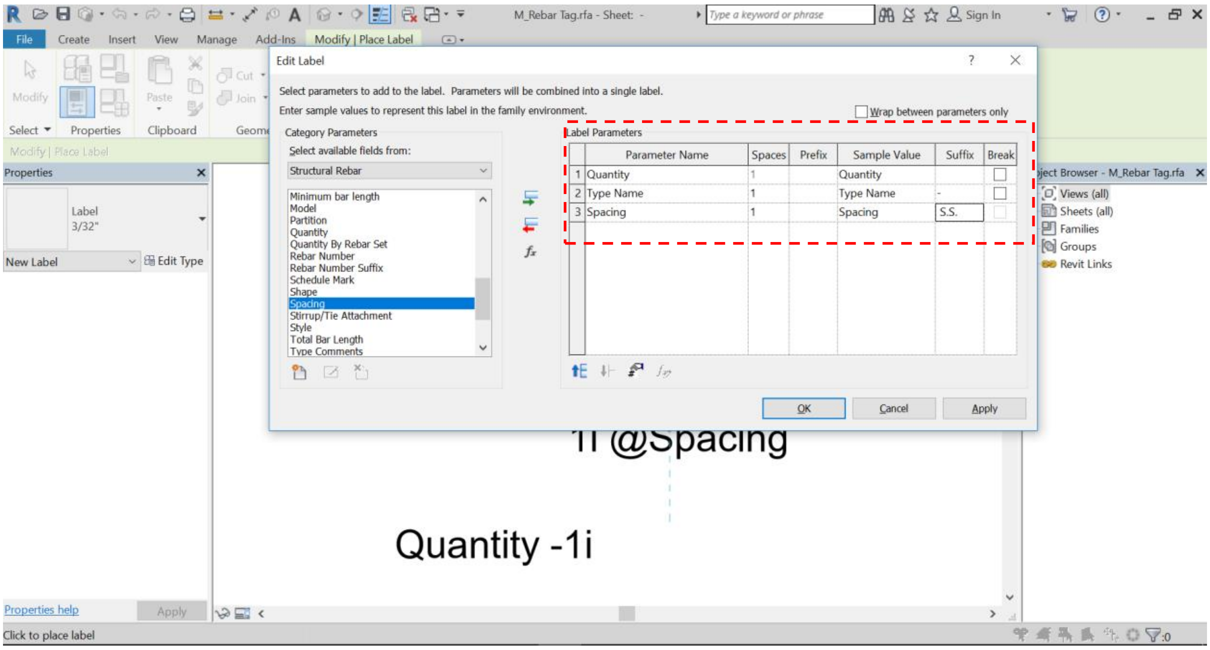
(5) Next, edit the links tag. Double click on it to edit its family.

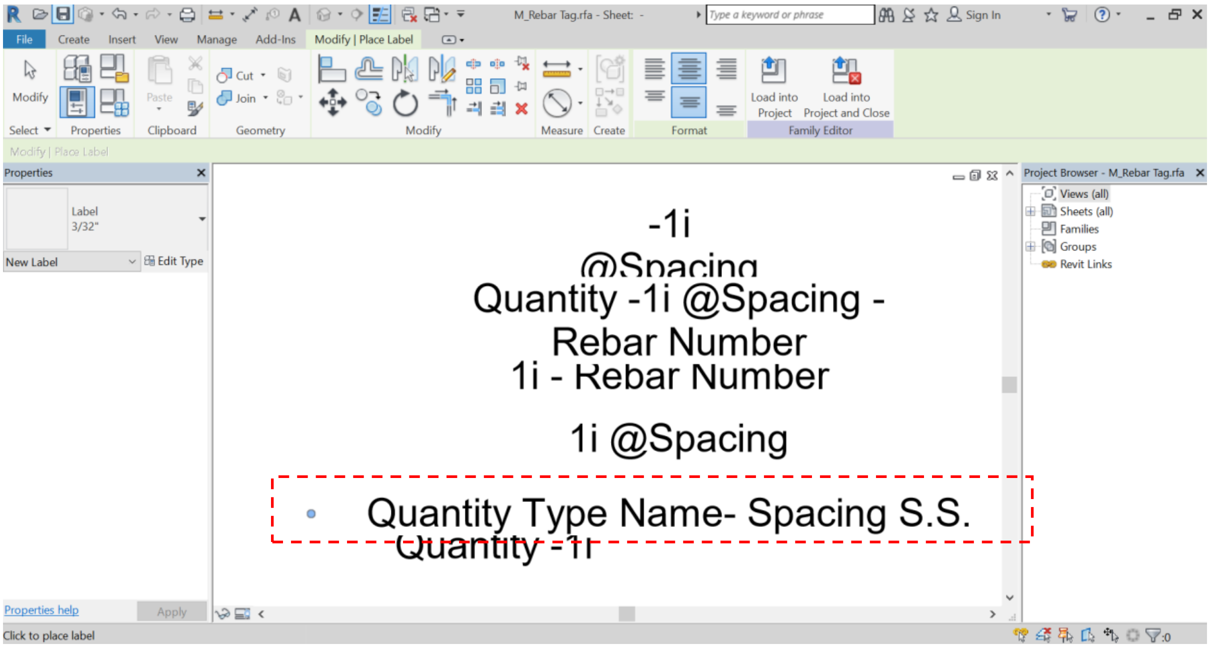
In the family editor, click "Label" to create a new label. Click a location and start editing the new label.

Add parameters "Quantity", "Type Name" and "Spacing" to the Label Parameters. Also type in "-" as Type Name's suffix and "S.S." as Spacing's suffix as shown below.

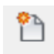
Click "OK" then stretch the label outline so that the text display in one line.



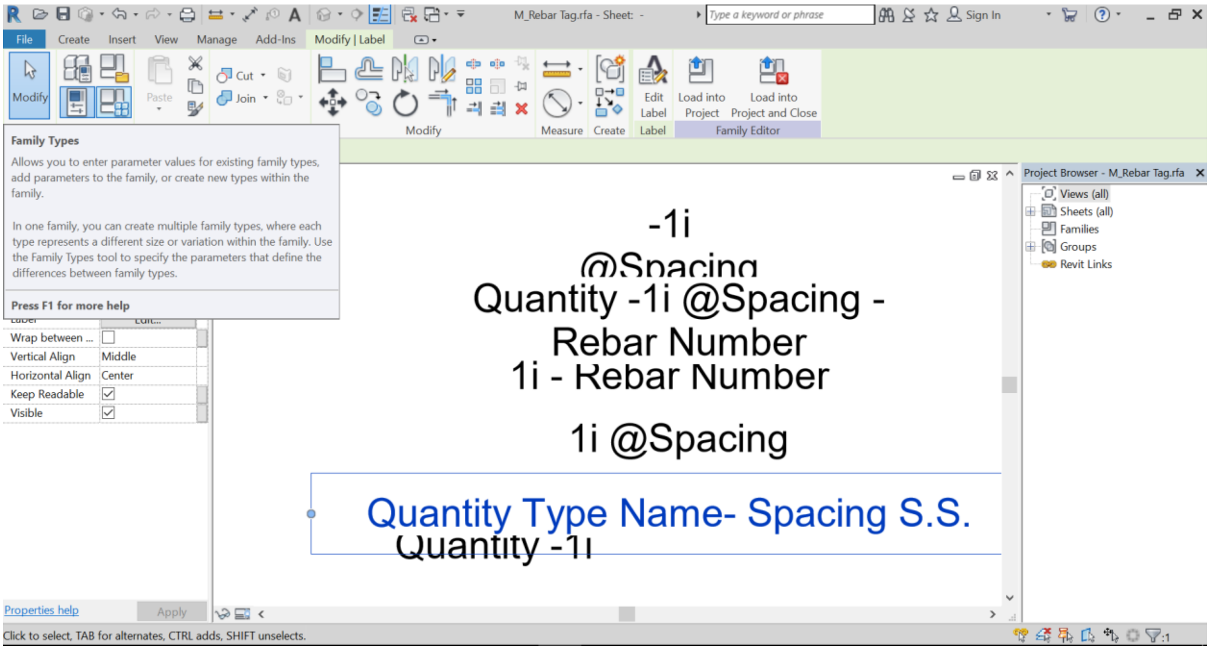


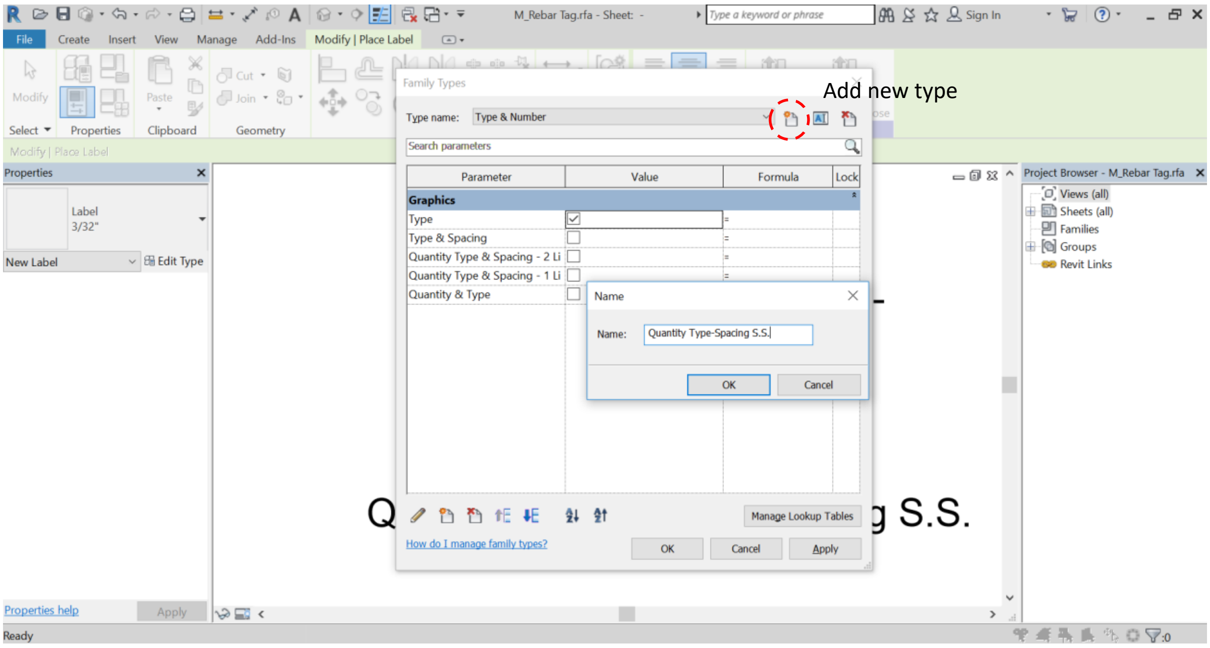


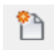
Before saving or loading the family into the project, you should add a new type to this family for selection.

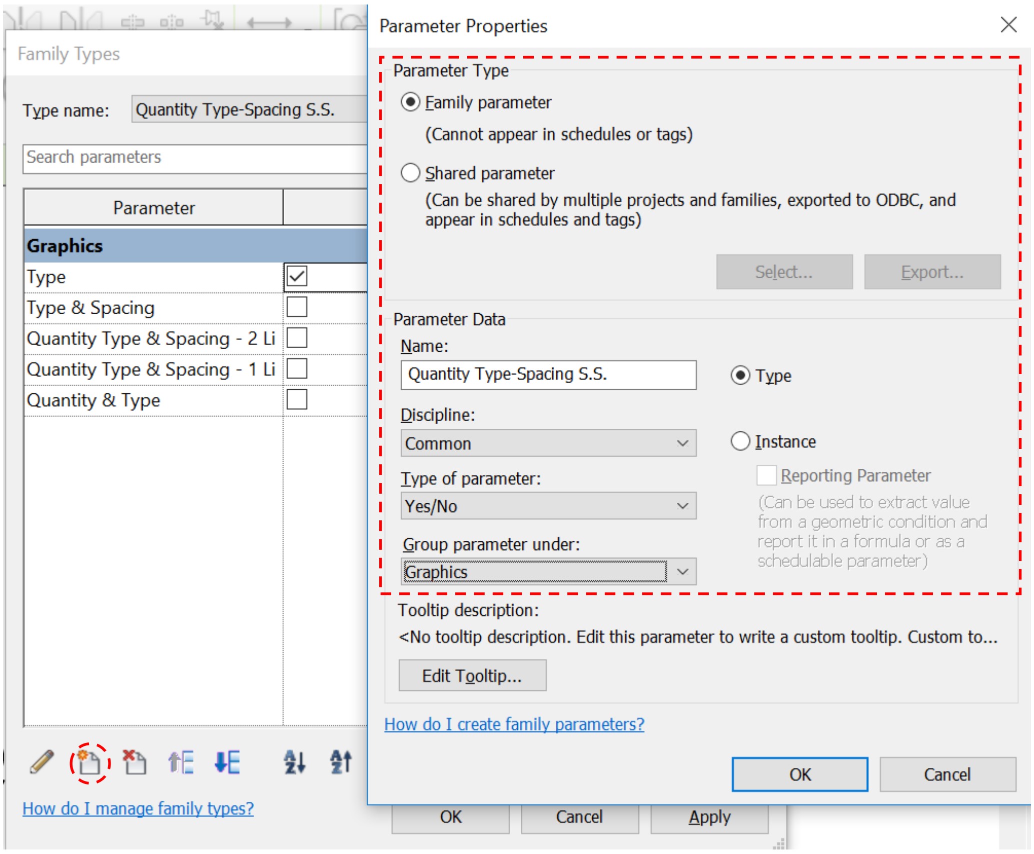


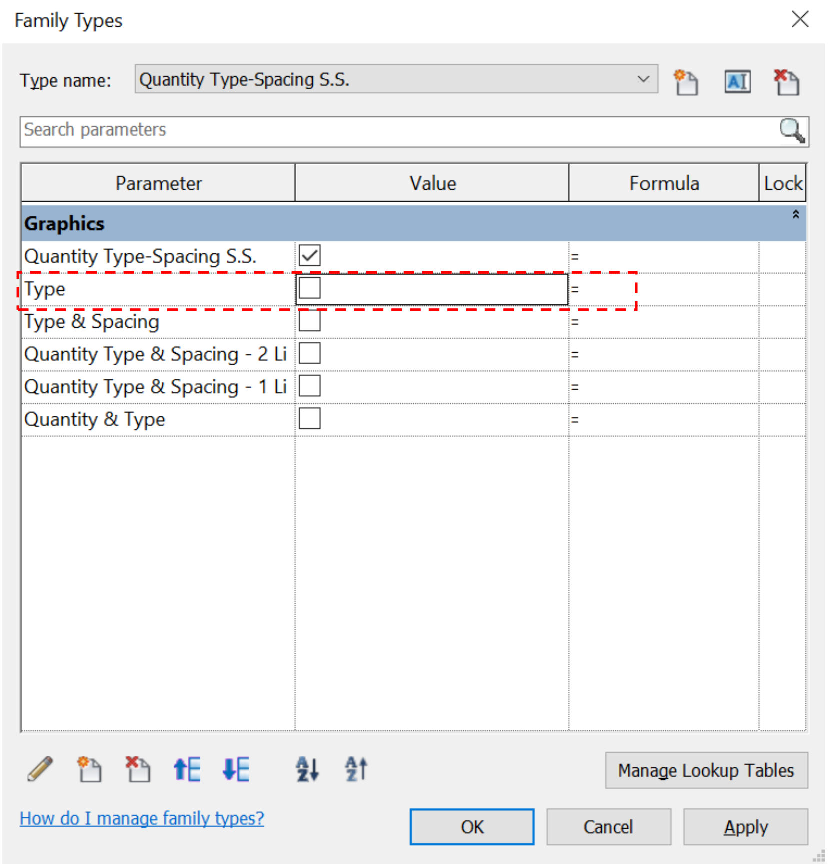
Select the new label and click “Family Types”. In the Family Types editor, click new type and name it as “Quantity Type-Spacing S.S.” and click “OK”.





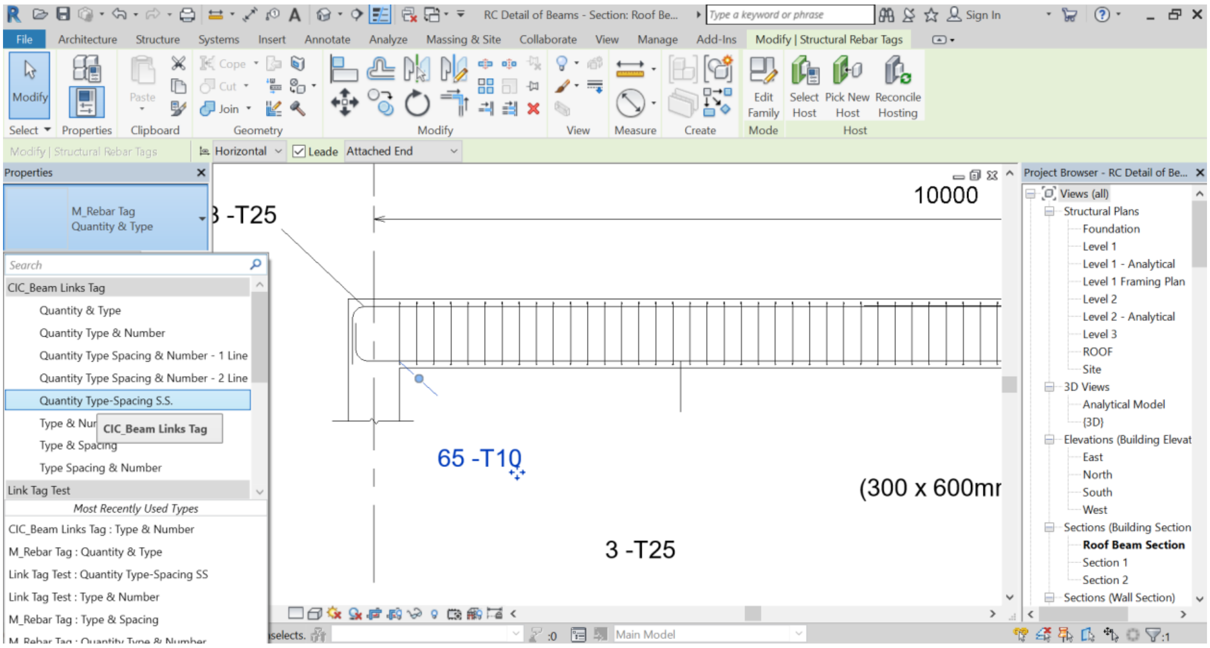
Next, add new parameter to this type by clicking the  icon at the bottom. Set the new parameter name as “Quantity Type-Spacing S.S.” and other settings as shown below. Click “OK” and then uncheck the “Type” parameter. Click “OK” to finish editing the new type.

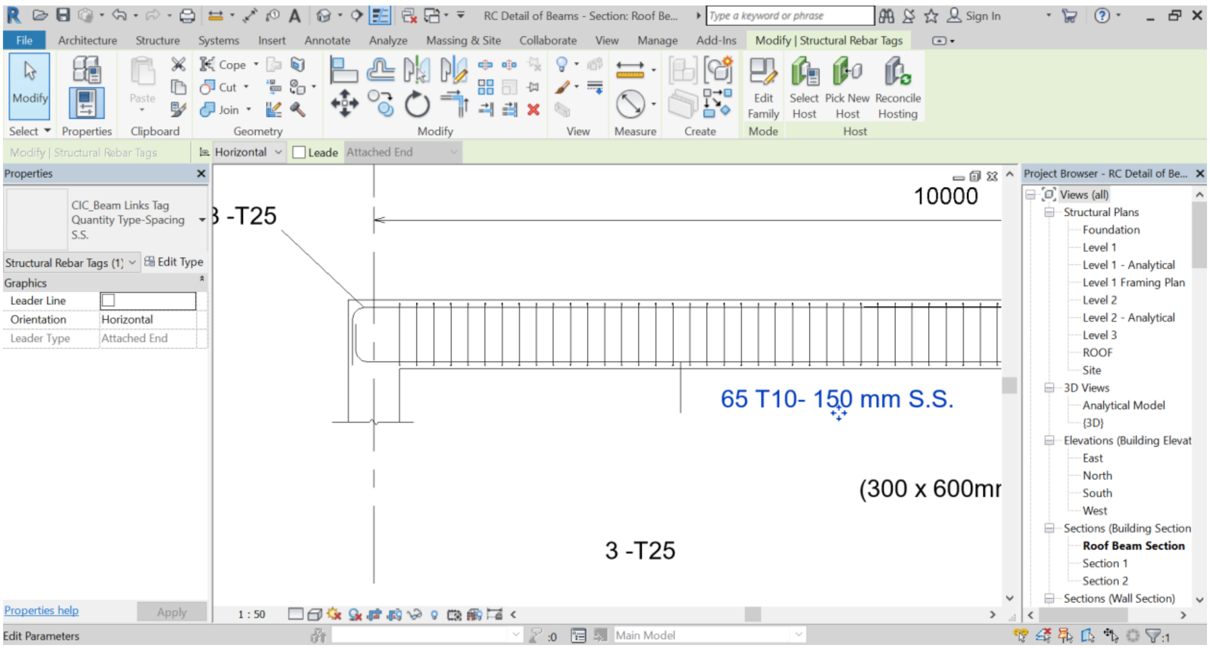




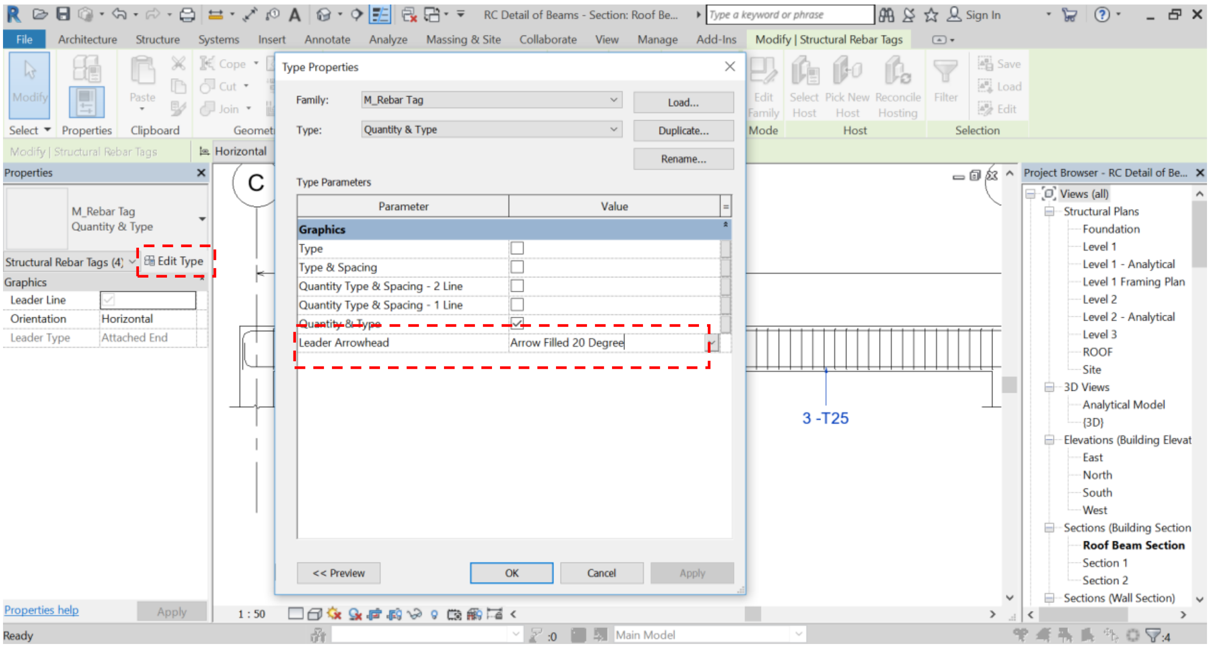
Click “Load into Project and Close” and click “Yes” to save change. Save the family as “CIC_Beam Links Tag.rfa”.

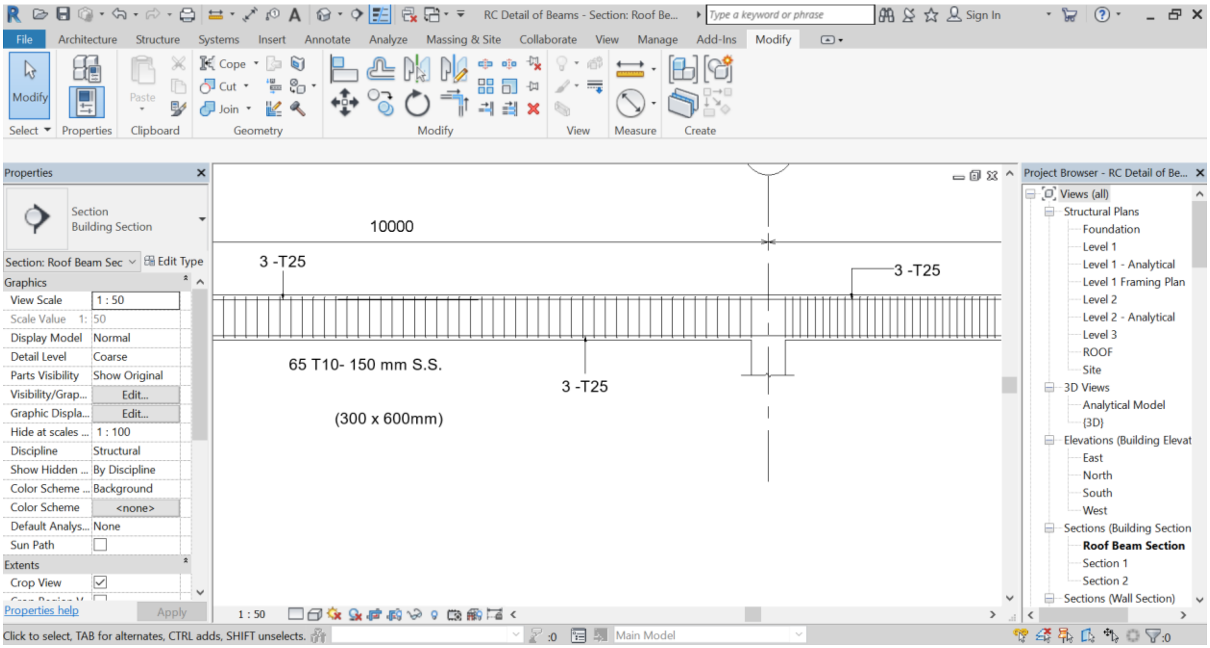
Now select the links tag and change its type to CIC_Beam Links Tag: "Quantity Type-Spacing S.S." from the Type Selector. And uncheck "Leader Line" and move the link tag to an appropriate location.





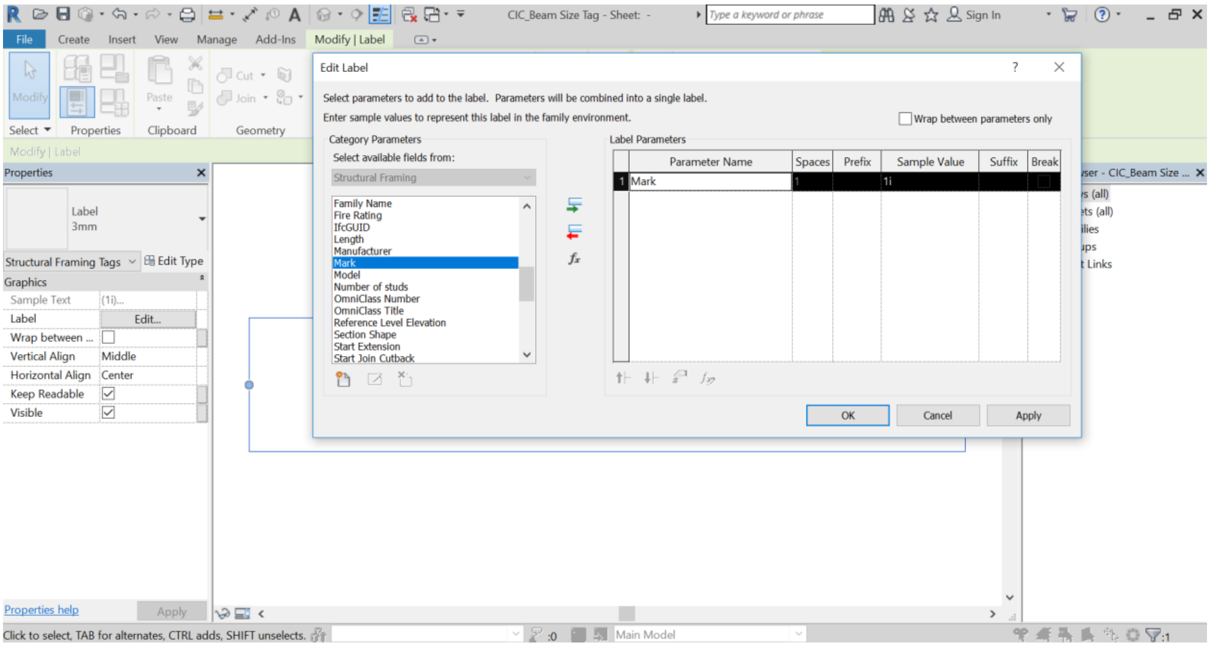
(6) For the main rebar tags, select them and click "Edit Type" and display arrow set to "Arrow Fill 20 Degree". Click "OK" and then move these tags to proper locations.



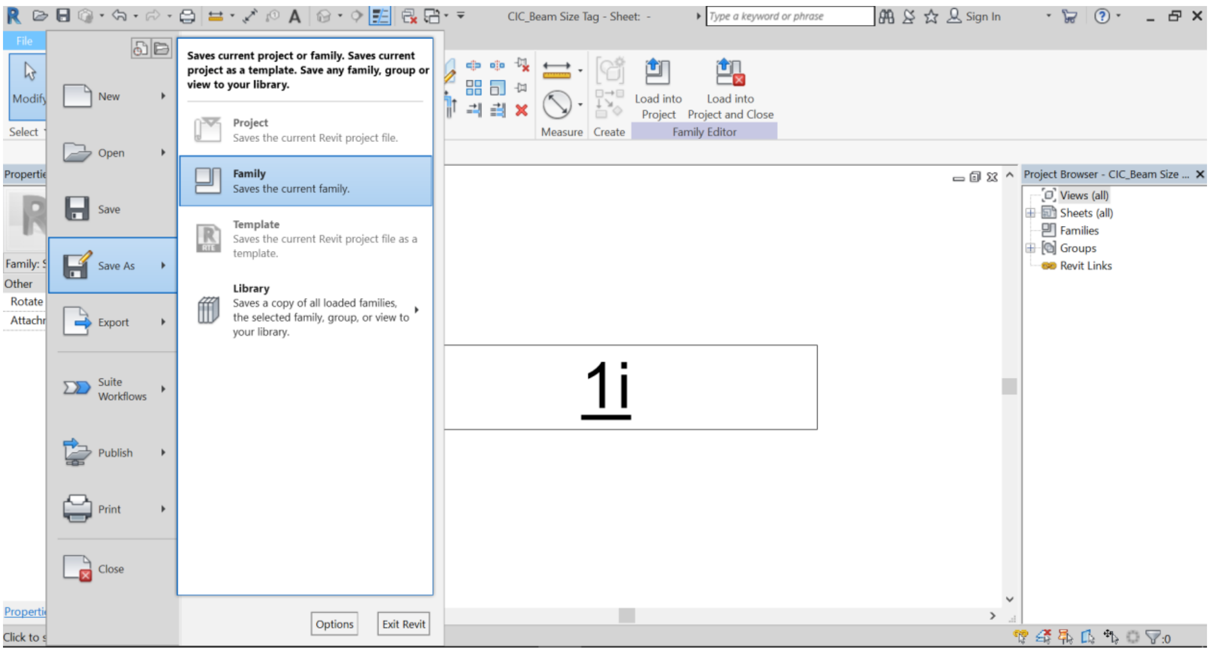


(7) Now, copy the beam size tag and double click to edit it. In the Family Editor, select the label and edit it.

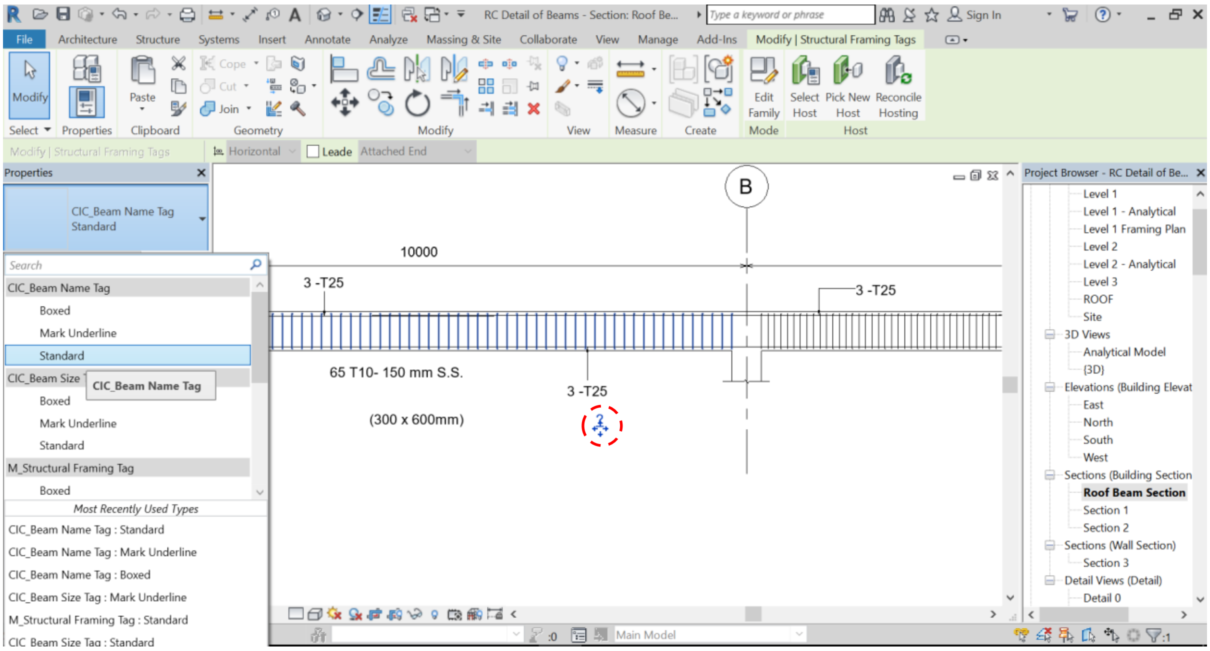
Remove the "Type Name" parameter and add the "Mark" parameter as shown below.



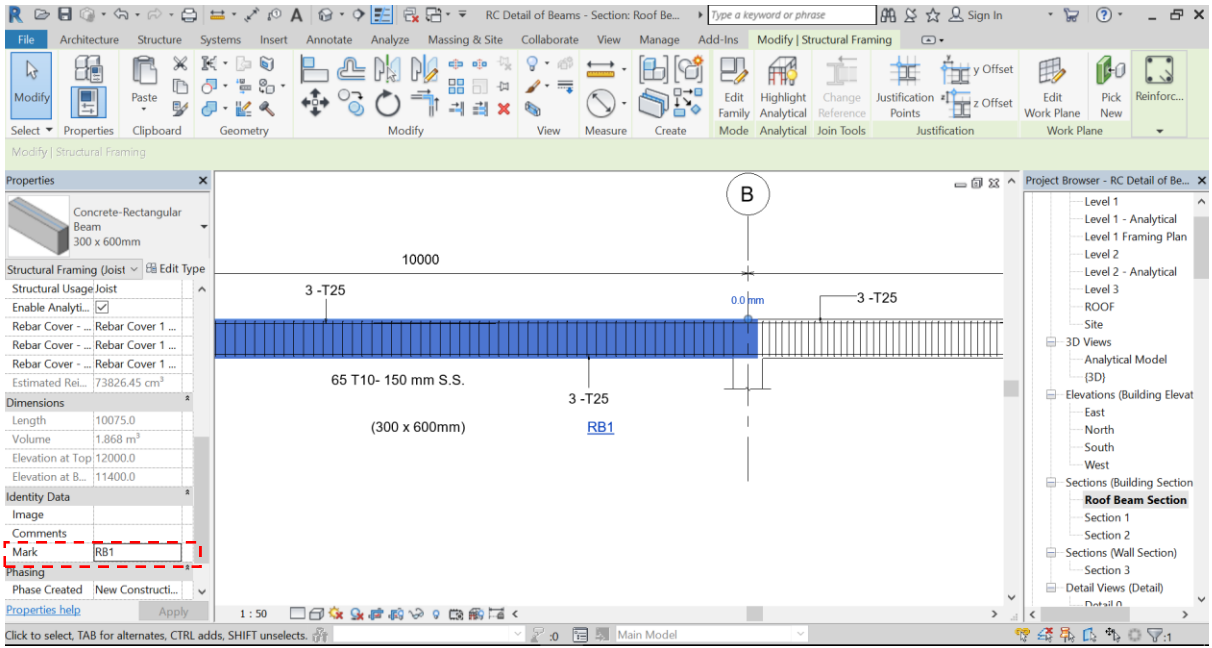
Save the edited family as "CIC_Beam Name Tag.rfa". Then click "Load into Project and Close".



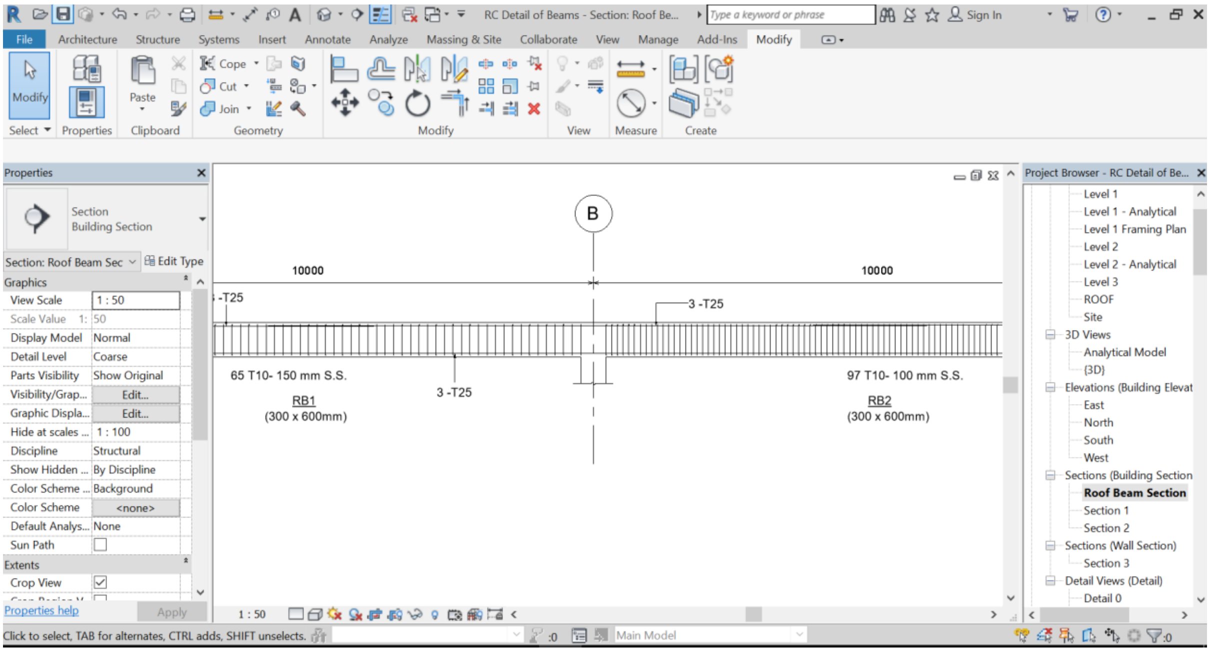
Select the beam tag and change its type to "CIC_Beam Name Tag: Standard". You will notice that Revit display the tag as "?". This is because you have not given the beam a "Mark".



To give a beam mark, select the left beam and scroll down to find "Mark" from its Properties Palette. Type in "RB1" for this beam and click "Apply".

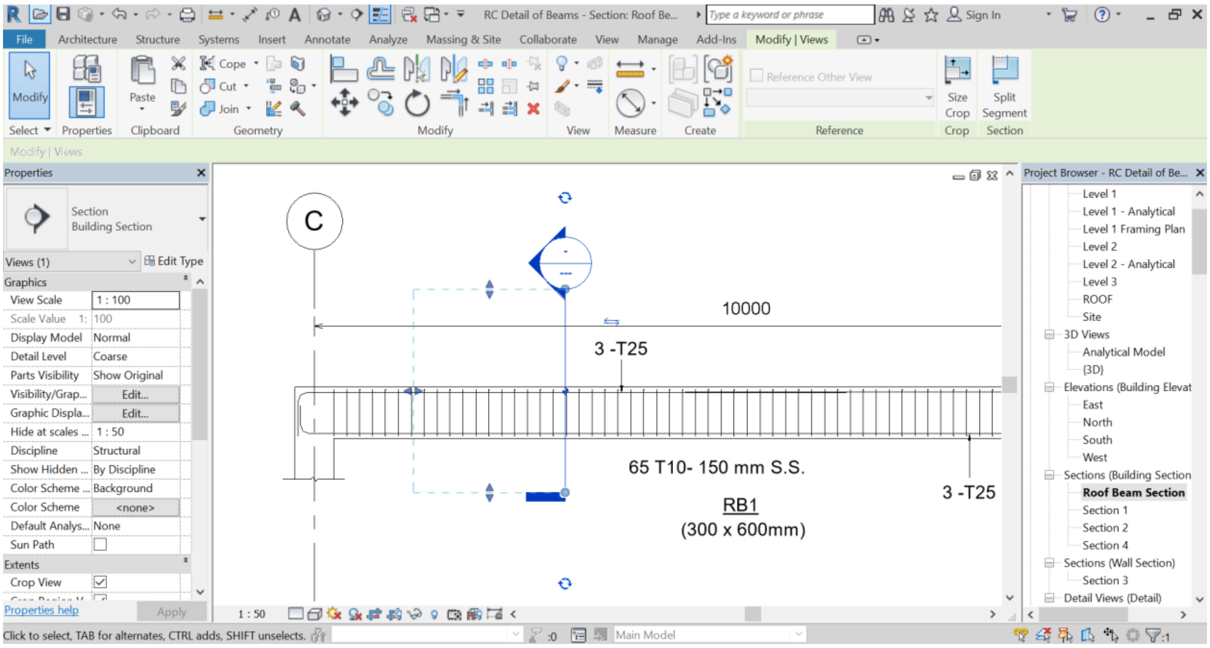


For the beam on the right, type in its mark as "RB2".

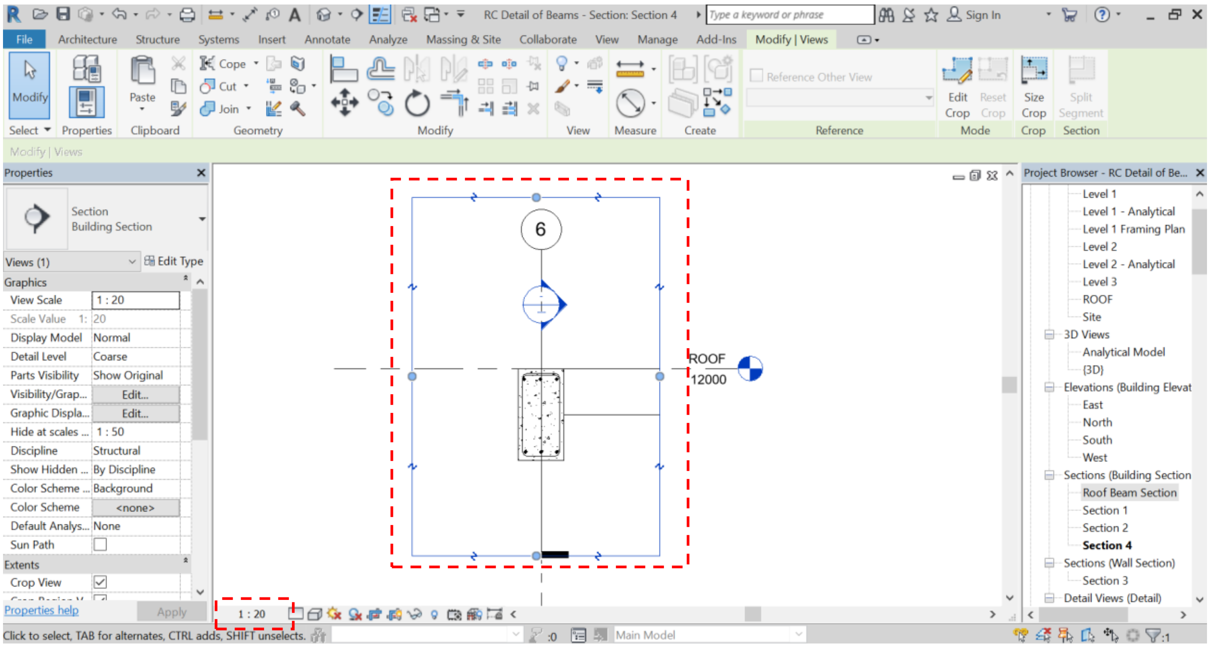


(8) The beam elevations have been from the above steps. Next, add a beam section for the beam RB1.

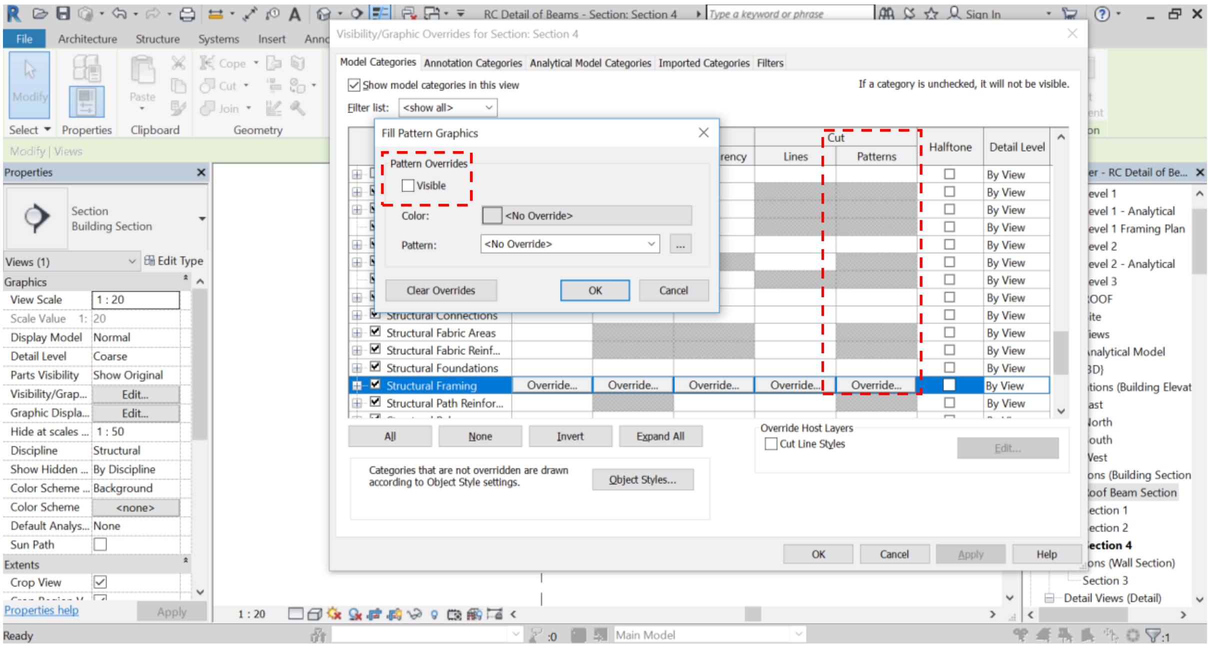
Click View tab ➢ Create panel ➢ Section and choose Building Section. Place the section as shown below.

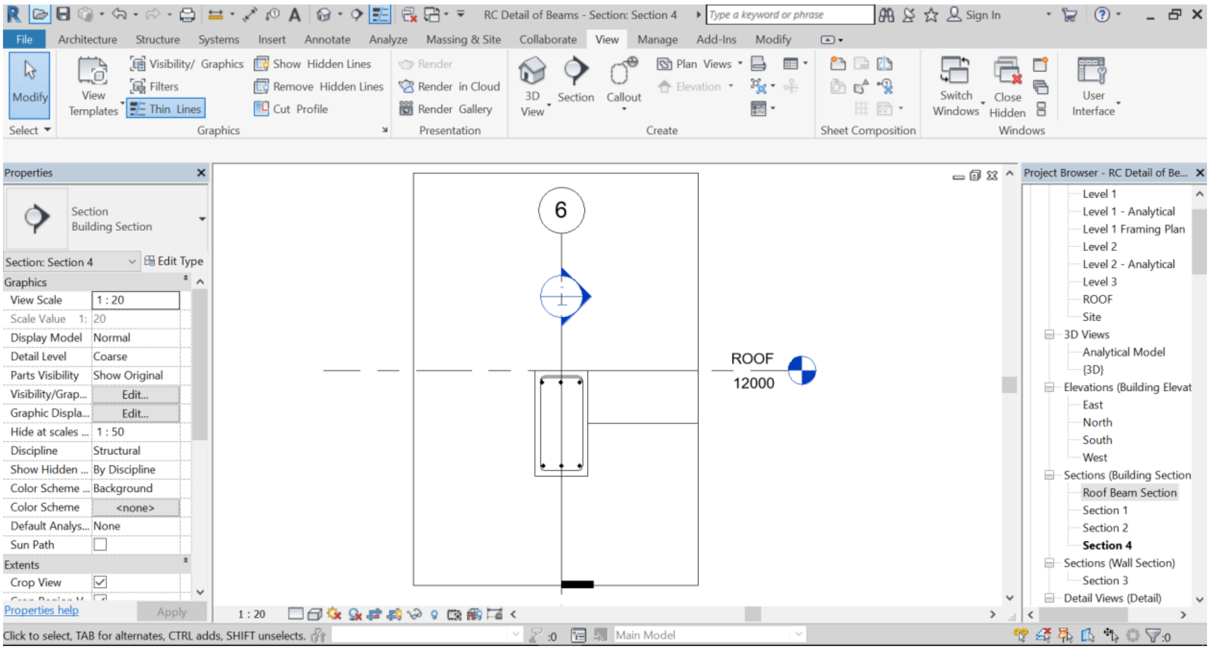


Resize the section view crop region and move Grid-6 closer as shown below. Also change the scale to 1:20.



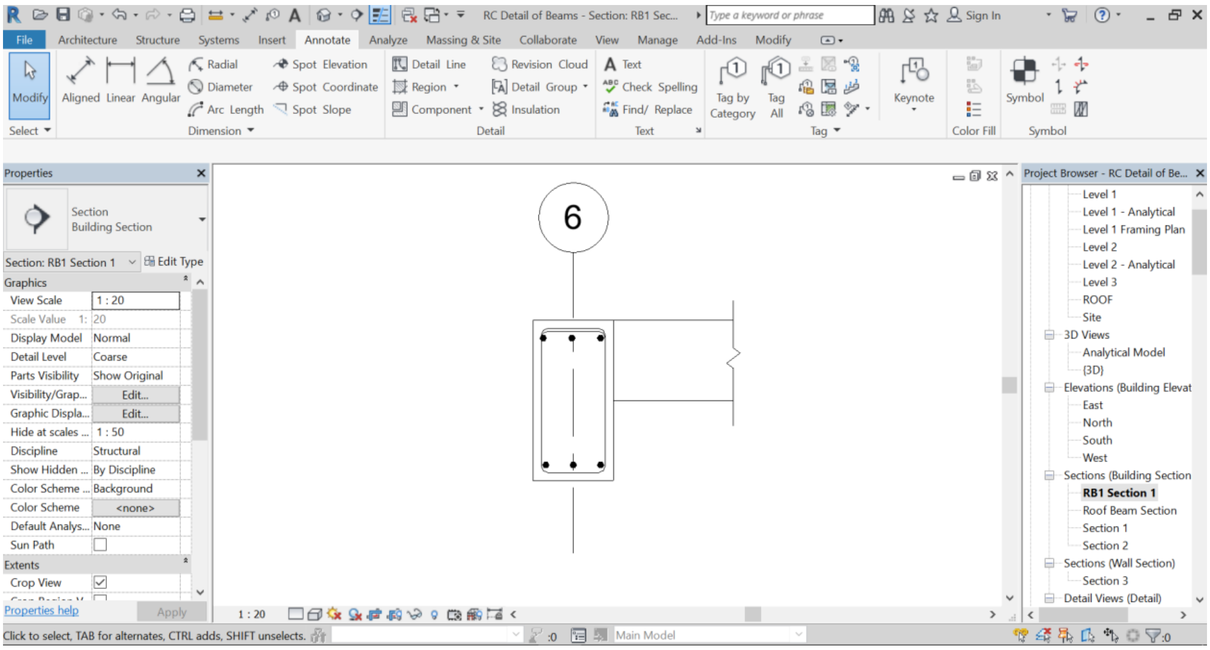
Next, change the visibility to show no hatching. Type “vv” for Visibility/Graphics control and set Cut pattern for Structural Framing to not visible.



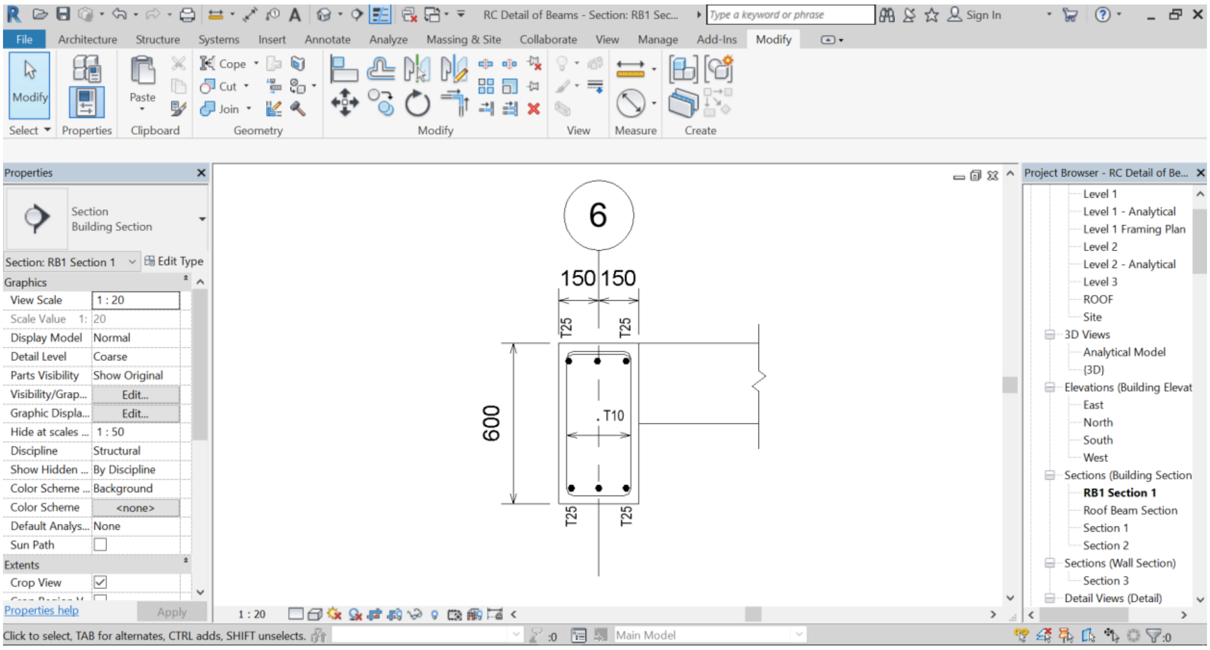


Tidy up this beam section view by:

- Hiding “Level” and “Section line” in this view.
- Add break line to the slab. (Break lines are Detail Components and can be found in Annotate tab ➢ Detail panel ➢ Component. Click “Load Family” and find the break line detail item “M_Break Line” from the path C:\ProgramData\Autodesk\RVT2018\Libraries\Hong_Kong\Detail Items\Div 01-General.)
- Hide Crop View.  (button is located at the bottom of the draw area.)
- Rename the section as “RB1 Section 1”.

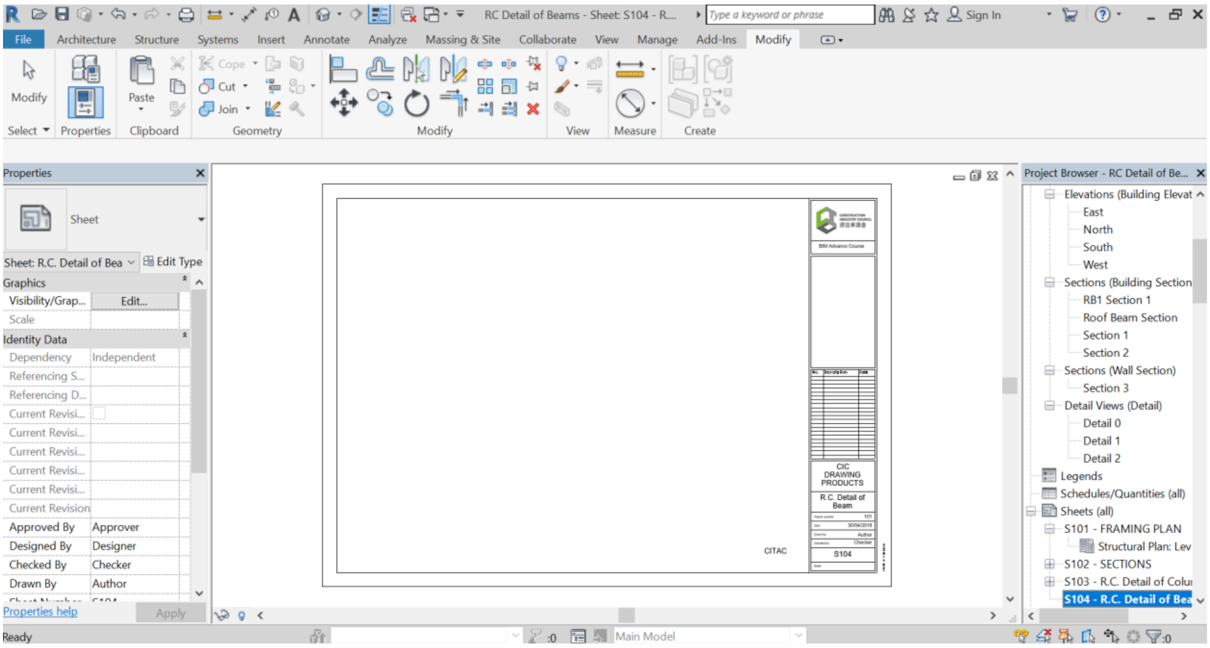


(9) Now, use the methods described in Steps (3) to (7) above, try to annotate the beam section of RB1 as shown below.

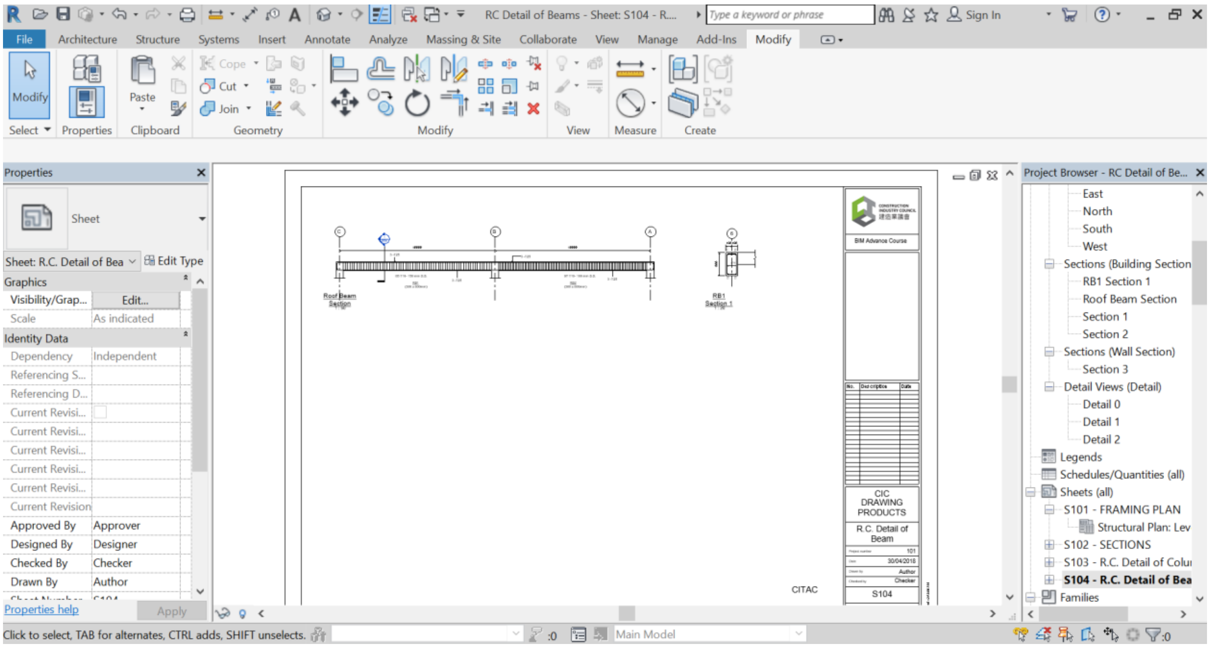


17 With the beam elevation and section created, you can now create a new sheet to place these details as a R.C. Detail of Beam drawing.

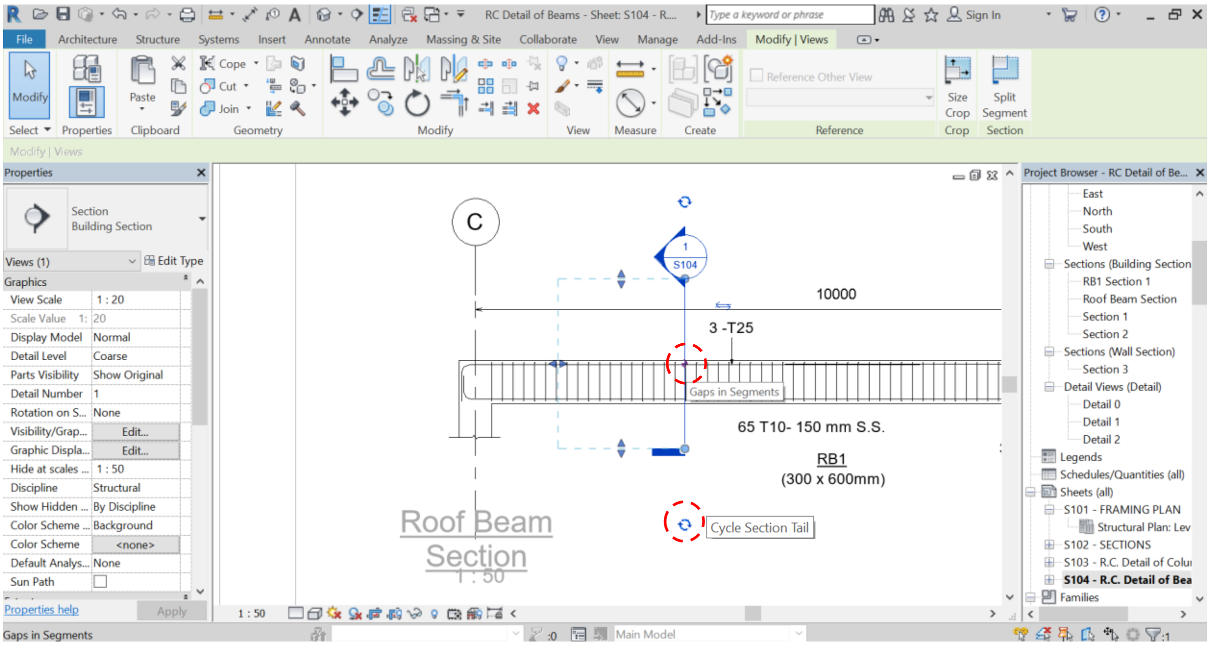
(1) To add a new sheet (right click on "Sheets (all)" and select "New Sheet"). Use the CIC Course Title Block then rename this sheet to "R.C. Detail of Beam" (right click and rename).

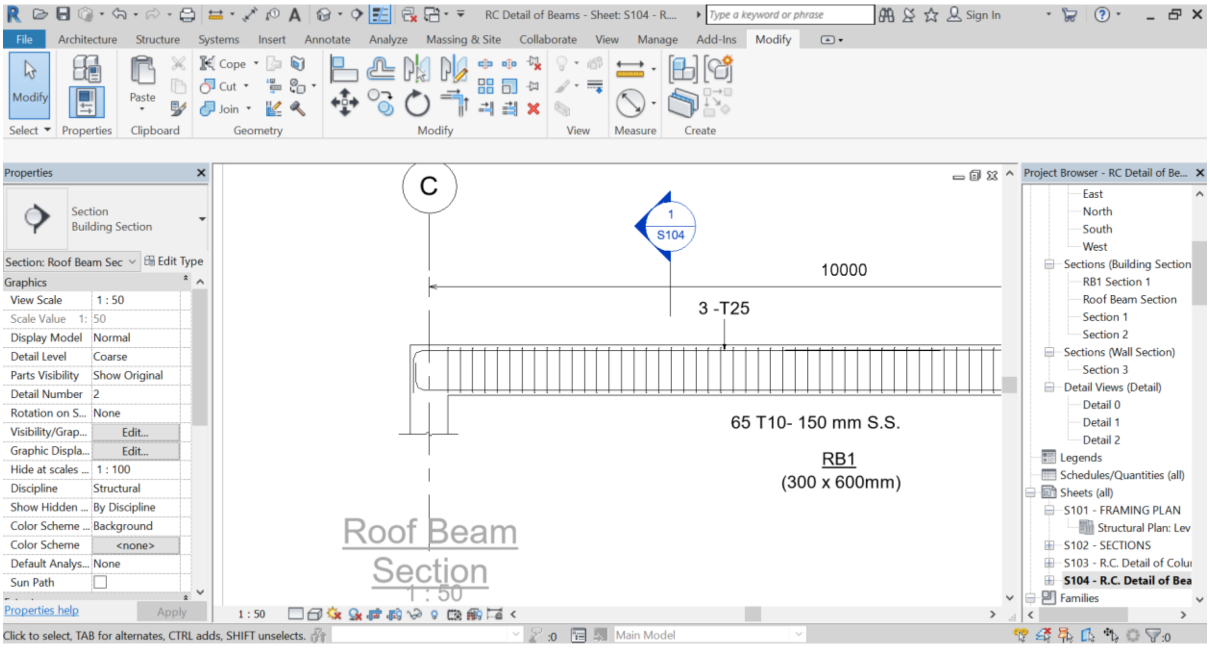


(2) With the sheet "R.C. Detail of Beam" still activated, select "RB1 Section" and drag it into the sheet. Do the same for "Roof Beam Section".

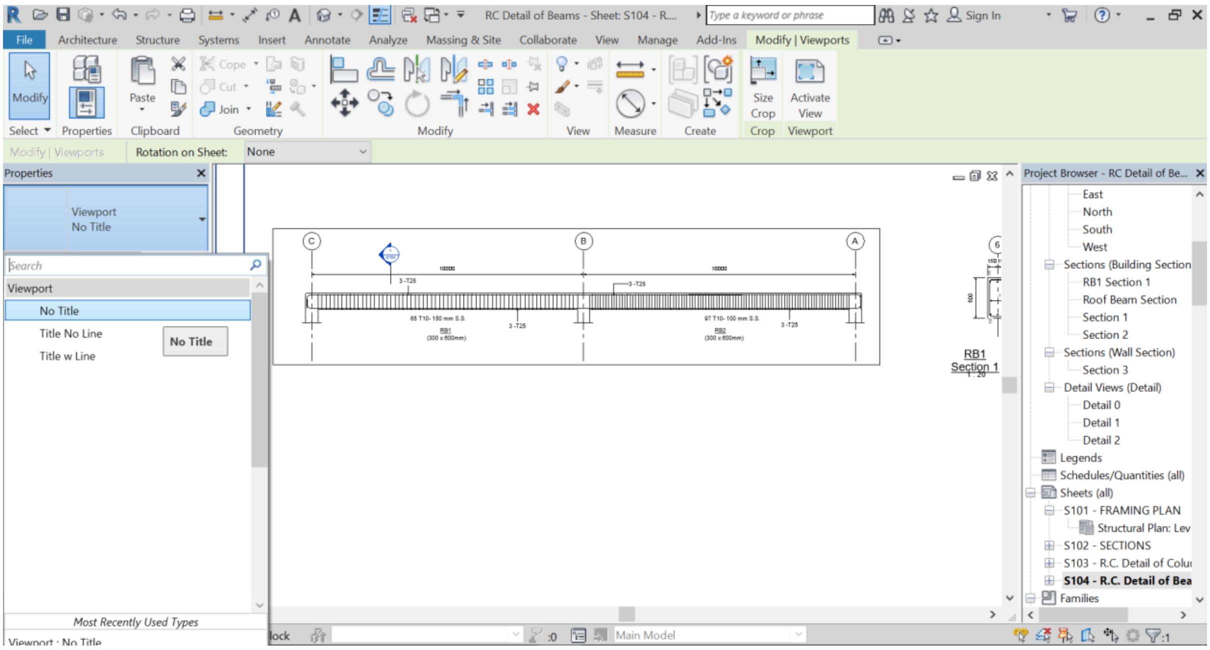


(3) Double click on the Roof Beam Section in the sheet. Then select the section line and click the "Gaps in Segments" symbol to provide gap to the section line. Also click the "Cycle Section Tail" symbol to hide the section tail.

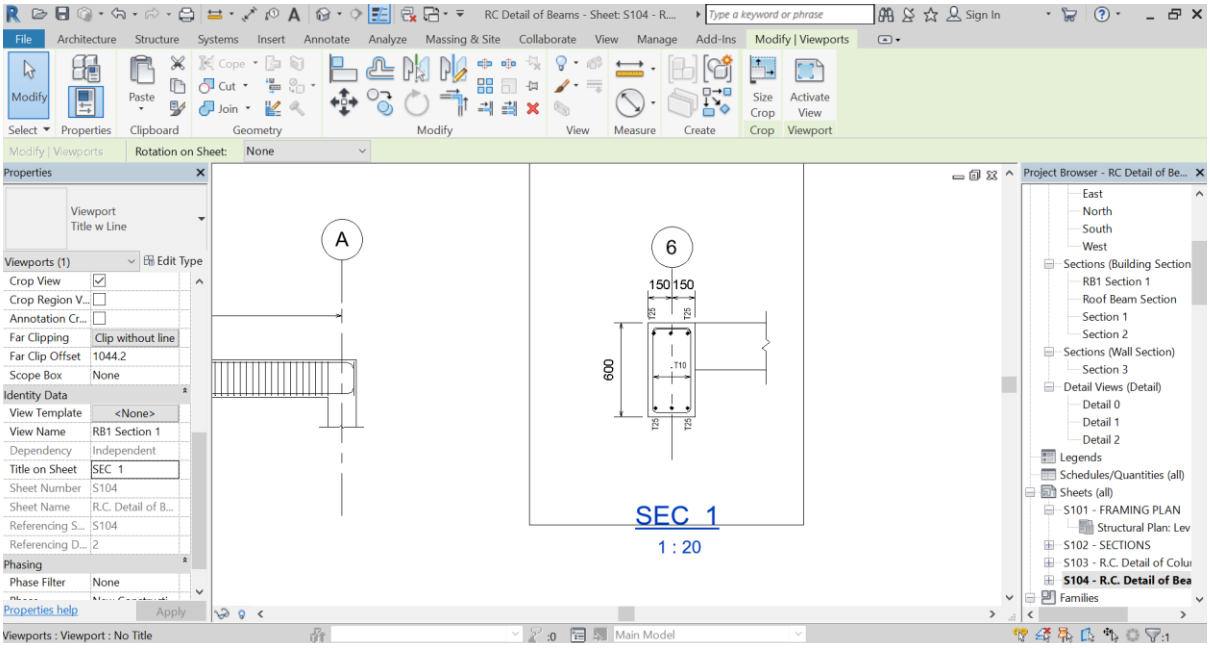


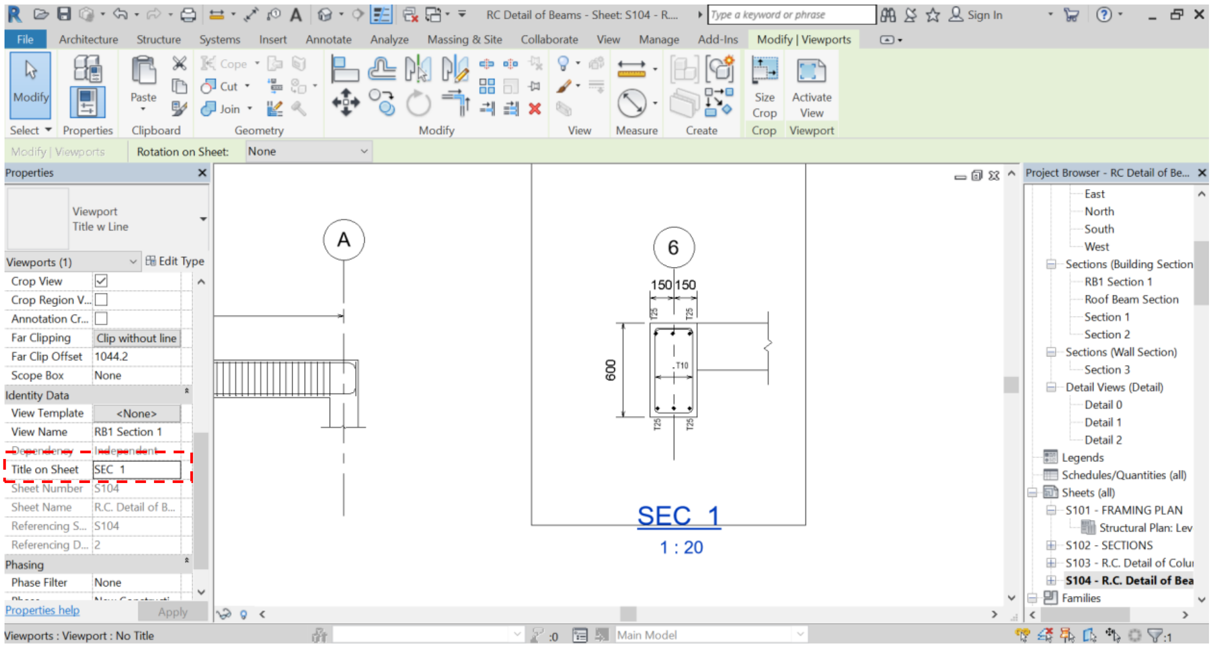


(4) Change the viewport type of the Roof Beam Section view to "Viewport No Title".

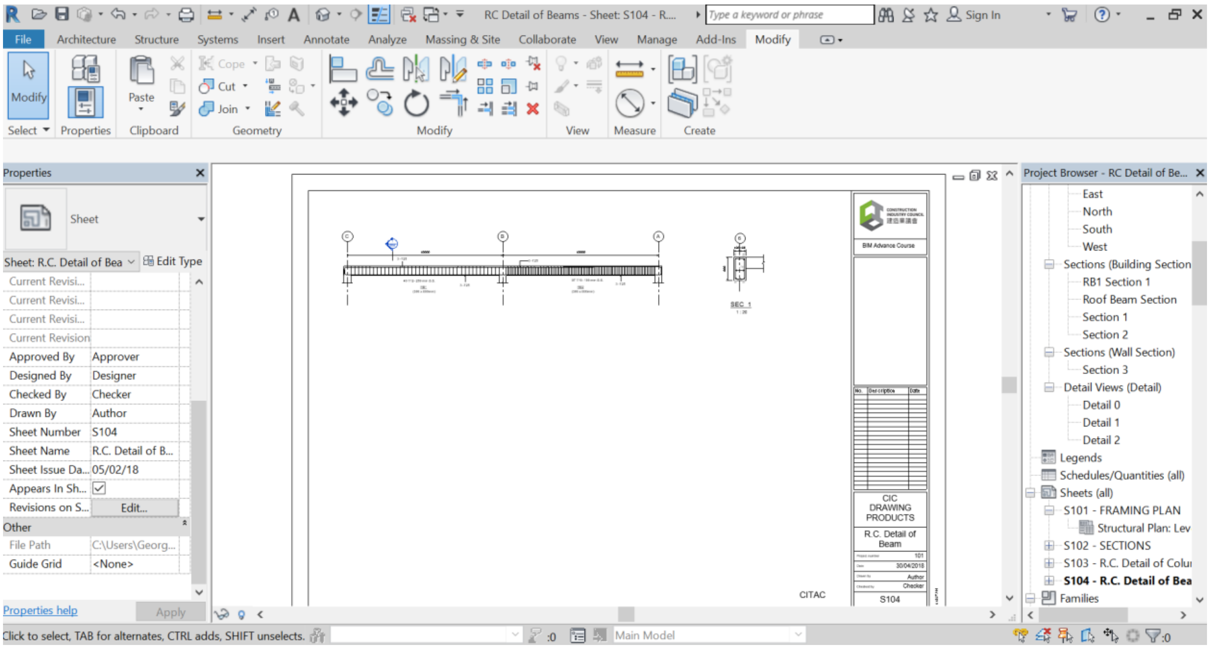


(5) Next, change the title of the "RB1 Section 1" viewport to "SEC 1".





Finally, you have created a R.C. Detail of Beam drawing.



You can follow the steps described above and draw the beam section for RB1 and try to annotate the support of the beam as well.

Note that the sections included in this drawing are parametric. Hence, if you change the size or rebars details of the model, then all related details in the drawing will be updated automatically. An example is shown below for reference.

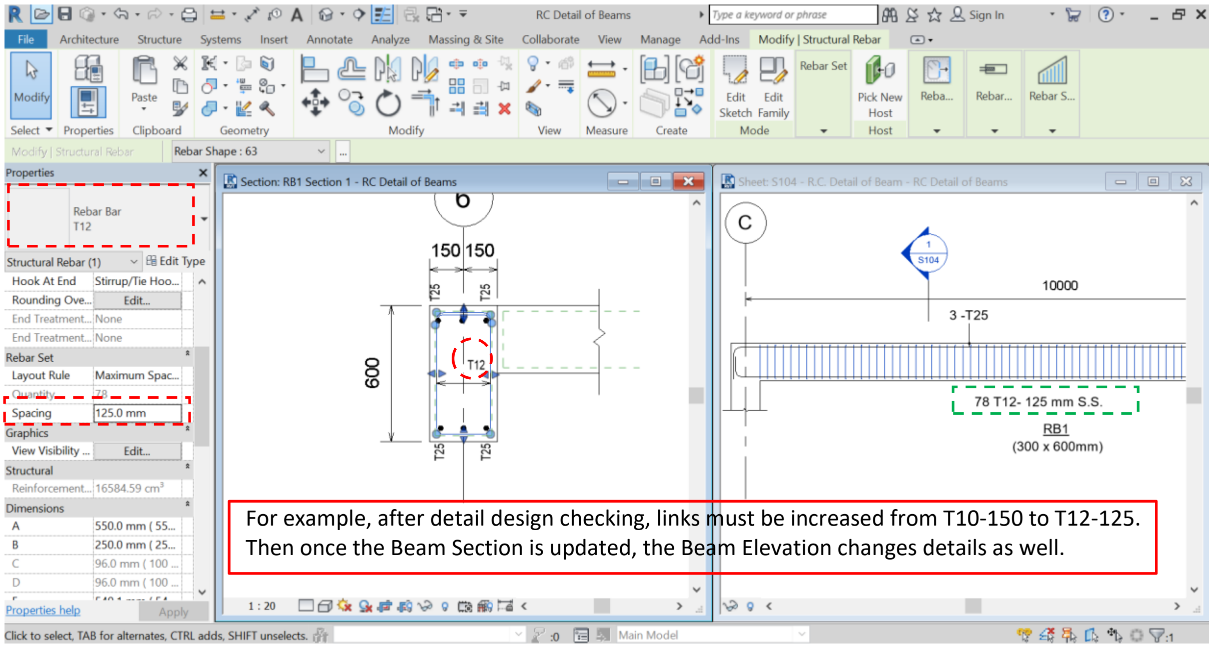


Table of Contents

CHAPTER 18 – Model Standard (Structural)

The BIM standard set out the methodology of modelling elements in Revit and other BIM software. The BIM uses provide a guideline for modelling following the Level of Developments (LOD). In this chapter, we will introduce the common standards used for BIM Modelling and their BIM uses in Structural Engineering.

The following two standards are commonly referred.

1. BIM Standards – CIC Building Information Modelling Standards (Phase One)
2. PAS 1192 by BSI from UK

Please refer to the chapters in each standard.

CIC BIM Standards – CIC Building Information Modelling Standards (Phase One)

Reference could be made on the following Chapters for modelling standards:

- Chapter 2 – Modelling Methodology
- Chapter 2.1.3. – Structural Modelling Guidelines
- Chapter 2.3. – Collaborations Procedures.
- Chapter 3.2.3. – LOD Responsibility Matrix – Structural Model
- Chapter 3.3.3. – LOD Specifications – Structural Model
- Chapter 4.1 – Folder Structure
- Chapter 4.3 – Drawing Production

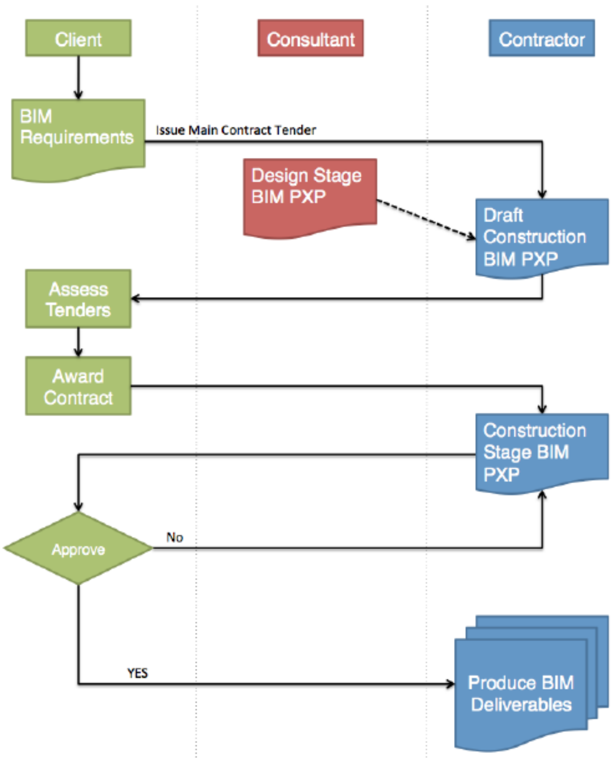


Table of BIM Uses by Project Phases

	Concept Design, Inception Feasibility & Planning	Preliminary & Scheme Design	Detailed Design	Submission to Approving Authority	Tender Stage	Construction	As-built, Facilities Management, Operation
Design Authoring	Y / N	Y / N	Y / N	Y / N	Y / N	Y / N	
Design Reviews	Y / N	Y / N	Y / N	Y / N	Y / N	Y / N	
Existing Conditions Modelling	Y / N	Y / N	Y / N			Y / N	Y / N
Site Analysis	Y / N	Y / N	Y / N	Y / N			
3D Coordination			Y / N	Y / N	Y / N	Y / N	
Cost Estimation	Y / N	Y / N	Y / N	Y / N	Y / N	Y / N	
Engineering Analysis			Y / N	Y / N	Y / N	Y / N	
Facility Energy Analysis			Y / N		Y / N	Y / N	
Sustainability Evaluation	Y / N	Y / N	Y / N	Y / N	Y / N	Y / N	
Space Programming	Y / N	Y / N	Y / N	Y / N			
Phase Planning (4D Modelling)			Y / N	Y / N	Y / N	Y / N	
Digital Fabrication			Y / N		Y / N	Y / N	
Site Utilization Planning					Y / N	Y / N	
3D Control and Planning						Y / N	
As-Built Modelling						Y / N	Y / N
Project Systems Analysis						Y / N	Y / N
Maintenance Scheduling						Y / N	Y / N
Space Management and Tracking						Y / N	Y / N
Asset Management						Y / N	Y / N

Each of the BIM Uses in the table are defined below. The competencies and resources needed for each BIM Use are included in Appendix D.

Engineering Analysis

A process which uses the BIM model to analyse and assess different design options to determine the most effective engineering solution to meet design codes and client requirements.

For structural analysis, the analytical modelling software uses the model to determine the behaviour of a given structural system.

Reference - "CIC Building Information Modelling Standards (Phase One)." 30 Sep. 2015, http://www.cic.hk/cic_data/pdf/about_cic/publications/eng/reference_materials/CIC%20BIM%20Standards_FINAL_ENG_v1.pdf.

Structural Analysis

DESIGN

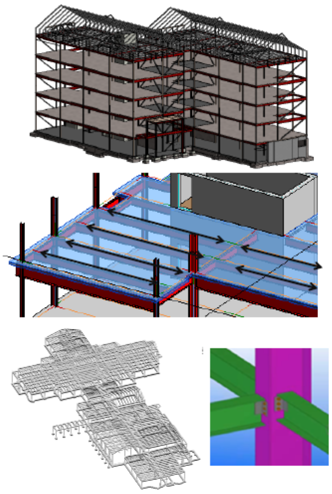
Description

A process in which analytical modeling software utilizes the BIM design authoring model so to determine the behavior of a given structural system. With the modeling minimum required standards for system design and analysis are used for optimization. Based on this analysis further development of the design takes place. The development of this information is the basis for what will be passed onto the digital fabrication and construction system design phases.

Value

- Improve specialized expertise and services offered by the design firm
- Achieve optimum efficient design solutions by applying various rigorous analyses
- Faster return on investment with applying audit and analysis tools for engineering analyses
- Improve the quality of the design analyses
- Reduce the cycle time of the design analyses

Examples



Code Validation

DESIGN

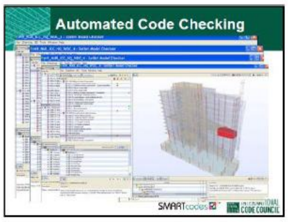
Description

A process in which code validation software is utilized to check the model parameters against project specific codes. Code validation is currently in its infant stage of development within the U.S. and is not in widespread use. However, as model checking tools continue to develop, code compliance software with more codes, code validation should become more prevalent within the design industry.

Value

- Validate that building design is in compliance with specific codes
- Reduces the chance of code design errors, omissions or oversights
- Continuous feedback on code compliance as design progresses
- Reduced turnaround time for 3D BIM model review by local code officials
- Save time on multiple checking for code compliance

Examples



PAS 1192

The production of co-ordinated design and construction information is a task- and time-based process, independent of which procurement route or form of contract is used. Each task needs to be carried out in a particular order for the mutual benefit of all those involved, otherwise known as "collaborative working". In a collaborative working environment, teams are asked to produce information using standardized processes and agreed standards and methods, to ensure the same form and quality, enabling information to be used and reused without change or interpretation. If an individual, office or team changes the process without agreement, it will hinder collaboration – a participant insisting on "my standard" is not acceptable in a collaborative working environment. (BSI, n.d.)

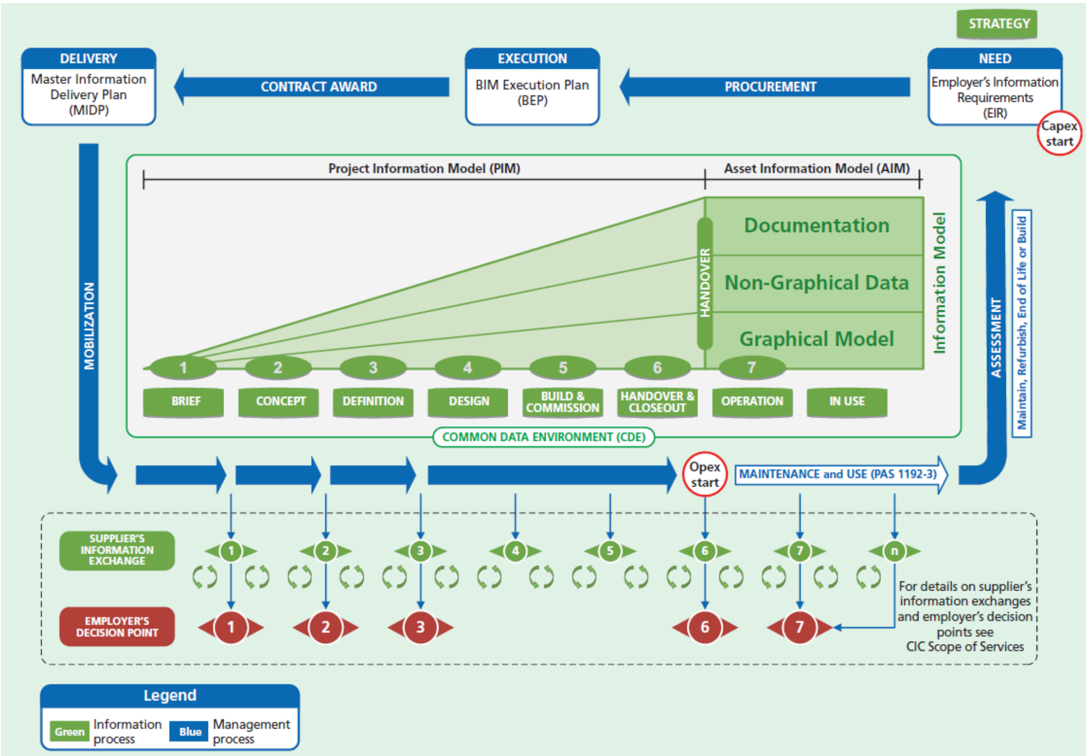
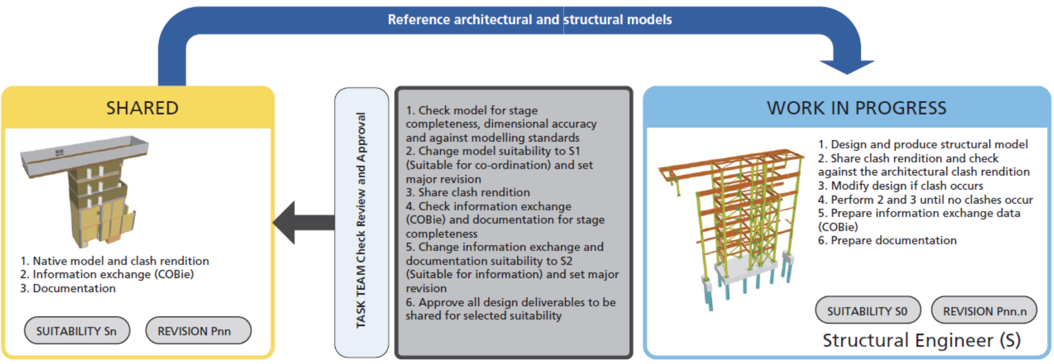


Figure 17 – Structural engineer's issue to SHARED



NOTE Copyright is claimed in this illustration. Reproduction of this illustration and making products from it might infringe that copyright. Details of the copyright owners can be found in the Foreword.

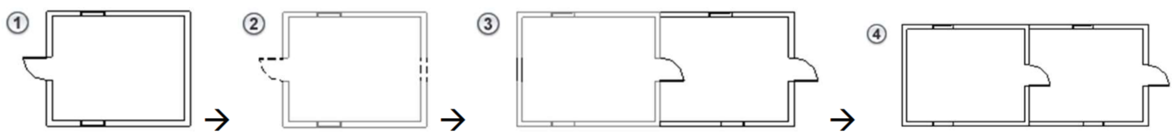
References - "Standards | BIM Level 2." http://bim-level2.org/en/standards/.

Project Phasing

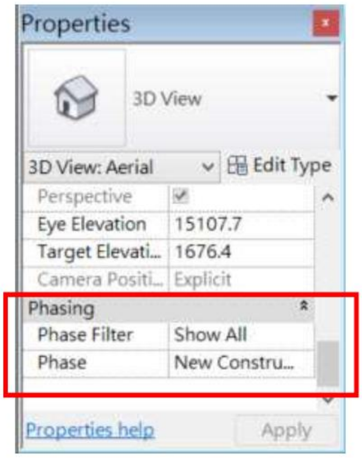
Projects phases represent a distinct time period in the life of the project. Revit tracks the phase for elements created or demolished. Phase filters are used to control the flow of building model information into views and schedules. This allows you to create phase-specific project documentation.

The following image shows a floor plan in different phases of a project: (1) existing

Construction, (2) demolition, (3) new construction, and (4) complete.



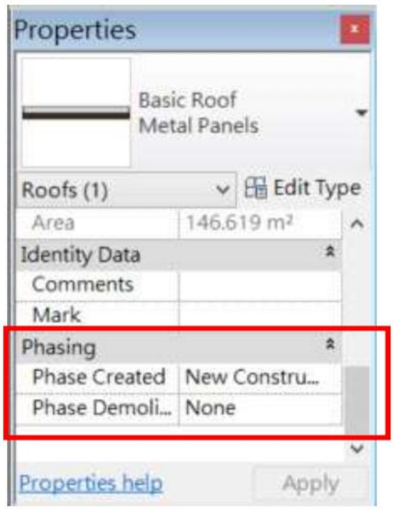
Phase Properties for Views



Each view in Revit has a Phase property and a Phase Filter property. The Phase property is the name of the view phase. When a view is opened or created, it automatically has a Phase value. You can copy a view and then select a different phase value for that view.

- For example, the original view has Phase 1; the copy has Phase 2. You create an element in Phase 1 and demolish it in Phase 2. The element displays as new in the original and as demolished in the copy.

- The Phase Filter property lets you control the display of elements in a view. For example, demolished walls may display in blue dashed lines, while existing elements display in solid black lines. You can apply a phase filter to a view to see elements from one or more specified phases.



Phase Properties for Elements

Each element that you add to a project has a Phase Created property and a Phase

Demolished property.

- The Phase Created property identifies the phase in which the element was added to the building model. The default value for this property is the same as the Phase value for the current view. You can specify a different value as needed.

- The Phase Demolished property identifies the phase in which the element was demolished. The default value is None. When you demolish an element, this property updates to the current phase of the view in which you demolished the element. You can also demolish an element by setting the Phase Demolished property to a different value.

Workflow for Phasing

This workflow describes a typical method of using phases and phase filters to manage a project.

1. Determine the phases of work that you want to track for the project, and create a phase for each. For example, depending on your needs, you might create phases named Phase 1, Phase 2, and Phase 3.

2. Create one or more copies of a view for each phase, and name these views appropriately.

For example, duplicate the Level 1 floor plan several times, and rename the views to

include the phase names: Level 1 - Phase 1, Level 1 - Phase 2, and so on. Tip: Organize the Project Browser by phase. The Project Browser sorts phases and views by name

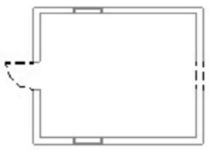
alphanumerically. Plan your phase names and view names carefully so they will be listed in the desired sequence.

3. For each view, on the Properties palette:

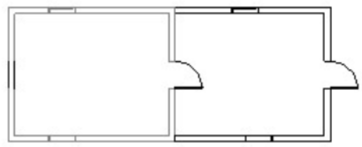
a) Set the Phase property to indicate the project phase to which the view corresponds.

b) Set the Phase Filter property to control the display of elements for that view.

Example:



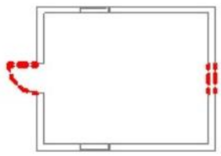
(1) Phase = Phase 1, Phase Filter = Demolition →



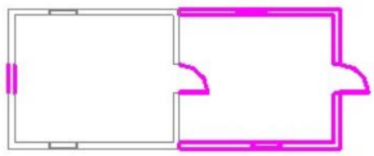
(2) Phase = Phase 2, Phase Filter = New + Existing

4. (Optional) Specify graphics for elements that are: existing, new, temporary, and demolished.

Example:



(1)Demolished elements = red dotted lines →



New elements = magenta solid lines

5. Specify the Phase property for each element. By default, elements are assigned to the phase that is specified for the view in which they are created. (For example, in the Level 1 - Phase 1 floor plan, you create a wall. By default, the wall is assigned to Phase 1.)

6. (Optional) Create phase-specific schedules for the project. (For example, you can create a Phase 1 door schedule that lists all doors to be removed or demolished, and a Phase 2 door schedule that lists all new doors to be purchased and installed.)

7. (Optional) Create phase-specific construction documents.

Phase Filters

A phase filter is a rule that you apply to a view to control the display of elements based on their phase status: new, existing, demolished, or temporary. Each view can show one or more phases of the construction. You can specify different graphic overrides for each phase status.

New. Element was created in the phase of the current view.

Existing. Element was created in an earlier phase and continues to exist in the current phase.

Demolished. Element was created in an earlier phase and demolished in the current phase.

Temporary. Element was created and demolished during the current phase.

Create a Phase Filters

Create phase filters can control the display of elements based on their phase status.

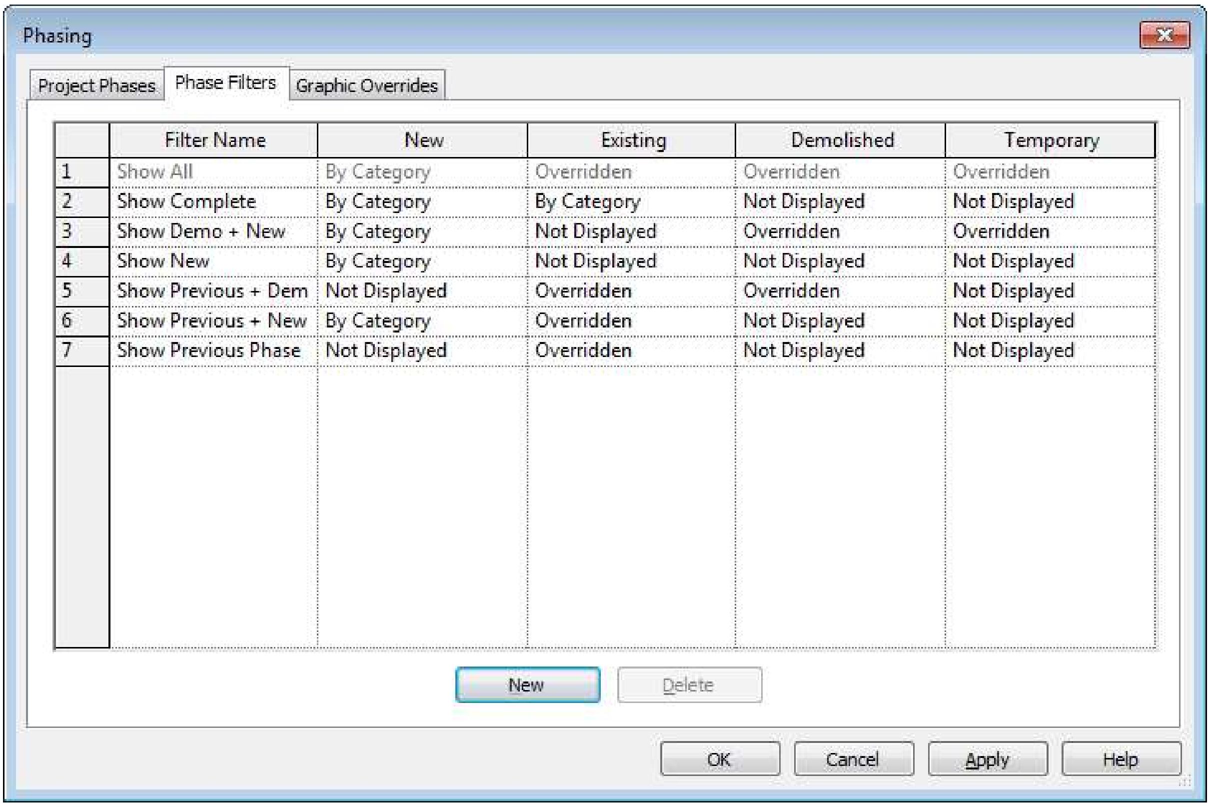
1. Click Manage tab ➤ Phasing panel ➤ (Phases).
2. In the Phasing dialog, click the Phase Filters tab.
3. Click New to insert a new phase filter. The filter is assigned a default name.
4. (Optional) Click in the Filter Name box to edit the name.
5. For each phase status column (New, Existing, Demolished, and Temporary), specify how you want to display the elements:

By Category. Displays elements as defined in the Object Styles dialog.

Overridden. Displays elements as specified in the Graphic Overrides tab of the Phasing dialog.

Not Displayed. Does not display the elements.

6. Click OK.



	Filter Name	New	Existing	Demolished	Temporary
1	Show All	By Category	Overridden	Overridden	Overridden
2	Show Complete	By Category	By Category	Not Displayed	Not Displayed
3	Show Demo + New	By Category	Not Displayed	Overridden	Overridden
4	Show New	By Category	Not Displayed	Not Displayed	Not Displayed
5	Show Previous + Dem	Not Displayed	Overridden	Overridden	Not Displayed
6	Show Previous + New	By Category	Overridden	Not Displayed	Not Displayed
7	Show Previous Phase	Not Displayed	Overridden	Not Displayed	Not Displayed

Apply a Phase Filters

Apply a phase filter to a view to control the display of elements for the view based on project phases.

1. Access view properties for the view.
2. On the Properties palette, for Phase Filter, select one of the following:
    a. A default phase filter.
    b. A phase filter that you created.
    c. None to apply no phase filter. (All elements are shown without any graphic overrides.)

Design Options

- Design options can be used to explore alternative designs in the building model. You can use design options to explore multiple designs as the project develops, all within the same project.

- At any time in the design process, you can have multiple sets of design options. Typically, each set of design options addresses a particular issue or area. After the final design is chosen, you can incorporate the chosen options into the main model and remove the alternatives.

Workflow for Design Options

This workflow describes a typical method of developing sets of design options for your building model.

1. Decide on the areas for which you want to develop design options.

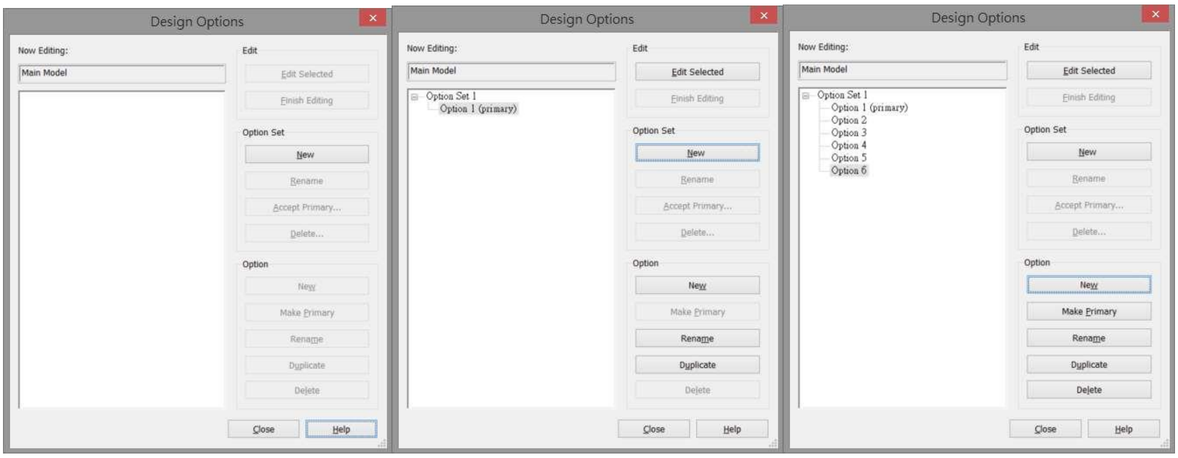
2. Create the building model, including all elements that will be common to all of the design options. (This is the main model.)

Note: If you add elements to a building and later decide that those elements should be part of a design option, you can move them to the design option.

3. Create a design option set for each area.

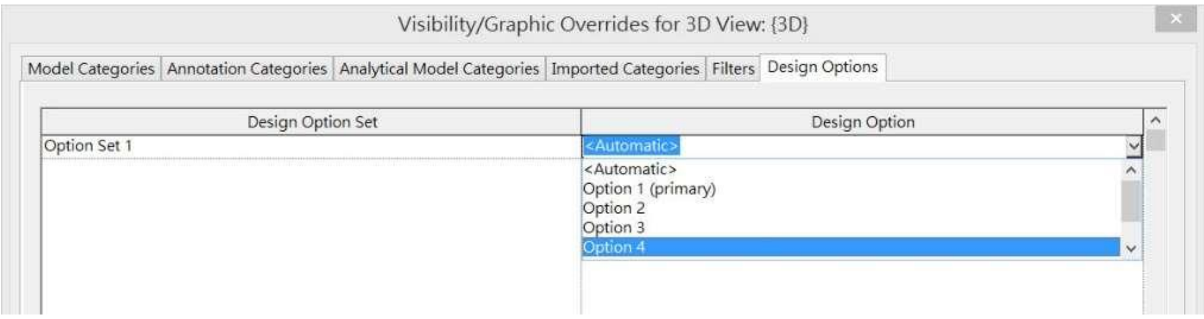
4. For each design option set, edit the primary option. When you create a design option set, Revit also creates a primary option for the set. The primary option is typically the preferred design that you think will be chosen. It will be displayed in project views by default. Other design options will appear in views only when you specify. Edit the primary option to add elements to the design as desired.

5. Create secondary options for each design option set. You can create one or more secondary options for each set. In general, any elements that will be modified or referenced in an option should belong to the design option instead of the main model.



6. Create views that display each design option.

By default, all project views display the main model with primary design options only. To see secondary options, create project views that show them. (These are called dedicated views.) You can then place these views on sheets to present the designs to clients.



7. Incorporate a design option into the main model.

After the client has selected the desired option for each option set, you can incorporate the selected designs into the main model. This process deletes the design option set, so the other options in the set are no longer available, and the selected option becomes part of the building model.

View Design Option

When you create a design option set, Revit displays the main model and the primary option in all project views by default. To see secondary options with the main model, you must edit the option, change a view's display option settings and dedicate views to the design option.

Determine the Active Option

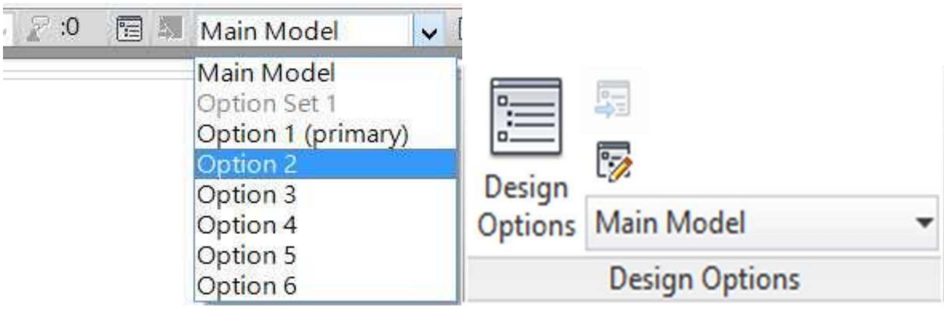
The active option is the design option that is currently being edited. If you are editing an option, the current view displays the main model and the active option. To determine whether you are currently editing a design option, use one of the following methods:

Status bar - Check the status bar. It indicates the active design option. If the status bar

displays Main Model, you are not currently editing a design option. If the status bar does not display the active design option, enable by:

View tab > Windows panel > User Interface drop-down > Status Bar - Design Options.

Ribbon - Click Manage tab Design Options panel. The drop-down list indicates the current design option. Changes that you make will affect the active option. If the drop-down list displays Main Model, you are not currently editing a design option.



Dedicate Views to a Design Option

To see secondary options with the main model, create duplicate views that are dedicated to those options. These views are called dedicated views. A dedicated view typically displays a specified design option for each set. You can dedicate all kinds of views (including schedules) to a specified design option. For example, you can create one schedule for the primary option and another schedule for a secondary option. Each schedule lists elements that are in the main model as well as elements that are in the design option.

To create a dedicate view to a design option, please follow the below steps:

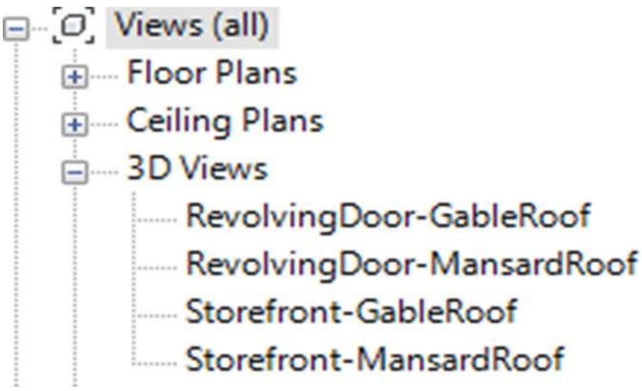
1. Open a view that you want to dedicate to a design option.

2. By default, the view displays the primary option with the main model.

3. In the Project Browser, right-click the view name, click Duplicate View > Duplicate

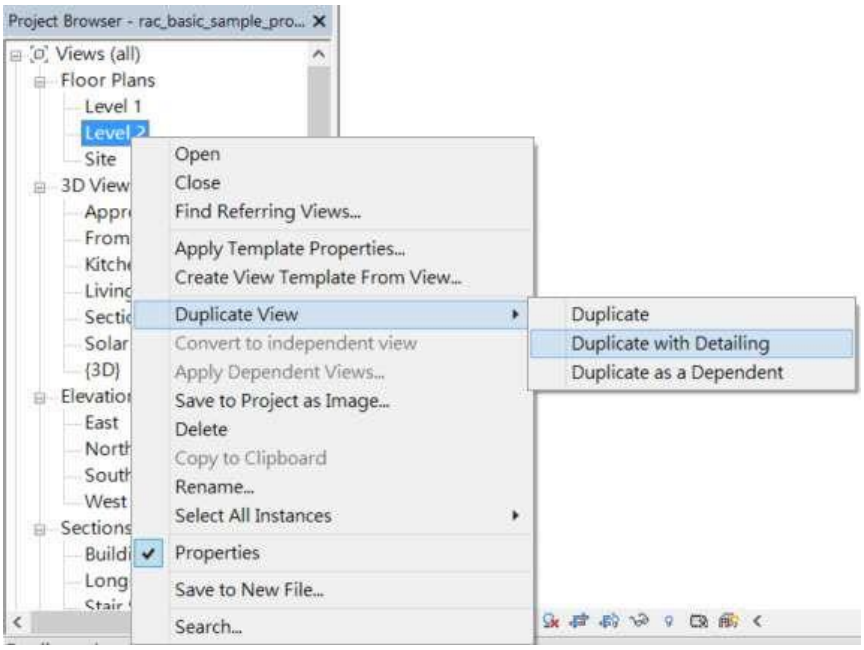
with Detailing.

4. Revit creates a duplicate view > rename the duplicate view.

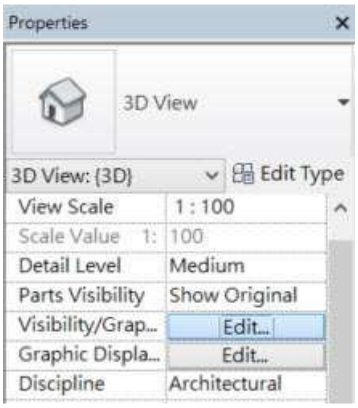


5. Specify design options for the view, as follows:

a) In the Project Browser, right-click the duplicate view name, and click Properties.



b) In the Properties palette, for Visibility/Graphics Overrides, click Edit.



c) The Visibility dialog displays the Design Options tab. Select the design option to display in this view. The view displays the main model and a selected design option for each set.

6. Repeat Steps 3–5 for each combination of design options that show in individual views.

Chapter 19 – Collaboration with Team

<u>Enable Worksharing</u>

Enabling worksharing involves creating a master project model, known as a central model, from an existing model.

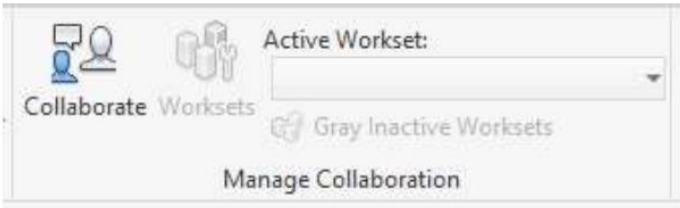
The central model stores the current ownership information for all worksets and elements in the project, and acts as the distribution point for all changes made to the model. All users should save their own local copy of the central model, edit locally in this workspace, and then synchronize with central to publish changes to the central model so that other users can see their work.

To enable worksharing and create a central model

1. Open the project file (RVT) to use as the central model.
2. Click Collaborate tab ➤ Manage Collaboration panel ➤ (Worksets).

   Note: If you have installed the Collaboration for Revit add-on, then:

   a. Click Collaborate tab ➤ Manage Collaboration panel ➤ Collaborate.



b. In the Collaborate dialog, select Collaborate within your network and click OK.

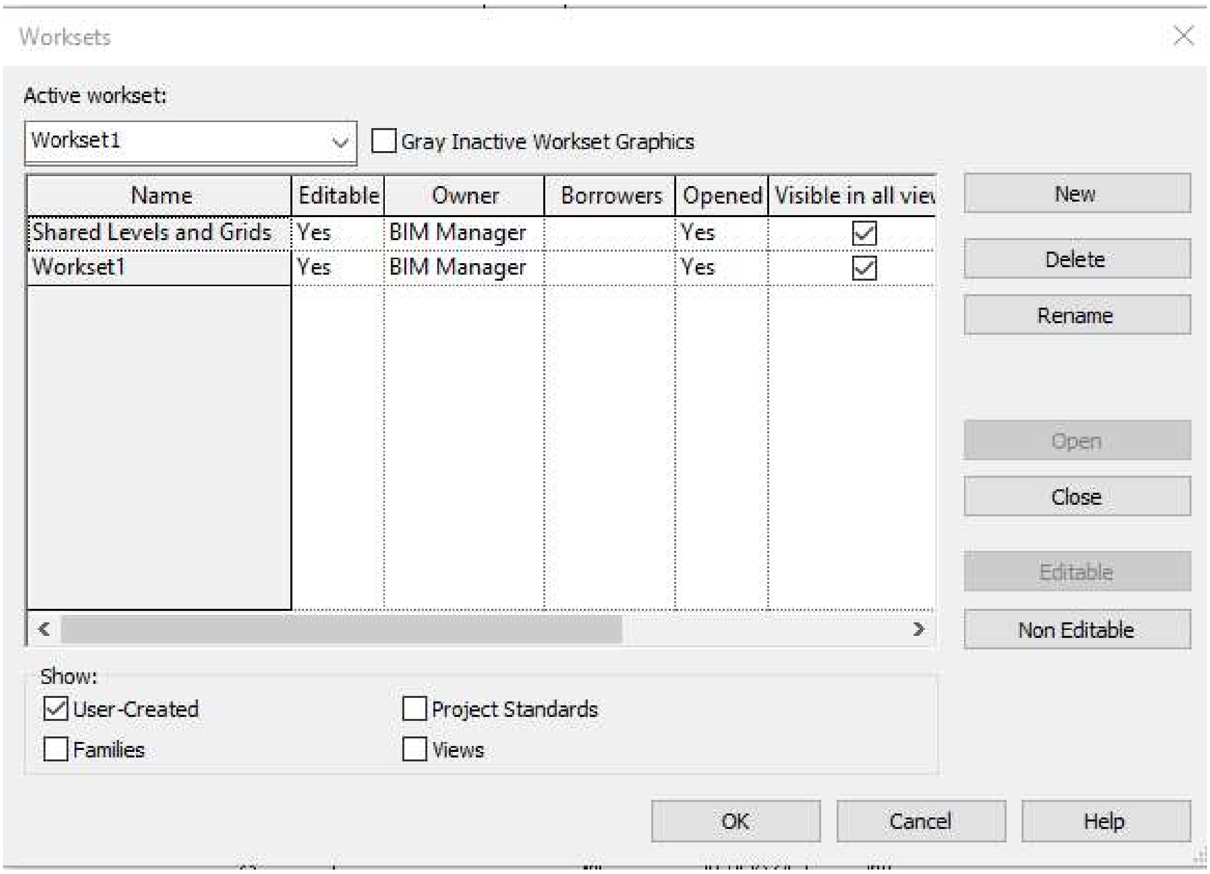
The Worksharing dialog displays, showing the default user-created worksets (Shared Levels and Grids, and Workset1).

3. If desired, rename the worksets.
4. In the Worksharing dialog, click OK. The Worksets dialog displays.
5. In the Worksets dialog, click OK.

   You do not need to create worksets at this point.

6. Click File tab ➤ Save As ➤ (Project).
7. In the Save As dialog, specify a file name and directory location for the central model.

When specifying a name for the central model, use a naming convention that identifies it as the central model (for example, OfficeBuilding_CentralModel.rvt).



Attention: When you save the central model, be sure that it is saved to a network drive to which all team members have access. If you are using server-based worksharing, Revit Server must be installed on any machine hosting a central model, and the Host role must be enabled in the RSN.ini file on that machine.

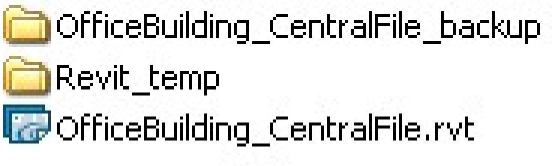
8. In the Save As dialog, click Options.
9. In the File Save Options dialog, select Make this a Central Model after save.

   Note: If this is the first time you have saved after enabling worksharing, this option is selected by default and cannot be changed.

10. Select a default workset for local copies.
11. Click OK.
12. In the Save As dialog, click Save.

The file is now the central model for the project. Revit creates the file in the directory you specified and creates a backup folder for the file.

For example, if your central model is called OfficeBuilding_CentralFile, you see the project file and the backup folder (OfficeBuilding_CentralFile_backup) in the directory.



The backup folder contains the backup information and editing permission information for the central model.

The Revit_temp folder contains files that provide progress information on operations (such as Synchronize with Central) to the Worksharing Monitor.

Create Worksets

After enabling a team project for worksharing, create worksets to organize elements into collections so that team members can display and work on selected worksets.

1. Open your local copy of the central model.
2. Click Collaborate tab ➤ Manage Collaboration panel ➤ (Worksets).
3. In the Worksets dialog, click New.
4. In the New Workset dialog, enter a name for the new workset.
5. To display the workset in all project views, select Visible in all views.

   Clear this option if you want the workset to display only in views where you specifically turn on its visibility.

   You can change the visibility of worksets later in the Visibility/Graphics dialog.

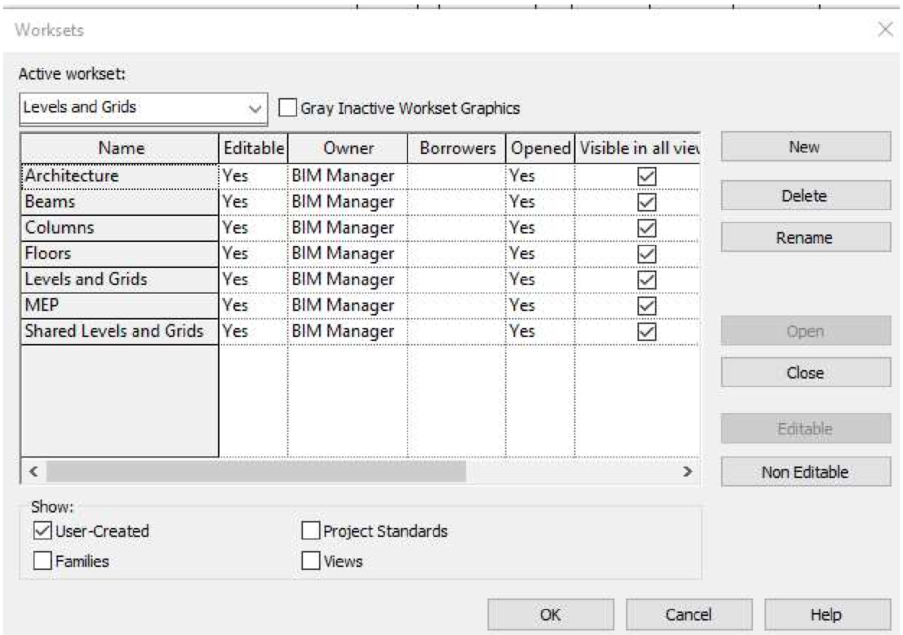
   Note: To improve performance, hide worksets that are not required for current work in the local model.

6. Click OK.

   The new workset displays in the list of worksets; it is editable, and your user name displays for Owner.

   If you are setting up a workshared model for your team and want to assign owners to each workset, each team member must open a local copy of the central model, select the workset in the Worksets dialog, and then select Yes in the Editable column.

7. When you finish creating worksets, click OK to close the Worksets dialog.
8. If you have added only one new workset, Revit prompts you to make the new workset active. Click Yes or No.

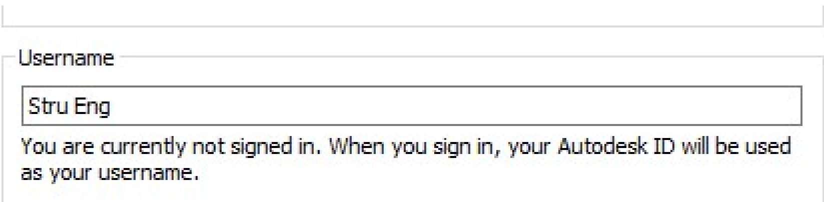


Create a Local Copy of a Central Model

It is good practice for users to create a local copy of the central model every day. Creating a new local copy ensures a local copy is always on your hard drive each time you begin modifying a project.

To open a model and create a local copy

Change the Username in Options such as Stru Eng

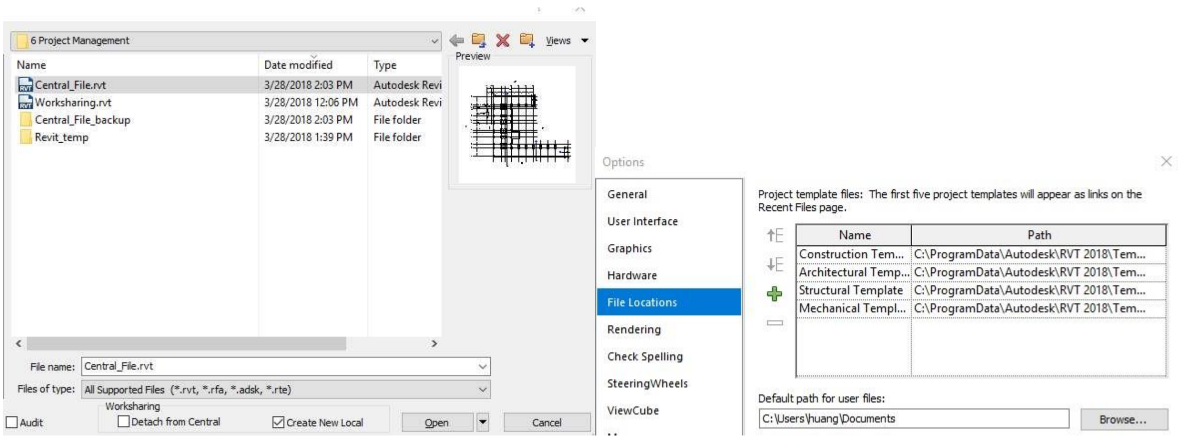


1. Click File tab > Open.

2. In the Open dialog, navigate to the folder where the central model resides and select it.

3. Under Worksharing, verify that Create New Local is selected.

Note: If you select Detach from Central, then Create New Local will be cleared. Clear both options to open the central model itself instead of a copy.



To create a local copy from an open central model

Use this procedure if you have already opened the central model, and you want to create a local copy of it.

1. Click File tab ➤ Save As ➤ (Project).
2. In the Save As dialog, navigate to the desired location on the Revit Server network or on your hard drive.
3. Enter a name for the file, and click Save.

Make Worksets Editable

When working on a workshared project, you can use various methods to make worksets editable.

To make worksets editable from the Worksets dialog

1. Open your local copy of the central model.

2. Click Collaborate tab > Manage Collaboration panel ➤ (Worksets).

3. In the Worksets dialog, to filter the display of worksets, make the appropriate selections under Show. For example, to see the Project Standards worksets, select Project Standards.

4. Click under Editable for the workset, and select Yes. Alternatively, you can select the

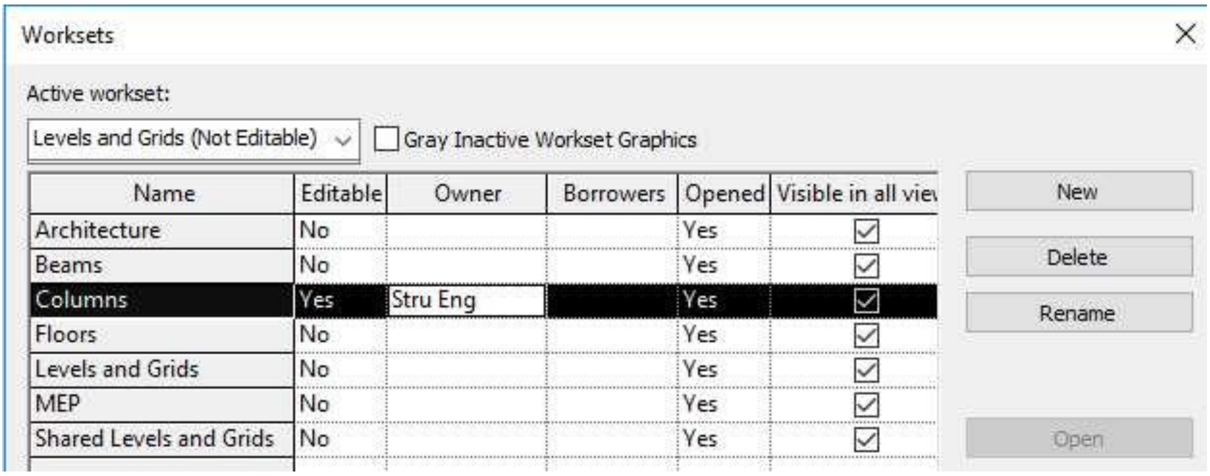
workset name, and click Editable.

Note: If a workset is owned by another user, you cannot change its editable status.

5. Repeat the previous step for as many worksets as you want to edit.

6. Click OK.

When you click OK, ownership information is communicated to the central model and to all local copies of the central model, so that all team members have the current ownership information.

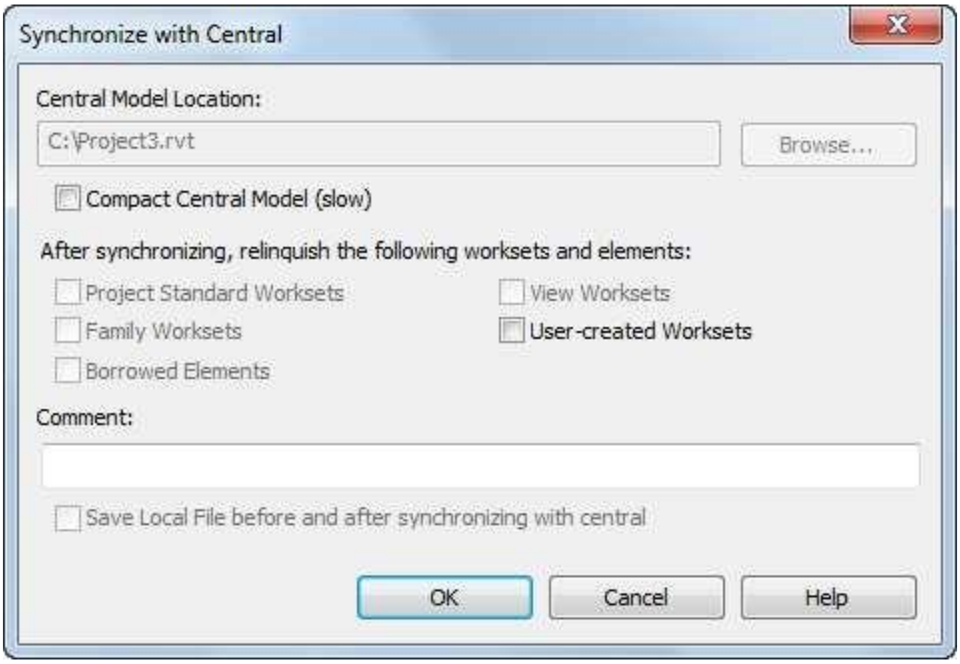


Synchronize with the Central Model

When working on a workshared project, use the Synchronize with Central tool to save your local changes to the central model.

To Synchronize with Central: Click Collaborate tab > Synchronize panel > Synchronize with Central drop-down ➤  (Synchronize Now).

If you want to modify the Synchronize with Central settings before you synchronize with central: Click Collaborate tab > Synchronize panel > Synchronize with Central drop-down >  (Synchronize and Modify Settings). The Synchronize with Central dialog displays.



1. In the Synchronize with Central dialog, verify the location of the central model.

If necessary, click Browse to specify a different path for the central model. Specify the new path in the Central File Location dialog, and click OK.

2. Select Compact Central Model to reduce file size when saving.

Note: Selecting this option increases the time needed to save.

3. Under After synchronizing, relinquish the following worksets and elements, select from the following options:

- To make your changed worksets and elements available to other users, select the appropriate checkboxes.
- To synchronize the changes to central but keep the worksets and elements editable, clear the appropriate checkboxes.

4. If desired, enter a comment that is saved to the central model.

5. Verify that Save local file before and after synchronizing with central is selected to ensure your local file remains synchronized with central.

6. Click OK.

Re-assign an Element to a Different Workset

When working on a team project that is enabled for worksharing, you can move an element from one workset to another.

1. In the drawing area, select an element.

2. On the Properties palette, locate the Workset parameter under Identity Data.

3. Click in the Value column for the parameter, and select a new workset.

In the Properties palette, the Workset parameter can be edited without first borrowing the element.

Identity Data	
Image	
Comments	
Mark	
Workset	Levels and Grids
Edited by	

Identity Data	
Image	
Comments	
Mark	
Workset	Columns
Edited by	Stru Eng

Add Elements to a Workset

When working on a team project that is enabled for worksharing, you can add elements to a workset.

1. Select the workset from the Active Workset drop-down on the status bar or the Collaborate tab > Manage Collaboration panel.

Note: You can select a non-editable workset as the active workset. If you place an element in a non-editable workset, the element becomes non-editable after you synchronize with central. If you add view-specific elements, such as detail lines or dimensions, they are added to the workset of the active project view.

2. If you want all elements that were not created in the active workset to display as gray, click Collaborate tab > Manage Collaboration panel (Gray Inactive Worksets).

3. Add the necessary elements to the drawing area.

Disable Worksharing

If needed, you can disable worksharing in a model that has been workshared.

1. Click File tab> Open.

2. Select a workshared model.

3. In the Open dialog, select Detach from Central.

4. Click Open.

- The default file name for the model is the original filename with "_detached" appended. You can change the default name when you save the model.

5. In the task dialog, specify whether to preserve or discard worksets.

6. Save the model.

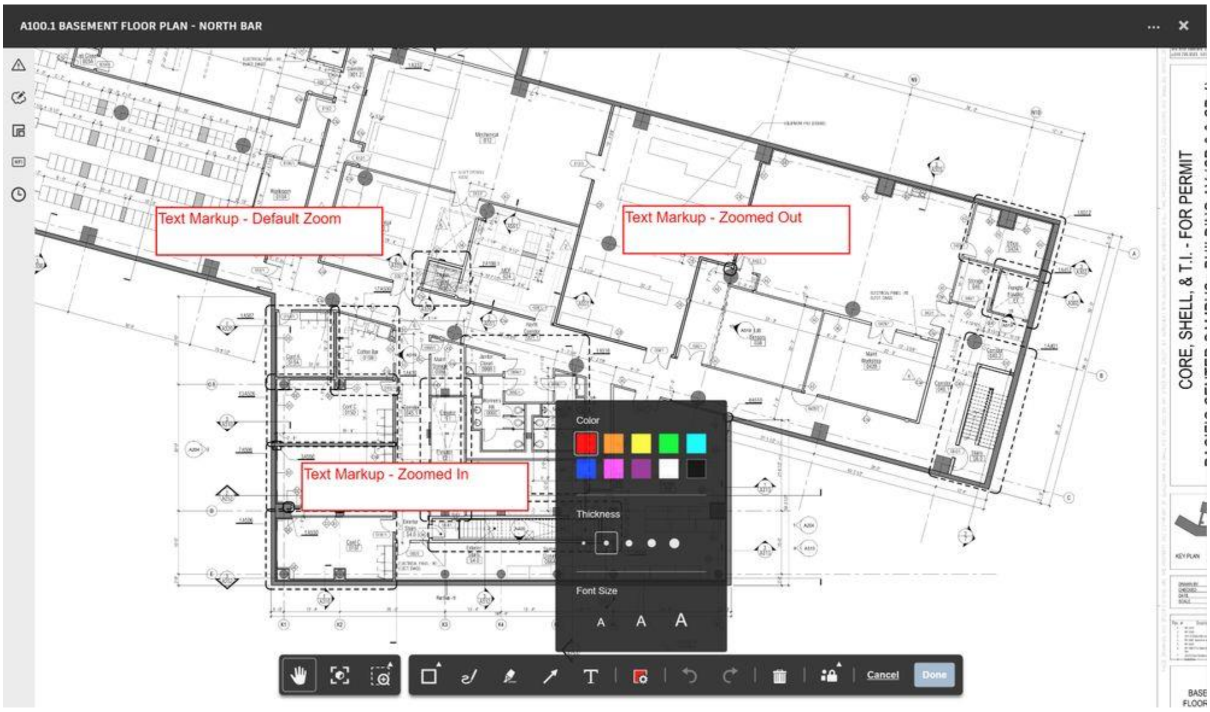
Cloud Collaboration Platforms

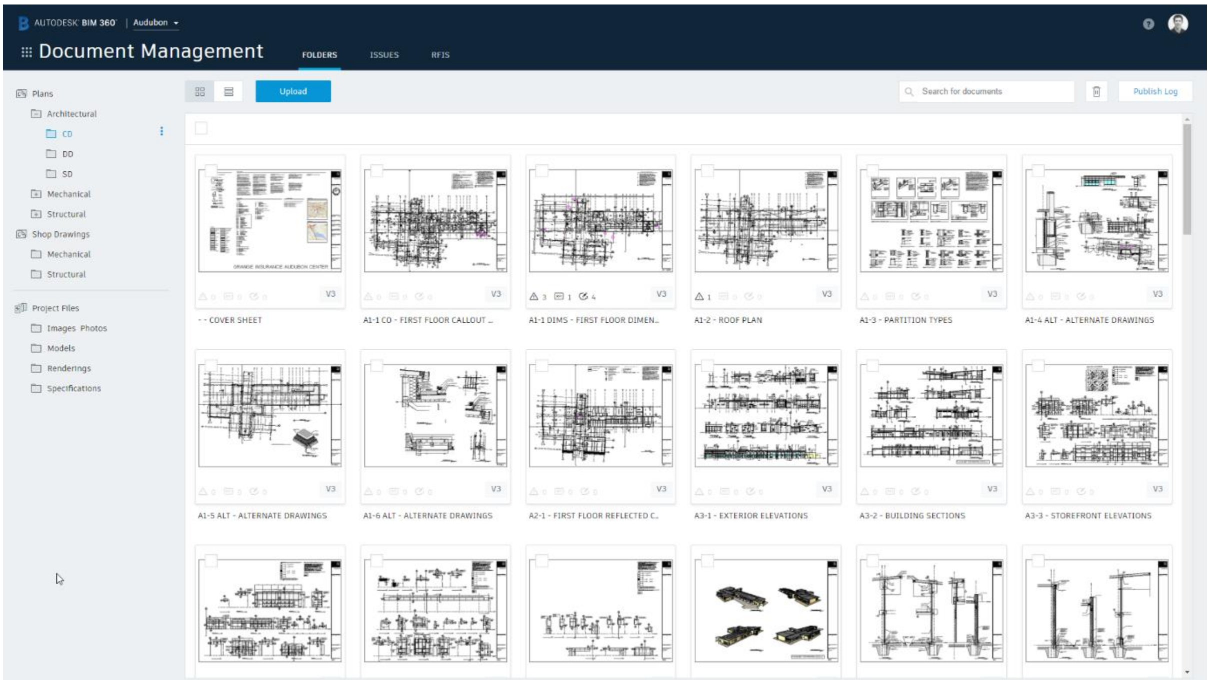
Keep the team on-track with the latest drawings, documents, and models.

1. Controlled sharing of information across the entire team
2. Perform constructability reviews with 2D & 3D markups
3. Reduce rework risks through versioning and permissions
4. Manage contract documents and track project activity

Workflow of Cloud platform for collaborations

- Get information published faster and organized logically with bulk upload, set creation, and automatic separation of sheets from design files.
- Organize construction drawings and documents into sets and quickly review changes between versions - in 2D or 3D.
- Maintain document control with five permission levels keeps the right information in the right hands throughout the project.
- Structural Engineers can work with Draftsman / Modeller to update model and drawing remotely on cloud and off-site anytime.





Reference - "BIM 360 Docs - Autodesk." https://docs.b360.autodesk.com/.