Revit Structure 2010

Imperial Tutorials

Part Number Date

Contents

Introduction

This introduction helps you get started with the Revit Structure 2010 tutorials and presents the fundamental concepts of the product, including:

- How Revit Structure works.
- An overview of the concepts and terms used in the product.
- How to perform common tasks in the product.

Using the Tutorials

1

This lesson provides information on how to get started with the Revit® Structure 2010 tutorials, including an overview of the building information model that you design in the tutorials, and where to find the training files required to complete the exercises.

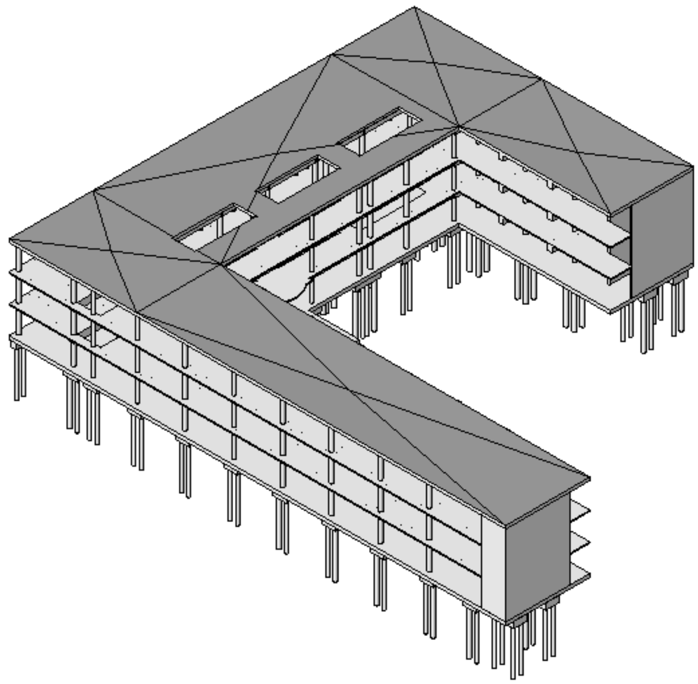
The Contents tab of the Revit Structure Tutorials window displays the available tutorial titles. Expand a title for a list of lessons in the tutorial. Expand a lesson title for a list of exercises in the lesson.

NOTE You may find it helpful to print a tutorial to make it easier to reference the instructions as you work in Revit Structure. The tutorials are also available in PDF format by clicking Help menu ➤ Documents on the Web in Revit Structure.

What is in the Tutorials

In these tutorials, you learn how to design a building information model (BIM) in Revit Structure. The technical training facility you design was a product of a design competition in Munich, Germany. This building is also used as the model in the Revit® Architecture and Revit® MEP 2010 tutorials.

Revit Structure Model



How the tutorials are organized

The tutorials are designed to follow the typical structural engineering workflow. You complete the following phases of design:

- Import the architectural floor plan in either a 2D (DWG) or a 3D (RVT) format.
- Add basic structural elements, such as beams, trusses, and slabs.
- Add more detailed modelling elements, such as area and path reinforcement.
- Analyze the structure using the analytical model.
- Create schedules, views, and sheets to document the project.
- Create detail views, annotations, and tags.

The tutorial exercises are designed to be basic and brief. You do not design the entire structure, but only enough of the building to learn how to use the tools and options in the product. For example, when you add the shape-modified slab, you only draw the slab on one wing of the structure.

When you open a training file, you may notice that structural elements are included that were not specifically added in an exercise. For example, to provide a richer and more finished design, elements such as additional beam systems, a metal deck on the entry roof, and additional foundation elements are incorporated into subsequent training files. These elements enhance the exercises you complete.

Accessing Training Files

Training files are Revit Structure projects, templates, and families that were created specifically for use with the tutorials. In this exercise, you learn where the training files are located, as well as how to open and save them.

Locate the training files

The Tutorials option on the Revit Structure 2010 Help menu provides a link to the installation website for the tutorial content and training files. When you install the training files as instructed, they are copied to the default location C:\Documents and Settings\All Users\Application Data\Autodesk\RST 2010\Training. Training files are grouped into 2 folders within the Training folder:

- Imperial: files for users working with imperial units. Imperial file names have an _i suffix.
- Metric: files for users working with metric units. Metric file names have an _m suffix.

NOTE Depending on your installation, your Training folder may be in a different location. Contact your CAD manager for more information.

IMPORTANT Content used in the tutorials, such as templates and families, is located and accessed in the training files location. Although this content may be installed in other locations on your system, all content used in the tutorials is installed in the training files location to ensure that all audiences access the correct files.

Use the training files

A training file is a Revit Structure project that defines a building information model and views of the model that are used to complete the steps in a tutorial. The tutorials include a Training File section that references the training file to be used with the tutorial.

The training files include a starting point for each tutorial exercise. So, you can complete any exercise without first completing the preceding exercises. On the Contents tab, the tutorials are grouped and presented in a recommended order for optimal learning; however, the exercises and lessons can be completed in any order.

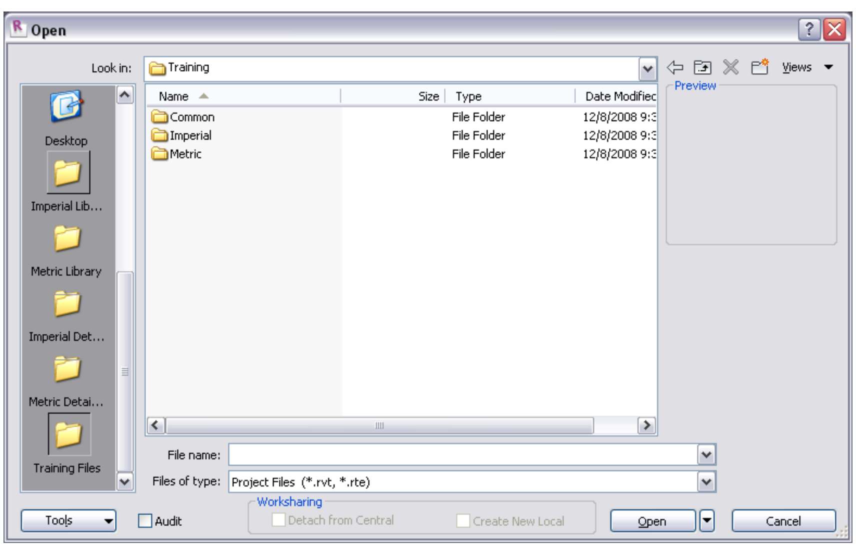
In some cases, a training file may include additional building

Open a training file

1 Click  ➤ Open ➤ Project.

2 In the left pane of the Open dialog, scroll down, and click the Training Files icon.

3 In the right pane, double-click Imperial or Metric, depending on the instructions in the tutorial.



4 Click the training file name, and click Open.

Save a training file

5 To save a training file with a new name, click ➤ Save As.

NOTE You are not required to save your work in a training file. A training file is provided as a starting point for each exercise.

---

6 Complete the information in the Save As dialog:

- For Save in, select the folder in which to save the new file.
  You can save the file in the appropriate Training Files folder or in another location.
- For File name, enter the new file name.
  A good practice is to save the training file with a unique name after you have made changes. For example, if you open settings.rvt and make changes, you should save this file with a new name such as settings_modified.rvt.
- For Files of type, verify that Project Files (*.rvt) is selected, and click Save.

Close a training file

7 Click ➤ Close.

8 If you have made changes, you are prompted to save the changes. You may close the file with or without saving changes.

Understanding the Basics

2

In this lesson, you learn what Revit Structure is and how its parametric change engine benefits you and your work. You begin with the fundamental concepts on which Revit Structure is built. You learn the terminology, the hierarchy of elements, and how to perform some common tasks in the product.

Understanding the Concepts

What is Revit Structure 2010?

The Revit Structure platform for building information modelling is a design and documentation system that supports the design, drawings, and schedules required for a building project. Building information modelling (BIM) delivers information about project design, scope, quantities, and phases when you need it.

In the Revit Structure model, every drawing sheet, 2D and 3D view, and schedule is a presentation of information from the same underlying building model database. As you work in drawing and schedule views, Revit Structure collects information about the building project and coordinates this information across all other representations of the project. The Revit Structure parametric change engine automatically coordinates changes made anywhere—in model views, drawing sheets, schedules, sections, and plans.

What is meant by parametric?

The term parametric refers to the relationships among all elements of the model that enable the coordination and change management that Revit Structure provides. These relationships are created either automatically by the software or by you as you work. In mathematics and mechanical CAD, the numbers or characteristics that define these kinds of relationships are called parameters; hence, the operation of the software is parametric. This capability delivers the fundamental coordination and productivity benefits of Revit Structure: Change anything at any time anywhere in the project, and Revit Structure coordinates that change through the entire project.

The following are examples of these element relationships:

- The outside of a door frame is a fixed dimension on the hinge side from a perpendicular partition. If you move the partition, the door retains this relationship to the partition.
- Rebar is spaced equally across a given elevation. If the length of the elevation is changed, the relationship of equal spacing is maintained. In this case, the parameter is not a number but a proportional characteristic.
- The edge of a floor or roof is related to the exterior wall such that when the exterior wall is moved, the floor or roof remains connected. In this case, the parameter is one of association or connection.

How does Revit Structure 2010 keep things updated?

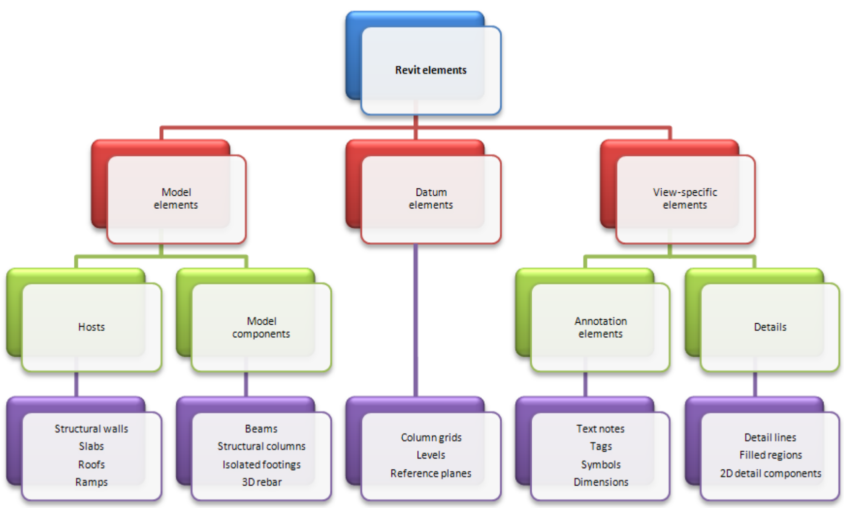
A fundamental characteristic of a building information modelling application is the ability to coordinate changes and maintain consistency at all times. You do not have to intervene to update drawings or links. When you change something, Revit Structure immediately determines what is affected by the change and reflects that change to any affected elements.

Revit Structure uses 2 key concepts that make it especially powerful and easy to use. The first is the capturing of relationships while the designer works. The second is its approach to propagating building changes. The result of these concepts is software that works like you do, without requiring entry of data that is unimportant to your design.

Element behavior in a parametric modeler

In projects, Revit Structure uses 3 types of elements:

- **Model elements** represent the actual 3D geometry of the building. They display in relevant views of the model. For example, structural walls, slabs, ramps, and roofs are model elements.
- **Datum elements** help to define project context. For example, column grids, levels, and reference planes are datum elements.
- **View-specific elements** display only in the views in which they are placed. They help to describe or document the model. For example, dimensions, tags, and 2D detail components are view-specific elements.



There are 2 types of model elements:

- **Hosts** (or host elements) are generally built in place at the construction site. For example, structural walls and roofs are hosts.
- **Model components** are all the other types of elements in the structural model. For example, beams, structural columns, and 3D rebar are model components.

There are 2 types of view-specific elements:

- **Annotation elements** are 2D components that document the model and maintain scale on paper. For example, dimensions, tags, and symbols are annotation elements.
- **Details** are 2D items that provide details about the structural model in a particular view. Examples include detail lines, filled regions, and 2D detail components.

This implementation provides flexibility for designers. Revit Structure elements are designed to be created and modified by you directly; programming is not required. If you can draw, you can define new parametric elements in Revit Structure.

In Revit Structure, the elements determine their behavior largely from their context in the structure. The context is determined by how you draw the component and the constraint relationships that are established with other components. Often, you do nothing to establish these relationships; they are implied by what you do and how you draw. In other cases, you can explicitly control them, by locking a dimension or aligning 2 walls, for example.

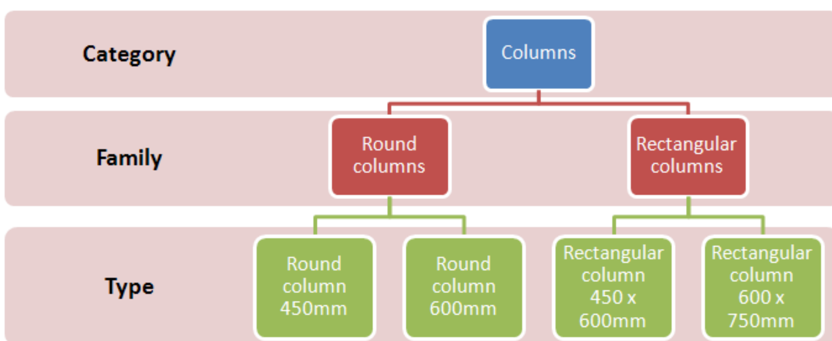
Understanding Revit Structure 2010 terms

Most of the terms used to identify objects in Revit Structure are common, industry-standard terms familiar to most engineers. However, some terms are unique to Revit Structure. Understanding the following terms is crucial to understanding the software.

Project: In Revit Structure, the project is the single database of information for your design—the building information model. The project file contains all information for the building design, from geometry to construction data. This information includes components used to design the model, views of the project, and drawings of the design. By using a single project file, Revit Structure makes it easy for you to alter the design and have changes reflected in all associated areas (plan views, elevation views, section views, schedules, and so forth). Having only one file to track also makes it easier to manage the project.

Level: Levels are infinite horizontal planes that act as a reference for level-hosted elements, such as roofs, slabs, and beams. Most often, you use levels to define a vertical height or story within a structure. You create a level for each known story or other needed reference of the structure; for example, first floor, top of wall, or bottom of foundation. To place levels, you must be in a section or elevation view.

Element: When creating a project, you add Revit Structure parametric building elements to the design. Revit Structure classifies elements by categories, families, and types.



Category: A category is a group of elements that you use to model or document a building design. For example, categories of model elements include beams and structural columns. Categories of annotation elements include tags and symbols.

Family: Families are classes of elements in a category. A family groups elements with a common set of parameters (properties), identical use, and similar graphical representation. Different elements in a family may have different values for some or all properties, but the set of properties—their names and meaning—is the same. For example, a truss could be considered one family, although the web supports that compose the family come in different sizes and materials.

There are 3 kinds of families:

- Loadable families can be loaded into a project and created from family templates. You can determine the set of properties and the graphical representation of the family.

- System families include slabs, dimensions, roofs, and levels. They are not available for loading or creating as separate files.
  - Revit Structure predefines the set of properties and the graphical representation of system families.
  - You can use the predefined types to generate new types that belong to this family within the project. For example, the behavior of a structural wall is predefined in the system. However, you can create different types of walls with different compositions.
  - System families can be transferred between projects.

- In-place families are custom families that you create in the context of a project. Create an in-place family when your project needs unique geometry that you do not expect to reuse or geometry that must maintain one of more relationships to other project geometry.
  - Because in-place families are intended for limited use in a project, each in-place family contains only a single type. You can create multiple in-place families in your projects, and you can place copies of the same in-place family element in your projects. Unlike system and standard component families, you cannot duplicate in-place family types to create multiple types.

Type: Each family can have several types. A type can be a specific size of a family, such as a 30” X 42” title block. A type can also be a style, such as default aligned or default angular style for dimensions.

Instance: Instances are the actual items (individual elements) that are placed in the project and have specific locations in the structure (model instances) or on a drawing sheet (annotation instances).

Using the Revit Structure User Interface

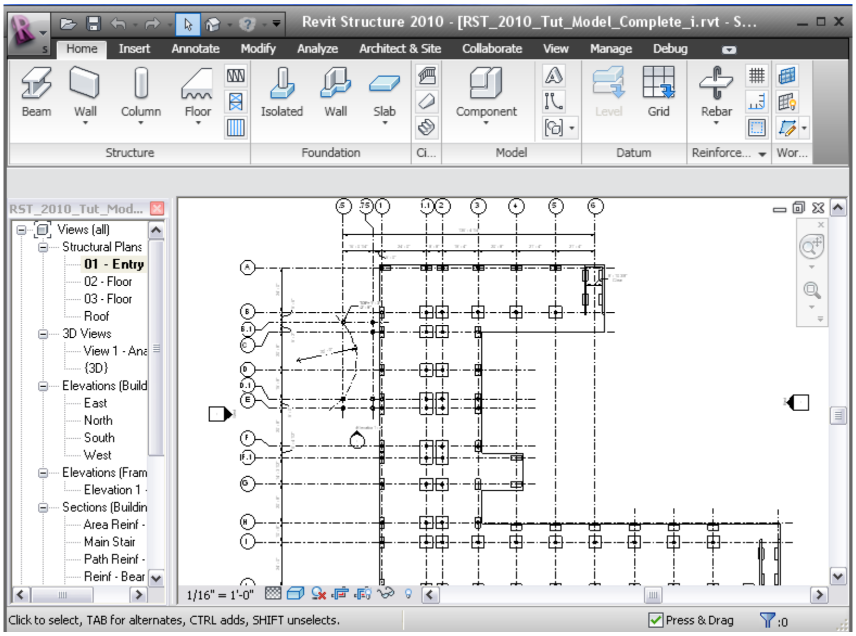
Revit Structure is a powerful CAD product for the Microsoft® Windows operating system. Its interface resembles those of other products for Windows featuring a ribbon that contains the tools used to complete tasks.

In the Revit Structure interface, many of the components (such as walls, beams, and columns) are available at the click of a button. You can drop these components into the drawing and immediately determine whether they meet your design requirements.

Parts of the Revit Interface

The Revit Structure interface is designed to simplify your workflow. With a few clicks, you can change the interface to better support the way that you work. For example, you can set the ribbon to one of the three display settings for optimum use of the interface. You can also display several project views at one time, or layer the views to see only the one on top.

Read the following topics to familiarize yourself with the basic parts of the Revit interface. Then experiment with them, hiding, showing, and rearranging interface components to support the way that you work.

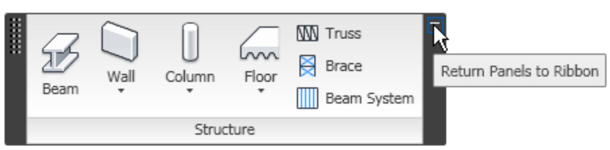


Ribbon Overview

The ribbon displays automatically when you create or open a file, and provides all the tools necessary to create your file. Customize the ribbon by changing the panel order, or moving a panel off the ribbon to your desktop. The ribbon can be minimized for maximum use of the drawing area.

To move panels:

- Click a panel label and drag the panel to a desired place on the ribbon.
- Click a panel label and drag the panel off the ribbon to the desktop. To return the panel to the ribbon, click the Return Panels to Ribbon button, or drag the panel back to its original ribbon tab.



To minimize the ribbon

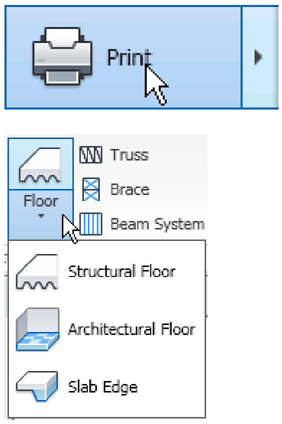
1. Click (Show Full Ribbon) to the right of the ribbon tabs.
2. The minimize behavior cycles through the following minimize options:

- **Show Full Ribbon:** Shows entire ribbon.
- **Minimize to Panel Tiles:** Shows tab and panel labels.
- **Minimize to Tabs:** Shows tab labels.

Ribbon Tabs and Panels

TIP When you see a button that shows a line dividing it into two sides, you can click the top (or left) side to access the tool you probably use most often. Click the other side to expose a list of other related tools.

Examples of buttons that can be clicked on two sides



The following table describes the ribbon tabs and the types of commands they contain.

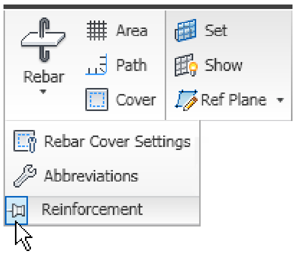
There are a number of button types on the ribbon

Ribbon Tab	Includes commands for...
Home	many of the tools you need to create the building model.
Create (family files only)	many of the tools you need to create and modify a family of elements.
Insert	tools to add and manage secondary items such as raster images, and CAD files.
Annotate	tools used for adding 2D information to a design.
Modify	tools used for editing existing elements, data and systems. When working on the Modify tab, select the tool first, then select what you want to modify.
Collaborate	tools for collaboration with internal and external project team members.
Analyze	tools used for running analysis on the current model.
View	tools used for managing and modifying the current view, and for switching views.
Manage	project and system parameters, and settings.
Architect & Site	architect and site-specific tools.

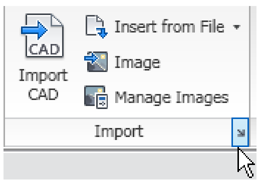
Ribbon Tab	Includes commands for...
Add-Ins	third-party tools used with Revit Structure 2010. The Add-Ins tab is enabled only when a third-party tool is installed.

Expanded Panels

A drop-down arrow at the bottom of a panel indicates that you can expand the panel to display additional tools and controls. By default, an expanded panel closes automatically when you click another panel. To keep a panel expanded, click the push pin icon in the bottom-left corner of the expanded panel.



A dialog-launcher arrow on the bottom of a panel opens a dialog.



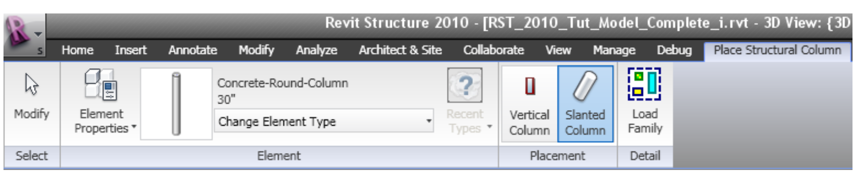
Contextual Ribbon Tabs

When you execute certain commands or select an element, a special contextual ribbon tab displays that contains a set of tools that relate only to the context of the command.

For example, when drawing walls, the Place a Wall contextual tab displays that has three panels:

- Select: contains the Modify command.
- Element: contains Element Properties and the Type Selector.
- Draw: contains the draw editors necessary for the wall sketch.

This contextual ribbon tab closes once you end the command.



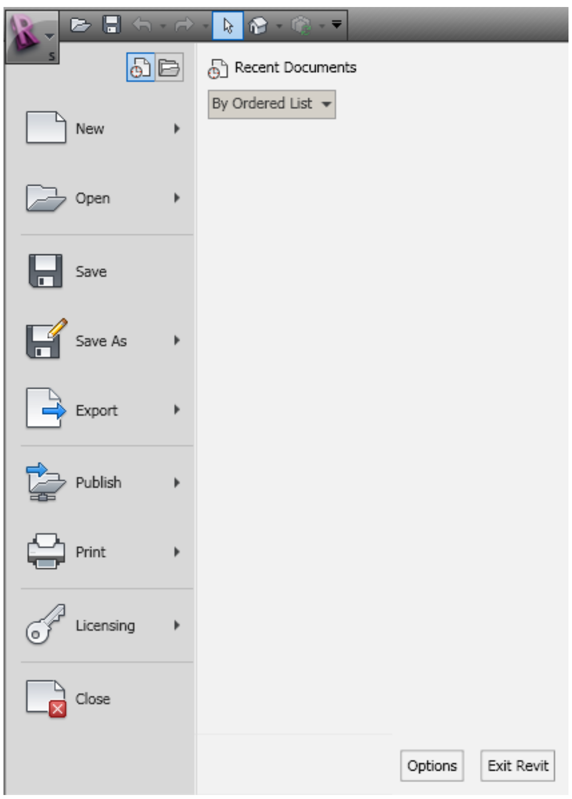
Application Frame Overview

The application frame contains tools and provides feedback to help you manage your Revit Structure projects.

The application frame consists of five main areas described in the following table:

Application Window Tool	Description
application button	opens the application menu (single -click). closes the application menu (double-click).
application menu	provides access to common tools.
Quick Access toolbar	displays frequently used tools.
InfoCenter	provides requested information.
Status Bar	displays information related to the current state of a Revit operation.

The Application Menu



The application menu provides access to many common file actions and also allows you to manage your files using more advanced commands, such as Export and Publish.

NOTE Revit Structure options are set from Options on the application menu.

Access Common Tools

Access common tools to start or publish a file in the application menu.

Click to quickly perform the following actions:

- Create a file
- Open an existing file
- Save a file
- Export a file to another file format
- Publish a file and place it in a central or shared location
- Print a file
- Access license information
- Close the application

You can quickly access the following dialogs from the application menu:

Click the left side of...	to open the...
(New)	New Project dialog
(Open)	Open dialog
(Publish)	DWF Publish Settings dialog
(Print)	Print dialog
(Licensing)	Product and License Information dialog

Using the Quick Access Toolbar

The Quick Access toolbar contains the following items by default:

Quick Access Toolbar Item	Description
(Open)	opens a project, family, annotation, or template file.
(Save)	saves a current project, family, annotation, or template file.
(Undo)	cancels the last action by default. Displays list of all actions taken during the session.
(Redo)	reinstates the last cancelled action also displays a list of all reinstated actions performed during the session.
(Modify Settings before Synchronizing)	synchronizes a local file with that on the central server.

Quick Access Toolbar Item	Description
(3D View)	provides views including Default 3D, Camera, and Walkthrough.
(Modify)	enters selection mode and ends the current operation.
(Customize Quick Access Toolbar)	customizes the items displayed on the Quick Access toolbar. To enable or disable a tool item, click next to it on the Customize Quick Access Toolbar drop-down. **NOTE** New displays on the Customize Quick Access Toolbar drop-down, but is not enabled by default.

To undo or redo a series of operations, click the drop-down to the right of the Undo and Redo buttons. This displays the command history in a list. Starting with the most recent command, you can select any number of previous commands to include in the Undo or Redo operation.

The Quick Access toolbar can display below the ribbon. Click Show Below the Ribbon on the Customize Quick Access Toolbar to change the display setting.

While in an edit mode (such as Place a Wall), or the Family Editor, items that are added to the Quick Access toolbar from the Create, Modify, Group, Clipboard, or View Graphics panel persist on the toolbar for that mode. However, when you switch to another editing mode, these items do not display and need to be re-added to the Quick Access toolbar.

NOTE There are some tools on contextual tabs that cannot be added to the Quick Access toolbar.

Status Bar

The status bar is located along the bottom of the Revit Structure application frame. When you are using a command, the left side of the status bar provides tips or hints on what to do. When you are highlighting an element or component, the status bar displays the name of the family and type.

Several other controls appear on the right side of the status bar

- Press & Drag: Allows you to click and drag an element without it selecting first.
- Editable Only: Filters selections to select only editable, workshared components.
- Active Only: Filters selections to select only active design option components.

- Exclude Options: Filters selections to exclude components that are part of a design option.
- Filter button: Displays how many elements are selected and refines the element categories selected in a view.

To hide the status bar, click View tab ➤ Windows panel ➤ User Interface drop-down. Clear the Status Bar check mark. To show the status bar again, repeat the command.

Getting Hints About What to Do Next

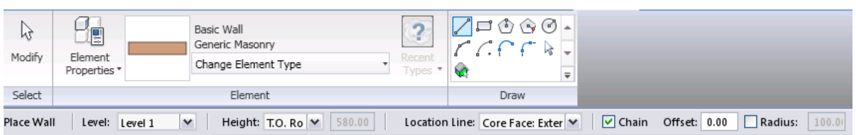
If you start a command (such as Rotate) and are not sure what to do next, check the status bar. It often displays tips or hints about what to do next for the current command. In addition, a tool tip appears next to the cursor, displaying the same information.

To cancel or exit the current command, do either of the following:

- Press *Esc* twice.
- On the Quick Access toolbar, click  (Modify).

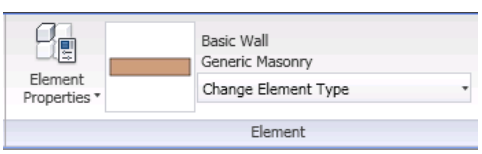
Options Bar

The Options Bar is located below the ribbon. Its contents change depending on the current command or selected element.

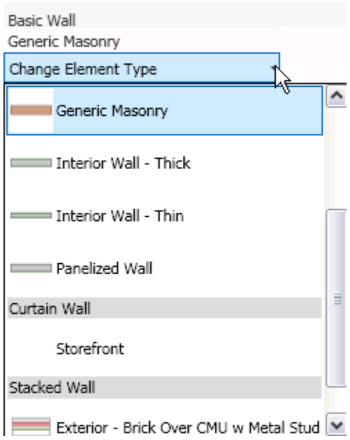


Type Selector

The Type Selector is located on the Element panel for the currently invoked tool, for example, Place a Wall. Its contents change depending on the current function or selected elements. When you place an element in a drawing, use the Type Selector to specify the type of element to add.



To change existing elements to a different type, select one or more elements of the same category. Then use the Type Selector to select the desired type.



View Control Bar

The View Control Bar is located at the bottom of the Revit window above the status bar. It provides quick access to functions that affect the drawing area, including the following:

- Scale
- Detail Level
- Model Graphics Style
- Show Rendering Dialog
- Shadows On/Off
- Show/Hide Rendering Dialog (Available only when the drawing area displays a 3D view.
- Crop Region On/Off
- Show/Hide Crop Region
- Temporary Hide/Isolate
- Reveal Hidden Elements

Modifying the View

In this exercise, you learn how to modify the views within your Revit Structure project. After you are familiar with these tasks, it will be easier to work in Revit Structure and focus on the lessons of each tutorial.

Use zoom commands to adjust the view

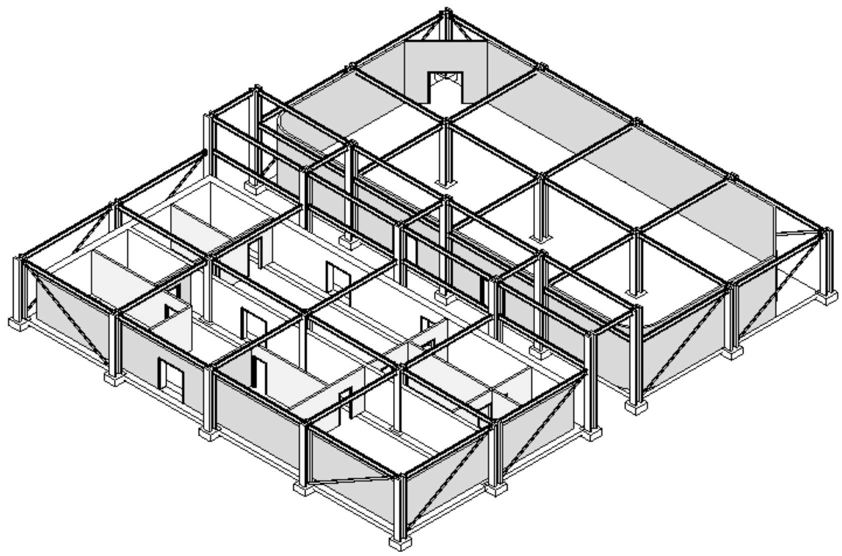
In the tutorials, you are instructed to use a zoom command to adjust the viewable area in the window. For example, you may be asked to zoom to a specific region of a view or to zoom to fit the entire structure or floor plan in the view. Understanding how to adjust the view will make it easier to work with the structural model in the window.

There are several ways to access zoom options. In the following steps, you open a training file and practice adjusting the view with the different zoom commands.

1 Click  ➤ Open ➤ Project.

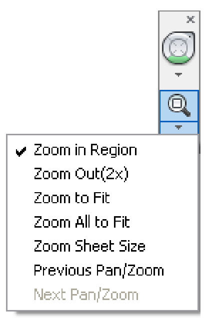
2 In the left pane of the Open dialog, click Training Files, and open Common\c_RST_Modify_Views.rvt.

The 3D isometric view displays:



3 Click Navigation bar ➤ Zoom In Region drop-down to display the zoom menu. (The Navigation bar is located in the top-right corner of the view.)

The zoom menu lists the zoom options.



NOTE Clicking the Zoom icon itself activates the currently-selected zoom command.

4 Click Zoom Out (2x).

In the drawing area, the view zooms out from the structural model.

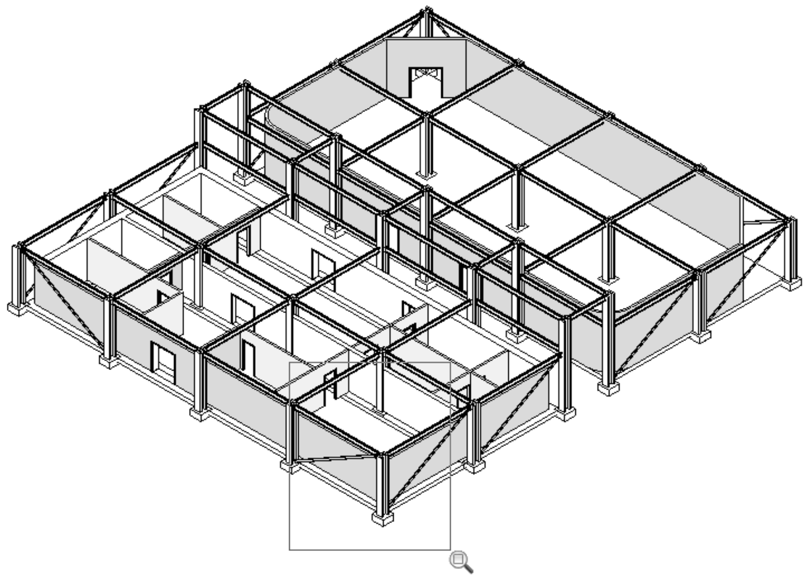
5 Click Zoom To Fit.

The view of the structural model is sized to fit the available window.

6 Click in the drawing area, and type the shortcut **ZR** to zoom in on a region.

The cursor becomes a magnifying glass.

7 Click the upper left corner and lower right corner of the region to magnify; this is referred to as a crossing selection.



When you release the mouse button, the view zooms in on the selected area.

8 If you use a mouse that has a wheel as the middle button, you can roll the wheel to zoom the view. Use the wheel mouse to zoom out to see the entire structure again.

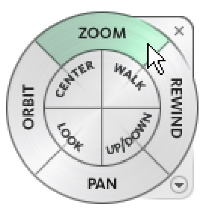
If you do not have a wheel mouse, use a zoom menu command or the toolbar option to zoom out.

NOTE As you zoom in and out, Revit Structure uses the largest snap increment that represents less than 2mm in the drawing area. To modify or add snap increments, click Manage tab ➤ Project Settings panel ➤ Settings drop-down ➤ Snaps.

Zoom is also available using SteeringWheels. SteeringWheels provide 2D and 3D navigation tools.

9 To display SteeringWheels, on the Navigation bar, click  .

The Full Navigation wheel displays in the drawing area.

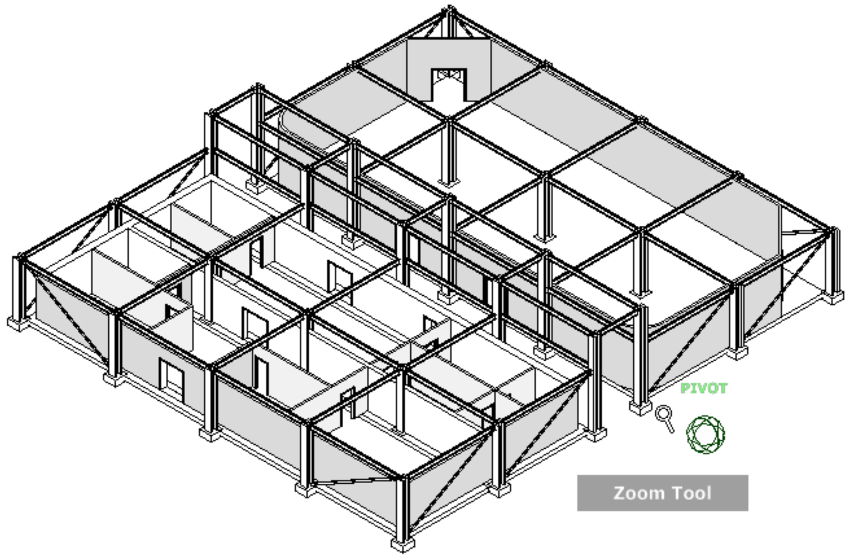


As you move the mouse, the wheel follows the cursor around the drawing area.

10 Move the cursor over the Zoom wedge of the wheel so that it highlights.

11 Click and hold the mouse button.

The cursor displays a pivot point for the Zoom tool.



12 Drag the cursor down or left to zoom out.

13 Drag the cursor up or right to zoom in.

You can change the pivot point by releasing the mouse button, moving the wheel to the desired location, and then using the Zoom tool again.

For more information about SteeringWheels, click the pull-down menu on the Full Navigation wheel, and click Help. To define settings for SteeringWheels, click Settings menu ➤ Options, and click the SteeringWheels tab.

14 To exit the wheel, press *Esc*.

15 Close the file without saving your changes.

Performing Common Tasks

In this exercise, you learn how to perform some of the common Revit Structure tasks that are included in the tutorials. After you are familiar with these tasks, it will be easier to work in Revit Structure and focus on the lessons of each tutorial.

Resize elements using the drag controls

1 Click  ➤ Open ➤ Project.

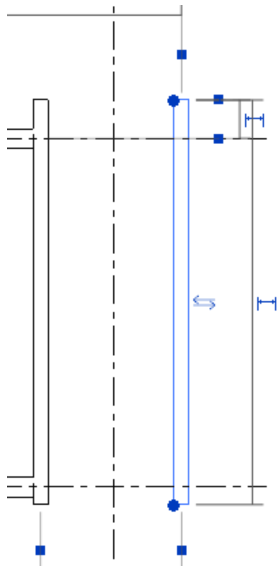
2 In the left pane of the Open dialog, click Training Files, and open Common\c_RST_Common_Tasks.rvt.

3 In the Project Browser, expand Views (all), expand Structural Plans, and double-click 02 - Floor.

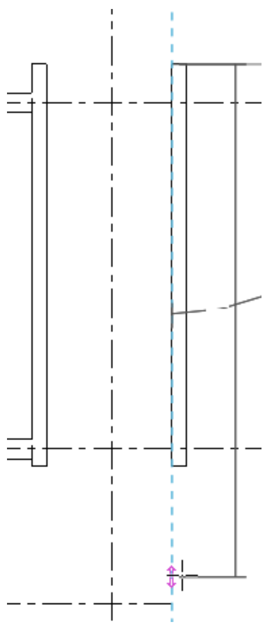
When drawing or modifying a structural model, it is important to understand how to adjust the size of components in the drawing area. Small blue dots, called drag controls, display at the ends of selected lines and structural walls in a plan view. Similar controls, referred to as shape handles, display along the ends, bottoms, and tops of selected walls in elevation views and 3D views.

4 Type **ZR**, zoom in on the upper-right corner of the structural floor plan, and select the wall, as shown.

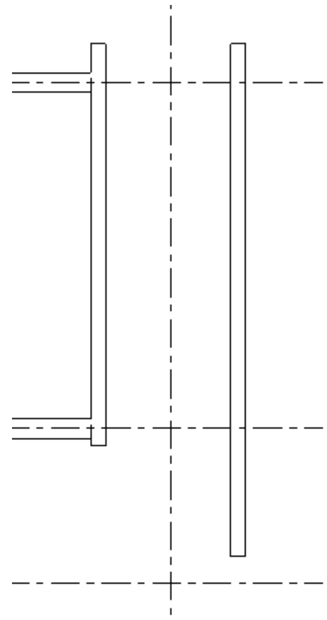
Notice the small blue dots that display at both ends of the wall. These are the drag controls.



5 Click and drag the bottom control, moving the cursor down vertically, to lengthen the wall.



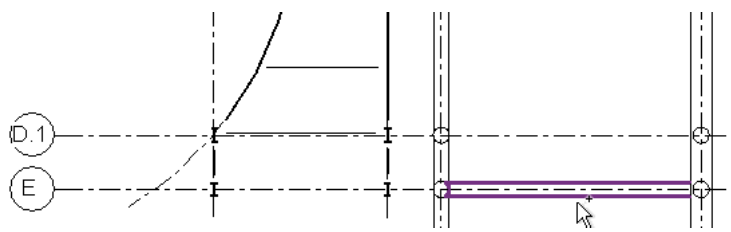
6 Click in the drawing area to deselect the wall.



Move an element

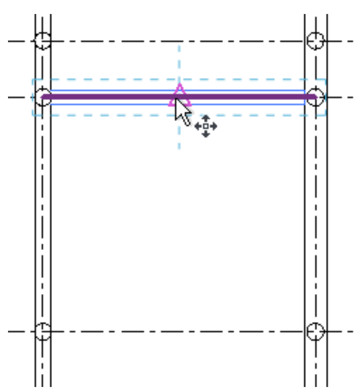
7 Scroll the view down so you can see the beam on grid line E.

8 Select the beam, and on the Modify tab, click Move

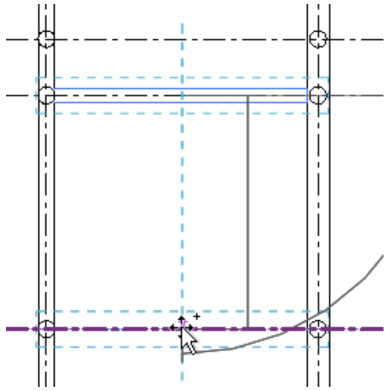


Some commands, such as Move and Copy, require 2 clicks to complete the command. After selecting the element to move, for example, click to specify the starting position, and click again to specify the ending position. In this case, you want to move the beam to grid line F.

9 Click the beam.



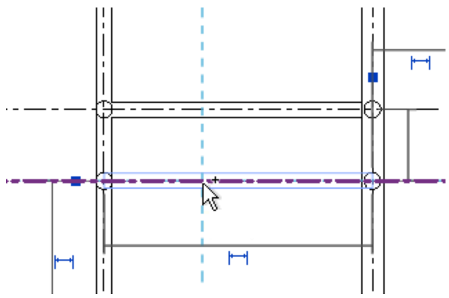
10 Click grid line F, as shown.



The beam moves down, and is placed at the move endpoint.

Another way to move an element is to select it and drag it to a new location.

11 Select the beam on grid line F, and drag it to grid line F.1



Undo commands

12 On the Quick Access toolbar, click the drop-down menu next to (Undo).

All changes you make to a project are tracked. The Undo command allows you to reverse the effects of one or more commands. In this example, you decide that you prefer the beam in its original position.

13 On the Undo drop-down, select the second item in the list, Move.

Selecting the second item in the list will undo the last 2 actions. All commands are canceled up to and including the selected command. The beam is returned to its original location.

NOTE To quickly undo the previous action, on the Quick Access toolbar, click the Undo command, or press *Ctrl+Z*.

End a command

14 Click Model tab ➤ Model line.

Some commands, such as the Lines command, stay active or current until you choose another command or end the current command.

15 Click in the drawing area to start the line, and click again to end it.

Notice that the Lines command is still active and you could continue to draw lines.

16 To end the command, use one of the following methods:

- Choose another command.
- Click Modify.

- Press *Esc* twice.

17 Close the file without saving your changes.

Starting a Project

Use the lessons in this tutorial to start the sample project, including:

- Importing an architectural drawing from Revit Architecture, and copying the grids and levels.
- Importing a 2D (DWG) drawing file, creating grids, and creating levels.

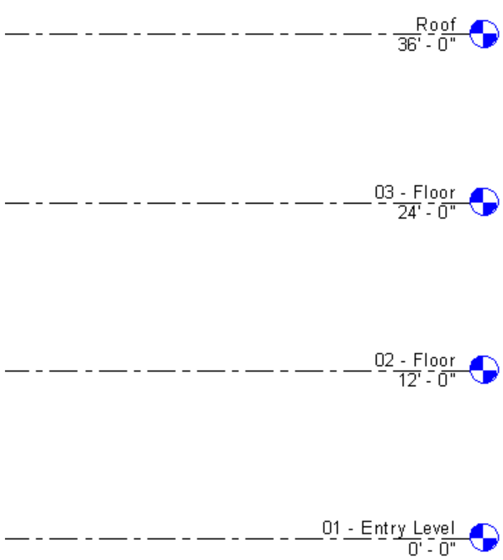
Importing a DWG File

3

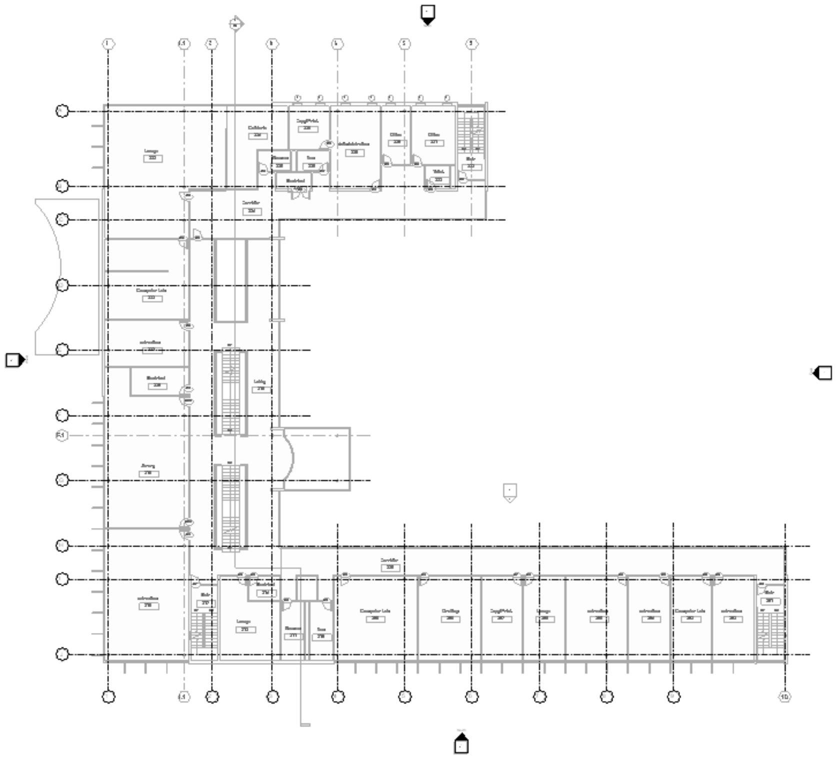
In this lesson, you begin your project in Revit Structure 2010, by importing and linking to a 2-dimensional (2D) architectural drawing (DWG format). The imported drawing is then used as a reference for creating your structure. You learn to:

- Open a new Revit Structure project, select a template file, and import a DWG file.
- Add new levels to the project.
- Add both vertical and horizontal column grids using the reference lines of the imported drawing.

New levels added to project



Column grids added to the project



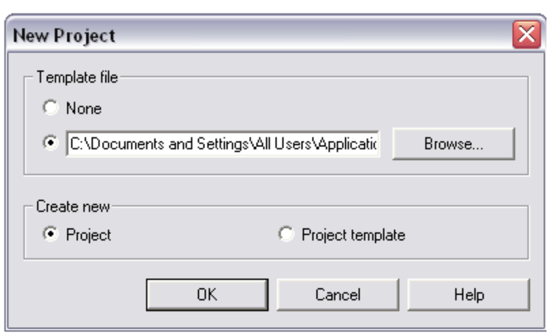
Importing/Linking a DWG File

In this exercise, you import and link a 2-dimensional (2D) architectural drawing (DWG format) to use as a background when you create your structure.

Open a new project

1 Click  ➤ New ➤ Project.

2 In the New Project dialog, under Template File, click Browse.



3 In the Choose Template dialog, open the Imperial Templates folder, select Structural Analysis-Default.rte, and click Open.

4 In the New Project dialog, under Create new, select Project, and click OK.

Import/Link a DWG file

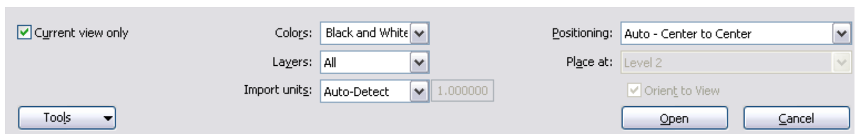
5 In the Project Browser, select Views (all) ➤ Structural Plans ➤ Level 2.

Level 2 displays as **bold** text because it is the active view that displays in the drawing area.

6 Click Insert tab ➤ Link panel ➤ Link CAD.

7 In the Link CAD Formats dialog:

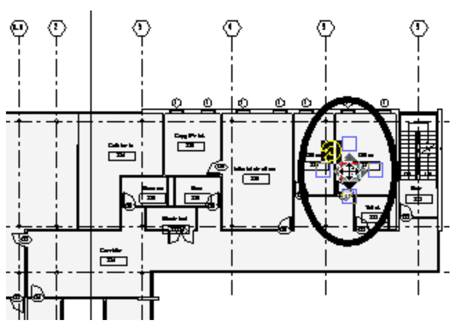
- In the left pane of the Import/Link dialog, click Training Files, and select Imperial\Technical_School-current.dwg.
- Select Current view only.
- For Colors, select Black and White.
- For Positioning, select Auto - Center to Center.
- Click Open.



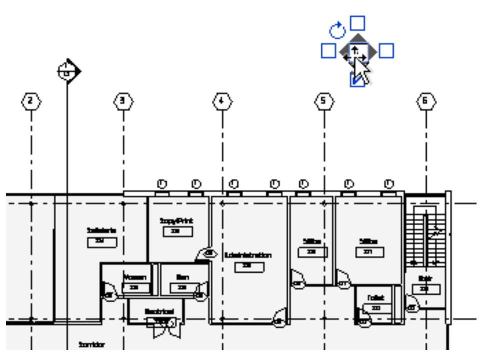
8 Enter **ZF** (Zoom to Fit).

Reposition elevation symbols

9 Select the North Elevation symbol as shown.



10 Click and drag the symbol to a position above the existing grids as shown.



11 Use the same method to relocate the remaining elevation symbols.

12 Close the file with or without saving it.

In the next tutorial, a new training file is supplied.

Adding New Levels

In this exercise, you create additional levels for your structure.

Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_BP_01_Add_Levels_i.rvt.

Modify existing levels

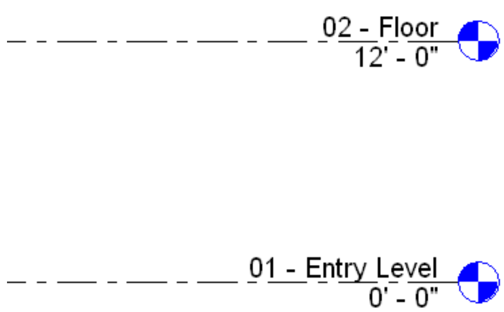
1 In the Project Browser, expand Elevations (Building Elevations), and double-click North.

2 Modify Level 1 as follows:

- Click the text for the Level 1 Elevation, and type **01 - Entry Level**.
- Press *Enter*.
- In the Revit dialog, click Yes to rename the corresponding views.

3 Modify Level 2 as follows:

- Click the text for the Level 2 Elevation, and type **02 - Floor**.
- Press *Enter*.
- In the Revit dialog, click Yes to rename the corresponding views.
- Click the value for the 02 - Floor elevation, type **12' 0"**, and press *Enter*.



Create 2 new levels

4 Create 2 additional levels as follows:

- Click Home tab ➤ Datum panel ➤ Level.
- Click Draw panel ➤ (Pick Lines).
- On the Options Bar, for Offset, type **12' 0"**.
- In the drawing area, click the 02 - Floor reference line; when a dashed line displays, click to create Level 3.
- Using the same method, create Level 4 with the offset value set to 12' 0".

5 Click Selection panel ➤ Modify.

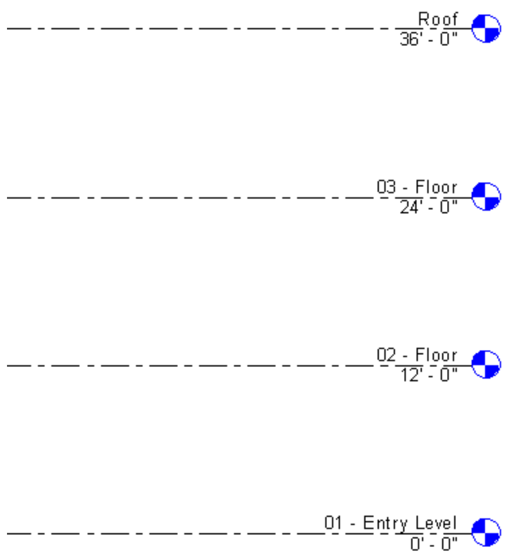
Modify text and values for the new levels

6 Modify Level 3 as follows:

- Click the text for the Level 3 Elevation, and type **03 - Floor**.
- Press *Enter*.
- In the Revit dialog, click Yes to rename the corresponding views.

7 Modify Level 4 as follows:

- Click the text for the Level 4 Elevation, and type **Roof**.
- Press *Enter*.
- In the Revit dialog, click Yes to rename the corresponding views.



8 Press *Esc*.

9 Close the file with or without saving it.

In the next tutorial, a new training file is supplied.

Adding Column Grids

In this exercise, you use the imported drawing as a background when creating both vertical and horizontal grids.

Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_BP_02_Add_Grids_i.rvt.

Place horizontal grids

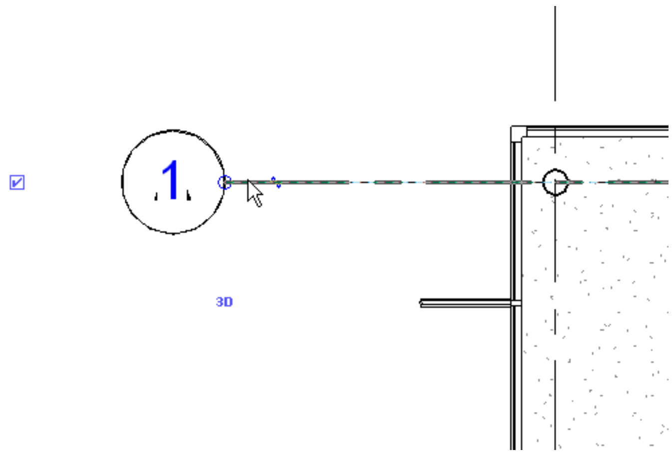
1 In the Project Browser, select Views (all) ➤ Structural Plans ➤ 02 - Floor.

2 Enter **ZF** (Zoom to Fit).

3 Click Home tab ➤ Datum panel ➤ Grid.

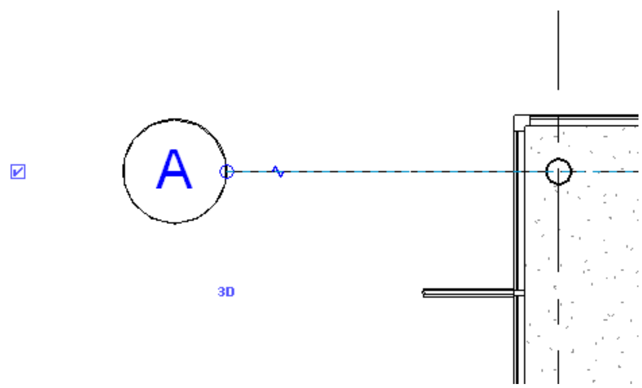
4 Click Draw panel ➤  (Pick Lines).

5 Select the first horizontal grid line.



The grid line is highlighted, and a value of 1 appears within the grid bubble.

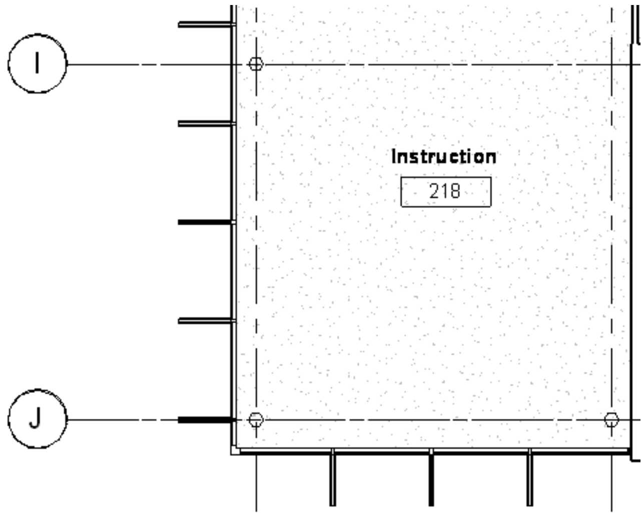
6 Select the grid bubble, type **A**, and press *Enter*.



7 Select the remaining horizontal grid lines using the zoom controls as needed.

As you select subsequent grid lines, labels display in alphabetical sequence.

The last horizontal grid line is labeled J.



8 Click Selection panel ➤ Modify.

Place vertical grids

9 Enter **ZF** (Zoom to Fit).

10 Click Datum panel ➤ Grid.

11 Click Draw panel ➤  (Pick Lines).

12 Zoom in on the southwest corner of the floor plan.

13 Select the first vertical grid line.

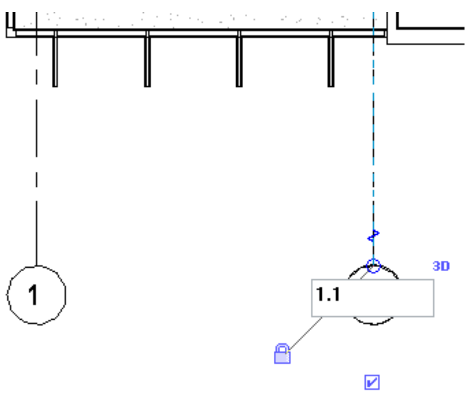
The grid line is highlighted, and a value of K appears within the grid bubble.

14 Click the value within the grid bubble, type **1**, and press *Enter*.

15 Select the second vertical grid line.

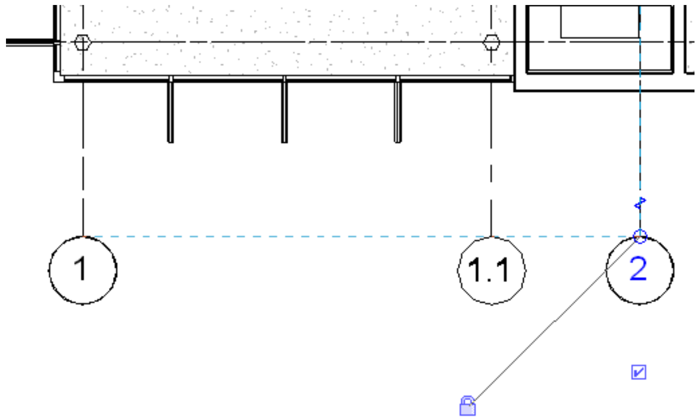
The sequencing changes from alphabetical to numeric.

16 Click the value within the grid bubble, type **1.1**, and press *Enter*.

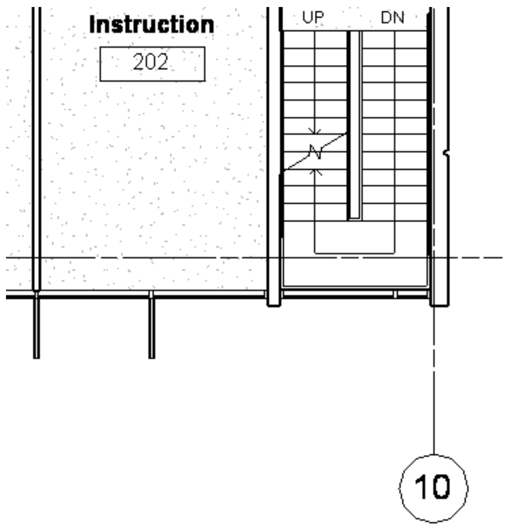


17 Select the vertical grid line to the right of grid 1.1.

18 Click the value within the grid bubble, type **2**, and press *Enter*.



19 Select the remaining vertical grid lines. The last grid line is labeled 10.



20 Click Selection panel ➤ Modify.

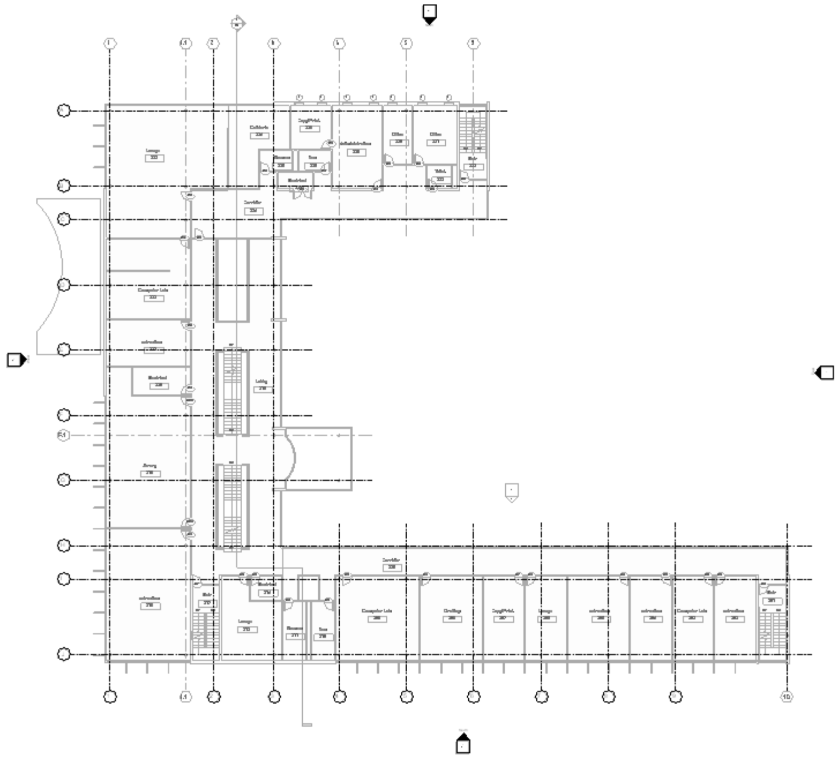
Change drawing visibility

21 Enter **ZF** (Zoom to Fit).

22 Click View tab ➤ Graphics panel ➤ Visibility/Graphics.

23 In the Visibility/Graphic Overrides dialog:

- Click the Imported Categories tab.
- Under Halftone, select the Technical_School-current.dwg file.
- Click Apply, and then OK.
  The halftone of the imported drawing is used as a background when placing additional structural elements.



24 Close the file with or without saving it.

In the next tutorial, a new training file is supplied.

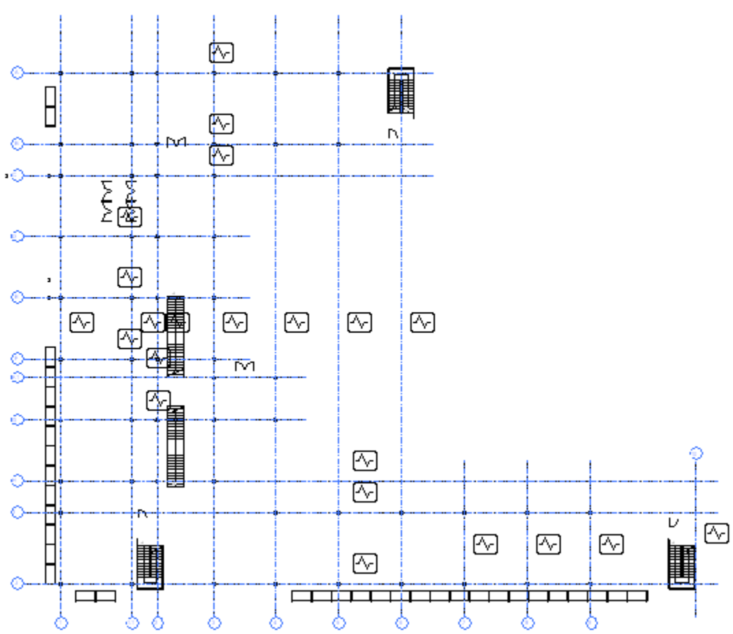
Importing an RAC File

4

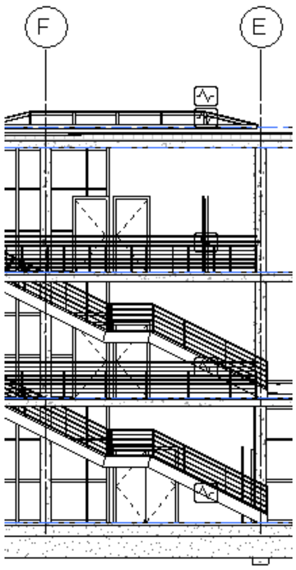
In this lesson, you begin your project in Revit Structure 2010, by importing and linking to an existing architectural project created in Revit Architecture. You learn to:

- Open a new Revit Structure project and select a template file.
- Import and link to the Revit Architecture file.
- Copy existing grids and levels from the imported file into the new project using the copy monitor feature.
- Choose structural views.

Copy existing grids



Copy existing levels



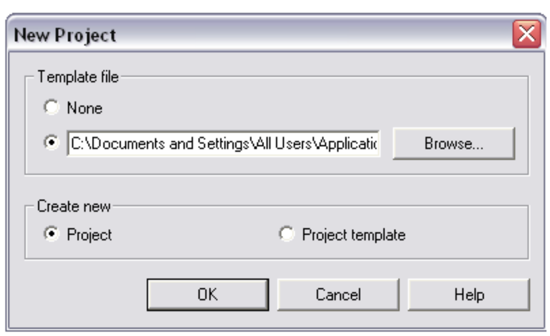
Linking a Revit Architecture File

In this exercise, you learn how to link your Revit Structure project to a 3-dimensional (3D) drawing created in Revit Architecture and use it as a background. You also learn how to use the Copy/Monitor feature of Revit Structure to provide project coordination between architects and structural engineers. After they are copied, all grids and levels are monitored for any changes made against the original file during the design review process.

Open a new project

1 Click  ➤ New ➤ Project.

2 In the New Project dialog, under Template File, click Browse.



3 In the Choose Template dialog, open the Imperial Templates folder, select Structural Analysis-Default.rte, and click Open.

4 In the New Project dialog, under Create new, select Project, and click OK.

Link a Revit Architecture file

5 Click Insert tab ➤ Link panel ➤ Link Revit.

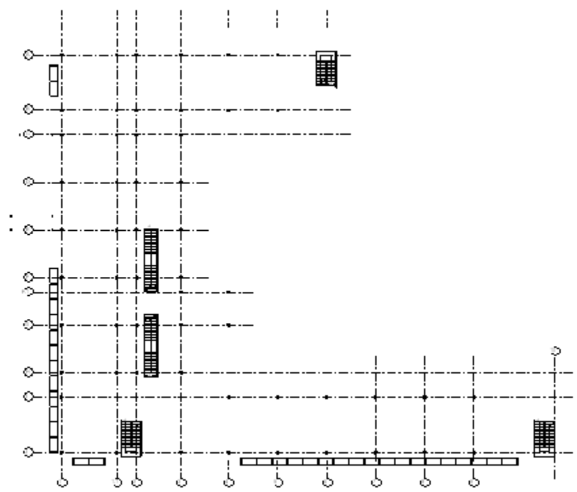
6 In the Import/Link RVT dialog:

- In the left pane of the open dialog, click Training Files.
- Select Imperial\Technical_School-current.rvt.
- For Positioning, select Auto - Origin to Origin.
- Click Open.

Exterior facade outlines, column grids, stairways, and a few basic interior walls display in the view.

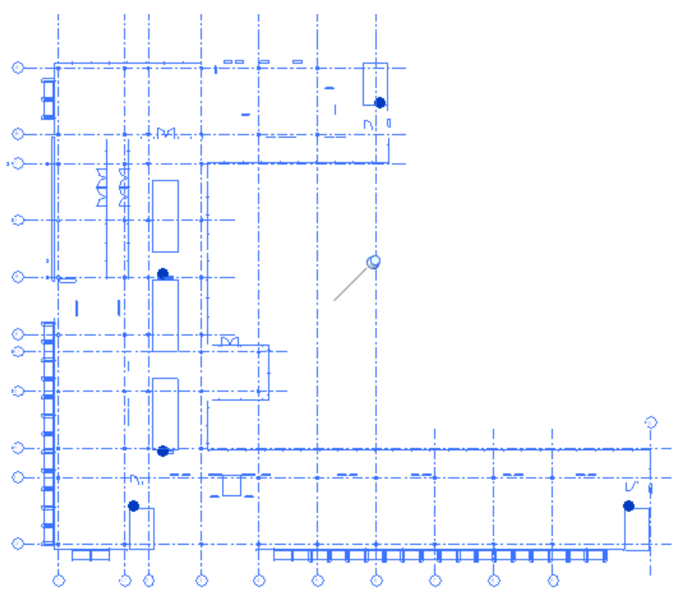
In the Project Browser, Level 2 is displayed in bold. This is the active view that displays in the drawing area.

7 Enter **ZF** (Zoom to Fit).



8 In the drawing area, click the linked file.

9 On the Modify RVT Links tab ➤ Modify panel ➤ Pin to pin the drawing. A pinned drawing cannot be moved within the Revit Structure drawing area.



Setup Copy/Monitor Option

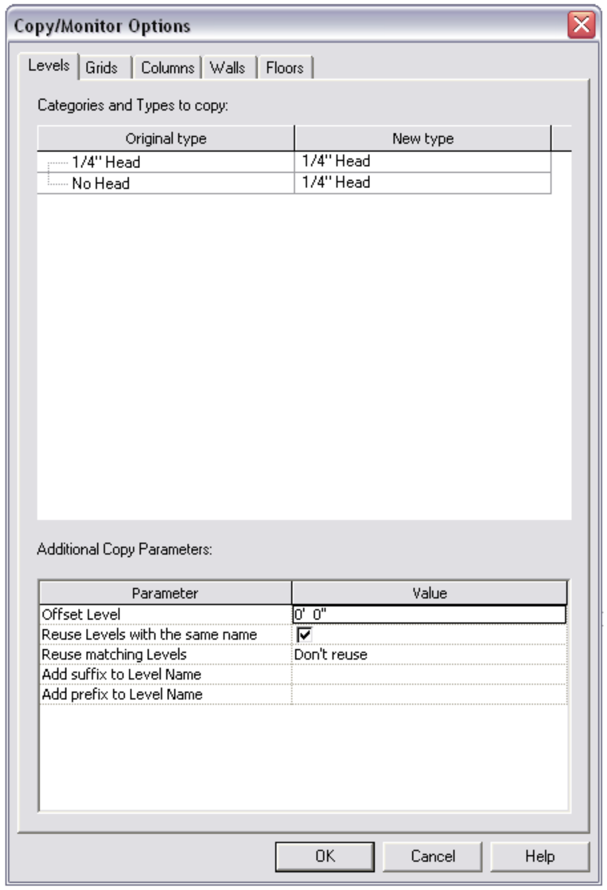
10 Click Collaborate tab ➤ Coordinate panel ➤ Copy/Monitor drop-down ➤ Select Link.

11 Click the imported Revit drawing.

NOTE The contextual tab displays options used to set up the Copy/Monitor feature.

---

12 Click Tools panel ➤ Options.



The Copy/Monitor Options dialog opens. Elements available for monitoring are separated into 5 categories (Levels, Grids, Columns, Walls, and Floors). On each tab the Original Type column lists the type for the Revit Architecture file, and the New Type column lists the corresponding element available in the template selected. Each tab provides various parameters that you can set for that specific element. Also, you can exclude element types that you do not want to copy.

13 Click OK.

Copy grids

14 In the Project Browser, under Structural Plans, select Level 2.

15 Click Tools panel ➤ Copy.

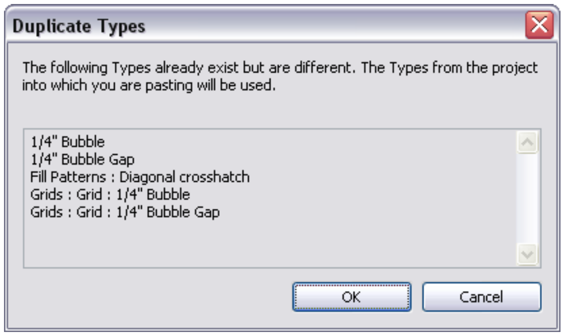
16 On the Options Bar, select Multiple.

17 While pressing *Ctrl*, select each grid on the Revit Architecture file.

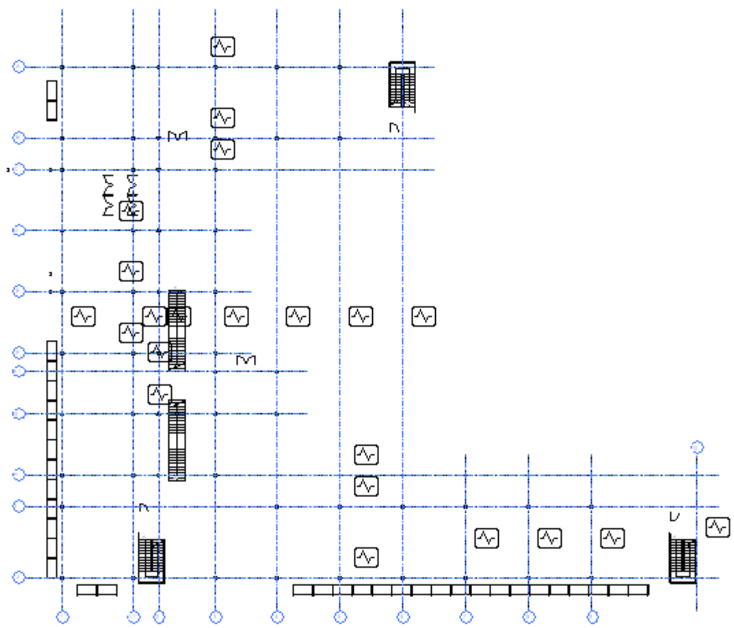
18 After selecting all grids, on the Options Bar, click Finish.

NOTE Ignore the Revit warning that indicates the loaded type has been renamed.

NOTE If you are prompted that the element type already exists in the project, and that the type from the new project will be used, click OK.



A copy/monitor symbol displays, indicating a relationship with the original element.



Copy levels

19 In the Project Browser, under Elevations (Building Elevation), click East.

20 Click Tools panel ➤ Copy.

21 On the Options Bar, select Multiple.

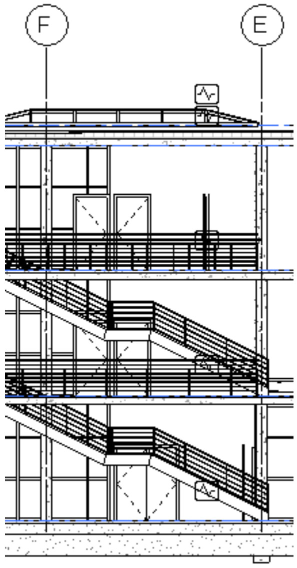
22 While pressing *Ctrl*, select each level on the Revit Architecture file.

23 After selecting all levels, on the Options bar, click Finish.

NOTE If you are prompted that the element type already exists in the project, and that the type from the new project will be used, click OK.

NOTE Ignore the Revit warning that indicates the loaded type has been renamed.

A copy/monitor symbol displays, indicating a relationship with the original element.



24 Click Copy/Monitor panel ➤ Finish.

Create new structural views

25 Click View tab ➤ Create panel ➤ Plan Views drop-down ➤ Structural Plan.

26 In the New Plan dialog:

- Under Floor Plan Views, select 01-Entry Level.
- While pressing *Shift*, select Roof.
- Click OK.

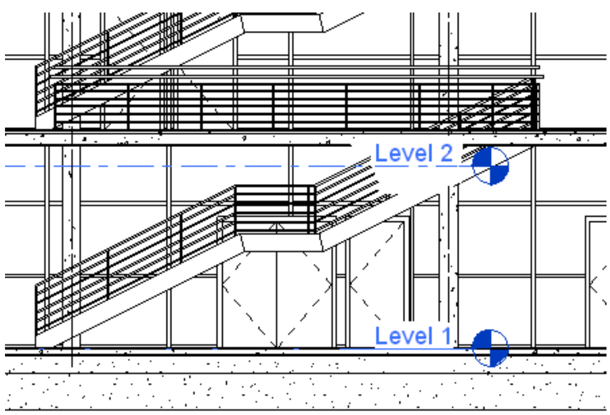
Notice that the new structural views display on the Design Bar.

Delete default levels

27 In the Project Browser, under Elevations (Building Elevation), double-click East.

28 Zoom to the center of the view.

29 Select Level 1, press *Ctrl*, and select Level 2.



30 Press *Delete*.

NOTE You can ignore the warning message that indicates the views for Levels 1 and 2 will be deleted. Click OK.

31 Close the file with or without saving it.

In the next tutorial, a new training file is supplied.

Modeling a Project

In this tutorial, you use a recommended workflow to create a Building Information Model (BIM) and refine the model.

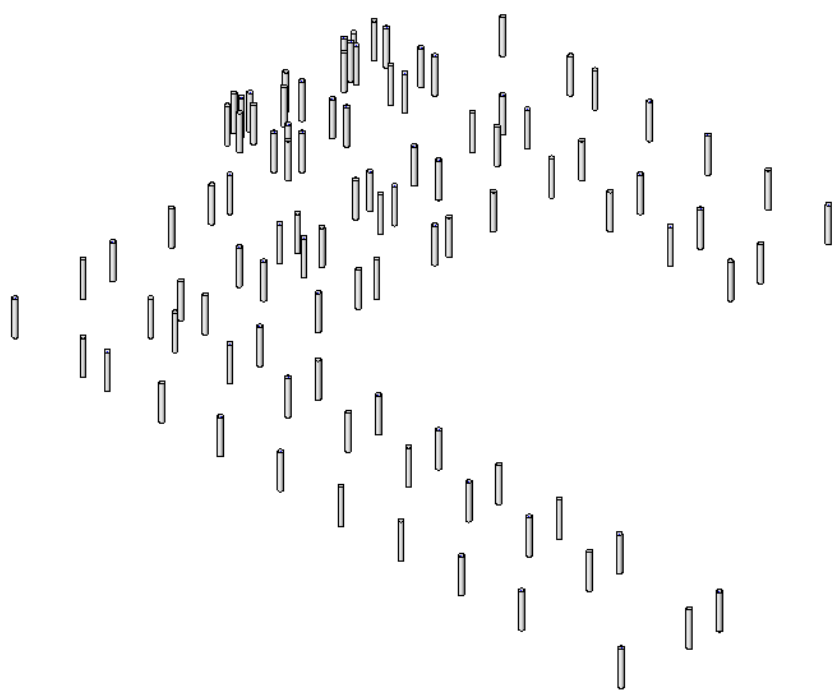
Adding Structural Columns

5

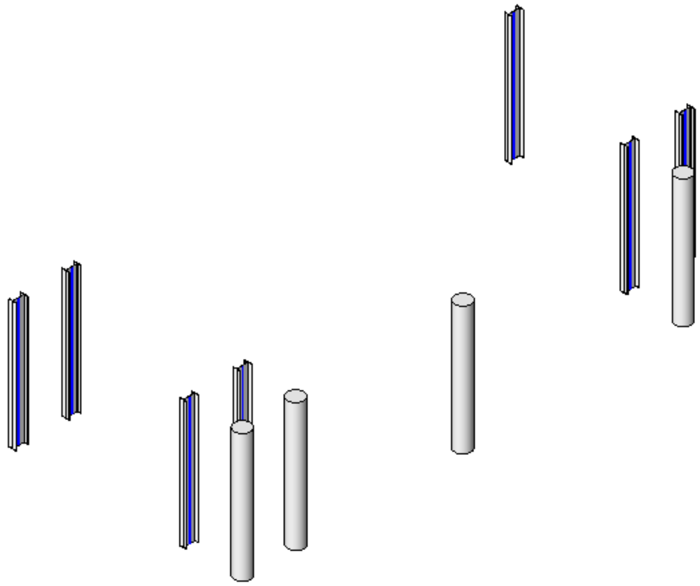
In this lesson, you add concrete columns to the main structure and steel columns to the entry way. You place columns using the grid intersection tool at locations specified in the architectural floor plan. You learn to:

- Add concrete columns to Level 2 of the structure.
- Add steel columns to the entry way.
- Verify the base offset dimension for all columns to align the base with the entry level of the structure.

Concrete columns added to the structure

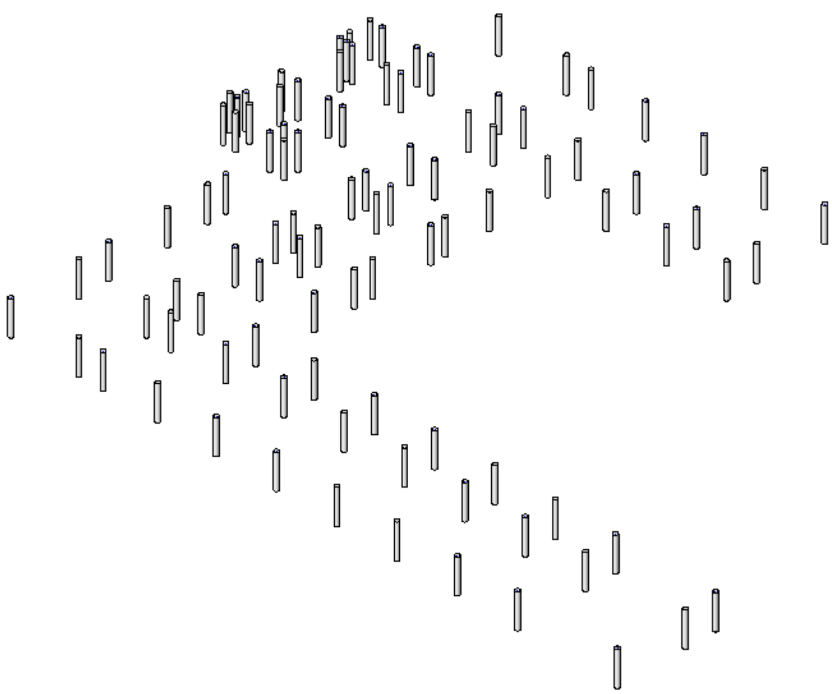


Steel columns added to the structure



Adding Concrete Columns

In this exercise, you add concrete columns to the structure. These columns extend from the entry level to Level 2.



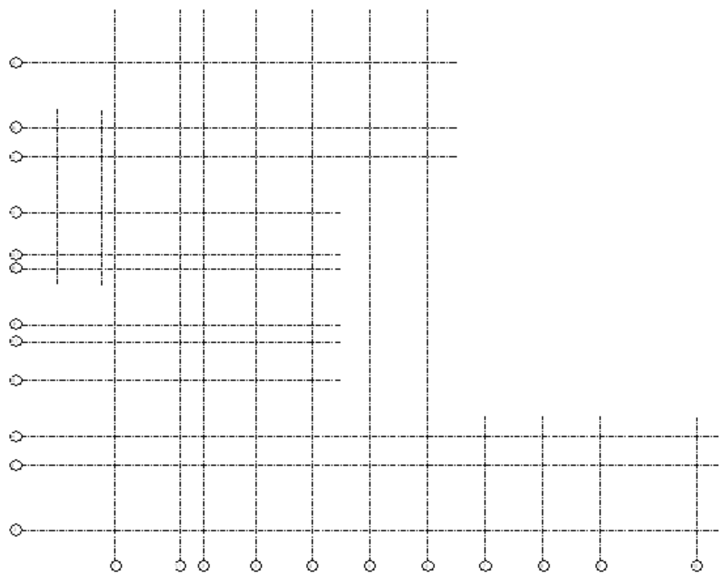
Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_COL_01_Concrete_i.rvt.

Open the structural plan

1 In the Project Browser, select Views (all) ➤ Structural Plans ➤ 02- Floor.

2 Enter **ZF** (Zoom to Fit).



Place concrete columns at grid intersections

3 Click Home tab ➤ Structure panel ➤ Column.

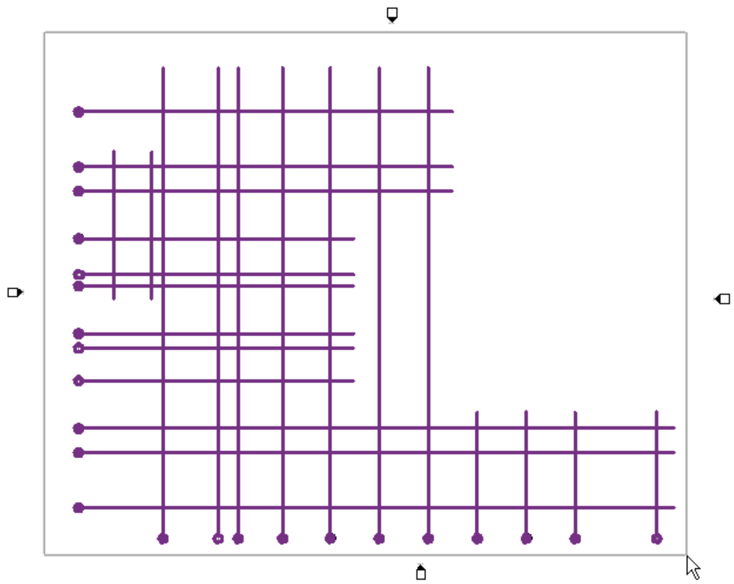
4 Click Element panel ➤ Change Element Type drop-down ➤ Concrete-Round-Column : 18”.

5 Click Multiple panel ➤ On Grids.

NOTE When you use the On Grids tool to place columns, Revit Structure automatically places the top level of the columns at the current level.

---

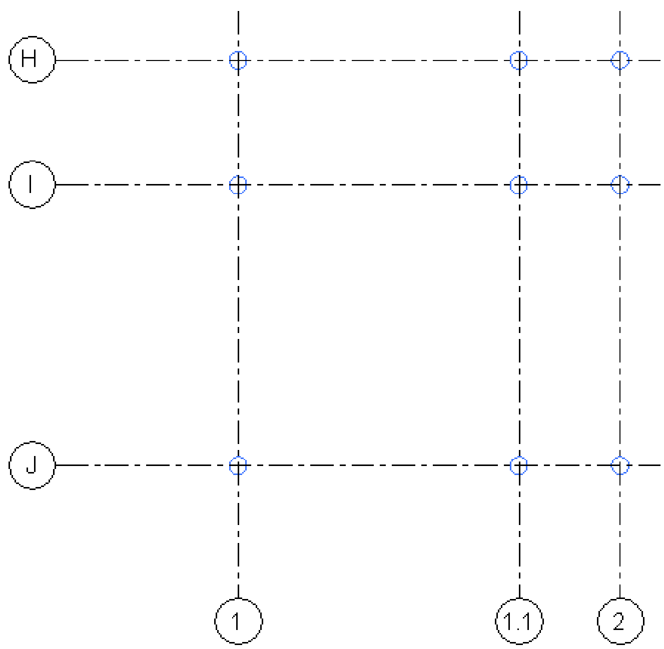
6 Draw a selection box around the grids by first clicking the upper-left corner of the drawing above the grid, and dragging the cursor to the lower-right corner as shown.



The grids are highlighted, indicating that they are selected.

7 Click Multiple Selection panel ➤ Finish Selection.

Concrete columns are placed at each grid intersection.



8 Press *Esc*.

Verify the column base level

Verify that the base level of all the columns is extended to the entry level of the structure.

9 Right-click the column at grid location A-1, and click Select All Instances.

All the columns are selected.

10 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

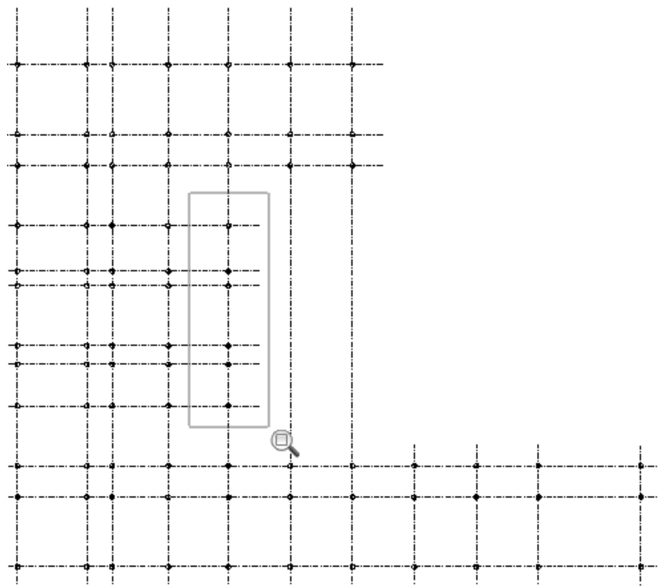
11 In the Instance Properties dialog:

- Under Constraints, for Base Level, select 01 - Entry Level.
- For Base Offset, type **0**' **0**".
- Click OK.

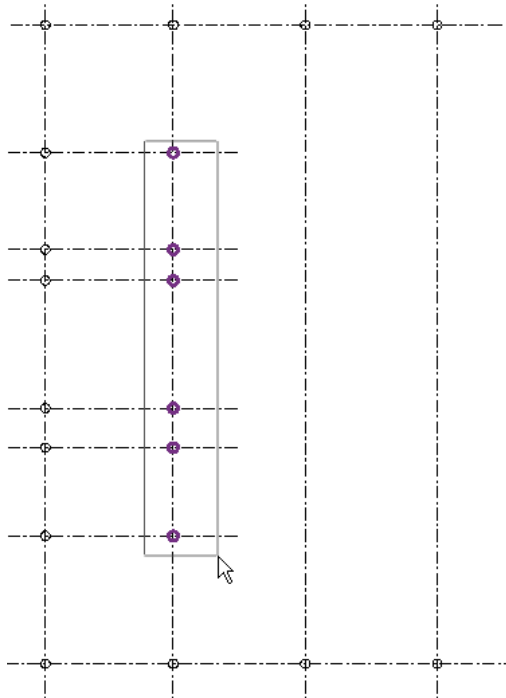
Delete the columns outside the drawing floor extents

During the placement of the columns, some were placed at grid locations outside of the floor extents of the architectural drawing and need to be deleted.

12 Zoom in on the columns on grid line 4 as shown.



13 Draw a pick box around the columns on grid line 4 as shown.



The columns are highlighted, indicating that they are selected.

14 Press *Delete*.

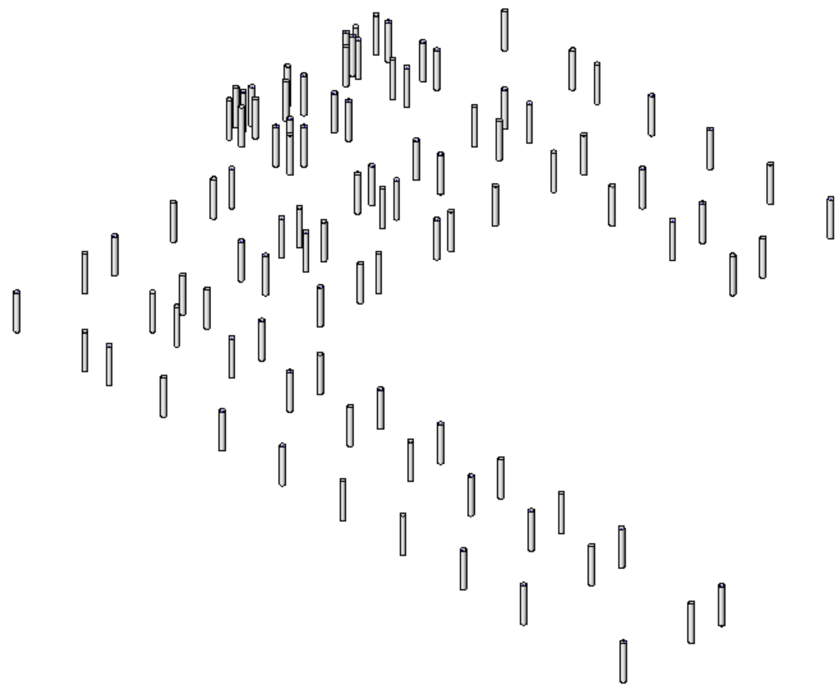
The columns are deleted from the plan view.

View the columns in 3D

15 In the Project Browser, click 3D Views ➤ 3D.

16 Enter **ZF** (Zoom to Fit).

17 On the View Control Bar, click Model Graphic Style: Shading w/Edges.

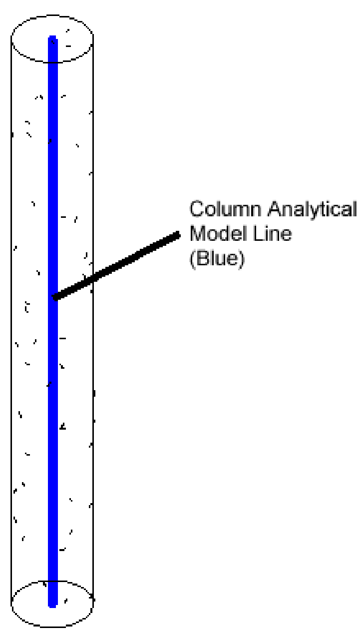


View the concrete column analytical model

As you create the model in Revit Structure, a simplified 3D representation of each element is created automatically. This simplified view is referred to as the analytical model. Each element is drawn with a different color to help identify specific analytical properties; for example, columns display as a blue line. You can export the analytical model to analysis and design applications.

18 On the View Control Bar, click Model Graphic Style: Wireframe.

19 Zoom in on one of the columns to see the blue line representing the analytical model.

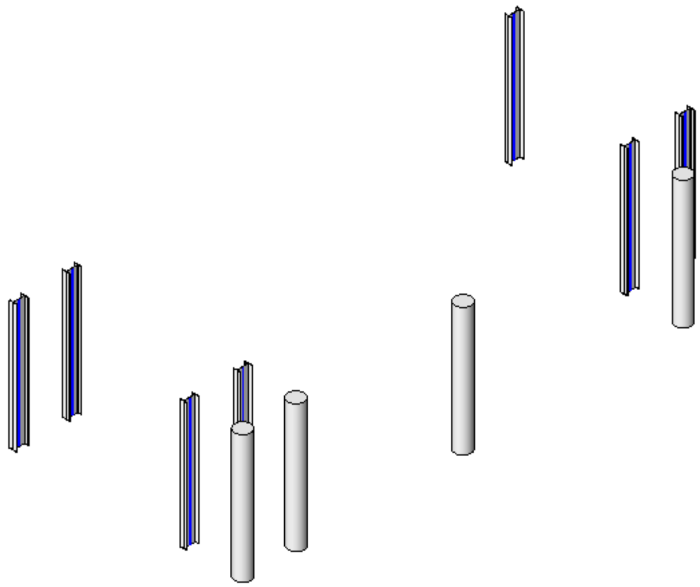


20 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Adding Steel Columns

In this exercise, you add steel columns to the entry way of the structure at specific grid intersections. The columns extend from the 01 Entry Level to Level 02 Floor.



Training File

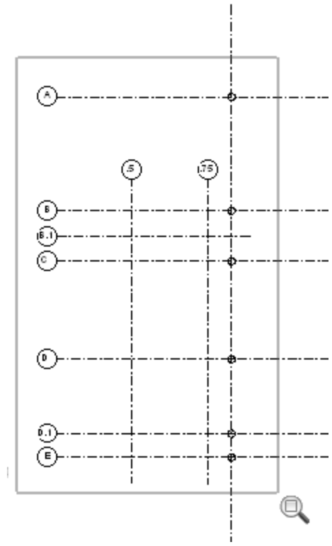
- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_COL_02_Steel_i.rvt.

Place columns at specific grid intersections

1 In the Project Browser, select Views (all) ➤ Structural Plans ➤ 02- Floor.

2 Enter **ZF** (Zoom to Fit).

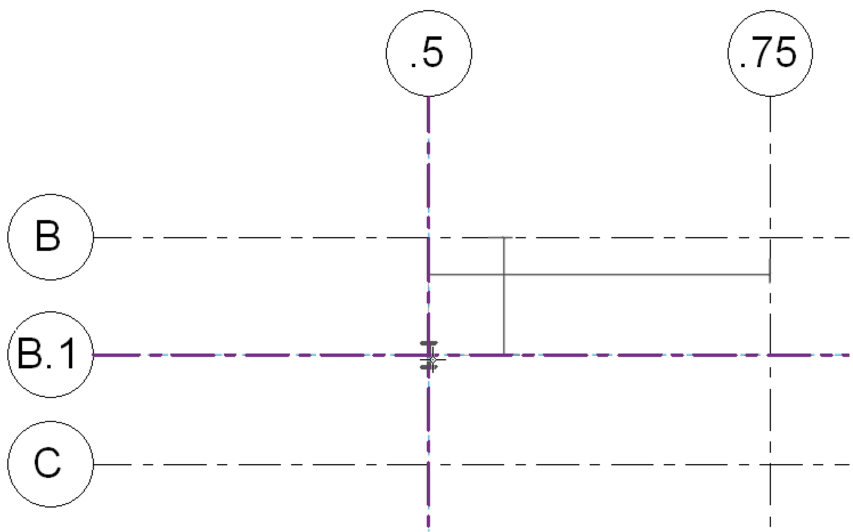
3 Zoom in on the entry way of the structure.



4 Click Structure panel ➤ Column.

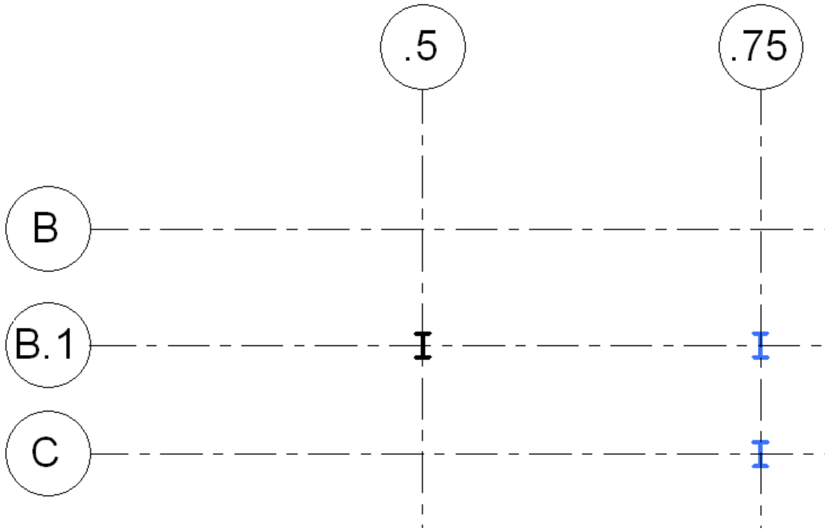
5 Click Element panel ➤ Change Element Type drop-down ➤ W-Wide Flange-Column : W14X43.

6 Click to place the first column at grid intersection B.1 / .5 as shown.



7 Using the same method, place a column at the following grid locations:

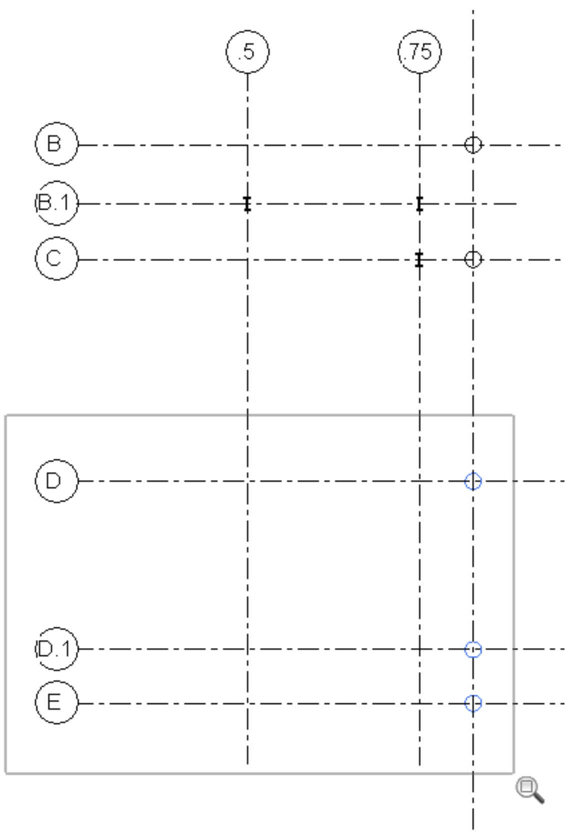
- B.1 / .75
- C / .75



8 Click Selection panel ➤ Modify.

Place remaining columns using the grid intersection tool

9 Zoom in around the lower portion of the entry way of the structure.



10 Click Home tab ➤ Structure panel ➤ Column.

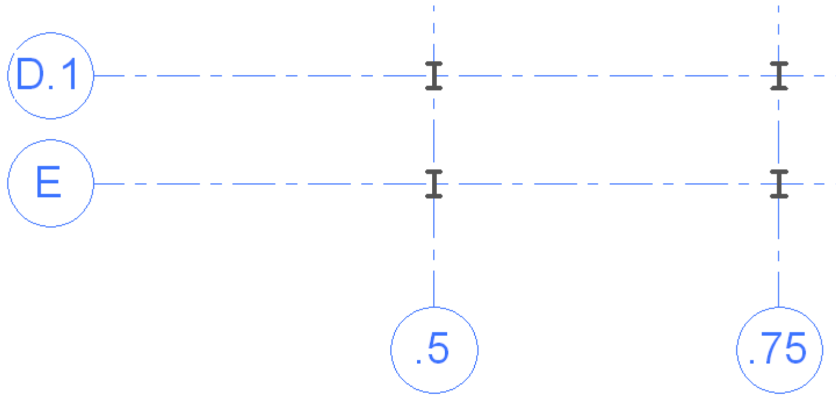
11 Click Element panel ➤ Change Element Type drop-down ➤ W-Wide Flange-Column : W14X43.

12 Click Multiple panel ➤ On Grids.

13 Select multiple grids as follows:

- Click grid E.
- While pressing *Ctrl*, select grid line D.1, .5, and .75.

The columns are placed automatically at these grid intersections.



14 Click Multiple Selection panel ➤ Finish Selection.

15 Click Selection panel ➤ Modify.

Set the column base level

The base level of all the columns is extended to the entry level of the structure.

16 Right-click the column at grid location E / .5, and click Select All Instances.

All the steel columns are selected.

17 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

18 In the Element Properties dialog:

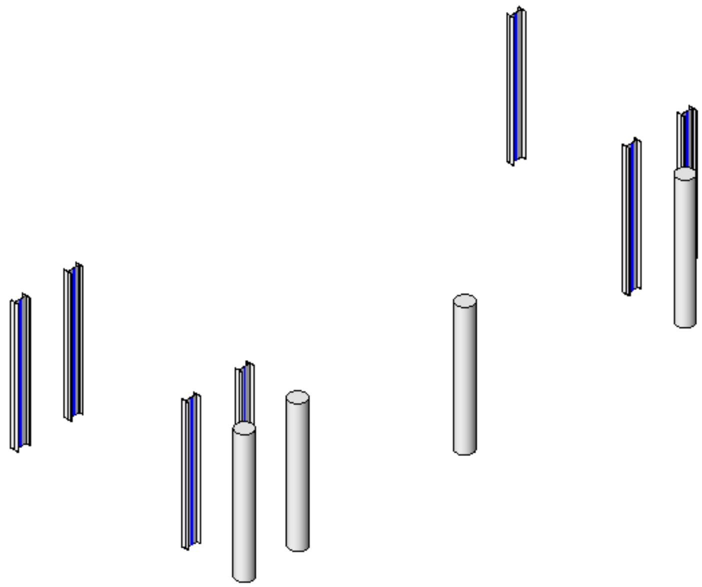
- Under Constraints, for Base Level, select 01 - Entry Level.
- For Base Offset, type **0**' **0**".
- Click OK.

19 Press *Esc*.

View the columns in 3D

20 In the Project Browser, click 3D Views ➤ 3D.

21 Zoom in on the steel columns.

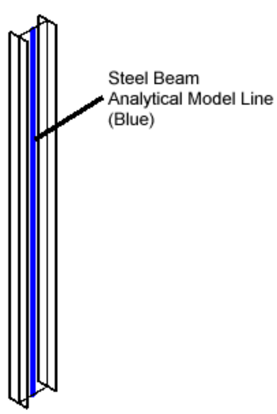


View the steel column analytical model

As you create the model in Revit Structure, a simplified 3D representation of each element is created automatically. This simplified view is referred to as the analytical model. Each element is drawn with a different color to help identify specific analytical properties; for example, columns display as a blue line. You can export the analytical model to analysis and design applications.

22 On the View Control Bar, click Model Graphic Style: Wireframe.

23 Zoom in on one of the columns to see the blue line representing the analytical model.



24 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

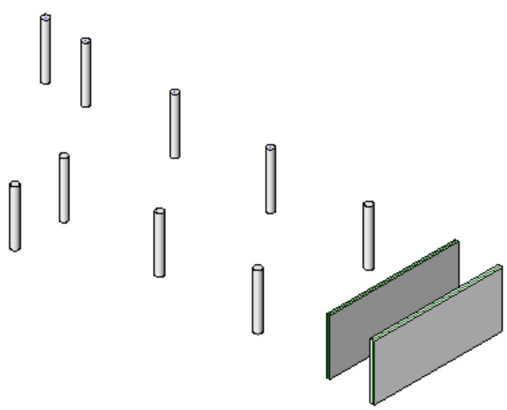
Adding Structural Walls

6

In this lesson, you add structural walls to the stairways in the northeast and southeast corners of the structure. You learn to:

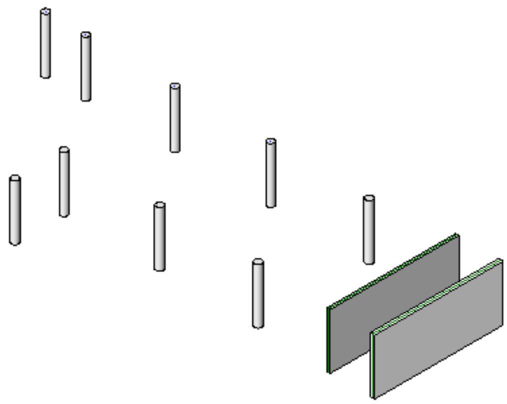
- Add a 12" concrete wall to the exterior surface of both stairways.
- Add a 8" concrete wall to the interior surface of both stairways.

Structural walls added to the northeast stairway



Adding Structural Walls

In this exercise, you add structural walls to the interior and exterior surfaces of the stairways located in the northeast and southeast corners of the structure.



Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_WAL_Add_Str_Walls_i.rvt.

Change the view visibility

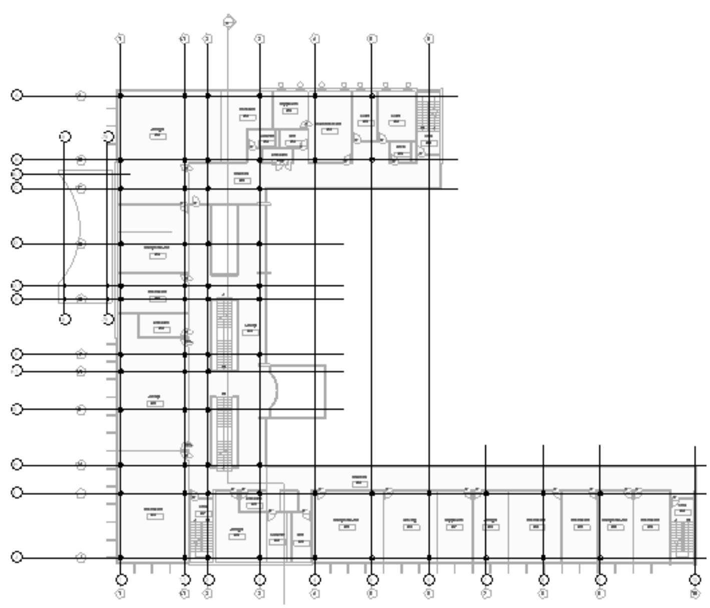
1 In the Project Browser, select Views (all) ➤ Structural Plans ➤ 02- Floor.

2 Enter **ZF** (Zoom to Fit).

3 Click View tab ➤ Graphics panel ➤ Visibility/Graphics.

4 In the Visibility/Graphic Overrides dialog:

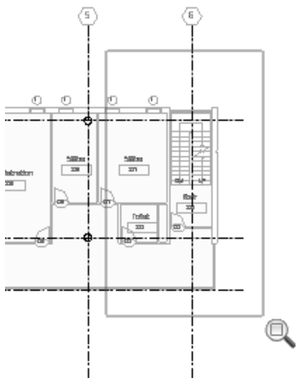
- Click the Imported Categories tab.
- Under Visibility, select Technical_School-current.dwg.
- Click Apply, and then OK.



The imported architectural drawing displays in the drawing area. You will use the drawing as a guide to trace the structural walls.

Place a structural wall using the Pick tool

5 Zoom in on the stairway located in the northeast corner of the architectural drawing.



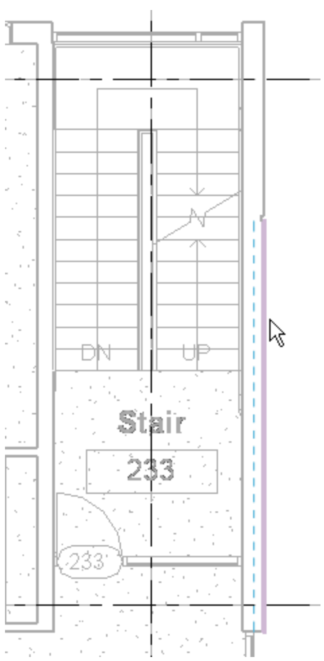
6 Click Home tab ➤ Structure panel ➤ Wall.

7 Click Element panel ➤ Change Element Type drop-down ➤ Basic Wall : Generic - 12" Masonry.

8 Click Draw panel ➤  (Pick Lines).

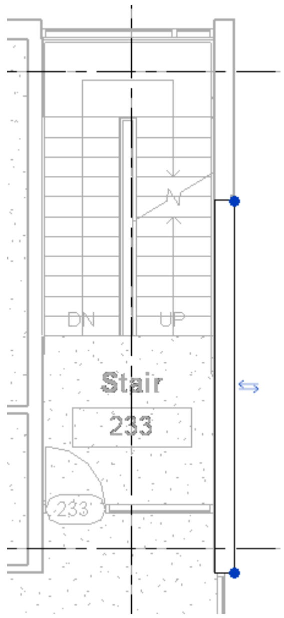
9 On the Options Bar, for Location Line, select Finish Face: Exterior.

10 Using the architectural drawing as a guide, select the line that represents the exterior face of the stairway.



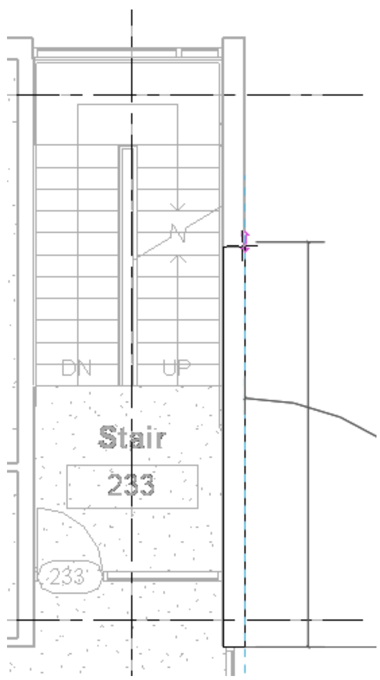
The blue dotted line represents the center line of the structural wall.

11 Click to place the first structural wall.



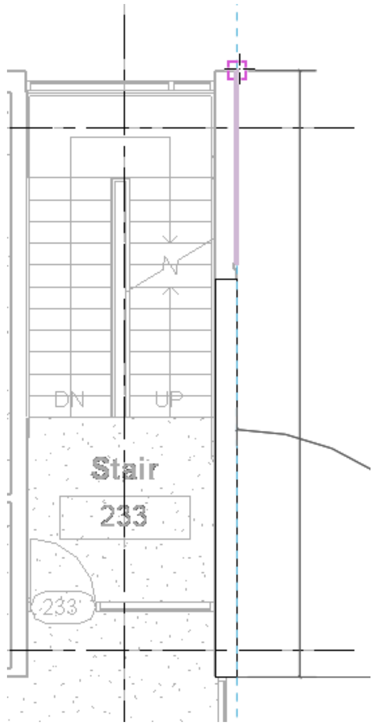
Extend the wall

12 Click the blue dot located on the structural wall as shown.



The blue dot represents the drag wall end command used to extend the length of the structural wall.

13 Drag the wall end up to the north end of the stairs as shown.



14 Press *Esc*.

Place a structural wall using the Draw tool

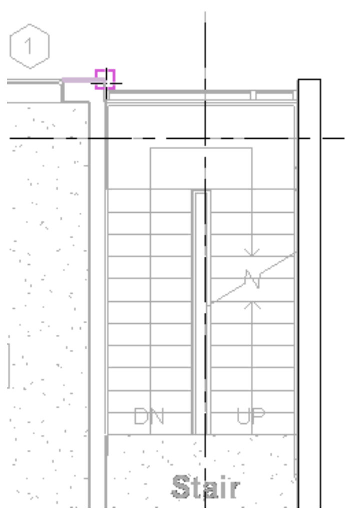
15 Click Home tab ➤ Structure panel ➤ Wall.

16 Click Element panel ➤ Change Element Type drop-down ➤ Basic Wall : Exterior - 8" Concrete.

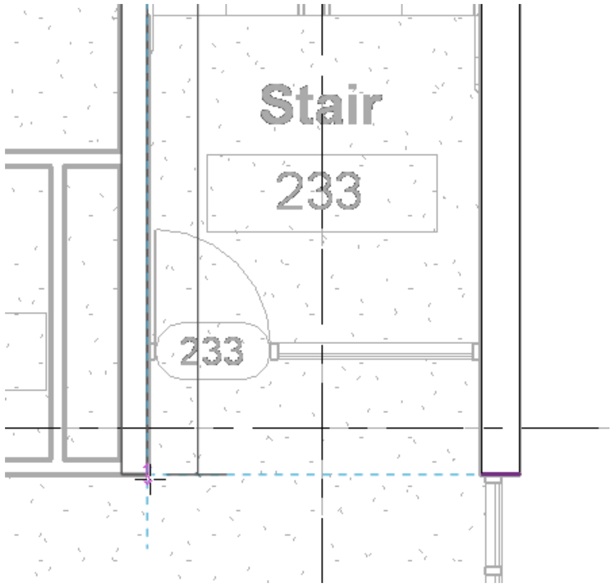
17 Click Draw panel ➤ (Line).

18 On the Options Bar, for Location Line, select Finish Face: Exterior.

19 Using the architectural drawing as a guide, click the exterior surface of the wall to establish the start point for the wall, as shown.

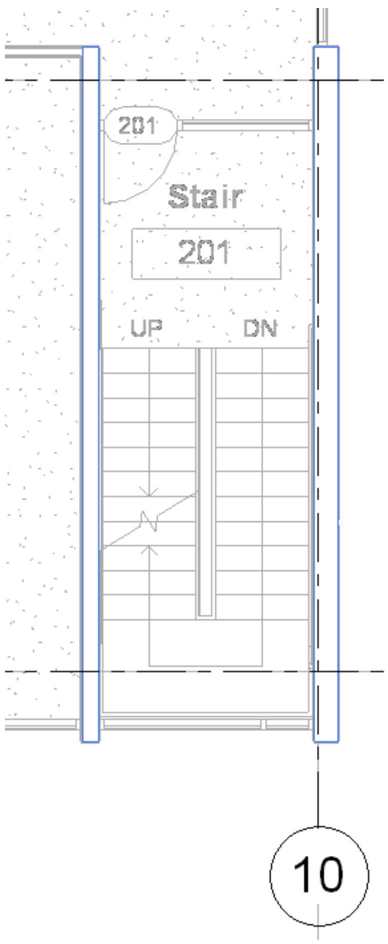


20 Drag the tool to the bottom edge of the stairway and click to place the second wall, as shown.



21 Press *Esc*.

22 Use the same method to place 2 structural walls around the southeast stairway, as shown.



Verify the wall base constraint

The base level of each wall is extended to the entry level of the structure.

23 Right-click the exterior wall of the northeast stairway, and click Select All Instances.

24 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

25 In the Element Properties dialog:

- Under Constraints, for Base Level, select 01 - Entry Level.
- For Base Offset, type **0**' **0**".
- Click OK.

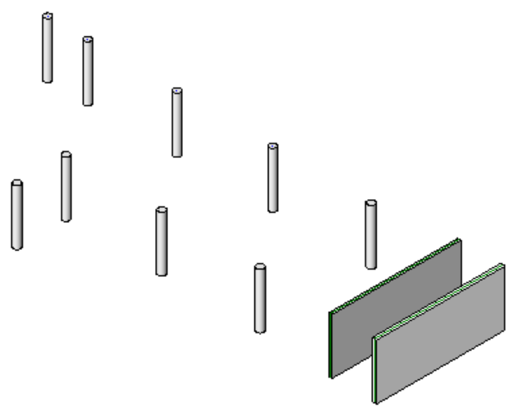
26 Use the same method to verify the base offset value for the interior walls of both stairways.

View the structural walls in 3D

27 In the Project Browser, select Views (all) ➤ 3D Views ➤ 3D.

28 On the View Control Bar select Model Graphic Style: Shading w/Edges.

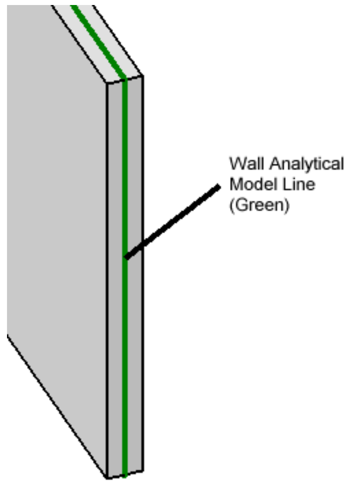
29 Zoom in on the the completed structural walls in the northeast corner of the architectural drawing.



View the wall analytical model

As you create the model in Revit Structure, a simplified 3D representation of each element is created automatically. This simplified view is referred to as the analytical model. Each element is drawn with a different color to help identify specific analytical properties; for example, walls display as a green line. You can export the analytical model to analysis and design applications.

30 Zoom in on one of the walls to see the green line representing the analytical model.



31 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

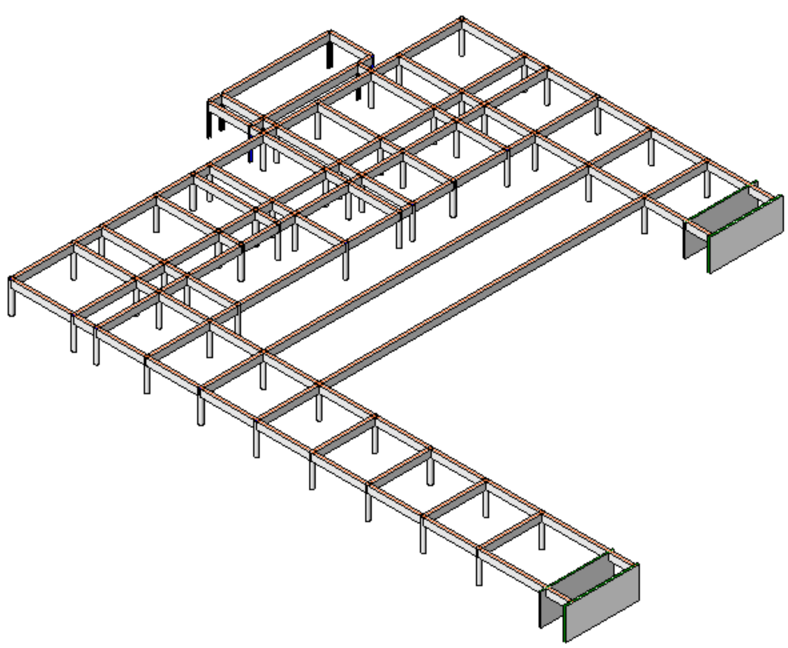
Adding Structural Beams

7

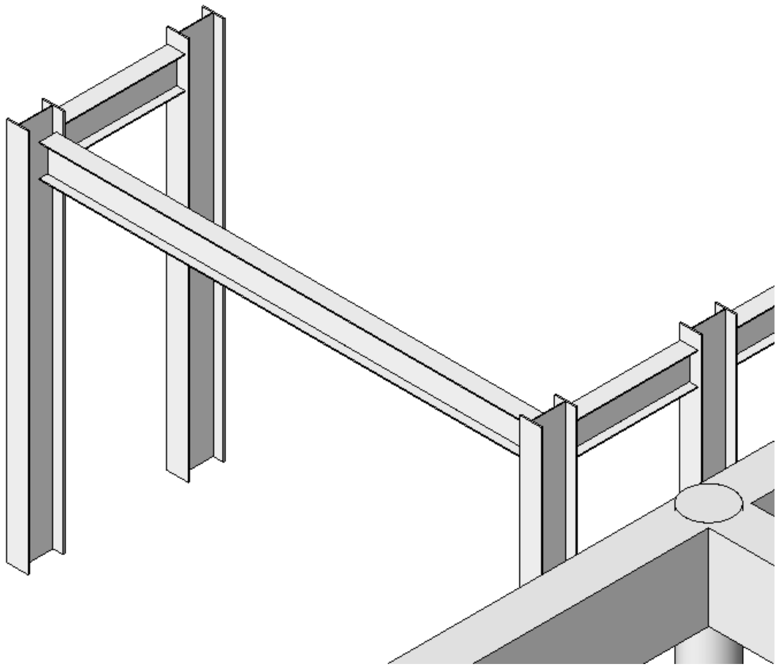
In this lesson, you add structural beams to the structure. Revit Structure determines the structural usage properties of a beam automatically, based on the structural elements that support the beam. Structural usage can be changed before or after the beam is placed. You learn to:

- Add concrete beams and create a new beam size.
- Add steel beams to the entry way, and set the beam offset to accommodate the slab that you add in a future exercise.

Concrete beams added to the structure

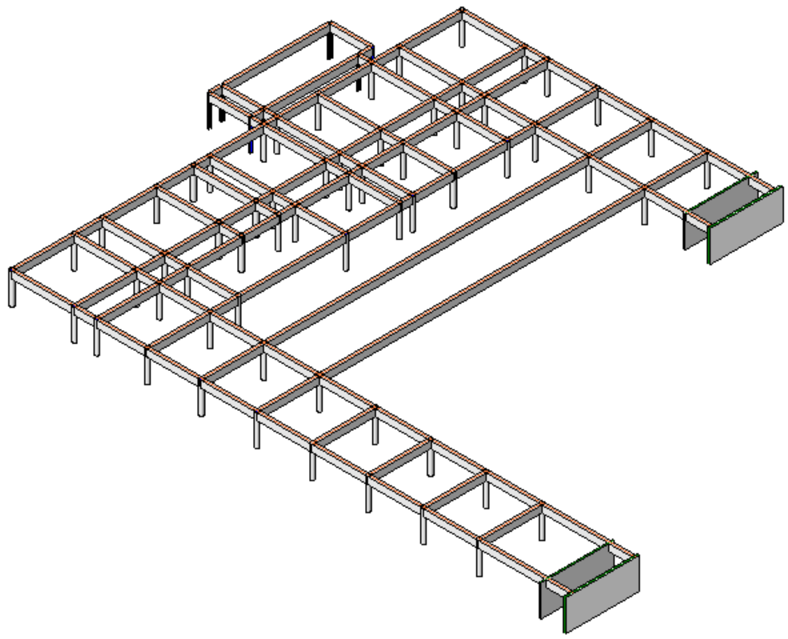


Steel beams added to the entry way



Adding Concrete Beams

In this exercise, you use the architectural drawing as a guide to automatically add concrete beams to Level 2, forming the framework of the structure. You also delete the beams that are placed outside the floor plan. Finally, you create a larger beam size and apply the new beam, providing additional support to the upper floors of the structure.

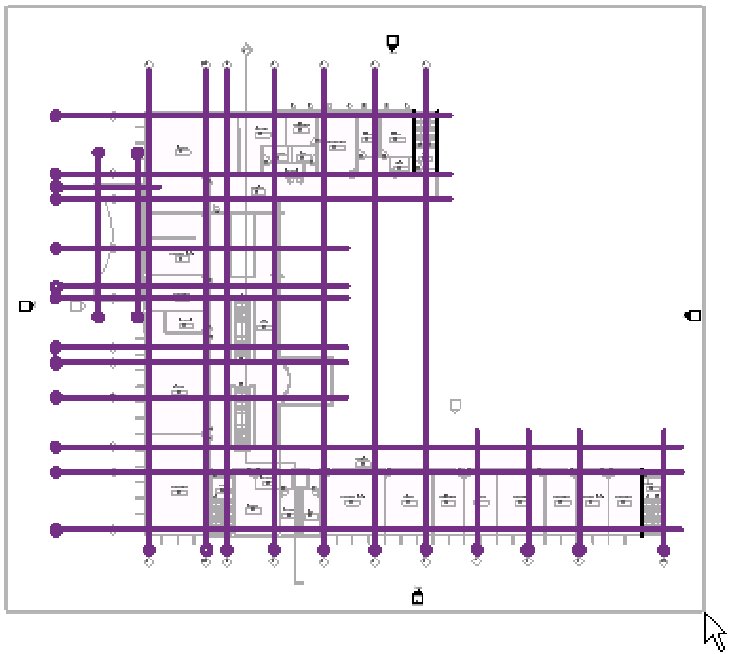


Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_BMS_01_Concrete_i.rvt.

Add concrete beams using the grid intersection tool

1. In the Project Browser, under Structural Plans, double-click 02 - Floor.
2. Enter **ZF** (Zoom to Fit).
3. Click Home tab ➤ Structure panel ➤ Beam.
4. Click Element panel ➤ Change Element Type drop-down ➤ Concrete-Rectangular Beam : 16 x 32.
5. Click Multiple panel ➤ On Grids.
6. Draw a selection box around the entire view to select all grid lines.



7 Click Multiple Selection panel ➤ Finish Selection.

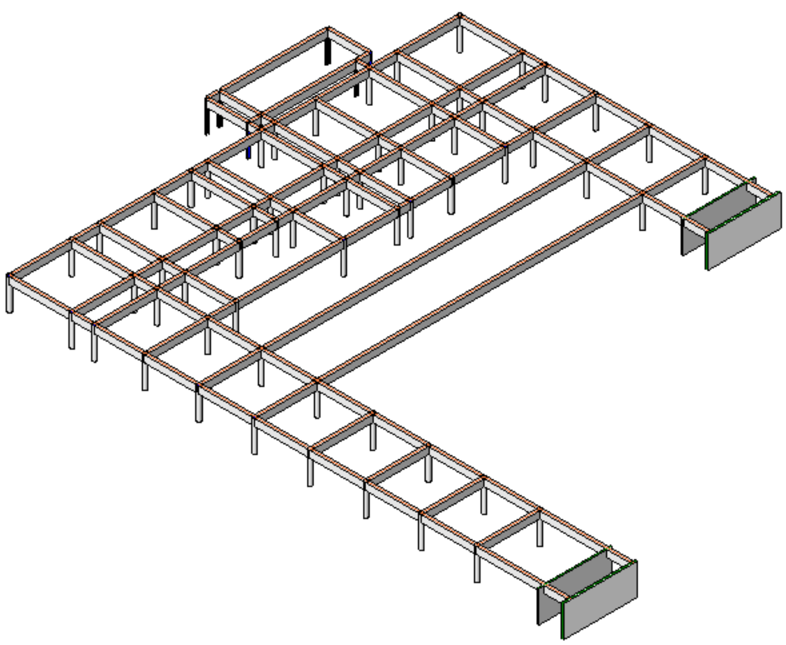
Beams are placed within the grid.

8 Click Selection panel ➤ Modify.

View the beams in 3D

9 Click View tab ➤ Create panel ➤ 3D View drop-down ➤ Default 3D.

10 Enter **ZF** (Zoom to Fit).

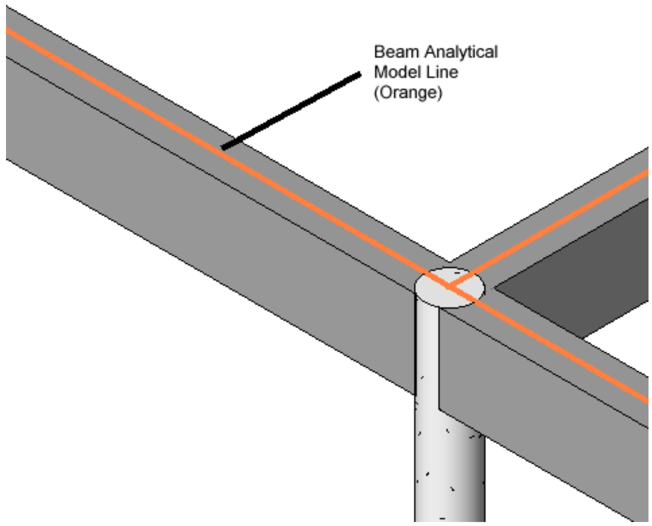


11 Press *Esc*.

View the beam analytical model

When you create the model in Revit Structure, a simplified 3D representation of each element is created automatically. This simplified view is referred to as the analytical model. Each element is drawn with a different color to help identify the specific analytical properties; for example, beams display as an orange line. You can export the analytical model to analysis and design applications.

12 Zoom in on one of the beams to see the orange line representing the analytical model.



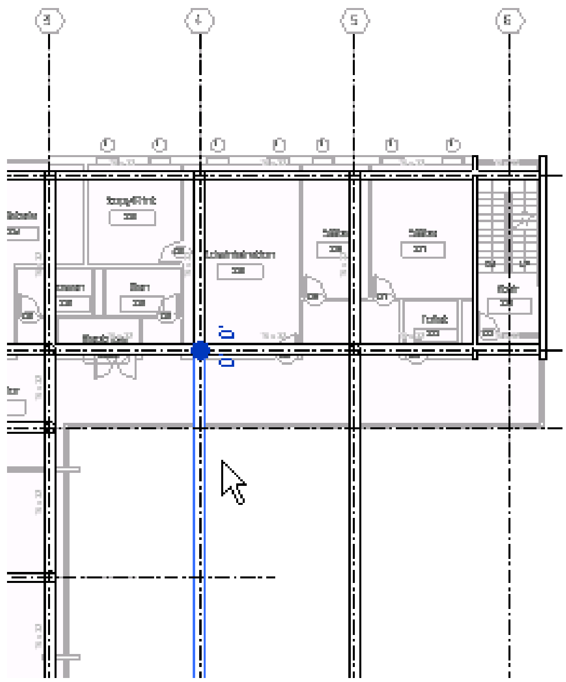
Delete the beams outside the drawing floor extents

Delete the two beams that were placed at grid locations outside of the floor extents of the architectural drawing.

13 In the Project Browser, under Structural Plans, double-click 02 - Floor.

14 Zoom in on the upper half of the structure.

15 Select the beam placed on grid line 4, as shown.



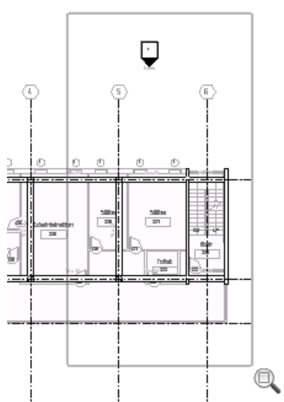
16 Press *Delete*.

17 Use the same method to delete the beam on grid line 5.

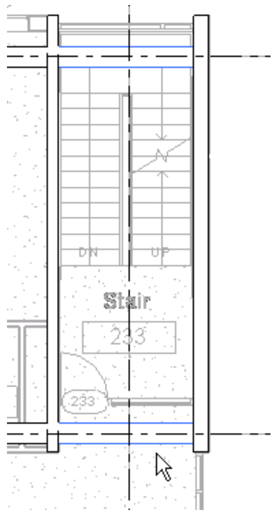
Delete beams placed inside the north and south stairways

Delete four beams that were placed at grid locations inside the structural walls of the north and south stairways.

18 Zoom in on the north stairway.



19 Select the beam placed on grid line A, press *Ctrl*, and select the beam placed on grid line B.



20 Press *Delete*.

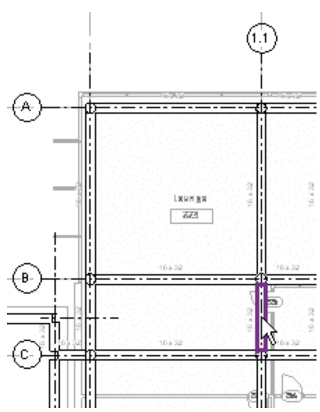
21 Using the same method, delete the beams placed inside the south stairway.

Create a new beam size

You create a new beam size to provide additional strength for a main support beam that runs the length of the structure.

22 Enter **ZF** (Zoom to Fit).

23 Select the concrete beam located on grid line 1.1, between grid line B and C as shown.



24 While pressing **CTRL**, select the remaining beams on grid line 1.1.

25 Click Element panel ➤ Element Properties drop-down ➤ Type Properties.

26 In the Type Properties dialog, click Duplicate.

27 In the Name dialog, type **24 x 36**, and click OK.

28 In the Type Properties dialog, under Dimensions:

- For b, type **2' 0**".
- For h, type **3' 0**".
- Click OK.

The new beam size is automatically applied to the selected beams.

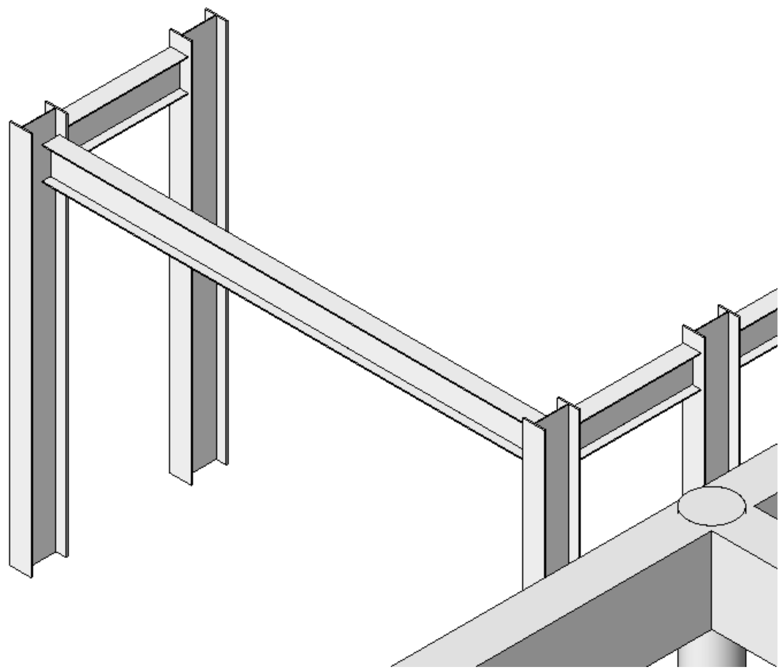
29 Press *Esc*.

30 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Adding Steel Beams

In this exercise, you add structural beams to the entry way of the structure. You also set the offset value of the steel beams to allow for the concrete slab that will be added in a later exercise.



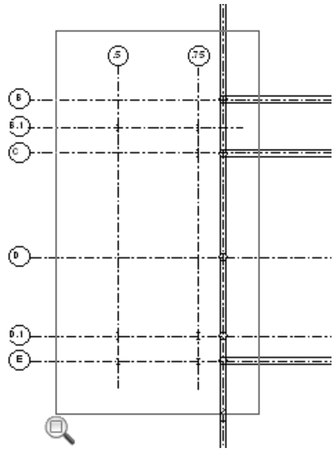
Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_BMS_02_Steel_i.rvt.

Add beams using the chain tool

1 In the Project Browser, under Structural Plans, double-click 02 - Floor.

2 Zoom in on the entry way of the structure.

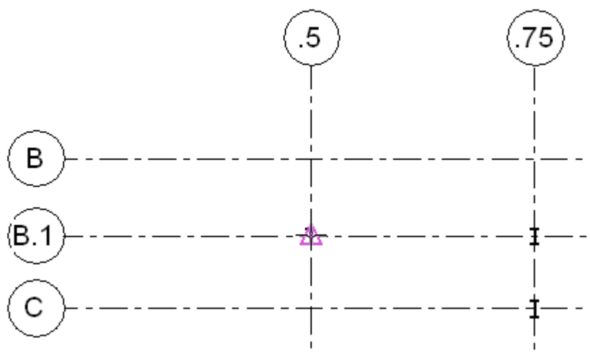


3 Click Home tab ➤ Structure panel ➤ Beam.

4 Click Element panel ➤ Change Element Type drop-down ➤ W-Wide Flange : W12X26.

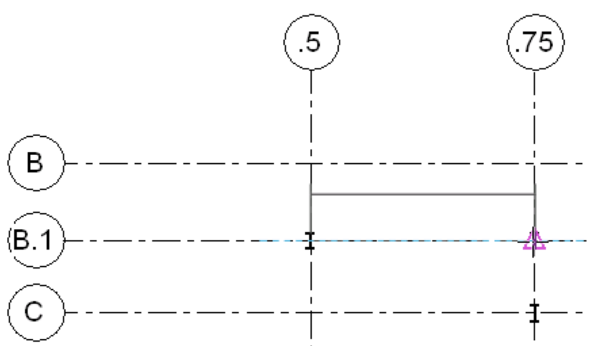
5 On the Options Bar, click Chain.

6 Move the cursor over the column at grid location B.1/.5 until a triangle displays, and then click to enter the beam start point.

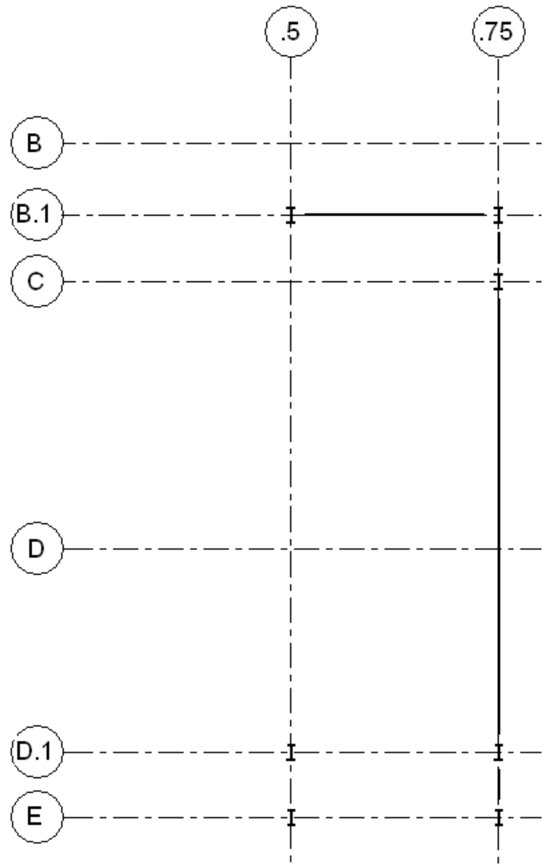


The triangle indicates the midpoint of the column.

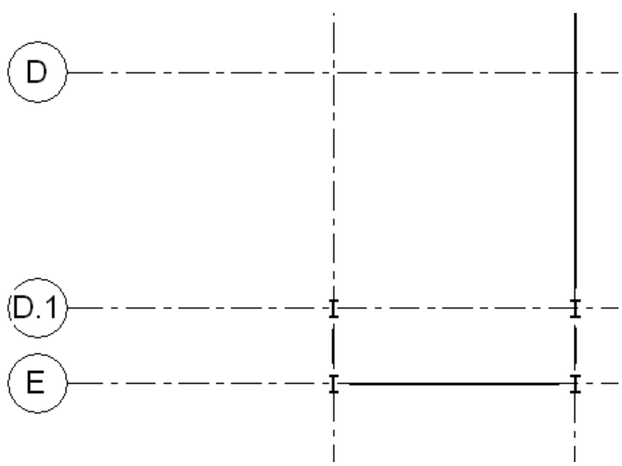
7 Click the midpoint of the column at grid location B.1/.75 to specify the beam endpoint.



8 Use the same method to place additional beams to frame the roof of the entry way of the structure. Click the midpoints of the columns (along grid line .75) at grid line C, D.1, and E.



9 Place the remaining beams by clicking the midpoint of the columns at grid locations E / .5 and D.1 / .5.

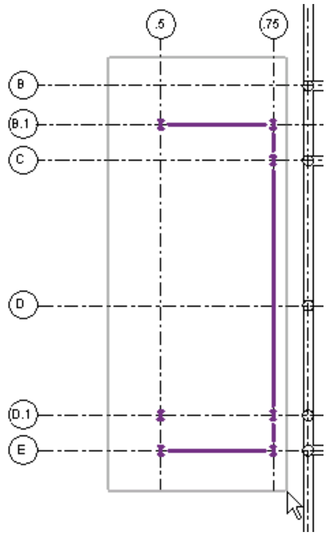


10 Click Selection panel ➤ Modify.

Set the beam offset value

Lower the offset value for all beams to accommodate the slab that you will add in a future exercise.

11 Draw a pick box around the entry way to select the columns and beams.



12 Click Filter panel ➤ Filter.

13 In the Filter dialog:

- Click Check None.
- Under Category, select Structural Framing (Girder).
- Click Apply, and then click OK.

Only the structural girders are highlighted.

14 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

15 In the Instance Properties dialog, under Constraints:

- For z-Direction Justification, select Other.
- For z-Direction Offset Value, type **-0' 5"**.
- Click OK.

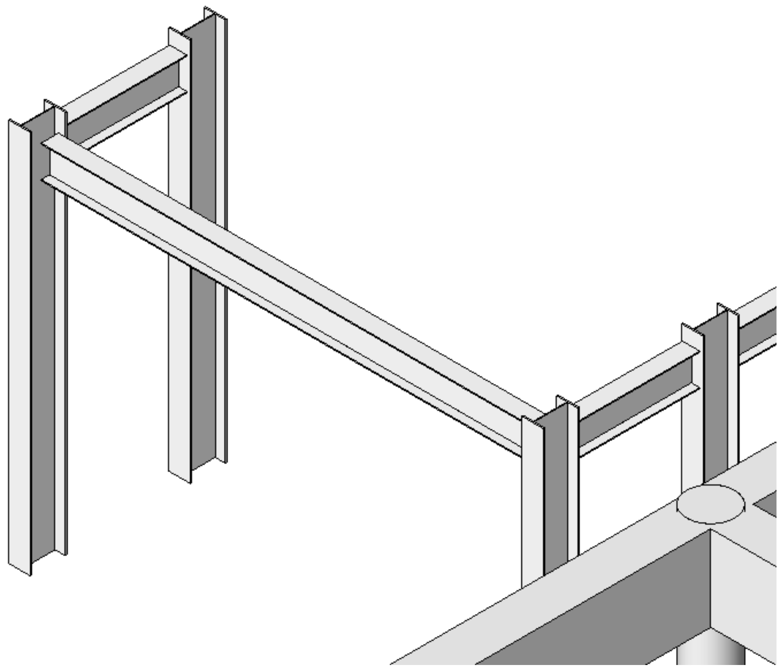
16 Press *Esc*.

View the beams in 3D

17 Click View tab ➤ Create panel ➤ 3D View drop-down ➤ Default 3D.

18 Zoom in on the entry way of the structure.

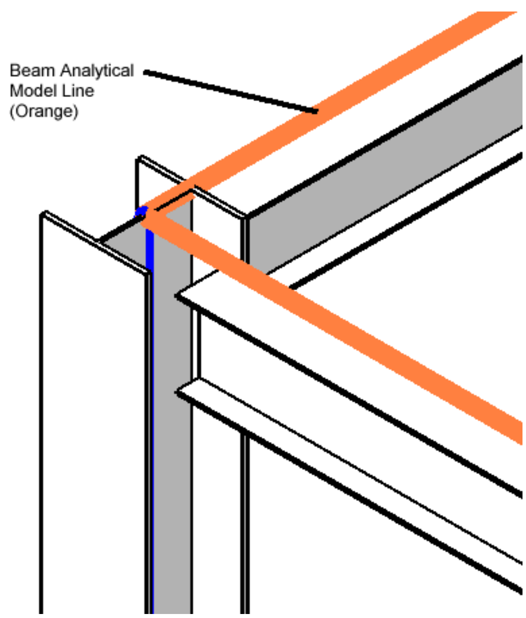
The beams have been lowered in the z-direction by the specified dimension.



View the steel beam analytical model

As you create the model in Revit Structure, a simplified 3D representation of each element is created automatically. This simplified view is referred to as the analytical model. Each element is drawn with a different color to help identify specific analytical properties; for example, beams display as an orange line. You can export the analytical model to analysis and design applications.

19 Zoom in on one of the steel beams to see the orange line representing the analytical model.



NOTE The model lines of the beam element do not appear to be joined to the structural columns. However, the analytical model lines of both elements are connected. The orange line of the beam analytical model is attached to the blue line representing the column analytical model.

20 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

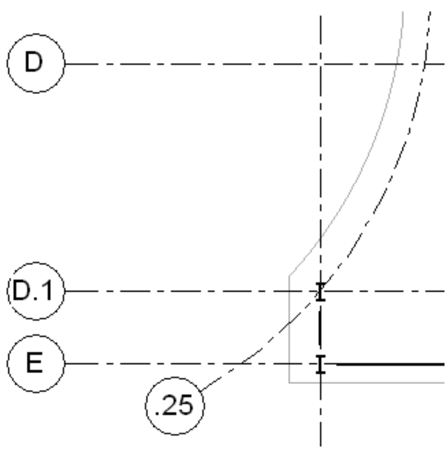
Adding Curved Beams

8

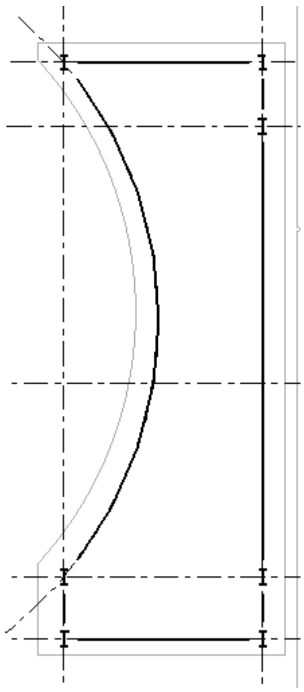
In this lesson, you add a curved beam to the steel frame of the entry way. Because the geometry of the curved beams can bend, you can draw it in both plan and elevation views. You learn to:

- Add a curved grid to the Level 2 plan view, providing snap points for placement of the curved beam.
- Add a curved steel beam to the frame of the entry way roof.

Curved grid added to the plan view



Curved beam added to the entry way roof frame



Adding a Curved Grid

In this exercise, you add a curved grid to the Level 2 plan view of the structure. The grid provides snap points for placement of the curved beam. Use the arc tool to draw the grid.

Training File

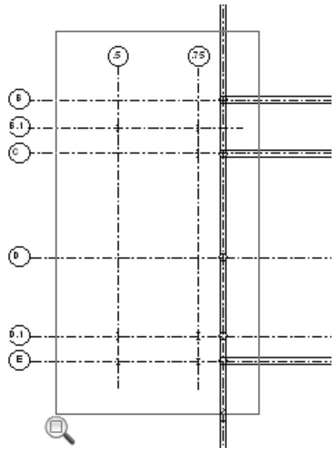
- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_CB_01_Add_Curved_Grid_i.rvt.

Draw a circular grid using the arc tool

1 In the Project Browser, under Structural Plans, double-click 02 - Floor.

2 Enter **ZF** (Zoom to Fit).

3 Zoom in on the entry way of the structure.

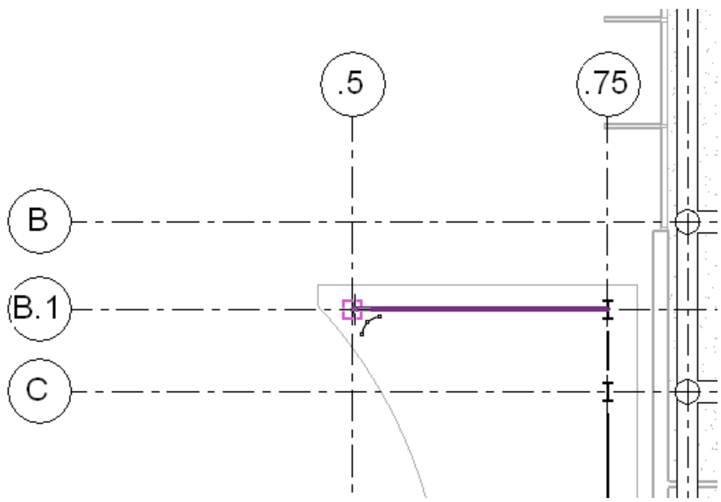


4 Click Home tab ➤ Datum panel ➤ Grid.

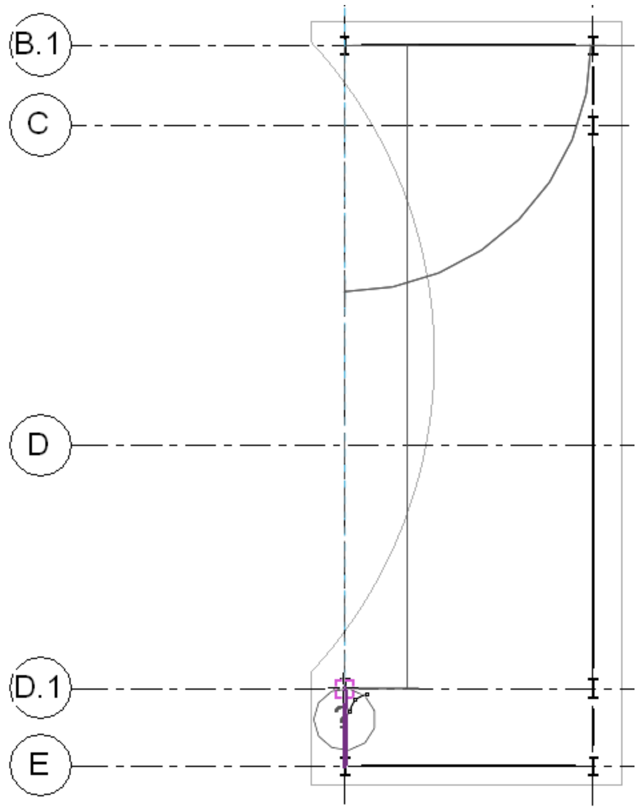
5 On the Draw panel, click (Start-End-Radius Arc).

6 On the Options Bar, click Radius, and type **32' 0"**.

7 Click the steel column at grid intersection B.1 / .5 to select the arc start point as shown.

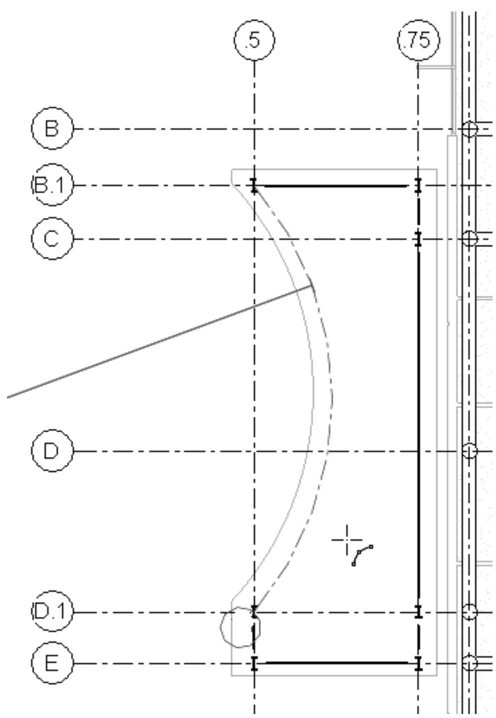


8 Click the steel column at grid intersection D.1 / .5 to select the arc endpoint.



The arc is positioned outside the boundaries of the entry way.

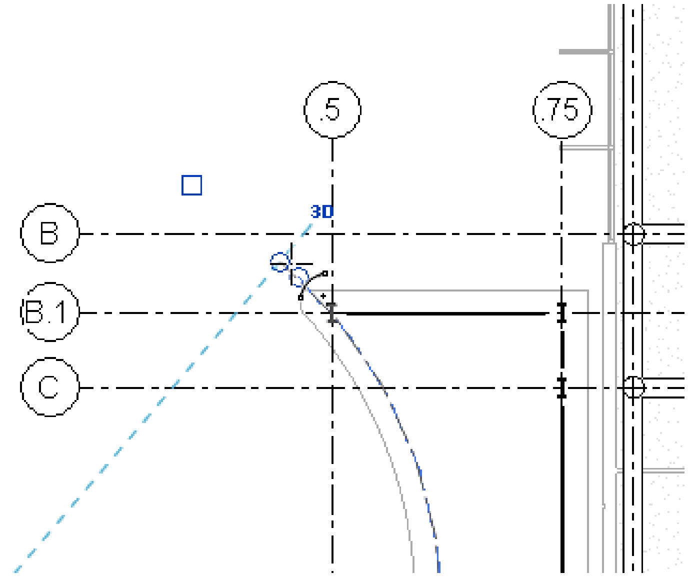
9 Click inside the entry way boundaries to place the circular grid.



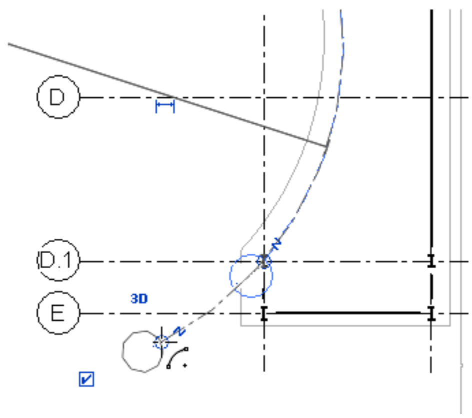
10 Click Selection panel ➤ Modify.

Extend grid

11 Click and drag the arc start point approximately as shown.



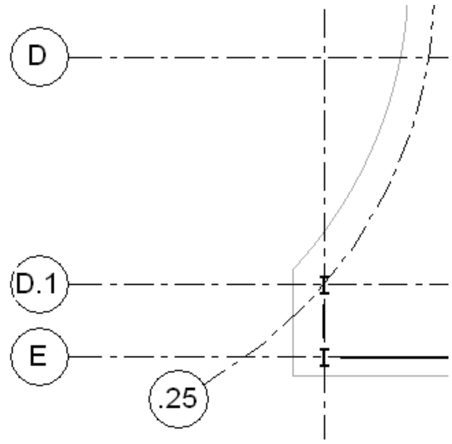
12 Click and drag the arc endpoint approximately as shown.



Rename the circular grid

13 Press *Esc*.

14 Click the grid balloon, and type **.25**.



15 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Adding Curved Beams

In this exercise, you add a curved beam to the entry way of the structure. The curved beam snaps to the circular grid placed in a previous exercise.

Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_CB_02_Add_Curved_Beam_i.rvt.

Zoom in on the entry way

1 In the Project Browser, under Structural Plans, double-click 02 - Floor.

2 Zoom in on the entry way.

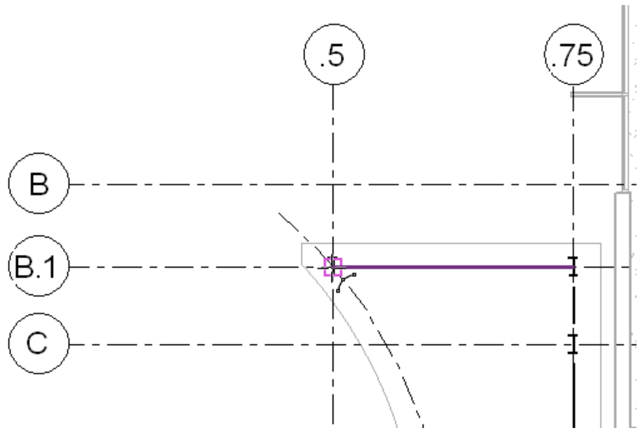
Place the curved beam

3 Click Home tab ➤ Structure panel ➤ Beam.

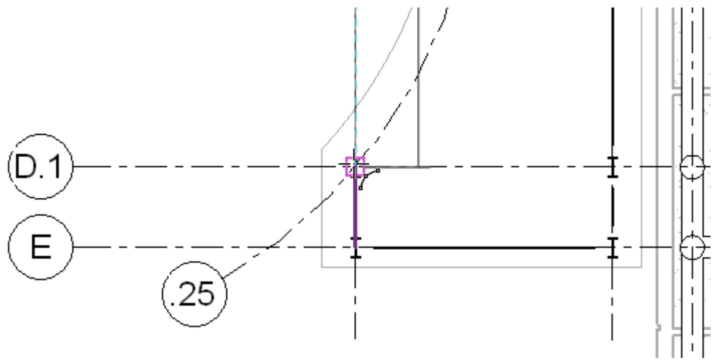
4 Click Element panel ➤ Change Element Type drop-down, verify that W-Wide Flange: W12X26 is selected.

5 On the Options Bar, select Chain, and click  (Start-End-Radius Arc).

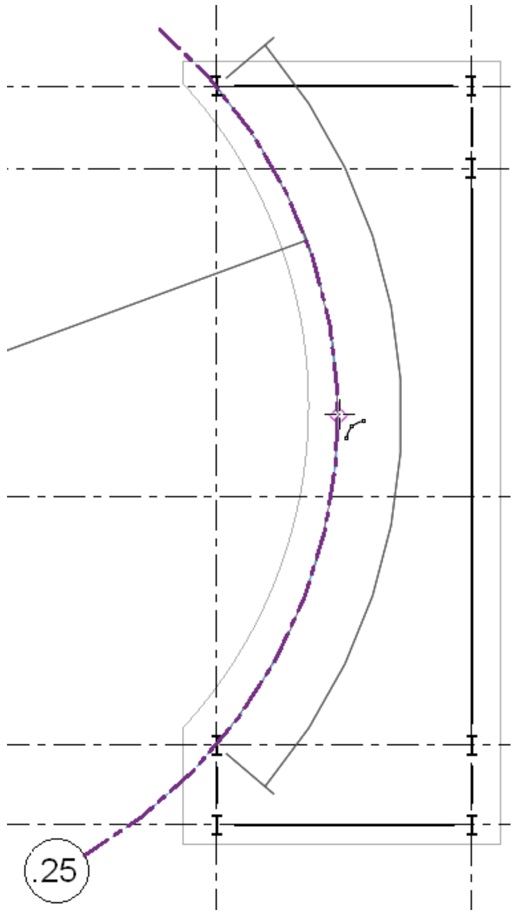
6 Click the steel column at grid intersection B.1 / .5 to select the beam start point.



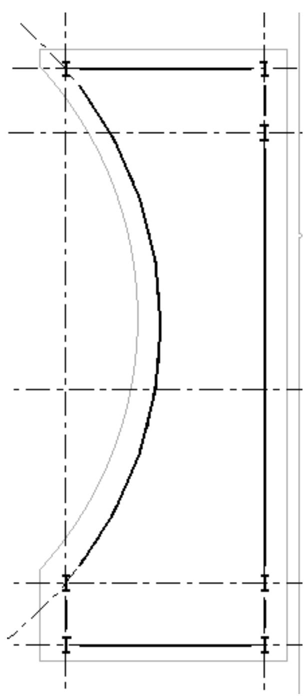
7 Click the steel column at grid intersection D.1 / .5 to select the beam end point.



8 Click the curved grid line, snapping the beam to the grid.



The curved beam is placed.



9 Press *Esc* twice.

Set the curved beam properties

10 Select the curved beam.

11 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

12 In the Instance Properties dialog:

- Under Constraints, for z-Direction Justification, select Other.
- Under Constraints, for z-Direction Offset Value, type **-0' 5"**.
- Click OK.

NOTE The z-Direction refers to the third axis (the z-axis) in the Revit Structure 3-dimensional coordinate system.

---

13 Press *Esc* twice.

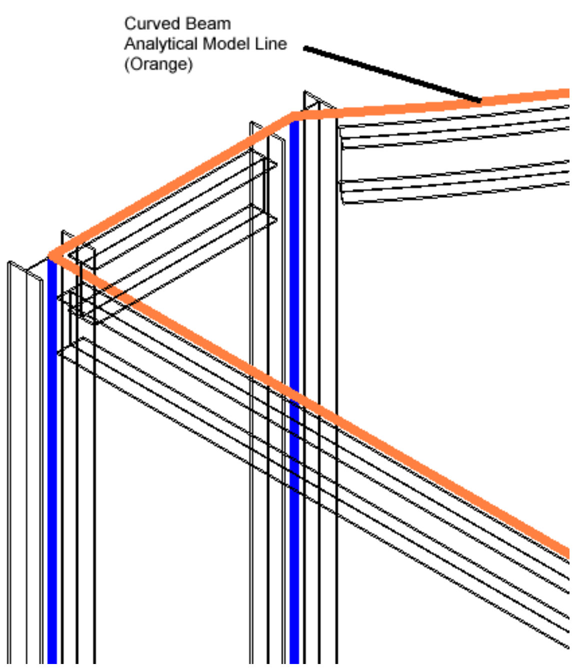
View the curved beam analytical model

As you create the model in Revit Structure, a simplified 3D representation of each element is created automatically. This simplified view is referred to as the analytical model. Each element is drawn with a different color to help identify specific analytical properties; for example, beams display as an orange line. The analytical model can be exported to analysis and design applications.

14 In the Project Browser, under 3D Views, double-click 3D.

15 On the View Control Bar, click Model Graphics Style: Wireframe.

16 Zoom in on the curved beam to see the orange line representing the analytical model.



NOTE The model lines of the curved beam element do not appear to be joined to the structural columns. However, the analytical model lines of both elements are connected. The orange line of the curved beam analytical model is attached to the blue line representing the column analytical model.

Approximate the curve of the analytical model

Curved beams have an analytical model that can be smooth or segmented. If the analytical model is segmented, you can adjust the number and location of the segments for more accurate analysis.

17 Select the curved beam.

18 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

19 In the Instance Properties dialog:

- Under Analytical Model, select Approximate Curve.
- For Maximum discretized offset, type **0' 4"**.
- Select Use hard-points.
- Click OK.

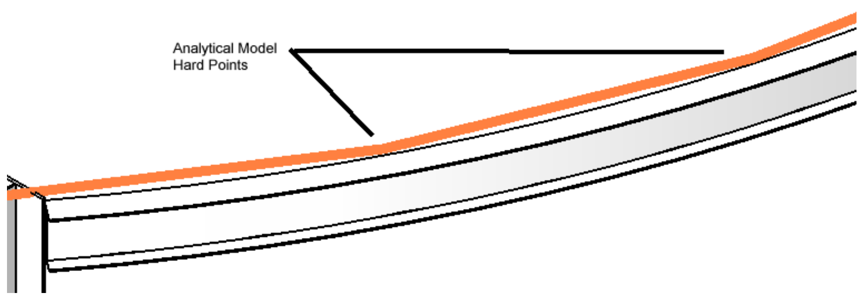
NOTE When you select Use hard-points, the analytical model will end at points on the curve where other framing members are joined. Additional line segments will be added to satisfy the Maximum discretized offset parameter.

---

20 Press *Esc*.

21 Use the ViewCube to rotate the 3D view until the curved beam is visible.

The analytical line of the curved beam is represented by segments, not by a single curved line.



22 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

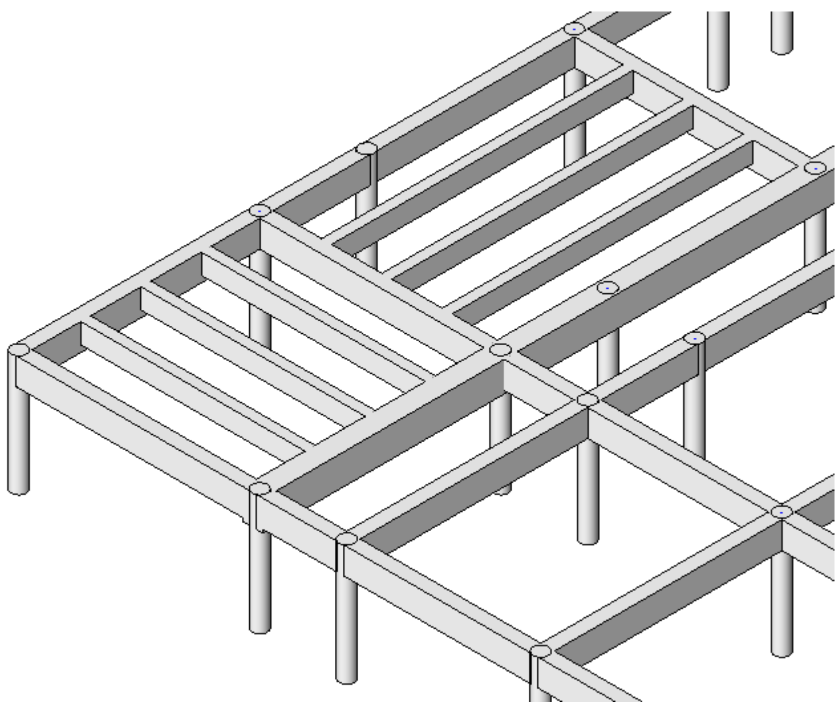
Adding Beam Systems

9

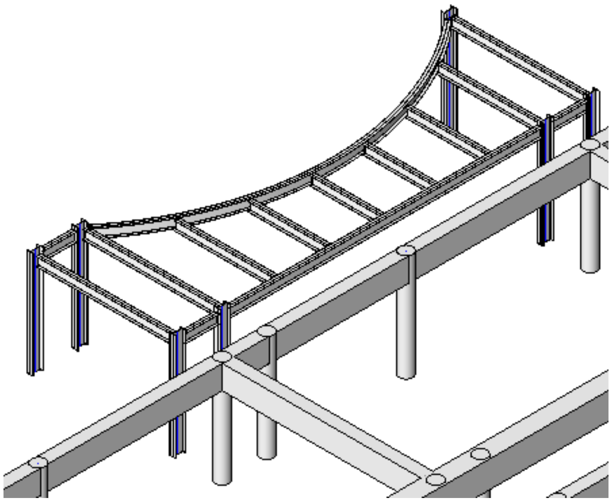
In this lesson, you add both steel and concrete beam systems to the structure using the tools available in Revit Structure. Structural beam systems create a single structural framing element that contains a series of individual beams placed in parallel. There are 2 ways to create a beam system, using the one-click method or trace mode. You learn to:

- Add 2 concrete beam systems by using the one-click method.
- Draw a steel beam system by tracing the outline of the entry way frame.

Concrete beam system

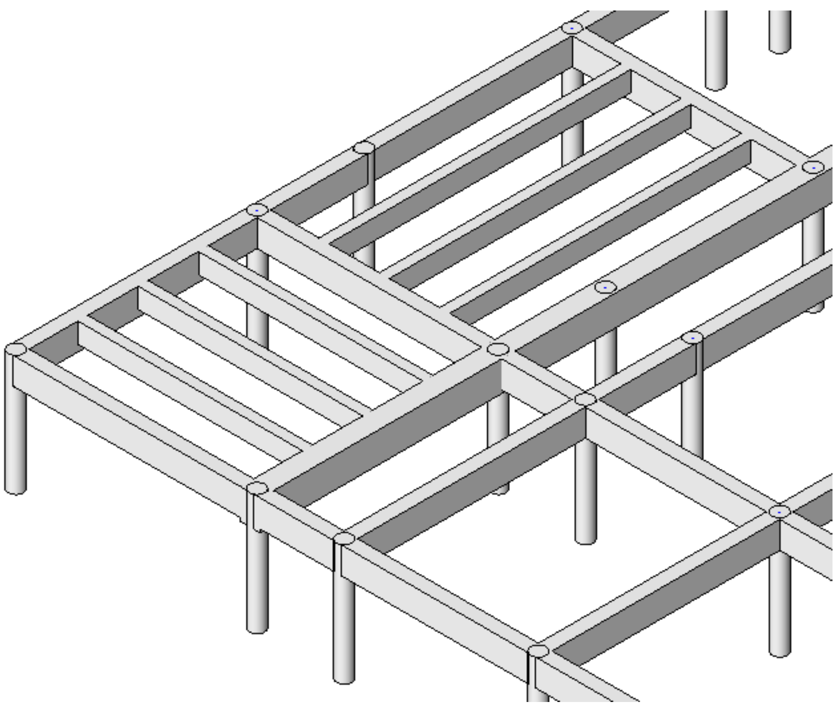


Steel beam system



Automatically Place Beam Systems

In this exercise, you add concrete beam systems to the frame of the structure.



Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_BS_01_Auto_Placement_i.rvt.

Automatically create multiple beam systems

1 In the Project Browser, under Structural Plans, double-click 02 - Floor.

2 Zoom in on the southwest corner of the structure.

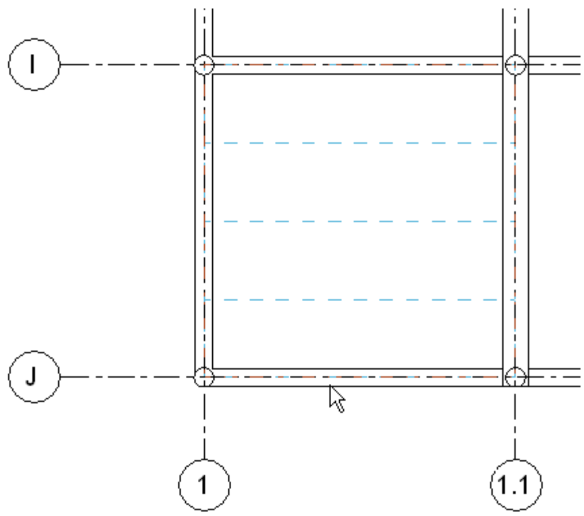
3 Click Home tab ➤ Structure panel ➤ Beam System.

4 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

5 In the Instance Properties dialog:

- Under Pattern, for Layout Rule, select Maximum Spacing.
- For Beam Type, select Concrete-Rectangular Beam : 12 x 24.
- Click OK.

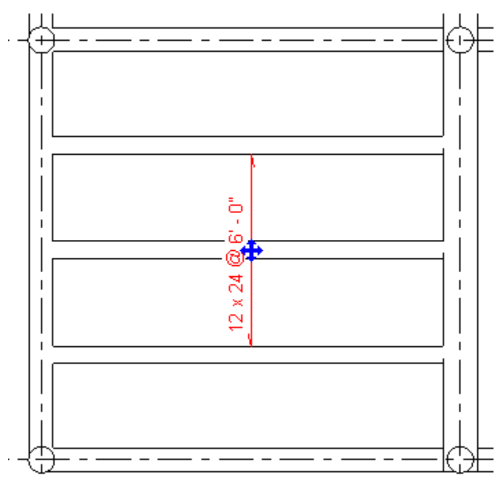
6 Select the beam located between grid line J1 and J1.1 to set the beam system direction.



The beam system location displays with multiple blue dashed lines.

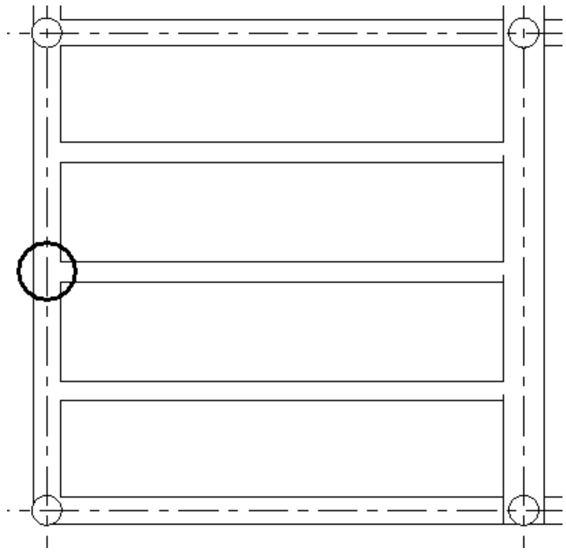
7 Click to place the beam system.

Notice the beam system tag displays on the plan view. This tag indicates the beam type and maximum spacing parameters.

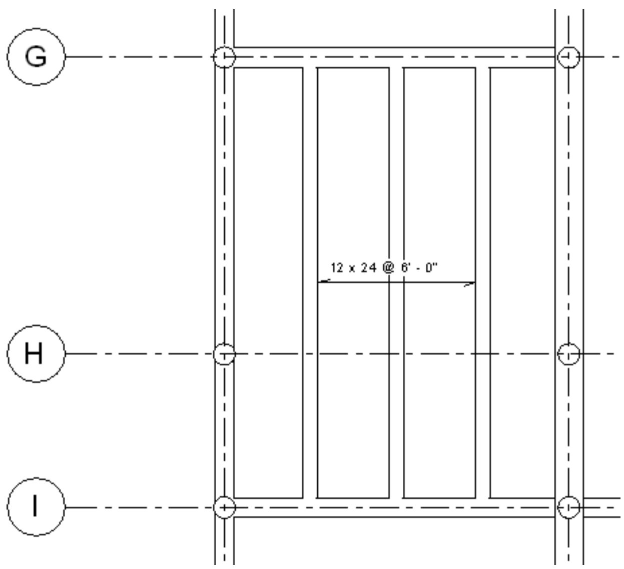


Also, notice that the concrete beam system is now joined to the concrete frame.

Beam system joined with concrete frame



8 Use the same method, select the beam between grids G and I to specify the beam direction, and place a beam system in the bay directly above the previous bay.

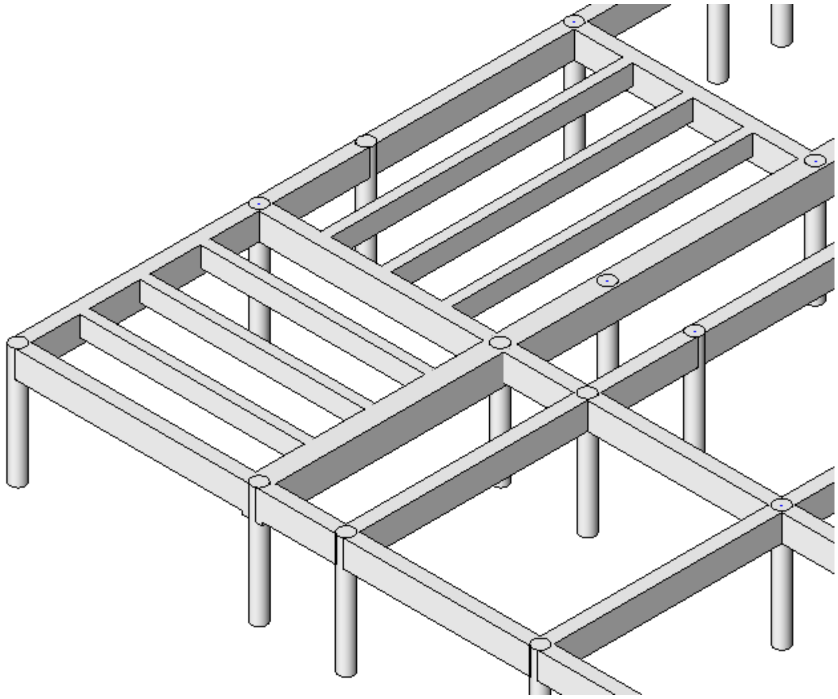


9 Press *Esc*.

View the beam system in 3D

10 In the Project Browser, under 3D Views, double-click 3D.

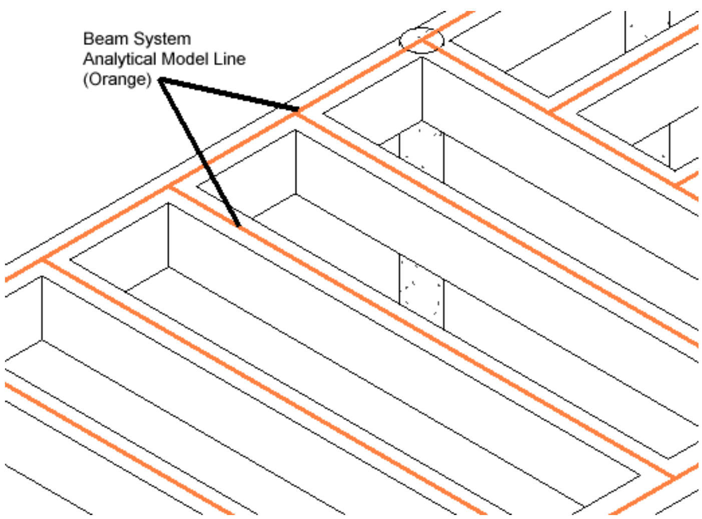
11 Zoom in on the southwest corner of the structure.



View the beam system analytical model

As you create the model in Revit Structure, a simplified 3D representation of each element is created automatically. This simplified view is referred to as the analytical model. Each element is drawn with a different color to help identify specific analytical properties; for example, beam systems display as orange lines. You can export the analytical model to analysis and design applications.

12 Zoom in on the beam system to see the orange lines representing the analytical model.

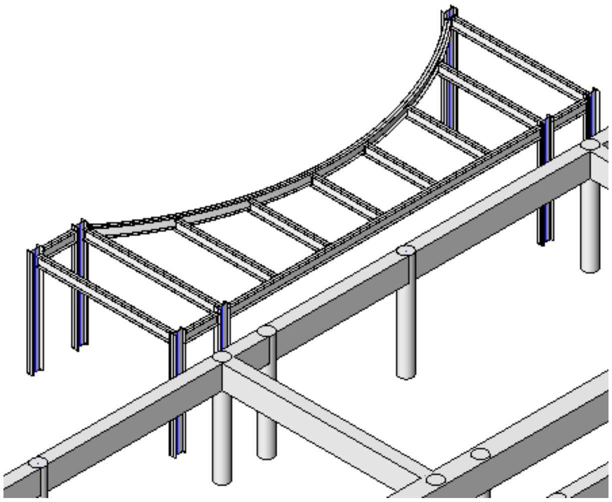


13 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Sketching a Beam System

In this exercise, you sketch a beam system to frame the entry way roof.



Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_BS_02_Sketch_i.rvt.

Zoom in on the entry way

1 In the Project Browser, under Structural Plans, double-click 02 - Floor.
2 Zoom in on the entry way of the structure.

Set the beam system properties

3 Click Home tab ➤ Structure panel ➤ Beam System.
4 Click Sketch panel ➤ Create Sketch.
You are now in sketch mode.

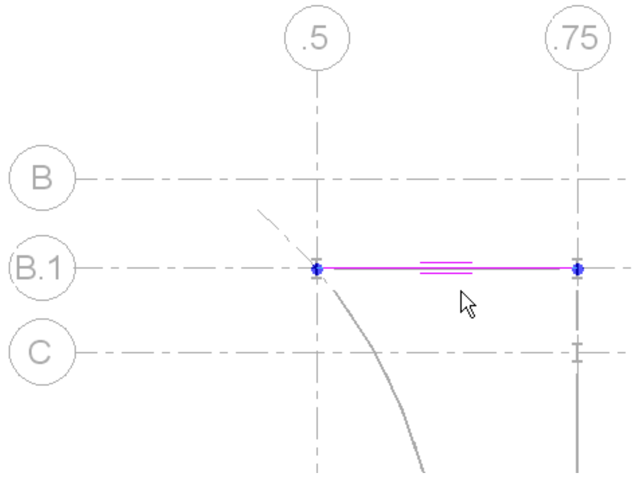
5 Click Create Beam System Boundary tab ➤ Element panel ➤ Beam System Properties.
6 In the Instance Properties dialog:

- Under Constraints, type **-0' 5"** for Elevation.
- Under Pattern, select Fixed Distance for Layout Rule.
- For Fixed Spacing, type **6' 0"**.
- For Justification, select Center.
- For Beam Type, select W-Wide Flange : W12X26.

7 Click OK.

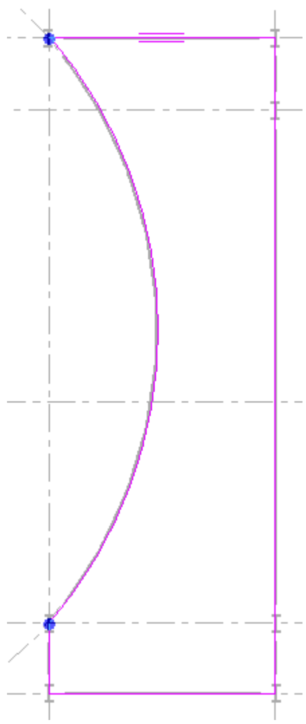
Sketch the beam system

8 Click Draw panel ➤ (Pick Supports).
9 Select the beam placed between grids B.1 / .5 and B.1 / .75.

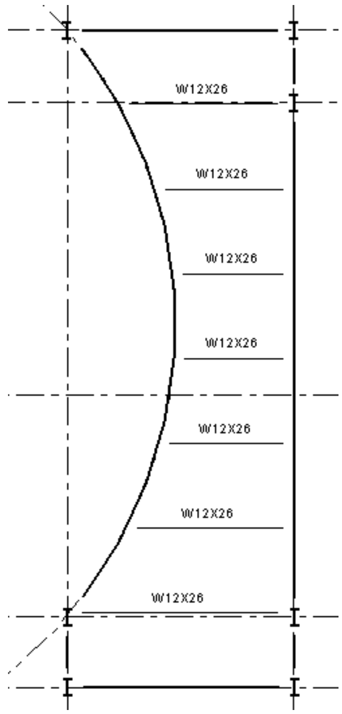


NOTE The 2 short lines adjacent to the beam represent the beam system direction. The longitudinal axis of the beam system members will be placed parallel to these lines.

10 Click the remaining beams that form the perimeter of the roof frame as shown.



11 Click Beam System panel ➤ Finish Beam System.



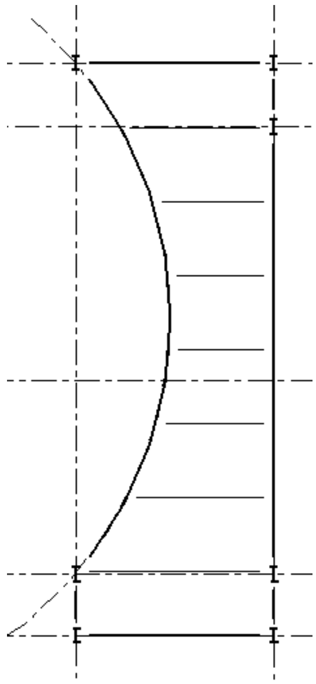
12 Press *Esc*.

Turn off framing tag visibility

13 Enter **VV** (Visibility/Graphic Overrides).

14 In the Visibility/Graphic Overrides dialog:

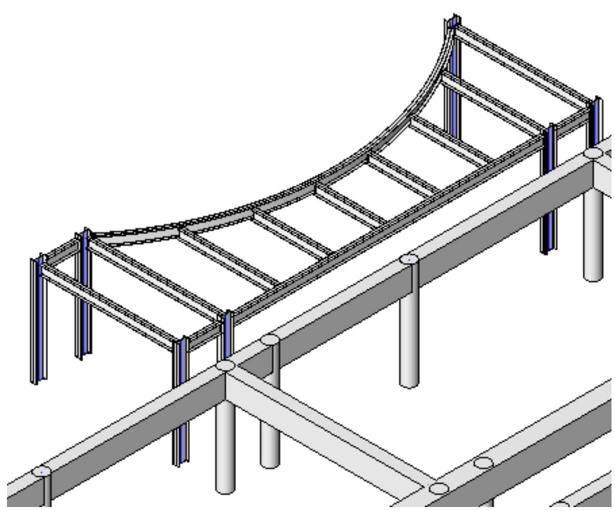
- Click the Annotation Categories tab.
- Under Visibility, clear the option for Structural Framing Tags.
- Click Apply, and then OK.



View the beam system in 3D

15 In the Project Browser, under 3D Views, double-click 3D.

16 Zoom in on the entry way of the structure.



View the beam system analytical model

As you create the model in Revit Structure, a simplified 3D representation of each element is created automatically. This simplified view is referred to as the analytical model. Each element is drawn with a different color to help identify specific analytical properties; for example, beam systems display as orange lines. You can export the analytical model to analysis and design applications.

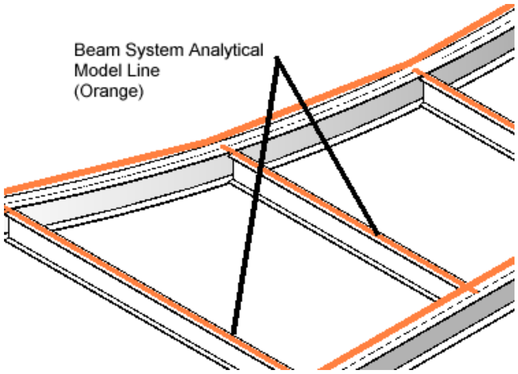
17 Enter **VV** (Visibility/Graphic Overrides).

18 In the Visibility/Graphic Overrides dialog:

- Click the Model Categories tab.

- Under Visibility, expand Structural Framing, and select Analytical Model.
- Click Apply, and then OK.

19 Zoom in on the beam system to see the orange lines representing the analytical model.



NOTE The model lines of the beam system element do not appear to be joined to the existing beams. However, the analytical model lines of the elements are connected. The orange line of the beam system analytical model is attached to the orange line representing the beam analytical model.

---

20 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

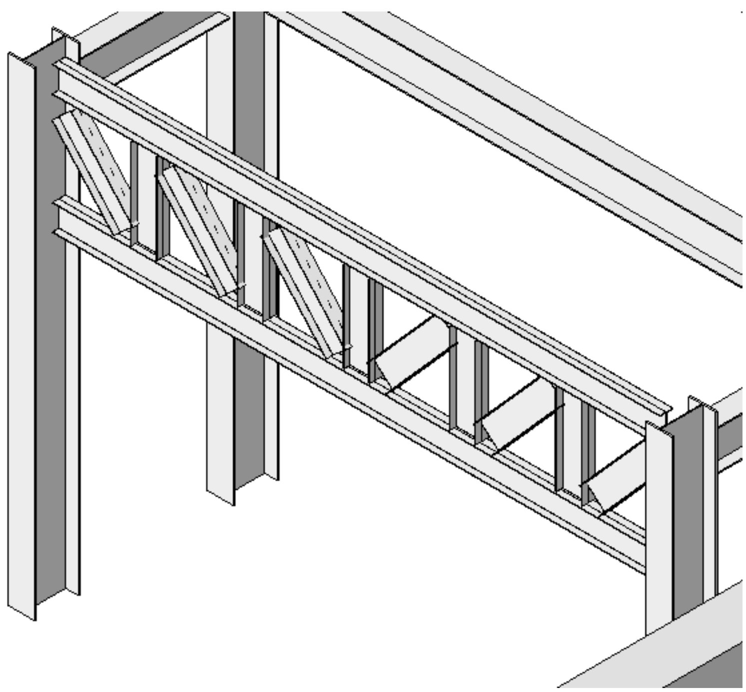
Adding Trusses

10

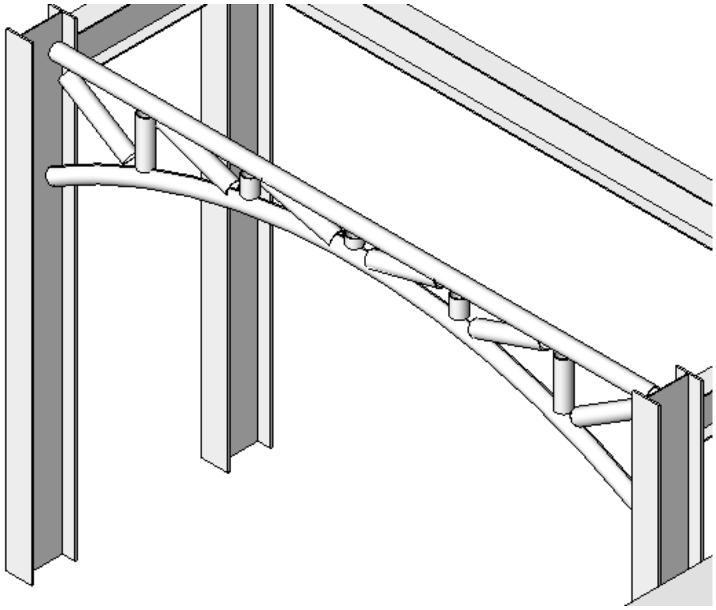
In this lesson, you add a steel truss to the entry way frame. The Revit Structure 2010 Truss tool creates a truss based on the parameters specified in the truss family type selected. Reference lines, included in the truss layout, determine the placement of the sub-elements of the truss element. Sub-elements include the top chord, bottom chord, and web members. You learn to:

- Add a steel truss to span the width of the entry way.
- Customize the truss profile by changing the top and bottom chord type, the vertical and diagonal web type, and the shape of the bottom chord.

Standard truss



Customized truss



Adding a Steel Truss

In this exercise, you add a standard pratt truss to the steel frame of the entry way roof. You draw the truss in the plan view by selecting the columns located on each side of the roof span.

Training File

- Click  ➤ Open.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_TRS_01_Add_Truss_i.rvt.

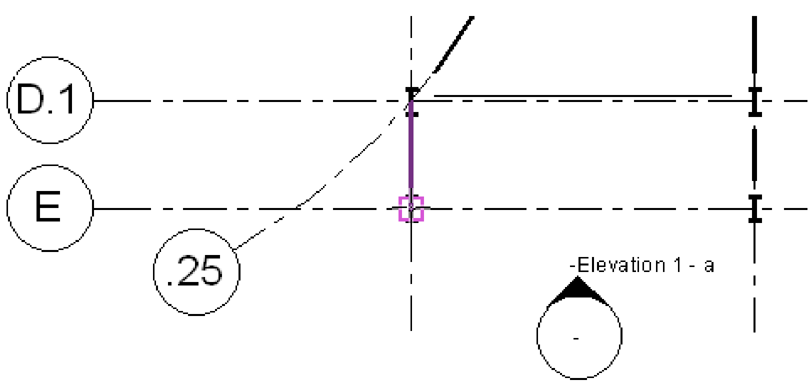
Defining the span

1 In the Project Browser, under Structural Plans, double-click 02 - Floor.

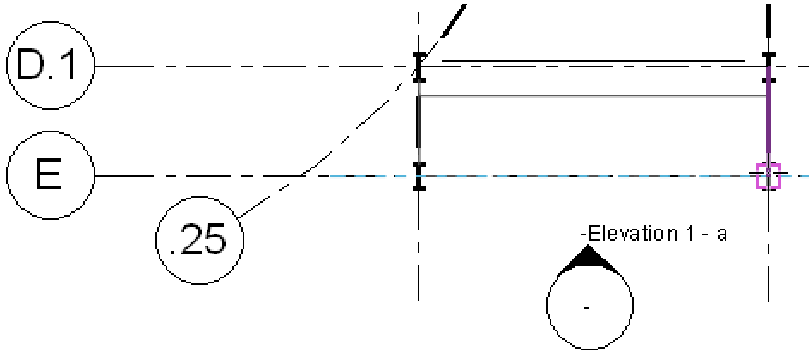
2 Click Home tab ➤ Structure panel ➤ Truss.

3 Click Element panel ➤ Change Element Type drop-down ➤ Pratt Flat Truss: Standard.

4 Click the column on grid intersection E / .5 to select the startpoint of the truss span.



5 Click the column on grid intersection E / .75 to select the end point of the truss span.



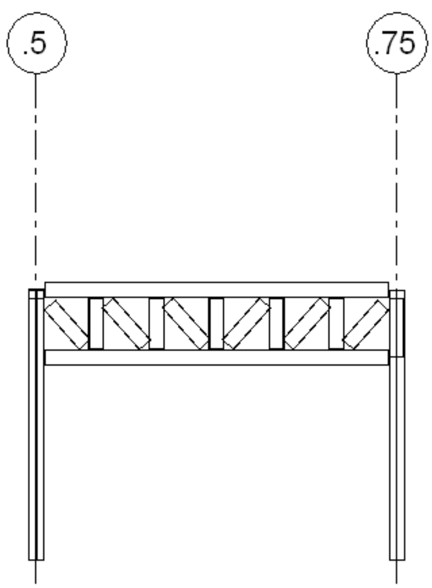
6 Select the truss, and click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

7 In the Instance Properties dialog:

- Under Constraints, for Start Level Offset and End Level Offset, type **-3' 0"**.
- Under Structural, for Bearing Chord, select Bottom.
- Under Dimensions, for Truss Height, type **3' 0"**.
- Click OK.

8 In the Project Browser, under Elevations (Framing Elevation), double-click Elevation 1 -a.

The truss is placed above the columns.



NOTE The standard pratt truss family includes 2 vertical webs that are not necessary for this span; you will delete them in the next section. Also, you will change the structural framing type for the truss chords (top and bottom) and webs (vertical and diagonal) in the next exercise.

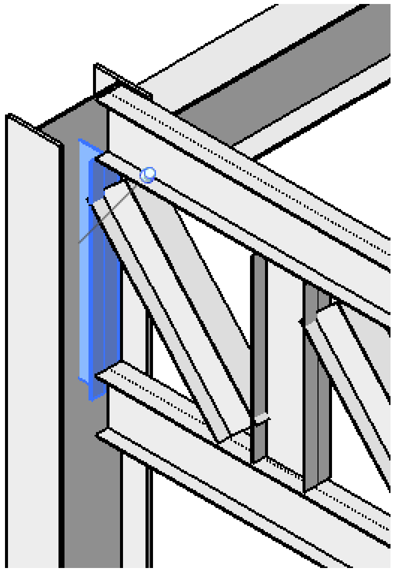
9 Press *Esc*.

Delete vertical webs

10 In the Project Browser, under 3D Views, double-click 3D.

11 Zoom in on the truss.

12 Click the left vertical web.



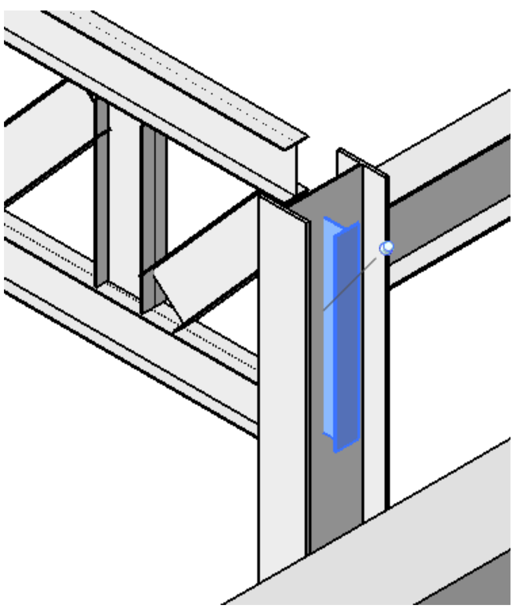
NOTE Check the Status Bar to make sure you select the left vertical web and not a truss chord or column.

---

13 Press *Delete*.

Close the Warning dialog that indicates the pinned object will be deleted.

14 Using the same method, delete the right vertical web on the opposite side of the truss.



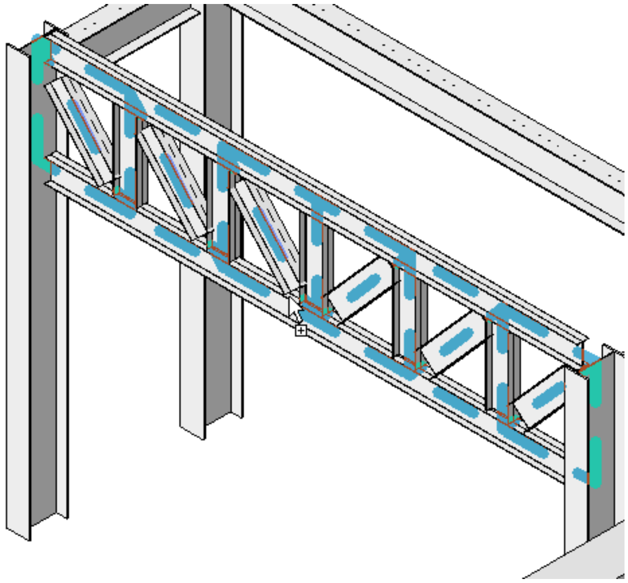
Change the truss level offsets

You change the level offset in order to lower the top level of the truss to the same level as the other beams in the entry way frame.

15 Select the truss.

NOTE Check the Status Bar to make sure you select the entire truss and not just the left vertical web or truss chord.

The truss element is represented by a dotted blue line.



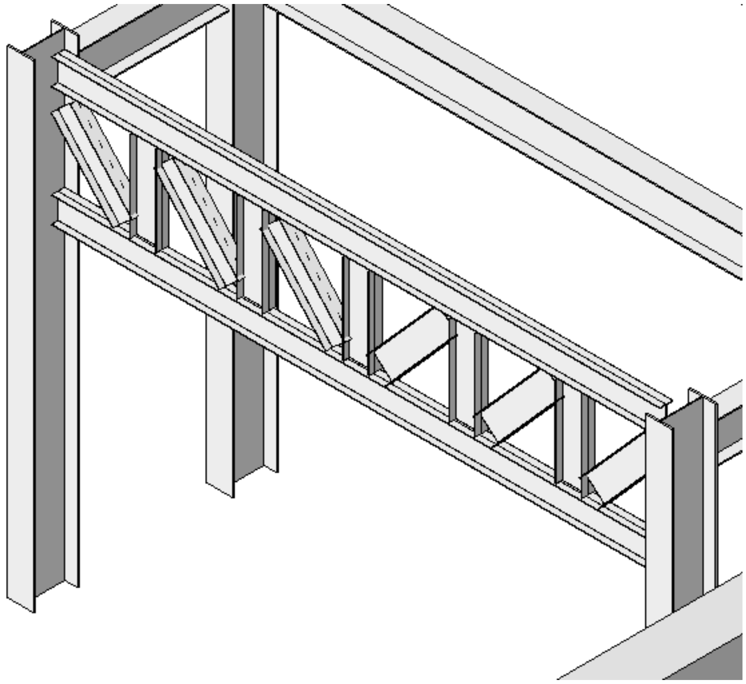
16 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

17 In the Instance Properties dialog:

- Under Constraints, for Start Level Offset, type **-3' 5"**.
- Under Constraints, for End Level Offset, type **-3' 5"**.
- Click OK.

18 Press *Esc*.

The truss is complete.



View the truss analytical model

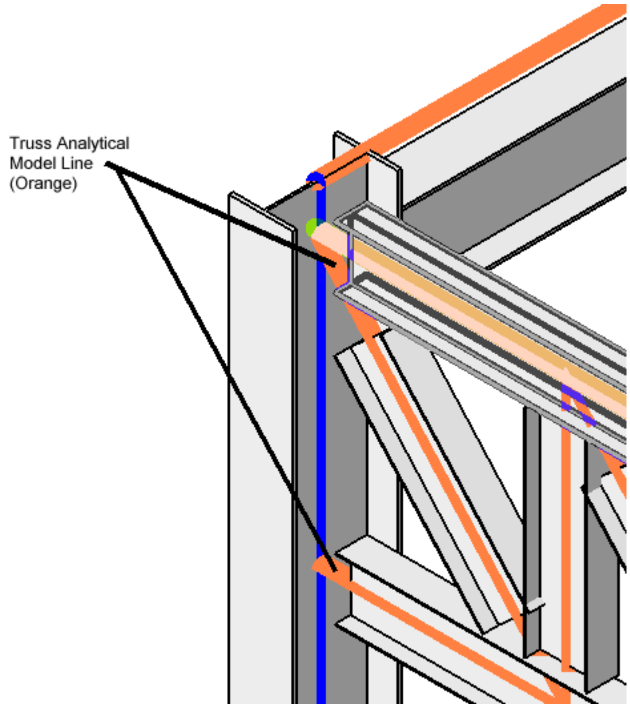
As you create the model in Revit Structure, a simplified 3D representation of each element is created automatically. This simplified view is referred to as the analytical model. Each element is drawn with a different color to help identify specific analytical properties; for example, trusses display as orange lines. You can export the analytical model to analysis and design applications.

19 Click View tab ➤ Graphics panel ➤ Visibility/Graphics.

20 In the Visibility/Graphics dialog:

- Click the Model Categories tab.
- Under Visibility, expand Structural Framing, and select Analytical Model .
- Under Visibility, expand Structural Columns and select Analytical Model .
- Click Apply, and then OK.

21 Zoom in on the truss to see the orange lines representing the analytical model of the truss.



NOTE The model lines of the truss element do not appear to be joined to the model lines of the structural columns. However, the analytical model lines of both elements are connected. The orange lines of the truss analytical model are attached to the blue line representing the column analytical model.

22 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Customizing Truss Parameters

In this exercise, you create a custom truss by changing the properties of the standard truss, creating an arc for the bottom chord. Then, you save the custom truss to the framing library in Revit Structure. You also copy the truss to the opposite side of the entry way roof frame.

Training File

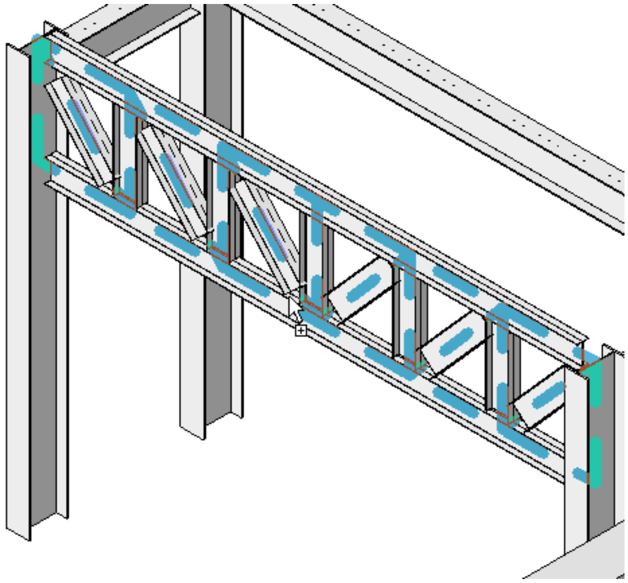
- Click File menu ➤ Open.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_TRS_02_Customize_Truss_i.rvt.

Create a new truss type for the entry way span

1 In the Project Browser, under 3D Views, double-click 3D.

2 Zoom in on the truss and select it.

The truss element is represented by a dotted blue line.



3 Click Element panel ➤ Element Properties drop-down ➤ Type Properties.

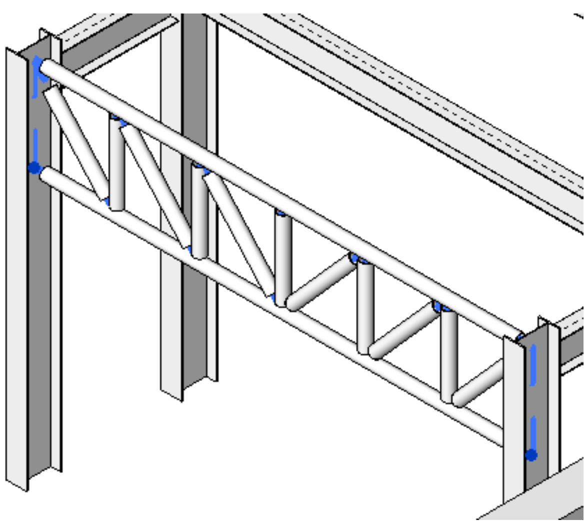
4 In the Type Properties dialog, click Duplicate.

5 In the Name dialog, type **Pratt Flat Truss - Short Span**, and click OK.

6 In the Type Properties dialog:

- Under Top Chords, for Structural Framing Type, select HSS-Round Structural Tubing:HSS4.500X0.337.
- Under Vertical Webs, for Structural Framing Type, select HSS-Round Structural Tubing:HSS4.500X0.337.
- Under Diagonal Webs, for Structural Framing Type, select HSS-Round Structural Tubing:HSS4.500X0.337.
- Under Bottom Chords, for Structural Framing Type, select HSS-Round Structural Tubing:HSS4.500X0.337.
- Click OK.

The truss framing elements reflect the specified changes. Also, the new truss type appears on the Element panel.



7 Press *Esc* twice.

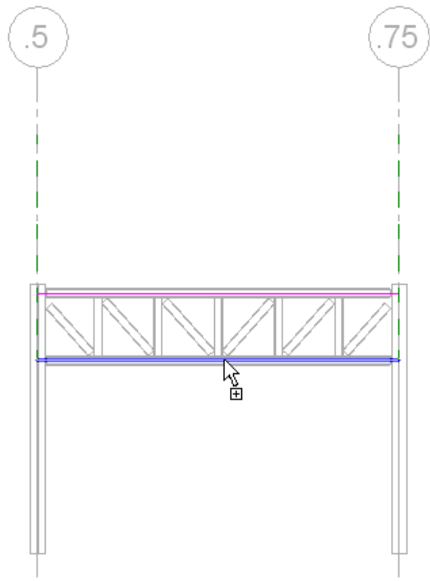
Change the shape of the bottom chord

8 In the Project Browser, under Elevations (Framing Elevation), double-click Elevation 1 -a.

9 Select the truss, and click Modify Truss panel ➤ Edit Profile.

You are now in sketch mode.

10 Select the model line for the bottom chord.

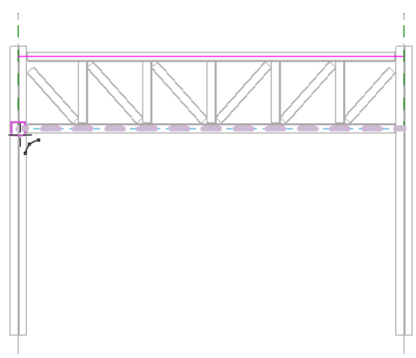


11 Press *Delete*.

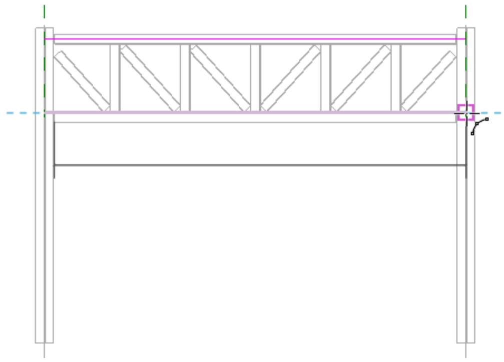
12 Click Draw Panel ➤ Bottom Chord.

13 On the Draw panel, click (Start-End-Radius Arc).

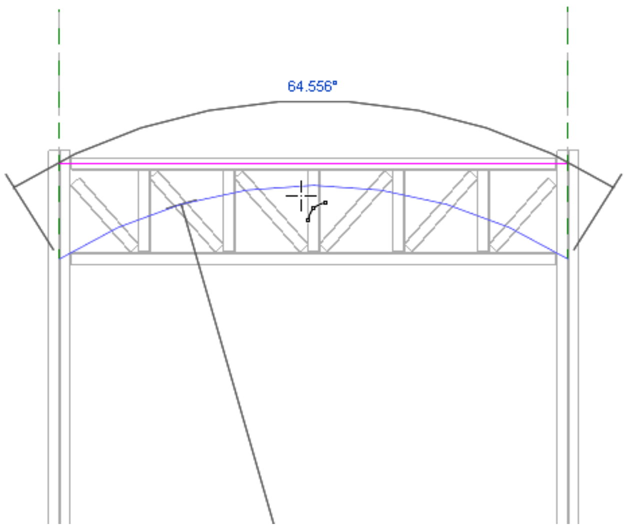
14 Click the column located at the intersection of grid line .5 and the bottom chord of the truss, to specify the startpoint for the arc.



15 Click the column on grid line .75 to specify the end point for the arc.



16 On the Options Bar, click Radius, and type **15' 0"**.

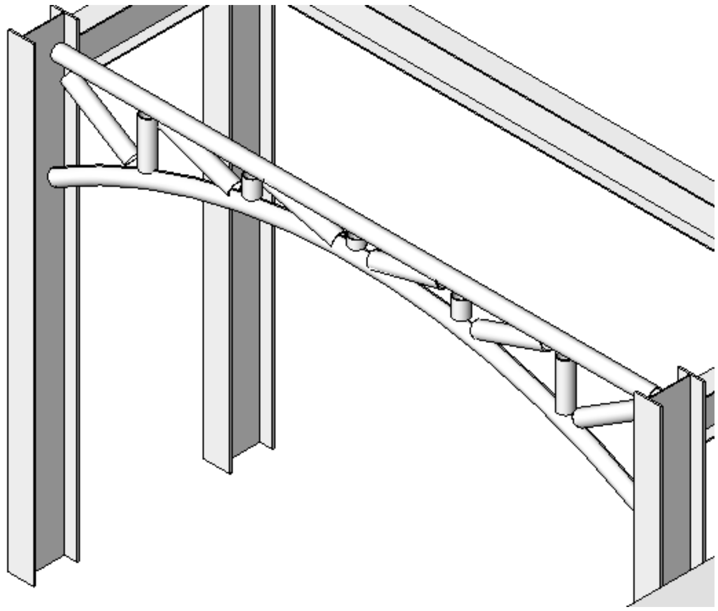


17 Click to place the arc.

18 Click Truss Profile panel ➤ Finish Truss.

View the truss in 3D

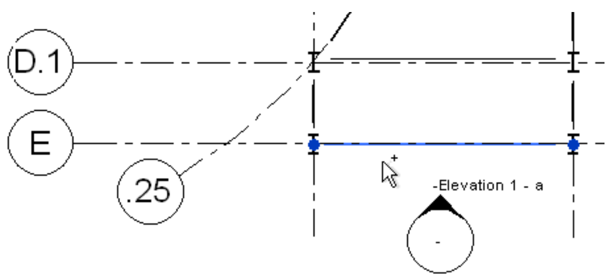
19 Click View tab ➤ Create panel ➤ 3D View drop-down ➤ Default 3D.



Copy the single truss

20 In the Project Browser, under Structural Plans, double-click 02 - Floor.

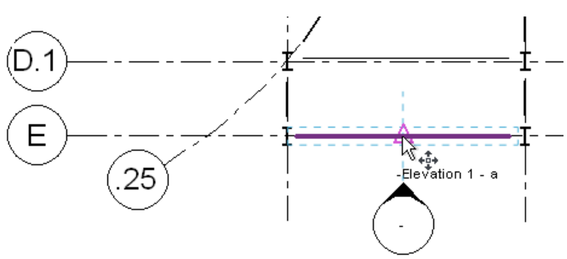
21 Select the truss placed in the previous steps.



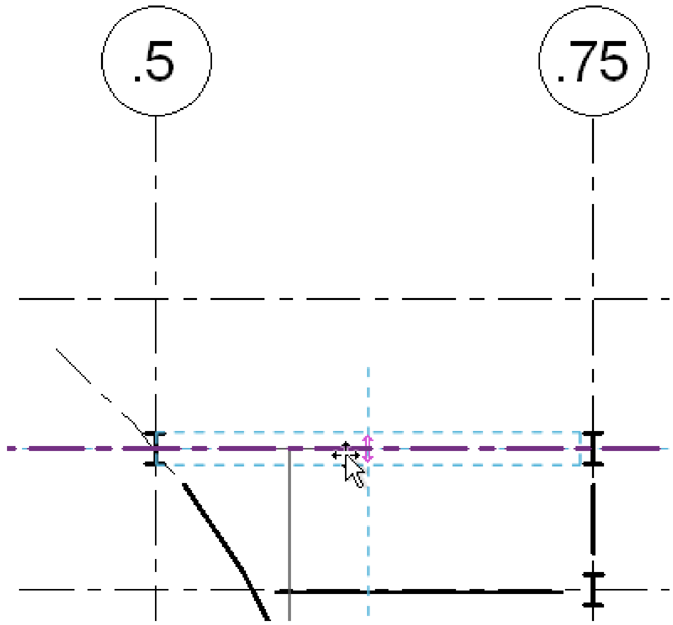
NOTE Press *Tab* and check the Status Bar to make sure you are selecting the truss and not the beam system.

22 Click Modify panel ➤ Copy.

23 Select the truss to specify the copy startpoint.



24 Click the open area on grid line B.1 between columns on grid lines .5 and .75, as shown, to specify the copy end point.

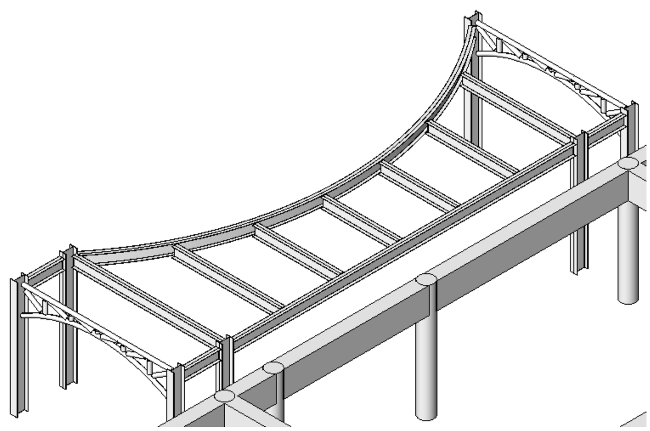


25 Press *Esc* twice.

View the trusses in 3D

26 In the Project Browser, under 3D Views, double-click 3D.

27 Zoom in on both trusses.



28 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

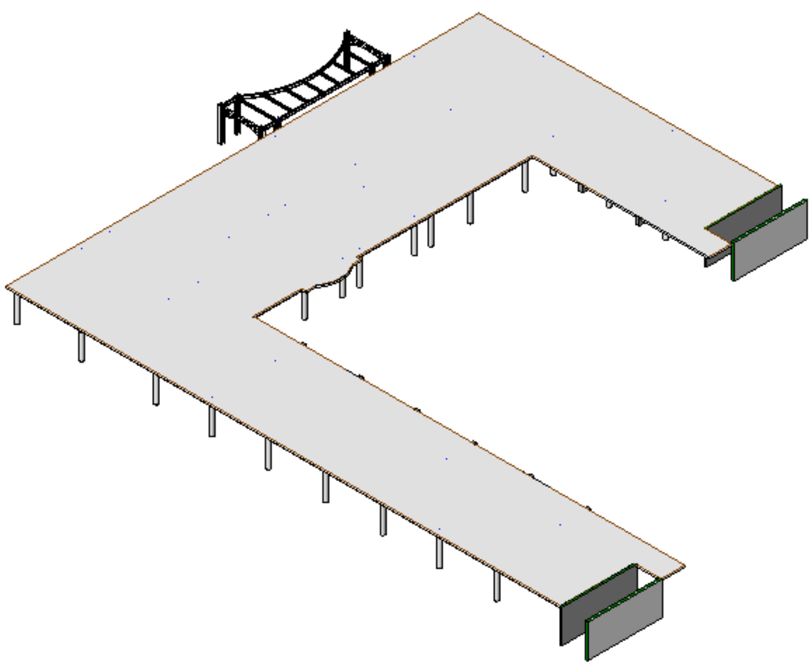
Adding Structural Slabs

11

In this lesson, you add a 6' concrete slab to level 2 of the structure. In Revit Structure, you use the line command to create the slab by tracing the outside edge of the drawing. The top of the slab is offset relative to the level at which it is placed. You learn to:

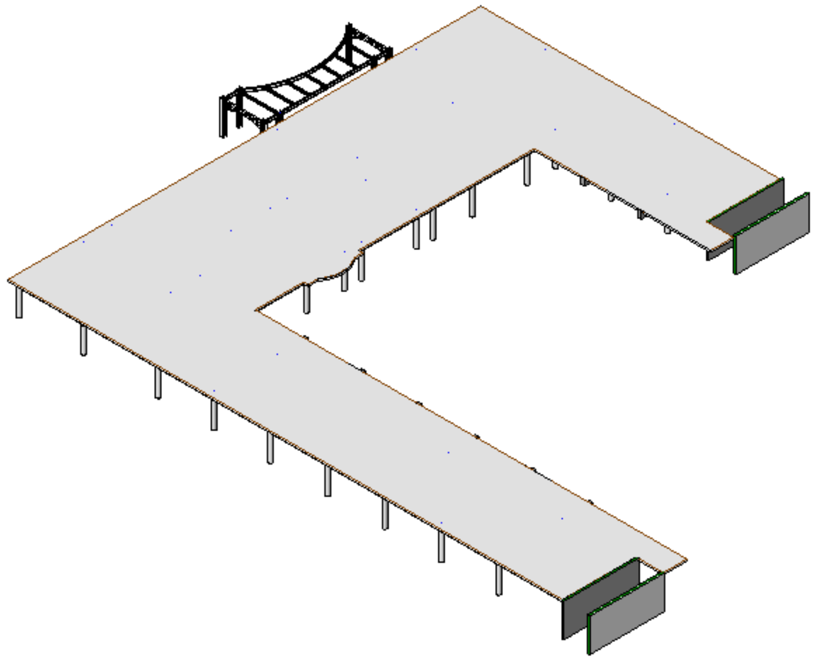
- Create the concrete slab by tracing along the outside edge of the architectural drawing.

Completed slab



Adding a Structural Slab

In this exercise, you create a concrete slab on level 2 of the structure by tracing the outside edge of the architectural drawing.



Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_Structural_Slabs_i.rvt.

Change the view visibility

You create the slab by tracing the outside edge of the architectural drawing. Therefore, the detail floor plan visibility must be turned on before you begin tracing.

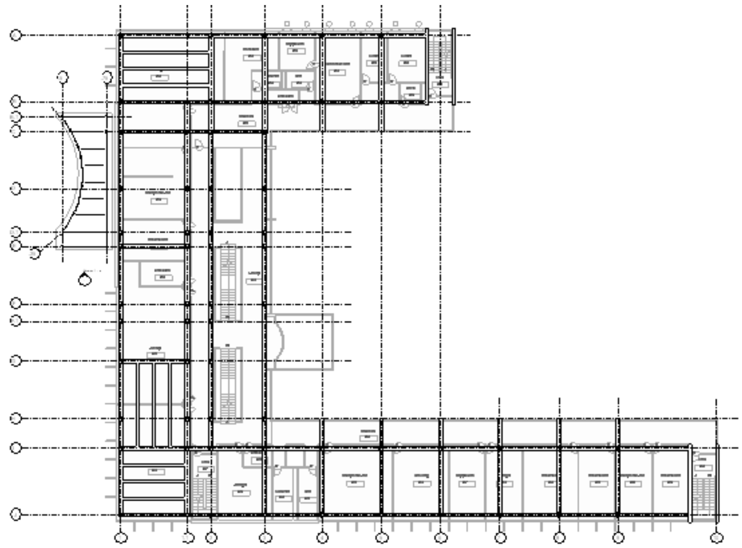
1 In the Project Browser, under Views (all) ➤ Structural Plans, double-click 02 Floor.

2 Click View tab ➤ Graphics panel ➤ Visibility/Graphics.

3 In the Visibility/Graphics dialog:

- Click the Imported Categories Tab.
- Under Visibility, click Technical_School-current.dwg.
- Click Apply, and then click OK.

The architectural drawing displays as a halftone image under the existing structural floor plan.



Set the slab properties

4 Click Home tab ➤ Structure panel ➤ Floor.

You are now in sketch mode.

5 Click Element panel ➤ Floor Properties.

6 In the Instance Properties dialog:

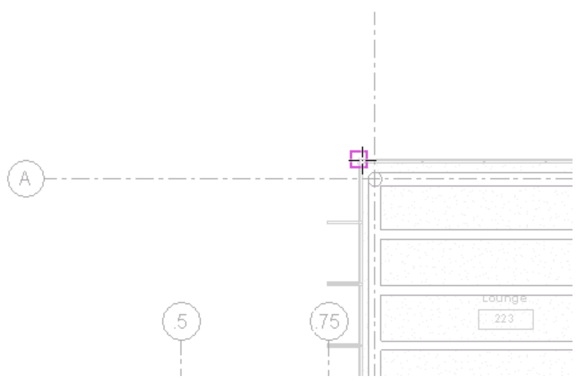
- For Type, select 6" Concrete.
- Click OK.

Trace the architectural drawing

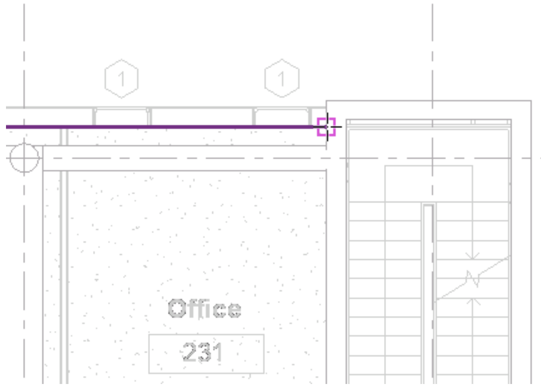
7 Click Draw panel ➤  (Line).

8 Zoom in on the northwest corner of the structure.

9 Click the outside edge of the floor outline to start the slab element.

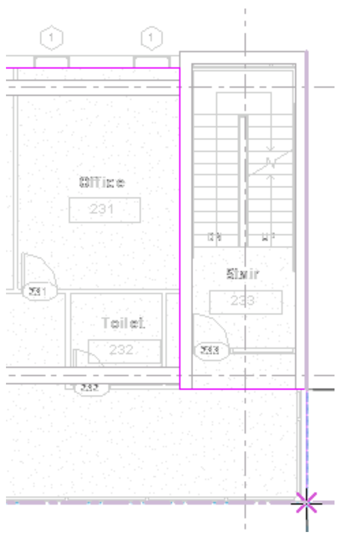


10 Place the first line along the outside edge of the architectural drawing, until you reach the interior concrete wall for the stairs on the northeast corner of the structure.

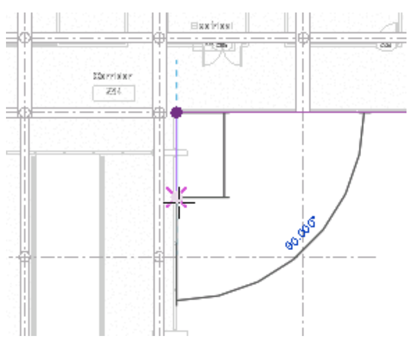


NOTE When drawing the slab, zoom in and out as needed, and make sure there are no duplicate lines. If necessary, click Edit panel ➤ (Trim/Extend Elements), and press *Delete* to remove unwanted lines and to clean up line intersections.

11 Continue drawing lines along the concrete walls that surround the northeast stairs.



12 Continue drawing lines along grid line C until you reach the interior surface at grid line 3.

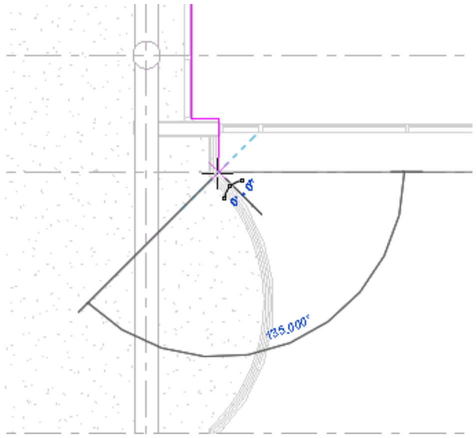


13 Continue drawing along grid line 3 until you reach the balcony.

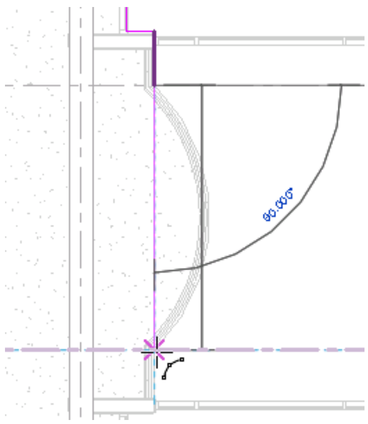
14 Trace the outline of the balcony as follows:

- Click Draw panel ➤ (Start-End-Radius Arc).

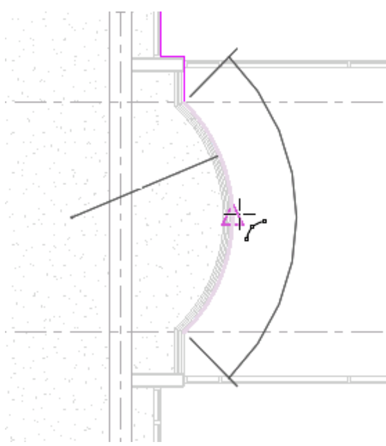
- Click to specify the arc start point.



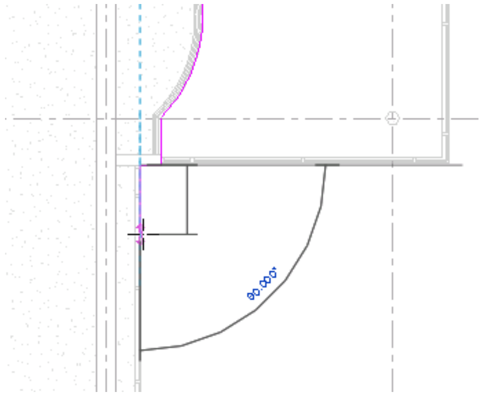
- Click to specify the arc endpoint.



- Click to specify the arc radius.



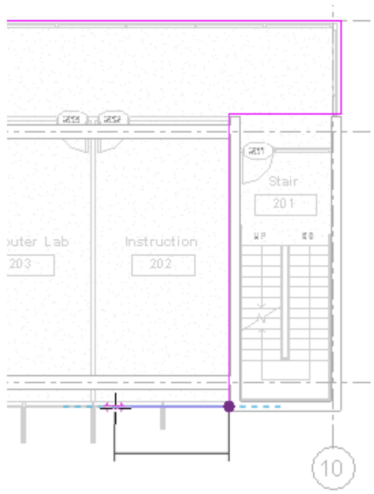
15 Click Draw panel ➤ (Start-End-Radius Arc).



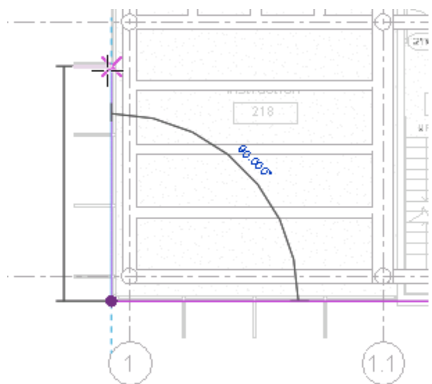
16 Continue tracing the slab along grid line 3 until you reach grid line H.

17 Trace along grid line H until you reach the southeast corner of the structure above the stairs.

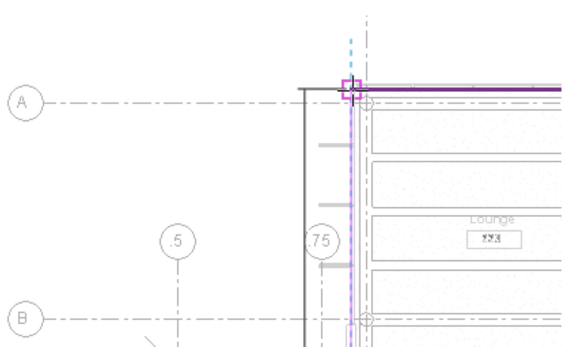
18 Continue drawing lines along the concrete walls that surround the southeast stairs.



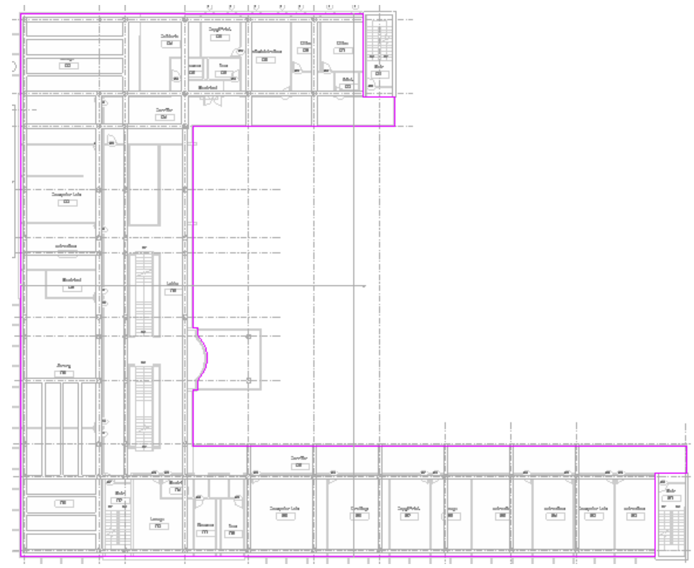
19 Trace along grid line K until you reach the southeast corner of the structure.



20 Continue drawing along grid line 1 until you reach the northwest corner of the structure, and click to complete the slab.



21 Enter **ZF** (Zoom to Fit).



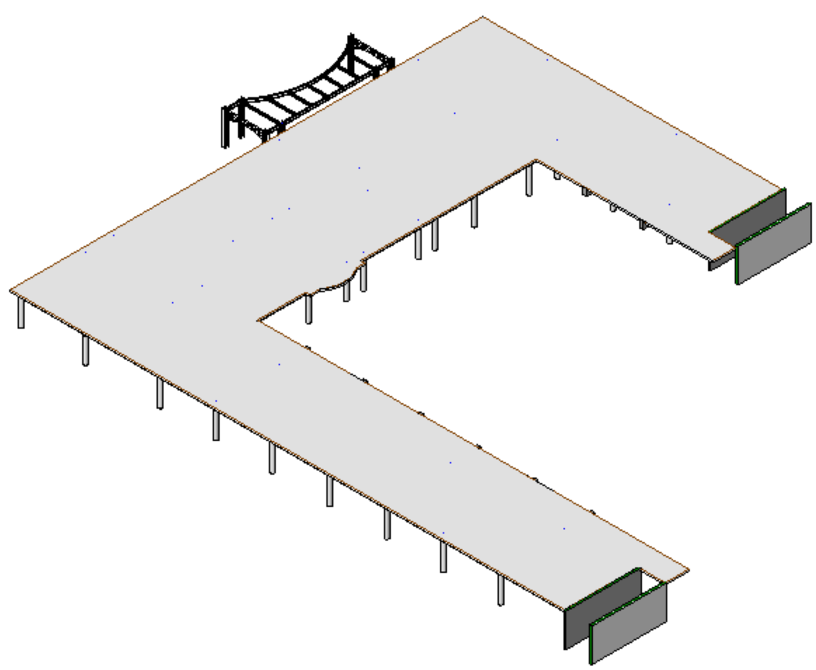
22 Click Floor panel ➤ Finish Floor.

In the Revit dialog, click No when asked if you would like walls that go up to this floor level to attach to its bottom.

View the slab in 3D

23 In the Project Browser, under Views (all) ➤ 3D Views, double-click 3D.

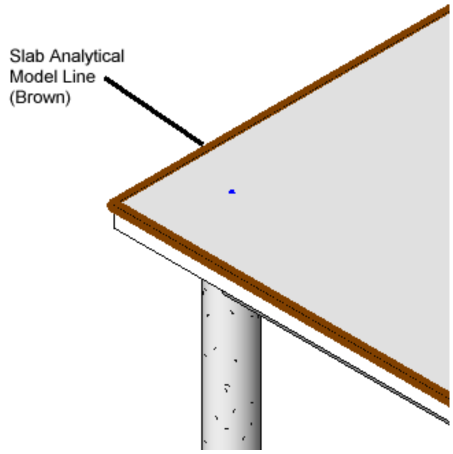
24 Press *Esc*.



View the slab analytical model

As you create the model in Revit Structure, a simplified 3D representation of each element is created automatically. This simplified view is referred to as the analytical model. Each element is drawn with a different color to help identify specific analytical properties; for example, slabs display as a brown line. You can export the analytical model to analysis and design applications.

25 Zoom in on the slab to see the brown line representing the analytical model.



26 Close the file with or without saving it.

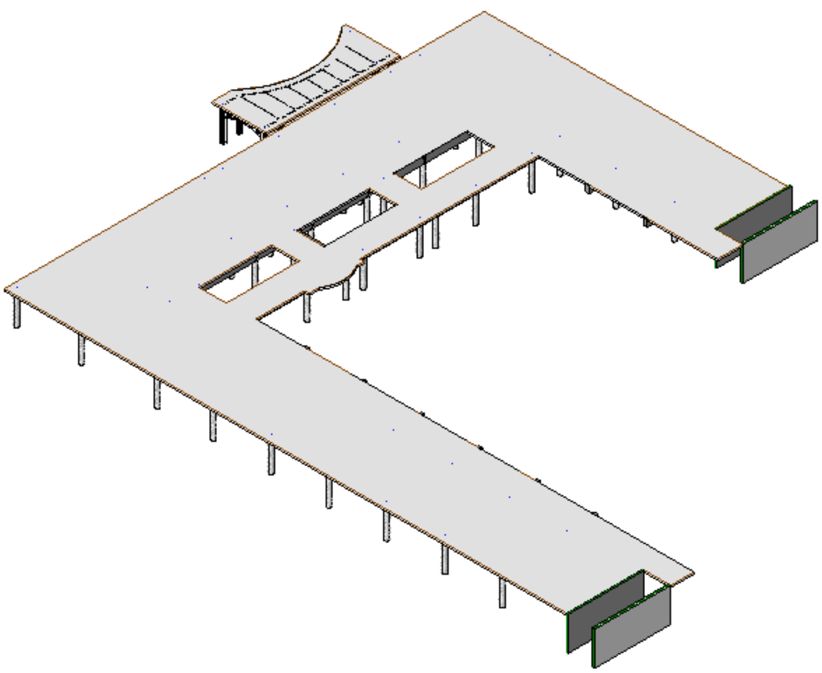
In the next exercise, a new training file is supplied.

Adding Openings

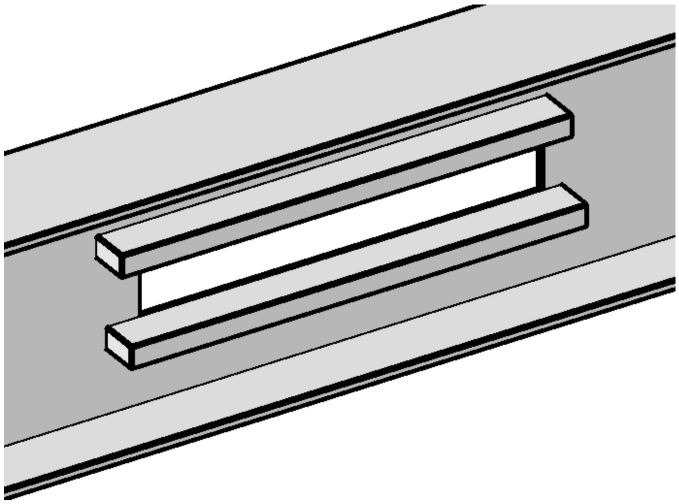
12

In this lesson, you add an opening in the concrete slab on level 2 of the structure. You also add an opening to the face of a steel beam in the entry way frame. You learn to:

- Create shaft openings in the concrete slab for the interior stairs and the skylights that will be installed on the roof of the structure.

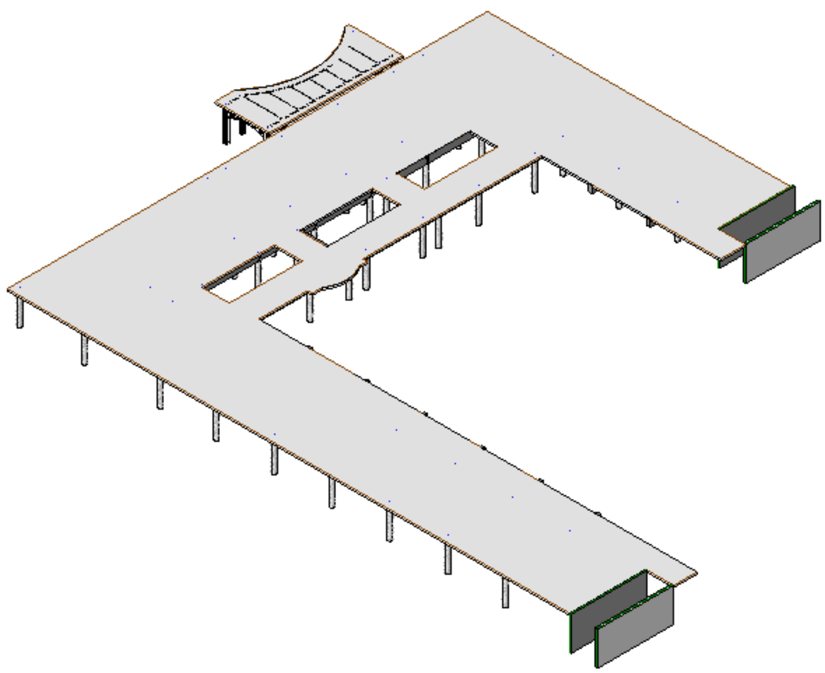


- Create an opening in the face of a steel beam in the entry way roof frame for installing electrical systems.



Adding Shaft Openings

In this exercise, you create shaft openings for the interior stairs and for the skylights that will be installed on the roof of the structure. You cut these openings in the slab on level 2 and then copy them to the remaining levels in a future exercise.



Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_OP_01_Add_Openings_Slab_i.rvt.

Change the view visibility

You create the shaft openings by tracing the outline for the interior stairs located on the architectural drawing. You must turn on the detail floor plan visibility before you can create the openings.

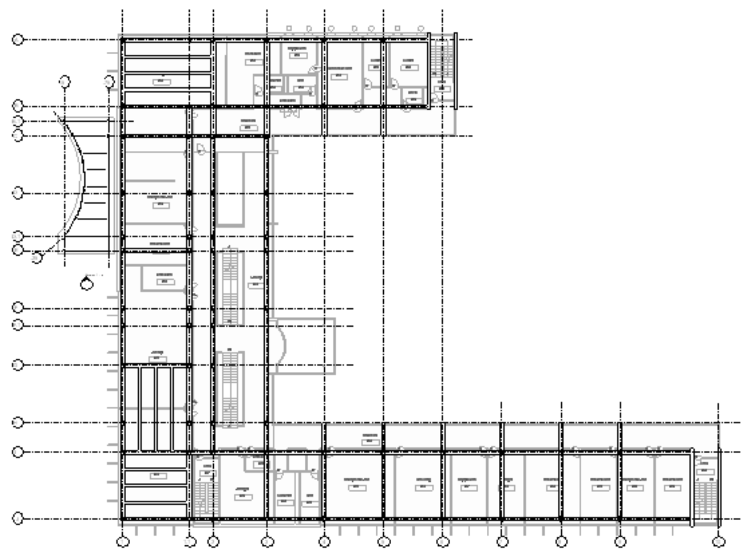
1 In the Project Browser, under Views (all) ➤ Structural Plans, double-click 02 Floor.

2 Click View tab ➤ Graphics panel ➤ Visibility/Graphics.

3 On the Visibility/Graphic Overrides dialog:

- Select the Revit Links tab.
- Under Visibility, select Technical_School-current.dwg.
- Click Apply, and then click OK.

The architectural drawing displays as a halftone image under the existing structural floor plan.



Set the opening properties

4 Zoom in around the stairs located between grid lines G and H.

5 Click Modify tab ➤ Opening panel ➤ Shaft.

6 Click Element panel ➤ Shaft Opening Properties.

7 In the Instance Properties dialog, under Constraints:

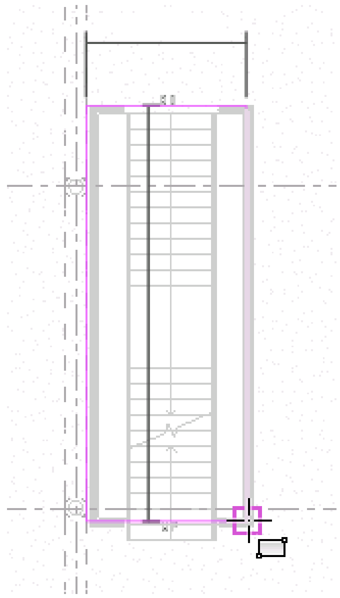
- For Base Constraint, select 02 - Floor.
- For Top Constraint, select Up to level: Roof.
- Click OK.

These settings ensure that the shaft opening extends through the roof of the structure without penetrating the foundation slab that will be added in a future lesson.

Create the first shaft opening

8 Click Draw panel ➤ (Rectangle).

9 Trace the outline for the interior stairs.



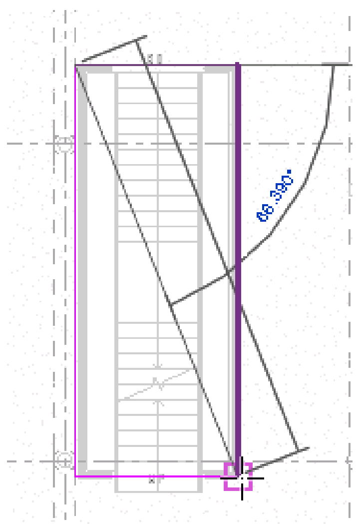
Add symbolic lines

Symbolic lines are added to identify the shaft opening in all related documentation.

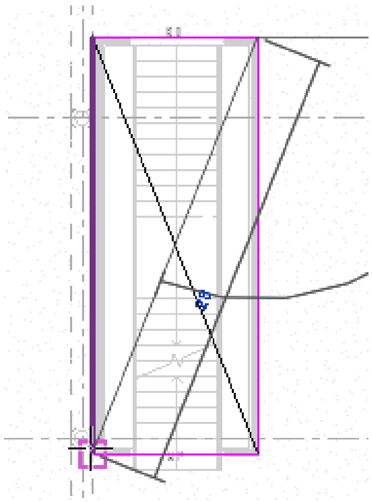
10 Click Draw panel ➤ Symbolic Lines.

11 Draw 2 lines that connect the opposite corners of the shaft opening.

Create the first diagonal line

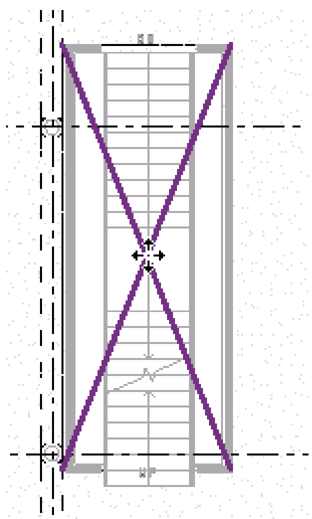


Create the second diagonal line



12 Click Shaft Opening panel ➤ Finish Opening.

Notice the shaft opening displays as an “X” created by the symbolic lines.



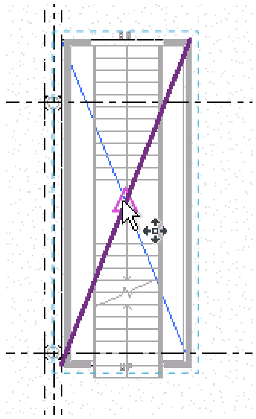
Copy the shaft opening

13 On the plan view, select the symbolic lines for the shaft opening.

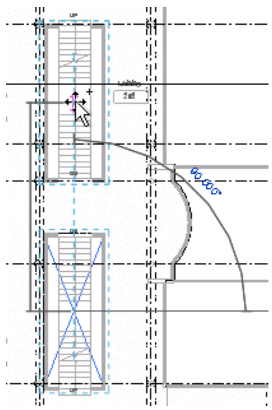
14 Click Modify panel ➤ Copy.

15 On the Options Bar, select Multiple.

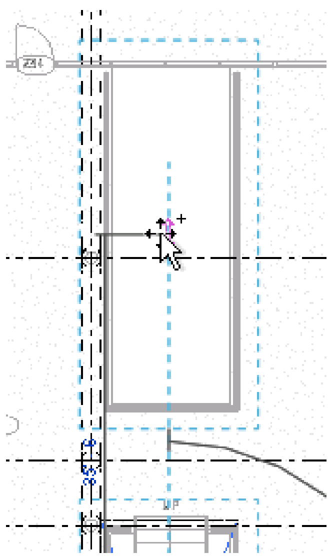
16 Click the center point of the first shaft opening.



17 Click the center point of the opening for the stairs located between grid lines E and F.1.



18 Click the center point of the opening for the stairs located between grid lines C and D.1.

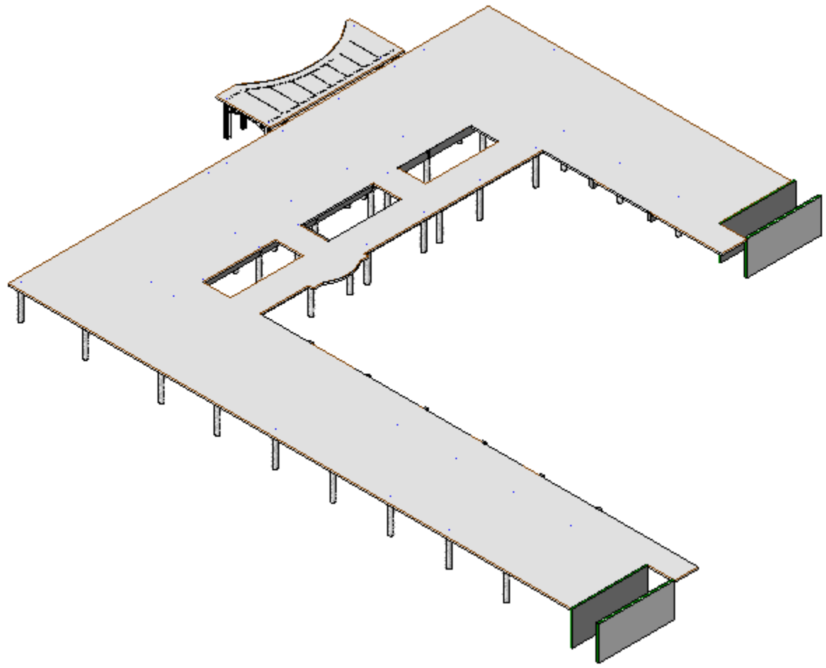


19 Press *Esc*.

View the openings in 3D

20 In the Project Browser, under 3D Views, double-click 3D.

The completed shaft openings are displayed.

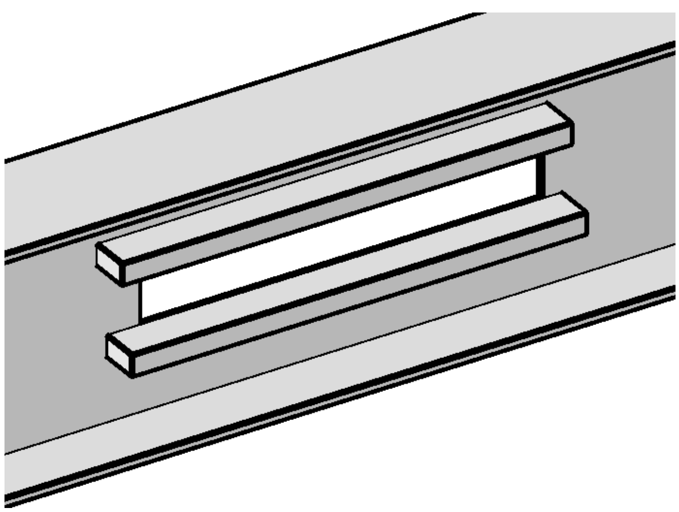


21 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Adding an Opening in a Beam

In this exercise, you add an opening to the face of a single beam and then add stiffener plates to the opening.



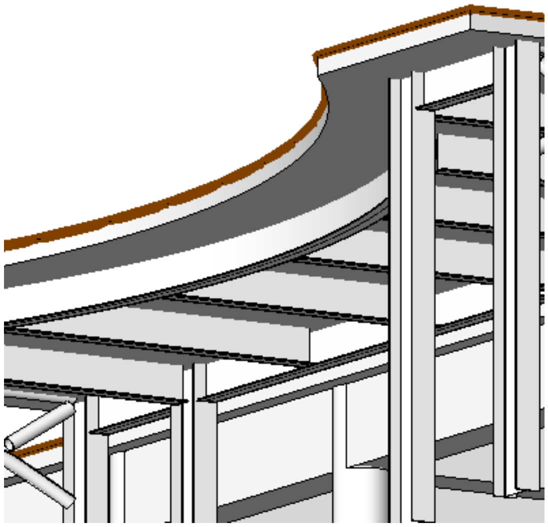
Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_OP_02_Add_Openings_Beam_i.rvt.

View the beam system

1 In the Project Browser, under 3D Views, double-click 3D.

2 Use the ViewCube to rotate the 3D view until the beams of the entry way steel frame are visible.

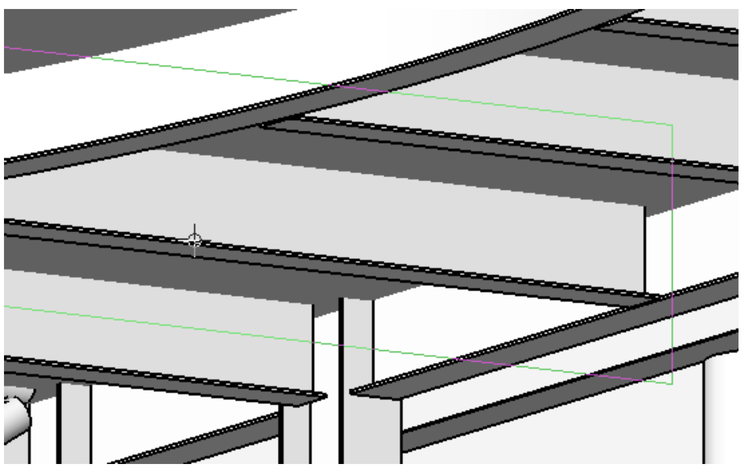


Add an opening to the face of the beam

3 Click Modify tab ➤ Opening panel ➤ By Face.

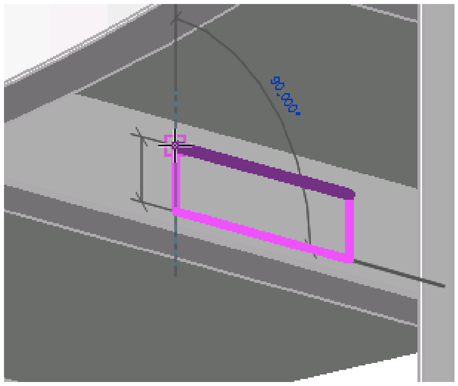
4 Select the planar face of one of the steel beams.

Press *Tab* to make sure you are selecting the beam and not the floor or beam system.



The opening will be cut perpendicular to the selected face.

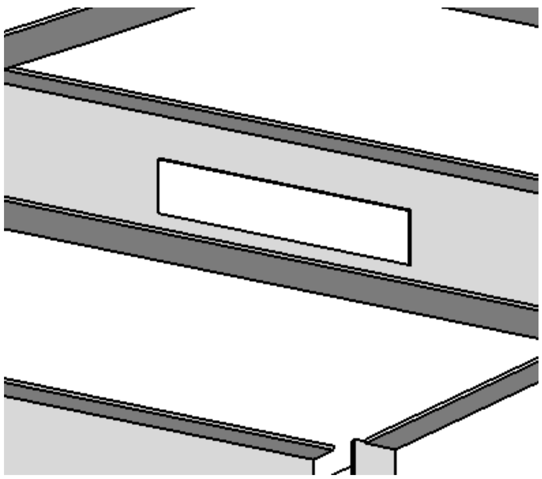
5 Using the tools available on the Draw panel, draw an opening in the beam approximately as shown.



Make sure the opening forms a closed loop.

6 Click Opening by Face panel ➤ Finish Opening.

7 Press *Esc*.



Add stiffener plates to the opening

You add stiffeners to strengthen the opening in the steel beam. Stiffeners are modelled as a component family with properties similar to steel beams. Use the Draw tool or the Pick tool to add the stiffener component to the project.

8 Click Home tab ➤ Model panel ➤ Component drop-down ➤ Place a Component.

9 Click Element panel ➤ Change Element Type drop-down ➤ Linear Stiffener-Plate: Standard.

Change the stiffener plate properties

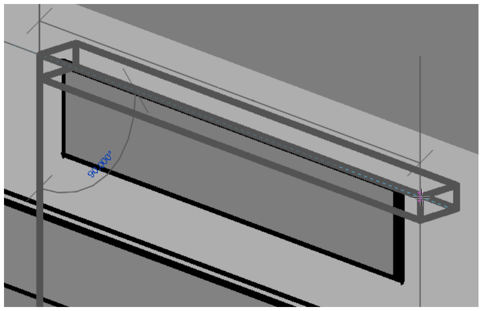
10 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

11 In the Instance Properties dialog:

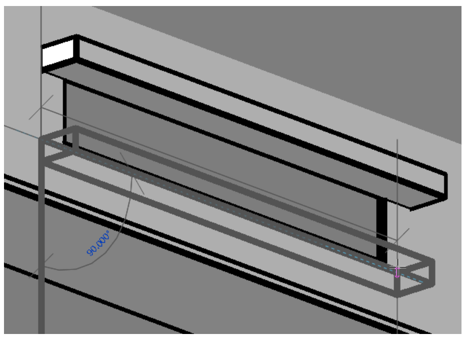
- Under Dimensions, for d, type **0' 2"**.
- Click OK.

12 Using the tools available on the Draw panel, draw a stiffener plate on both horizontal surfaces of the opening.

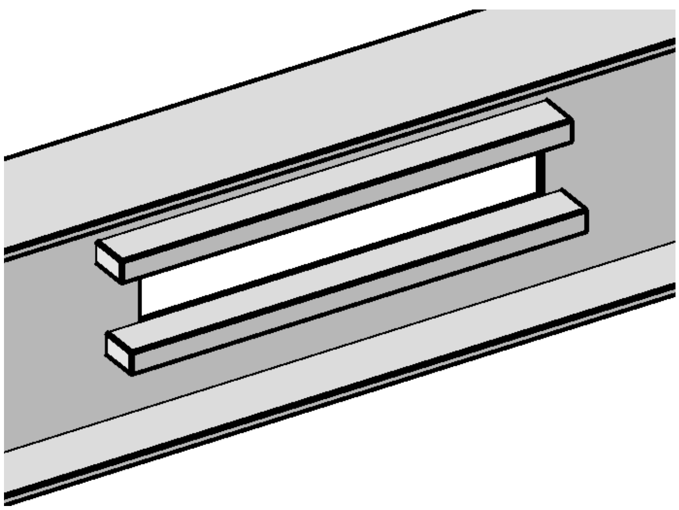
Draw the top horizontal stiffener



Draw the bottom horizontal stiffener



13 Press *Esc*.



14 Close the file with or without saving it.

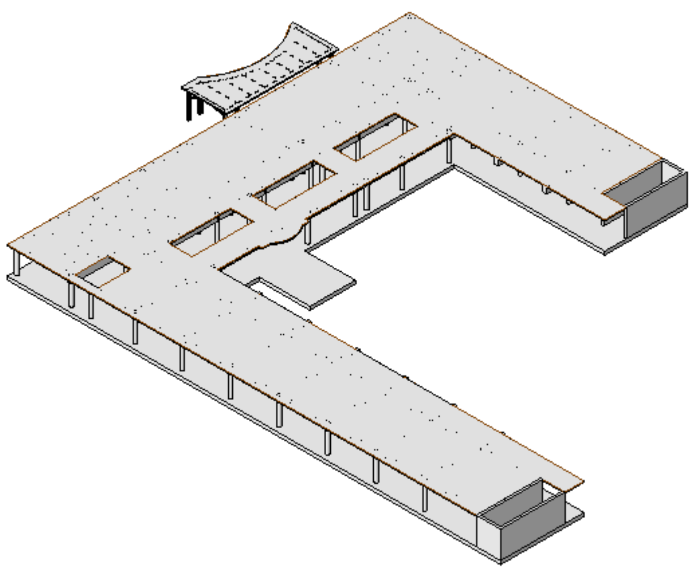
In the next exercise, a new training file is supplied.

13

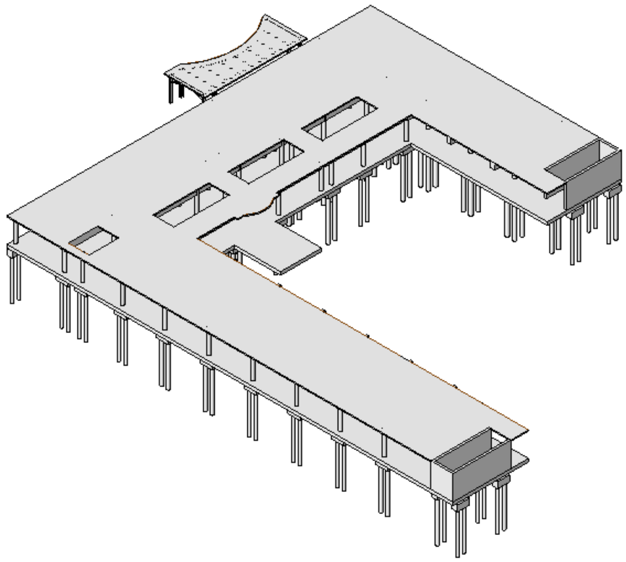
Adding Structural Foundations

In this lesson, you add structural foundations to the project. In Revit Structure, you can add different types of foundations, such as slab, wall, or isolated. Each foundation type includes properties that can be changed to match the requirements of the project. You learn to:

- Add a slab foundation to the entry level of the structure, and then modify the shape of the slab to include the stairs on the northeast and southeast corners.

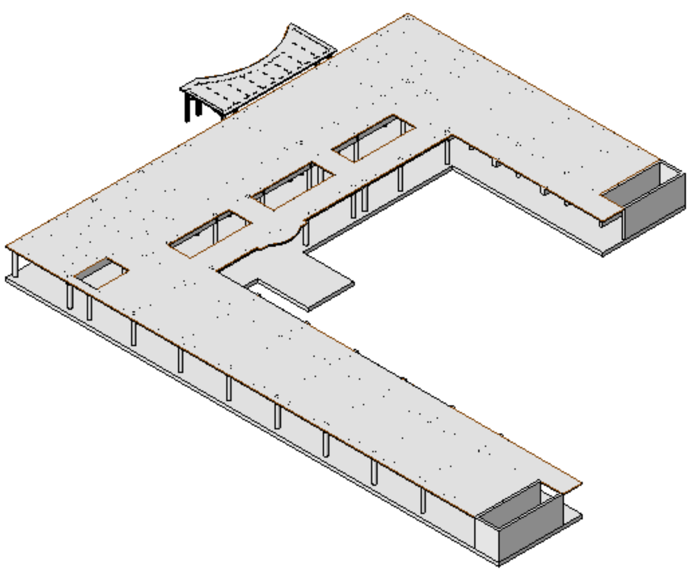


- Add 2 isolated pile caps of different sizes at specific column locations under the slab foundation.



Adding a Slab Foundation

In this exercise, you add a slab foundations to the entry level of the structure by first copying the slab from level 2 and then creating a new thicker slab size.



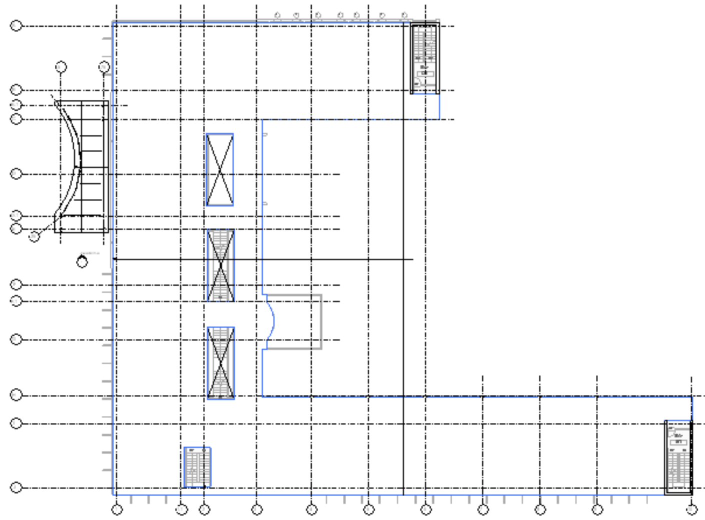
Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_FDN_01_Slab_Foundation_i.rvt.

Copy the slab from level 2

1 In the Project Browser, under Structural Plans, double-click 02 - Floor.

2 Select the slab.

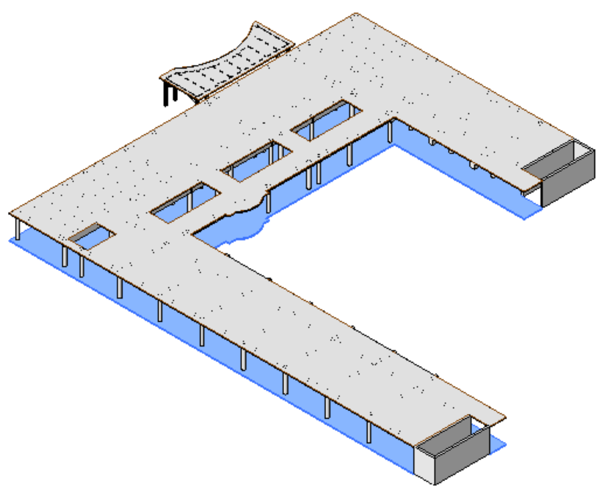


3 Click Clipboard panel ➤ Copy.

4 Click Clipboard panel ➤ Paste Aligned drop-down ➤ Select Levels.

5 In the Select Levels dialog, select 01 - Entry Level, and click OK.

6 In the Project Browser, under 3D Views, double-click 3D.



The slab is copied to the entry level of the structure. However, it does not support the structural walls of the northeast and southeast stairs or the balcony glass frame. In the next few steps, you modify the slab footprint so the new foundation supports these elements.

7 Press *Esc*.

Modify the foundation slab footprint

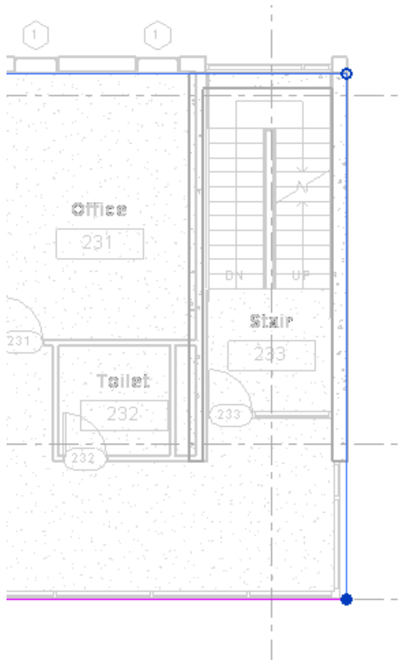
8 In the Project Browser, under Structural Plans, double-click 01 - Entry Level.

9 Select the slab, and click Edit panel ➤ Edit Boundary.

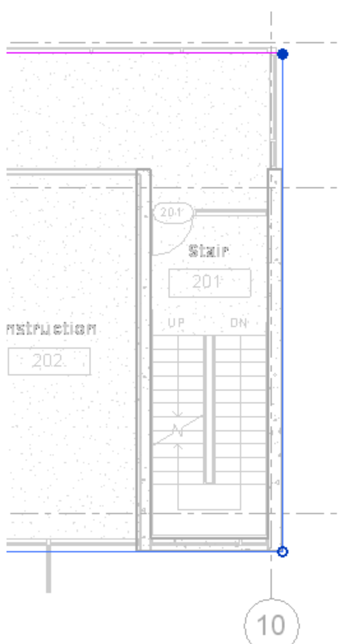
You are now in Sketch mode.

10 Modify the slab sketch as follows:

- Zoom in to the northeast corner of the structure.
- Delete the 2 inside sketch lines.
- Click and drag the outside sketch lines; extending them to the corner of the structure, as shown.

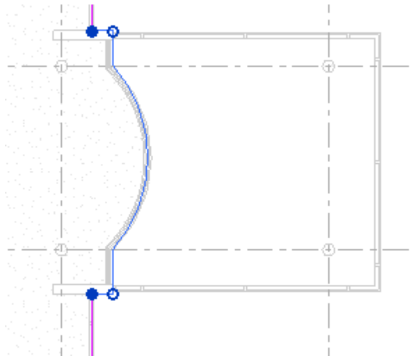


- Using the same method, modify the southeast corner of the slab as shown.

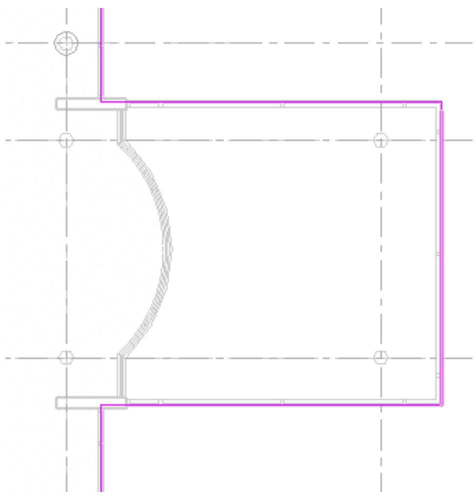


- Zoom in to the balcony located between grid lines F and G.

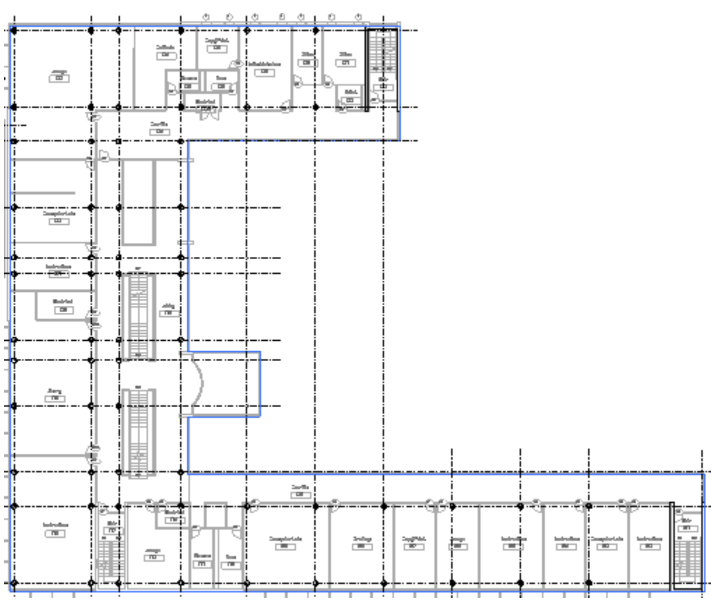
- Select the sketch lines that form the outside edge of the balcony and press *DELETE*.



- Click Draw panel ➤ Boundary Line ➤ (Line).
- Using the architectural drawing as a guide, sketch the outline of the balcony as shown



11 Click Floor panel ➤ Finish Floor.



Create a new slab size

12 With the slab still selected, click Element panel ➤ Element Properties drop-down ➤ Type Properties.

13 In the Type Properties dialog, click Duplicate.

14 In the dialog, type **18" Concrete**, and click OK.

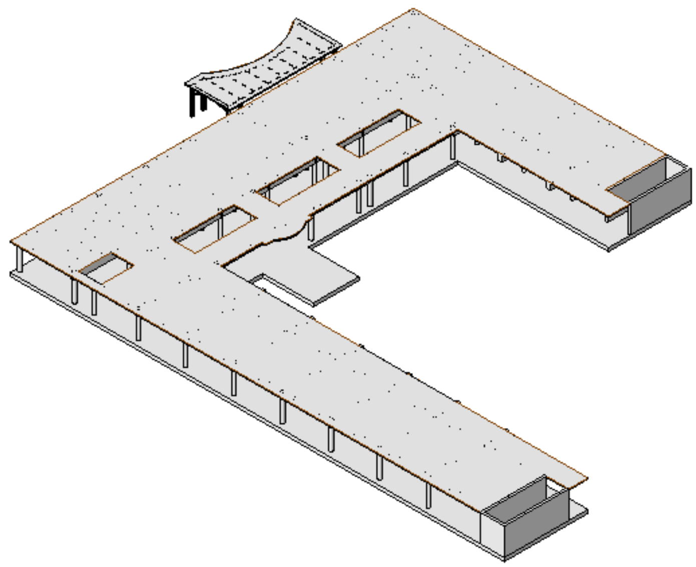
15 In the Type Properties dialog, under Construction, for Structure, click Edit.

16 In the Edit Assembly dialog, under Layer 2, for Thickness, type **1' 6"**, and click OK.

17 In the Type Properties dialog, click Apply, and click OK.

18 Press *Esc*.

19 In the Project Browser, under 3D Views, double-click 3D.

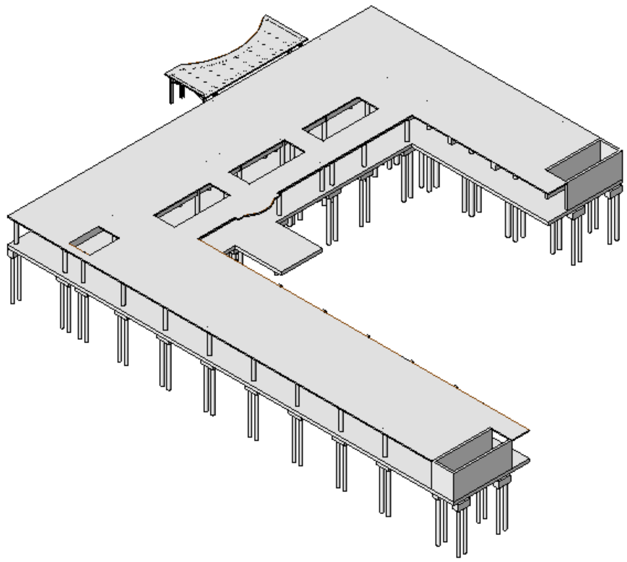


20 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Adding Isolated Foundations

In this exercise, you add isolated foundations with 2 piles and 4 piles, to the entry level of the structure. The pile caps are placed on grid locations. They directly support existing structural columns.

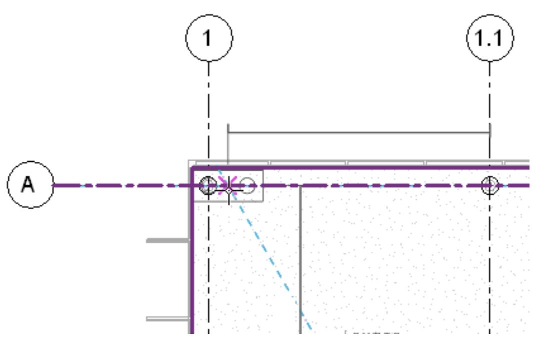


Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_FDN_02_Isolated_Foundations_i.rvt.

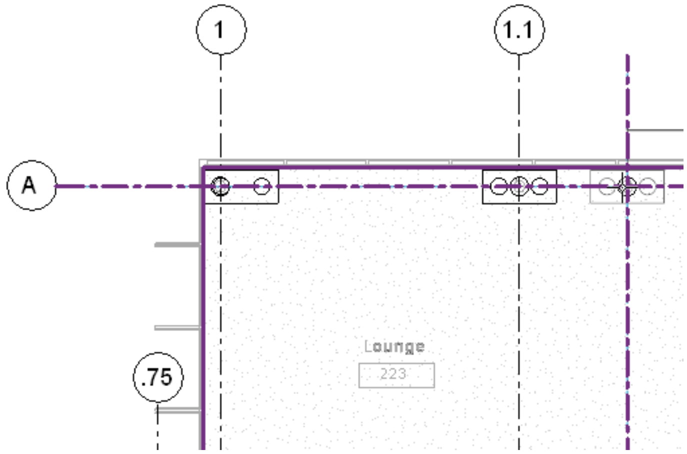
Place isolated foundations

1. In the Project Browser, under Structural Plans, double-click 01 - Entry Level.
2. Zoom in to the northwest corner of the structure.
3. Click Foundation panel ➤ Isolated.
4. Click Element panel ➤ Change Element Type drop-down ➤ Pile Cap-2 Pile : 31"x71"x35".
5. In the drawing area, click the column at grid location A1 to place the first isolated foundation.

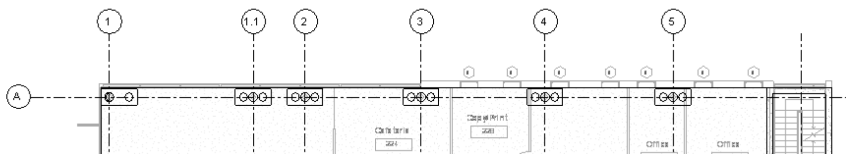


Notice that the pile cap snaps to the center of the column.

6. Click the columns at grid locations A1.1 and A2 to place additional pile caps.



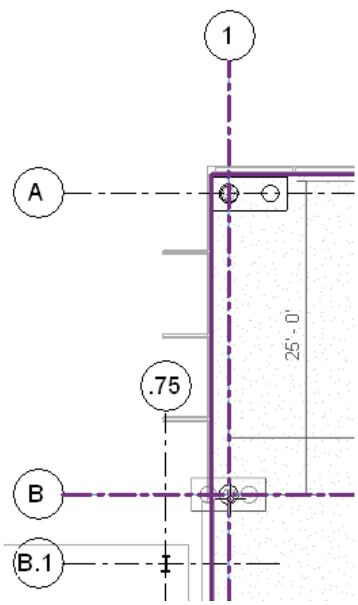
7 Using this same method, place pile caps along grid line A



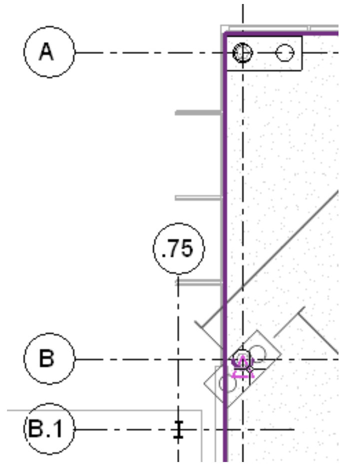
Place foundations on vertical grids

You can rotate isolated foundations placed on non-horizontal grid lines in increments of 90 degrees.

8 Position a pile cap over grid location B, but do not place it.

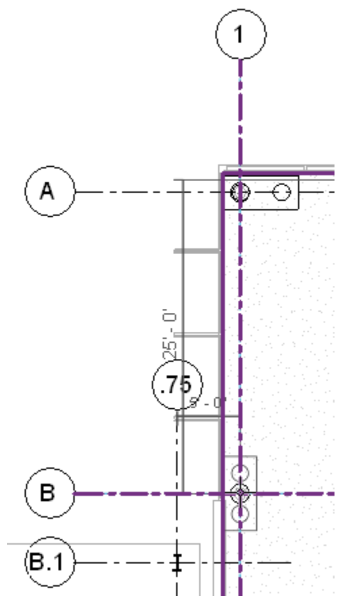


9 Press the *Spacebar* once.



The foundation is rotated 45 degrees.

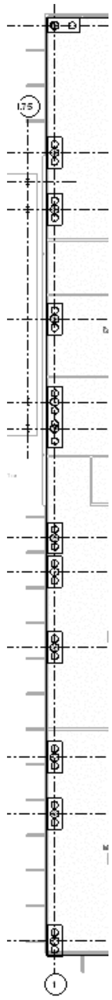
10 Press the *Spacebar* again.



The foundation is rotated to 90 degrees.

11 Click to place the isolated foundation.

12 Place additional pile caps along grid line 1 as shown.



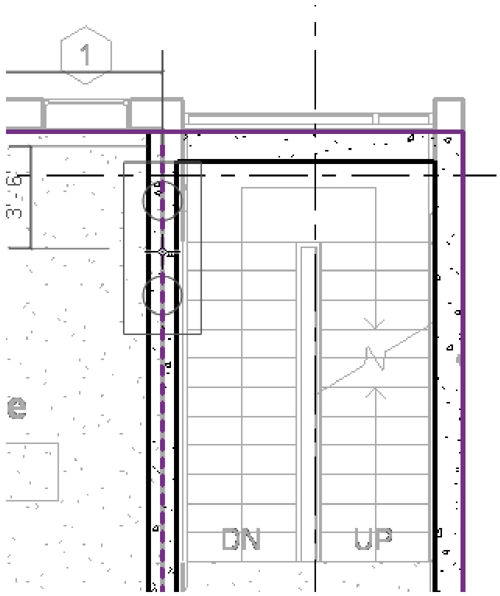
13 Click Selection panel ➤ Modify.

Place pile caps on non-grid locations

The stairs located in the northeast and southeast corners of the structure require isolated foundations, that are placed on non-grid locations.

14 Zoom in on the northeast corner of the structure.

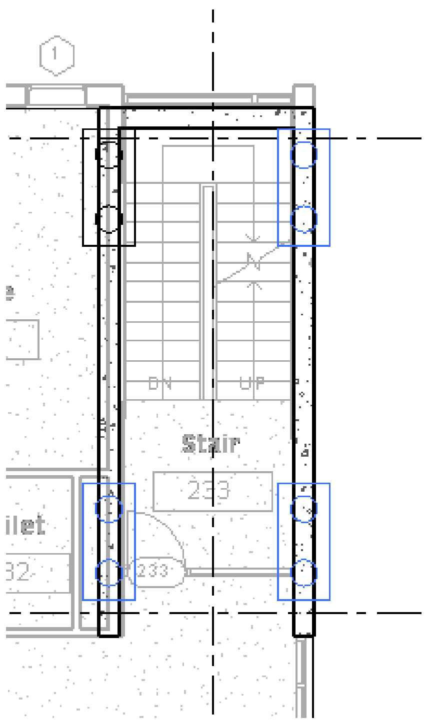
15 Position the pile cap over the concrete wall of the stairs until the center line of the wall highlights.



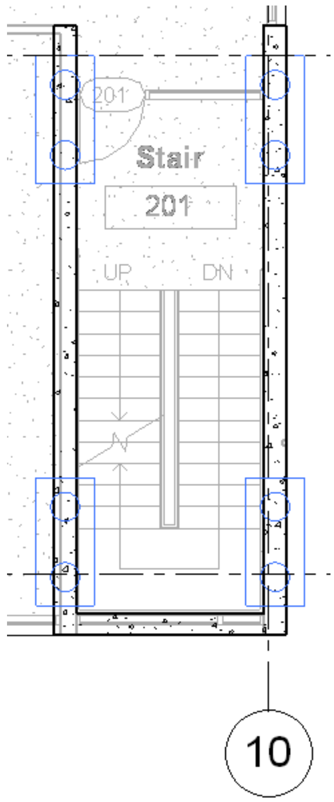
16 Click to place the pile cap.

17 Place additional pile caps along the walls of both sets of stairs, as shown.

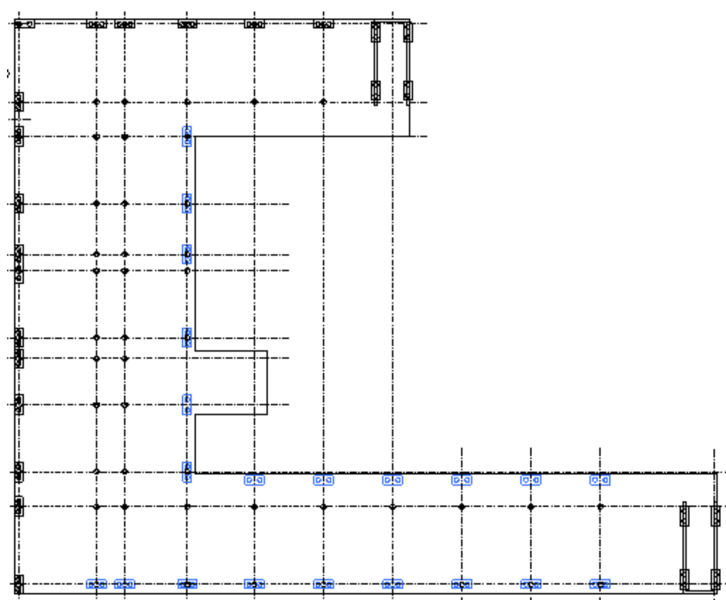
Northeast stairs



Southeast stairs



18 Using the same method, place additional pile caps along grid lines J and 3, and just below grid line H. Rotate the pile caps as needed.



19 Click Selection panel ➤ Modify.

Place isolated foundations on interior grid locations

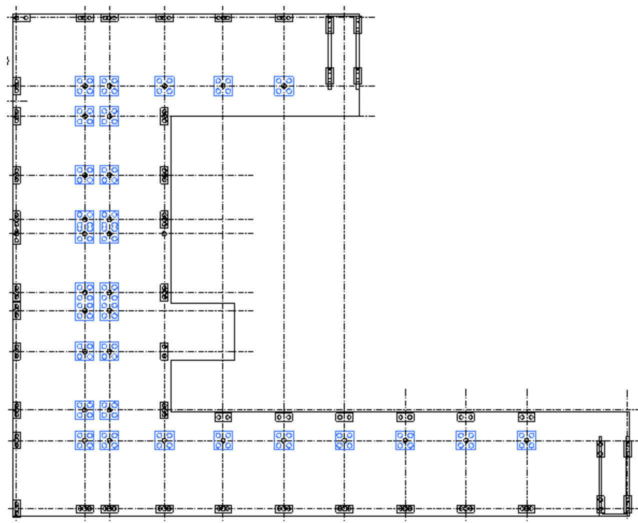
The columns located on the interior grid locations require an isolated foundation with a 4-pile cap to maintain the stability of the structure.

20 Click Foundation panel ➤ Isolated.

21 Click Element panel ➤ Change Element Type drop-down ➤ Pile Cap-4 Pile : 79"x79"x35".

22 Using the same method, place isolated foundations at grid intersections along the following grid lines:

- Grid line B.
- Grid line I.
- Grid line 1.1.
- Grid line 2.



23 Press *Esc* twice.

Change the offset distance for all isolated foundations

The offset distance sets the top surface of the isolated foundation in relation to the 01 - Entry Level. It is changed to position each foundation below the structural slab.

24 Draw a pick box around the entire structural plan.

25 Click Filter panel ➤ Filter.

26 In the Filter dialog:

- Click Check None.
- Under Category, select Structural Foundations.
- Click OK.

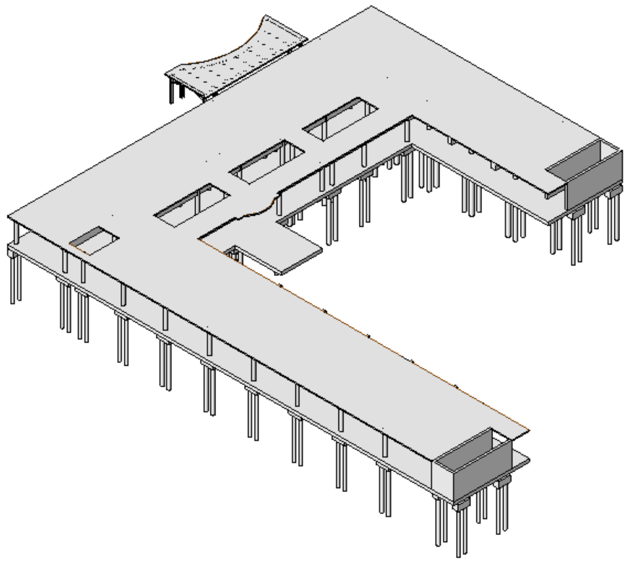
27 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

28 In the Instance Properties dialog, under Constraints, for Offset, type **-1' 6"**, and click OK.

29 Press *Esc* twice.

View the isolated foundations in 3D

30 In the Project Browser, under 3D Views, double-click 3D.



31 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Analyzing a Project

In this tutorial, you use the tools provided in Revit Structure 2010 to analyze your project before completing the model, and make necessary adjustments to the design.

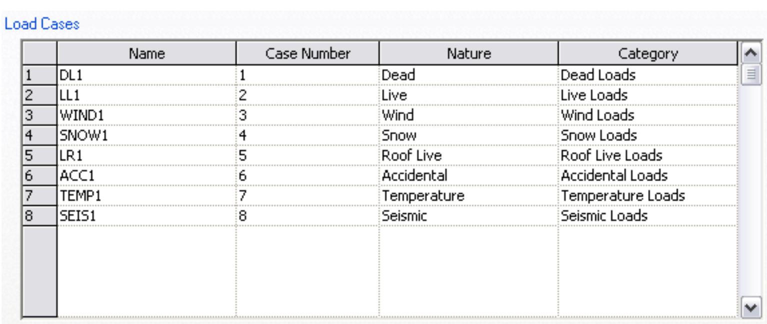
Analyze the Model

14

In this lesson, you learn some basic concepts about the analytical model that develops simultaneously while you build the physical model. In Revit Structure, an analytical model is a simplified 3D representation of the full engineering description of a structural physical model. The analytical model consists of those structural components, such as geometry, material properties, and loads, that together form an engineering system. The analytical model is created automatically as you create the physical model, and can be exported to analysis and design applications. You learn to:

- View the analytical model and run basic analytical checks.
- Create load cases that are applied to the structural model.
- Create load combinations that are applied to the structure.
- Transfer project standards (load types and combinations) from an existing project.
- Add a boundary condition to the foundation.

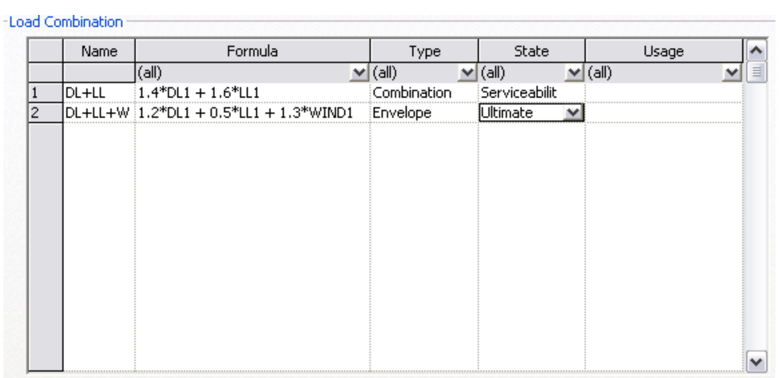
Load cases



Load Cases

	Name	Case Number	Nature	Category
1	DL1	1	Dead	Dead Loads
2	LL1	2	Live	Live Loads
3	WIND1	3	Wind	Wind Loads
4	SNOW1	4	Snow	Snow Loads
5	LR1	5	Roof Live	Roof Live Loads
6	ACC1	6	Accidental	Accidental Loads
7	TEMP1	7	Temperature	Temperature Loads
8	SEIS1	8	Seismic	Seismic Loads

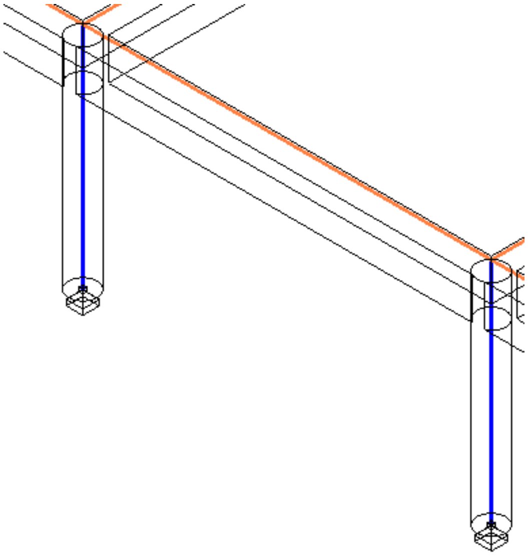
Load Combos



Load Combination

	Name	Formula	Type	State	Usage
		(all)	(all)	(all)	(all)
1	DL+LL	1.4*DL1 + 1.6*LL1	Combination	Serviceabilit	
2	DL+LL+W	1.2*DL1 + 0.5*LL1 + 1.3*WIND1	Envelope	Ultimate	

Boundary conditions



Analytical Checks

In this exercise, you learn how to check member supports, and run an analytical consistency check on the analytical model. You need to run analytical checks at different phases of the project and adjust the model accordingly.

Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_AYP_01_Analytical_Checks_i.rvt.

The analytical view template

When working with the analytical model, Revit Structure View Templates provide initial conditions for a view. You can apply a template to an existing or new view. You can also apply the view properties of an existing view by using the Apply View Template command. The view inherits view properties, such as View Scale, Discipline, Detail level, and the visibility settings of categories and subcategories. You use View Templates to standardize the appearance of all views.

Views and View Templates are not linked. All views created with that template are not automatically updated. To update a view, you can reapply the modified template.

Because each view in Revit Structure consists of different properties, the View Template saves and applies only common properties. You can save the View Template from one type of view and apply those same properties to any other geometric view.

For example, if you save a template from a plan view, you can apply that same template to a 3D view. Though the View Range property applies only to plan views, the template is still applied to the 3D view, because Revit Structure applies only the properties that apply to both.

Explore the view template

1 Click View tab ➤ Graphics panel ➤ View Templates drop-down ➤ Apply new template to current view.

2 In the Apply View Template dialog:

- Under Show type, select <all>.
- Under names, select Structural Analytical Normal.
  The Structural Analytical Normal view template includes structural components in a new analytical plan view and displays both the analytical model and the physical model.

- Under names, select Structural Analytical Stick.
  The Structural Analytical Stick view template presents structural components in a new analytical plan and displays only the analytical model.

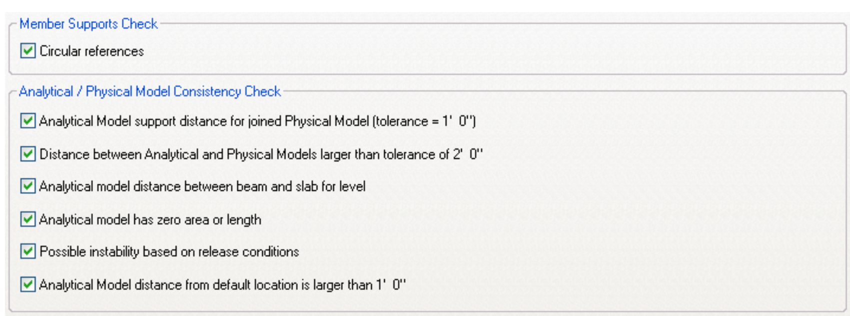
- Click Cancel.

View the analytical model settings

3 Click Manage tab ➤ Project Settings panel ➤ Structural Settings.

4 In the Structural Settings dialog:

- Click the Analytical Model Settings tab.
- To perform the checks automatically, under Automatic Checks, click Member Supports and Analytical/Physical Model Consistency.
  In this exercise, both the Member Supports Check and the Analytical/Physical Model Consistency Check verify all elements selected from the Structural Settings dialog.



- Click OK.

Run the member support checks

In the following steps, the member support checks are run to demonstrate the ability of Revit Structure to identify possible unsupported elements within the model. This exercise is not intended to resolve all issues displayed in the training file.

5 Click Analyze tab ➤ Analytical Model Tools ➤ Check Supports.

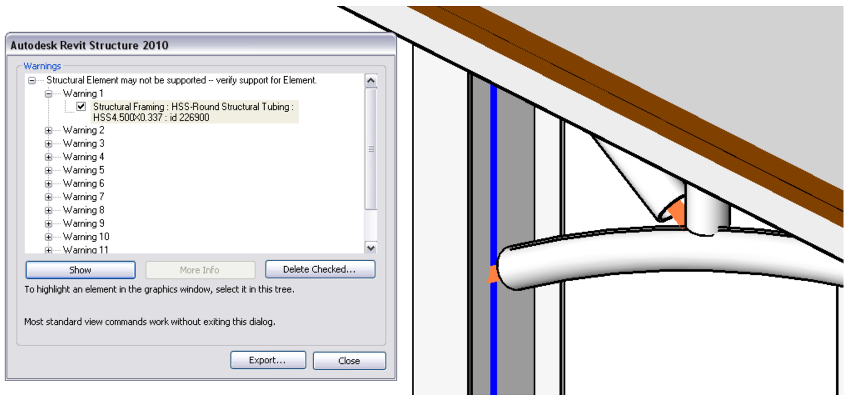
A Revit dialog displays the results of the member support analysis of the model.

6 In the Revit dialog, click (Expand warning dialog), to expand the warning message.

7 In the Warnings dialog:

- Expand Warning 1, and select Structural Framing: HSS-Round Structural Tubing.
- Click Show.
  In the Error Handling dialog, click Close.

  Click inside the drawing area, and use the ViewCube to rotate the view until the unsupported element displays.



  Using the same method, review the remaining member support warnings.

  After reviewing each warning, click Close in the Warnings dialog.

Run a consistency check

In the following steps, a consistency check is run to demonstrate the ability of Revit Structure to identify possible inconsistencies between the physical and analytical model. This exercise is not intended to resolve all issues displayed in the training file.

8 Click Analytical Model Tools ➤ Consistency Checks.

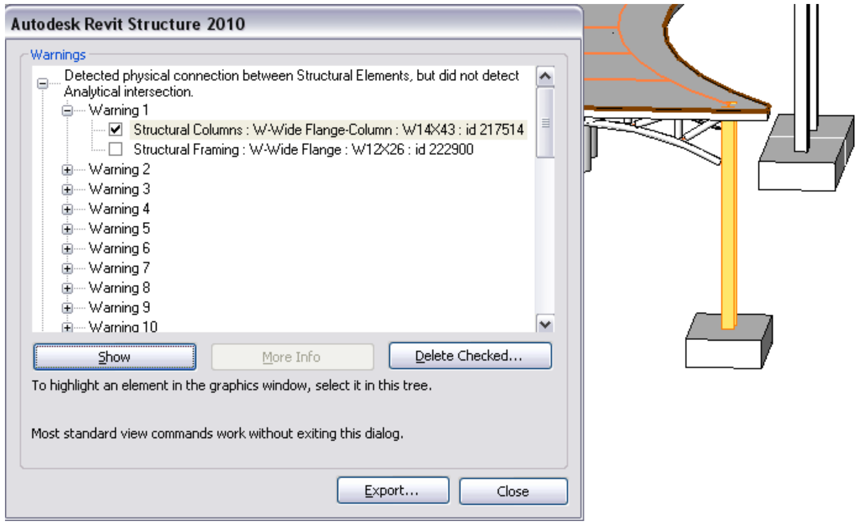
A Revit dialog displays the results of the analysis of the model consistency.

9 In the Revit dialog, click (Expand warning dialog), to expand the warning message.

10 In the Warnings dialog:

- Expand Warning 1, and select Structural Columns: W-Wide Flange-Column.
- Click Show.
  In the Error Handling dialog, click Close.

  Click inside the drawing area, and use the ViewCube to rotate the view until the inconsistent element displays.



Using the same method, review the remaining analytical warnings.

After reviewing each warning, click Close in the Warnings dialog.

11 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Load Cases

In Revit StructureY, you can apply point, line, and area loads . Each of these three load geometries is a family that contain instance and type parameters. Loads can be applied by sketching or by selecting a host component, such as a slab or beam. You edit and add load cases in the Structural Settings dialog.

The top portion of the dialog displays the Load Cases table. Revit Structure provides several default load case types that you can use to generate loads in the model. Use this table, to add, edit, or delete load cases.

Load Cases

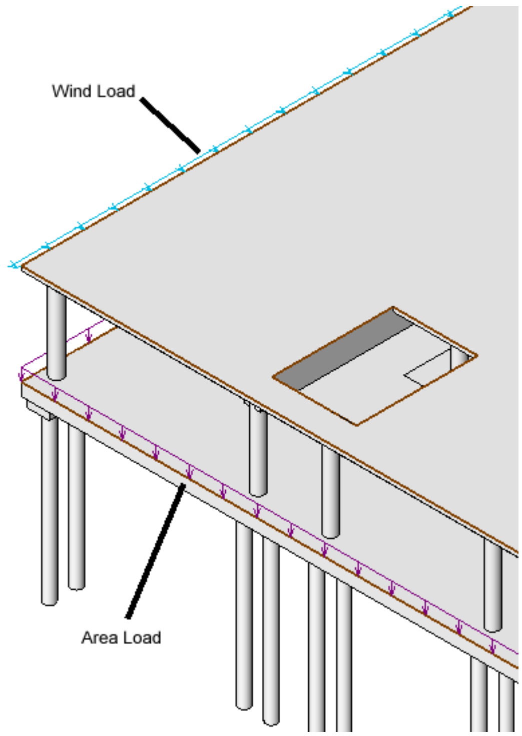
	Name	Case Number	Nature	Category
1	DL1	1	Dead	Dead Loads
2	LL1	2	Live	Live Loads
3	WIND1	3	Wind	Wind Loads
4	SNOW1	4	Snow	Snow Loads
5	LR1	5	Roof Live	Roof Live Loads
6	ACC1	6	Accidental	Accidental Loads
7	TEMP1	7	Temperature	Temperature Loads
8	SEIS1	8	Seismic	Seismic Loads

The bottom portion of the dialog displays the Load Natures table. Use this table to add or delete load natures.

Load Natures

	Name
1	Dead
2	Live
3	Wind
4	Snow
5	Roof Live
6	Accidental
7	Temperature
8	Seismic

In this exercise, you add both a wind load and area load to the structural model.



Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_AYP_02_Load_Cases_i.rvt.

Add a Load Case

1 Click Manage tab ➤ Project Settings panel ➤ Structural Settings.

2 In the Structural Settings dialog:

- Select the Load Cases tab.
- Under Load Cases, click Add.
  A new load case is added to the bottom of the table, and the Add button changes to Duplicate.
- Click the new load case, and type **Roof Hung**.

NOTE The Case Number column of the table is read-only. Revit Structure automatically enters a default number.

- Under Category, select Dead Loads.

NOTE You can also create a new load case with the Duplicate command. Select an existing load case record in the table, then click Duplicate. The selected load case is duplicated and copied in the table.

Add a load nature

3 In the Structural Settings dialog:

- Under Load Natures, click Add.
  A new load nature record is added to the bottom of the table.
- Click the new load nature, and type **Roof Hung**.

NOTE The new load nature displays and is now available under the Nature column of the Load Cases table.

- Click OK.

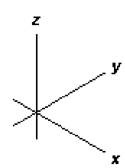
Add an area load to the entry level slab

1 In the Project Browser, expand 3D Views, and double-click 3D.

2 Click Analyze tab ➤ Loads panel ➤ Loads.

Load modelling accuracy depends on the type of coordinate system specified for the project. Revit Structure uses several coordinate systems for loads, such as:

- The Project coordinate system (X, Y, Z) that displays in the view when you click Loads.
- The work plane that is the current plane of object placement.
- The host work plane that is the plane in which the element you select to host a load resides.

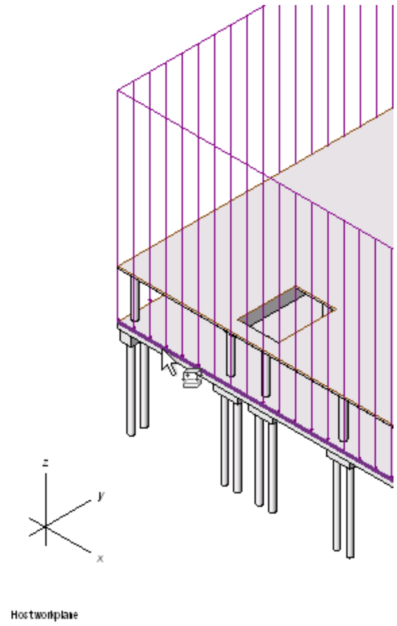


Host workplane

3 Click Loads panel ➤ Hosted Area Load.

4 Click Element panel ➤ Change Element Type drop-down ➤ Area Loads: Area Load 1.

5 Select the edge of the slab on the entry level.

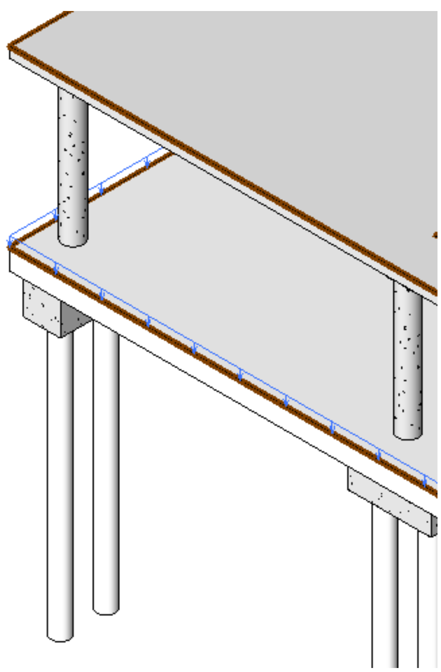


6 Click Selection panel ➤ Modify.

Modify the load parameter

7 Right-click the load, and click Element Properties.

8 In the Instance Properties dialog, type **-0.0300 ksf** for the Fz 1 parameter, and click OK.

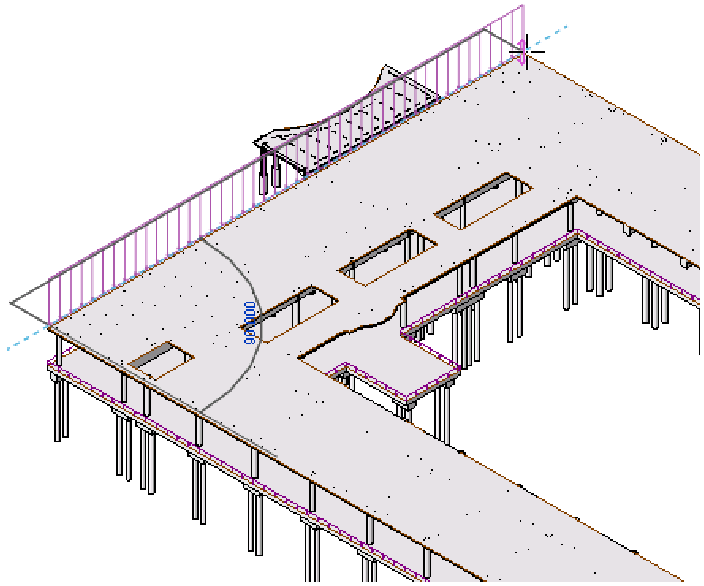


Add line load to Level 2

9 Click Analyze tab ➤ Loads panel ➤ Loads.

10 Click Loads panel ➤ Line Load.

11 Sketch a line load from one corner of the slab to the other, as shown.



12 Click Selection panel ➤ Modify.

13 Right-click on the Line Load 1, and click Element Properties.

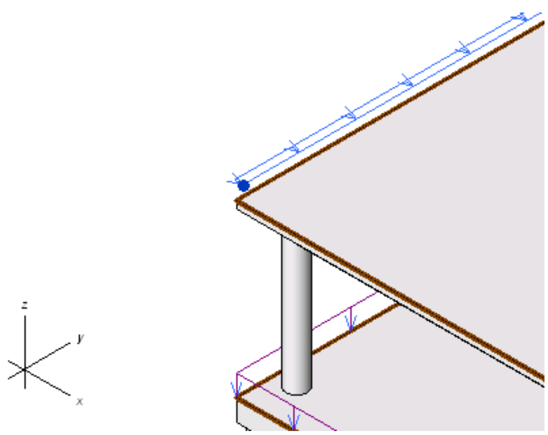
TIP You may need to press *Tab* to highlight the line load. Watch the Status Bar to be sure you highlight Line Load 1.

14 In the Instance Properties dialog, click Edit Type.

15 In the Type Properties dialog, click Duplicate, for Name, type **Wind Load**, and click OK twice.

16 In the Instance Properties dialog:

- Type **0 kip/ft** for Fz 1.
- Type **0.040 kip/ft** for Fx 1.
- Select WIND1(3) for Load Case, and click OK.

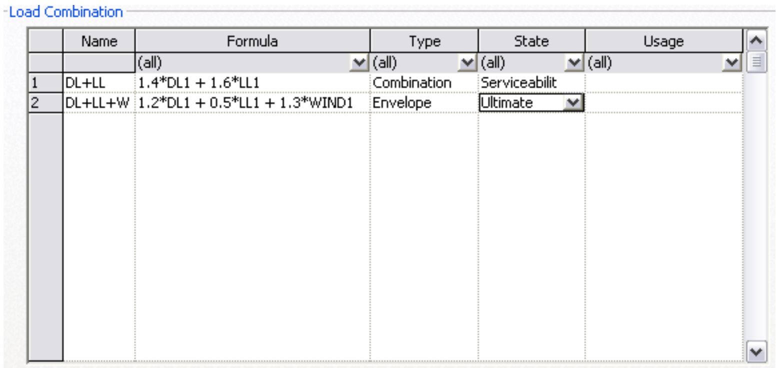


17 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Load Combinations

In this exercise, you add a load combination to your model for use by the analysis and design software.


Load Combination

	Name	Formula	Type	State	Usage
		(all)	(all)	(all)	(all)
1	DL+LL	1.4*DL1 + 1.6*LL1	Combination	Serviceabilit	
2	DL+LL+W	1.2*DL1 + 0.5*LL1 + 1.3*WIND1	Envelope	Ultimate	

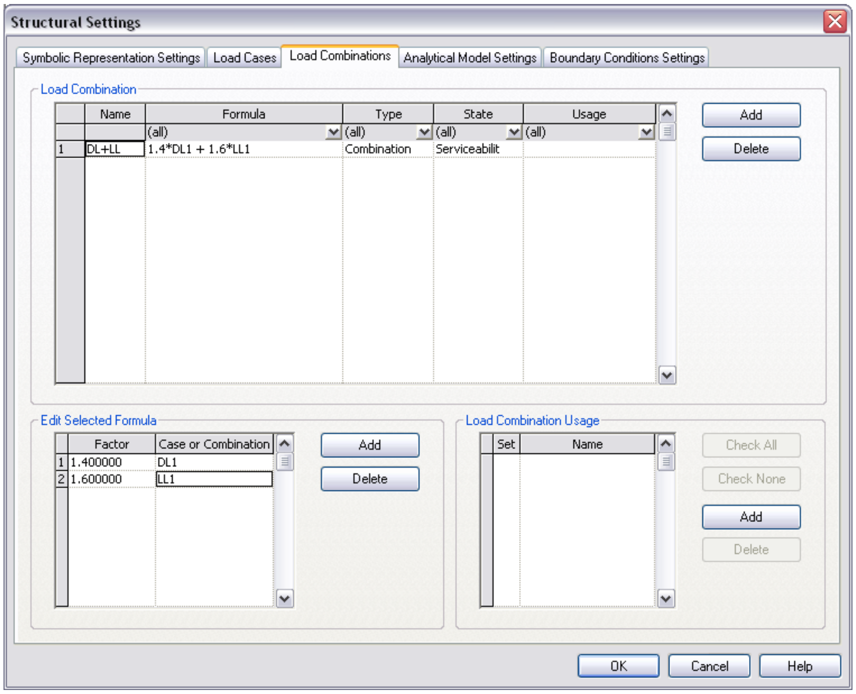
Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_AYP_03_Load_Combos_i.rvt.

Add a DL + LL combination

1. Click Manage tab ➤ Project Settings ➤ Structural Settings.
2. In the Structural Settings dialog, click the Load Combinations tab.
3. Under Load Combination, click Add.
4. Under Name, for New Combination, type **DL+LL**.
5. Under Edit Selected Formula, click Add.
6. Under Factor, type **1.4**.

   The Formula field under Load Combination automatically displays the new factor.
7. Under Case or Combination, select DL1.
8. Click Add.
9. Under Factor, type **1.6**.
10. Under Case or Combination, select LL1.



Add a DL + LL + wind combination with a factor

11 Under Load Combination, click Add.

12 Under Name, for New Combination, type **DL+LL+WIND**.

13 Under Edit Selected Formula, click Add.

14 Under Factor, and type **1.2**.

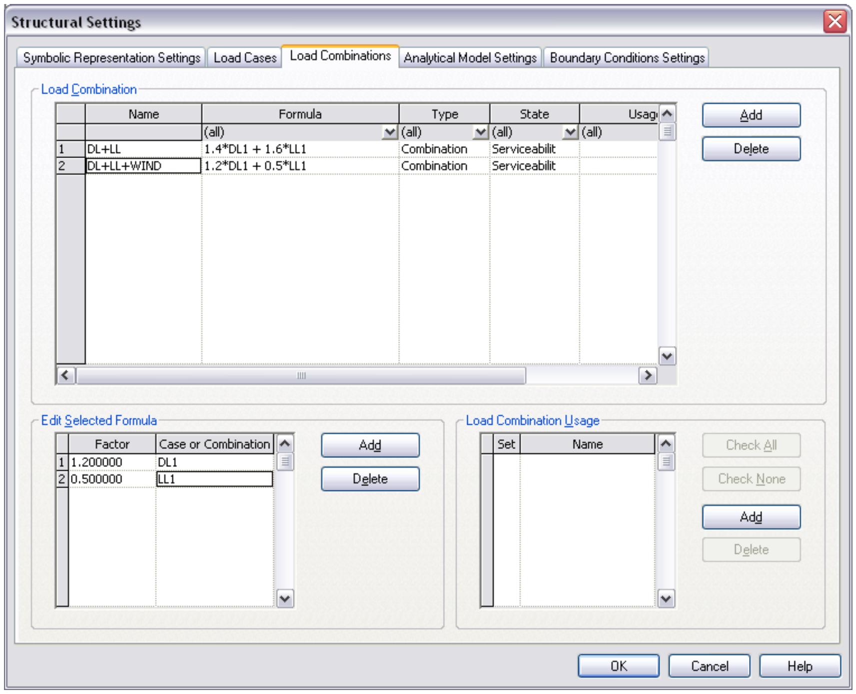
15 Under Case or Combination, select DL1.

Add a LL1 combination with a factor

16 Under Edit Selected Formula, click Add.

17 Under Factor, type **0.5**.

18 Under Case or Combination, select LL1.



Add a WIND1 combination with a factor

19 Under Edit Selected Formula, click Add.

20 Under Factor, type **1.3**.

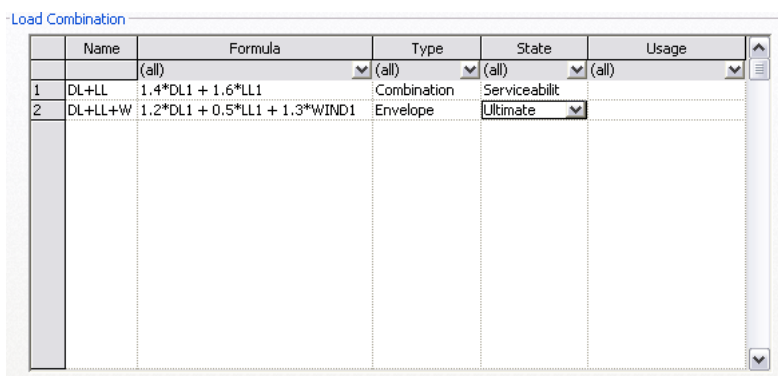
21 Under Case or Combination, select WIND1.

22 Under Load Combination, select Row 2.

23 Under Type, select Envelope.

Setting the load combination to Envelope gives maximum and minimum results for a group of load combinations.

24 Under State, select Ultimate.



25 In the Structural Settings dialog, click OK.

26 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Transfer Project Standards

In this exercise, you transfer the load combination table to a new project.

Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_AYP_04_Project_Standards_i.rvt.

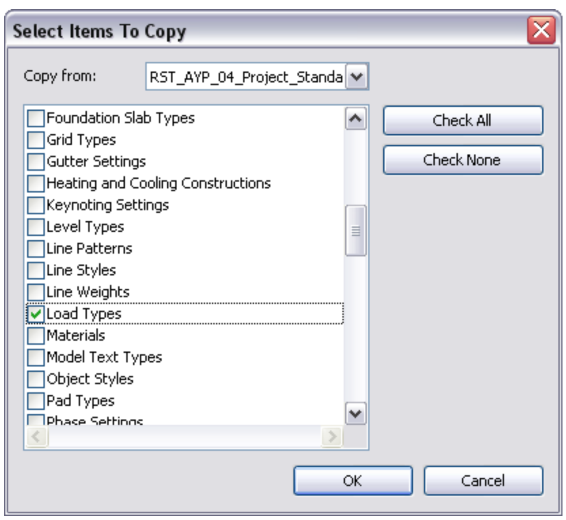
Importing load types and combinations into a new project

1 Click  ➤ New ➤ Project.

2 In the new project, click Project Settings panel ➤ Transfer Project Standards.

3 In the Select Items to Copy dialog:

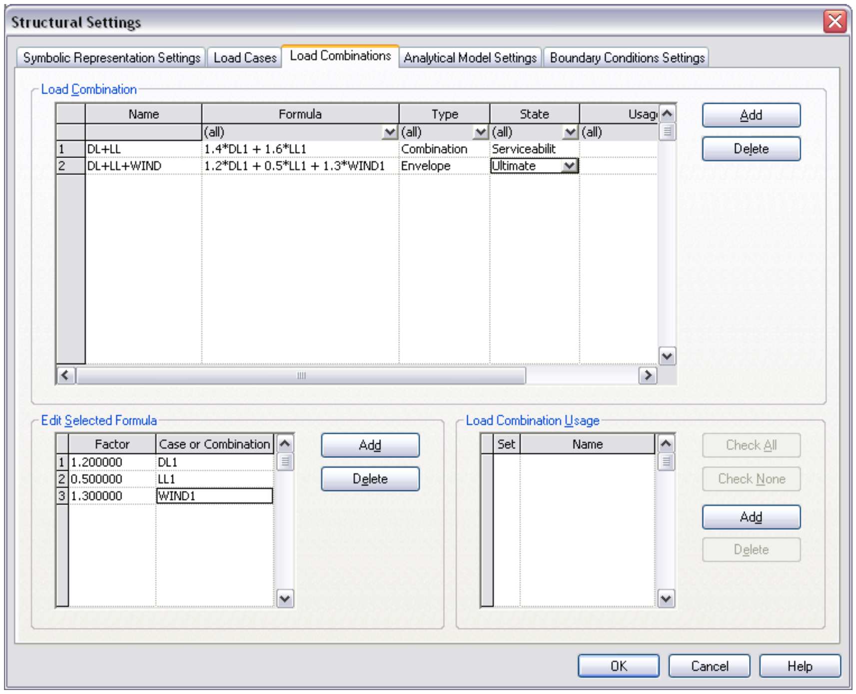
- Under Copy from, select RST_AYP_04_Project_Standards_i.rvt.
  This file contains the project standards that you want to copy into the new project.
- Click Check None.
- Scroll down the list of items and select Load Types.
- Click OK.



4 Click Project Settings panel ➤ Structural Settings.

5 In the Structural Settings dialog, click the Load Combinations tab.

The transferred load combinations display.



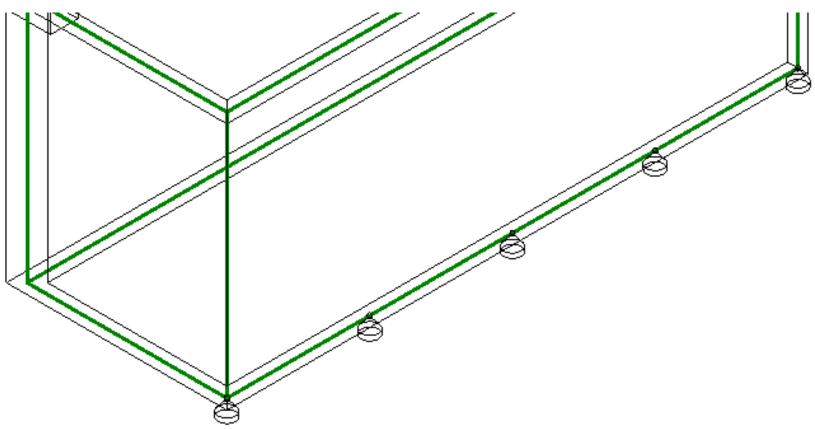
6 Click OK.

7 Close both open files with or without saving them.

In the next exercise, a new training file is supplied.

Boundary Conditions

In this exercise, you add a boundary condition to your model. Boundary conditions are analytical model elements that define the support conditions of a structural element by its surrounding environment. These elements communicate engineering assumptions about support conditions to analysis software packages. Boundary conditions are also known as supports or restraints in some analysis packages.



Training File

- Click  ➤ Open ➤ Project.

■ In the left pane of the Open dialog, click Training Files, and open Imperial\RST_AYP_05_Boundary_Conditions_i.rvt.

Change the view visibility

1 In the Project Browser ➤ 3D Views, double-click View 1 - Analytical.

2 Type **ZF** (Zoom to Fit).

3 Click View tab ➤ Graphics panel ➤ Visibility/Graphics.

4 In the Visibility/Graphic Overrides dialog, under Visibility, clear Structural Foundations and Floors, click Apply, and then click OK.

5 Zoom in around the columns located on the lower-right corner of the structure.

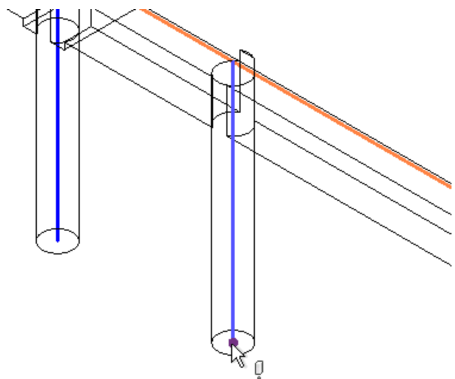
Add boundary conditions to the base of the structural columns

6 On the Modelling tab of the Design Bar, click Boundary Conditions.

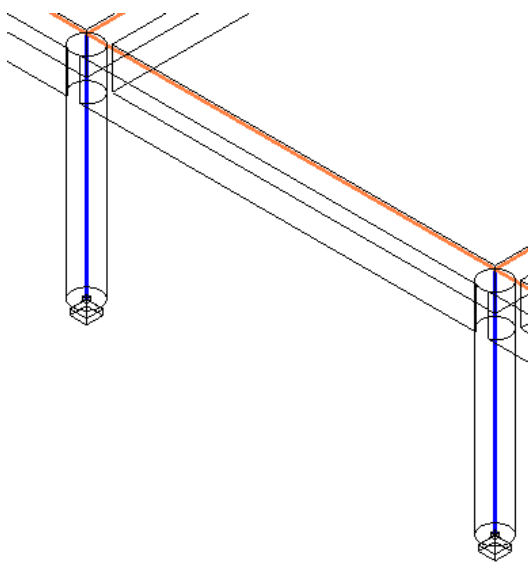
7 Click Analyze tab ➤ Boundary Conditions panel ➤ Boundary Conditions.

8 Click Boundary Conditions panel ➤ Point.

9 Click the end of the blue line (representing the analytical model of the beam) to place the point boundary condition.



10 Repeat this technique to add a fixed boundary condition to the next closest beam as shown.



11 Click Selection panel ➤ Modify.

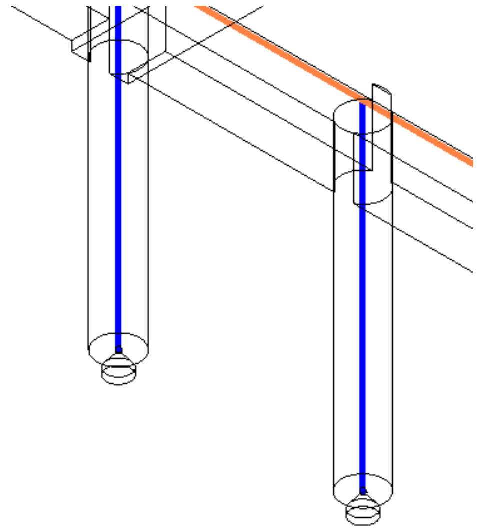
Change the boundary condition properties

12 Click a boundary condition symbol, press *Ctrl*, and select the other symbol.

13 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

14 In the Instance Properties dialog, under Structural Analysis, for State, select Pinned, and then click OK.

15 Press *Esc*.



The boundary conditions display a pinned state symbol.

Add a line boundary condition to the base of the structural wall

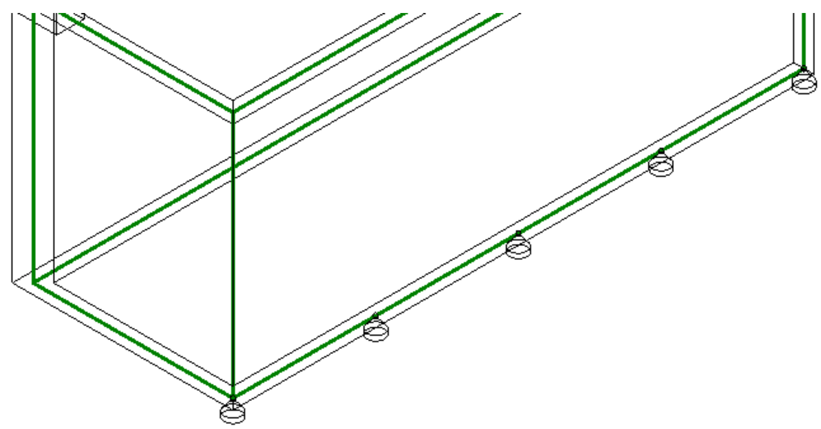
16 Click or drag the ViewCube to rotate the view until the foundation wall displays.

17 Click Boundary Conditions panel ➤ Boundary Conditions.

18 Click Boundary Conditions panel ➤ Line.

19 On the Options Bar, for State, select Pinned.

20 Click the green line (representing the analytical model of the foundation) to place the line boundary condition.



21 Click Selection panel ➤ Modify.

22 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Completing a Project

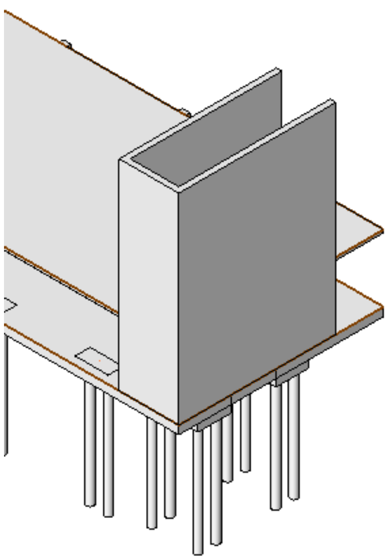
In this tutorial, you complete the project by extending the model to the roof level and adding a slope modified slab.

Completing the Structure

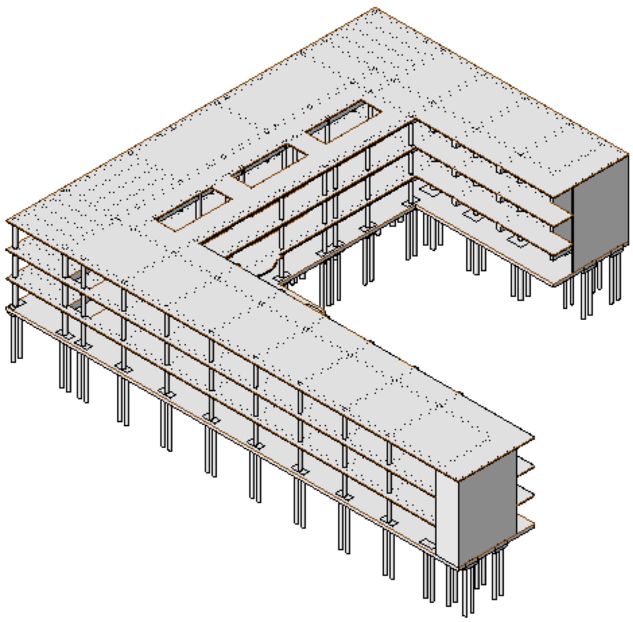
15

In this lesson, you complete the structure by extending all structural elements to the roof level. You learn to:

- Extend the concrete walls of the northeast and southeast stairs to the roof level of the structure by changing the top constraint.

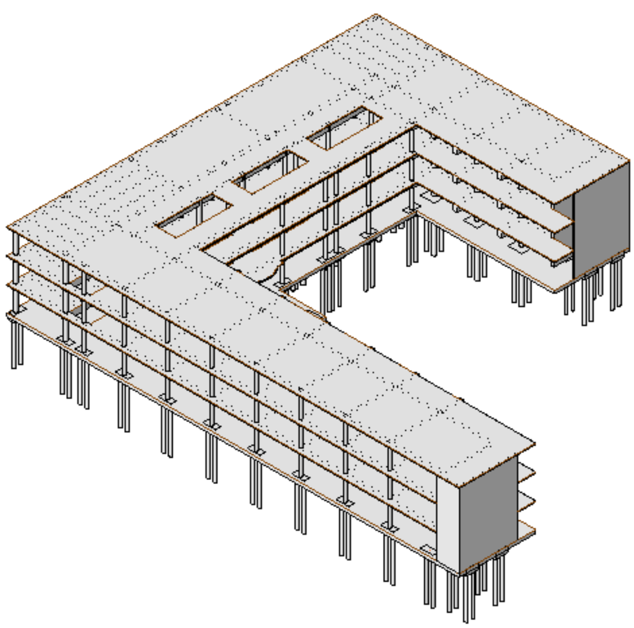


- Extend all structural elements to the roof level using the filter option.



Extending the Structure to the Roof Level

In this exercise, you copy the structural walls, columns, beams, beam systems, and the slab from Level 2 to Level 3 and the Roof. You then edit the slabs on both levels to complete the structure.



Training File

- Click ➤ Open ➤ Project.

■ In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_Extend_Structure_i.rvt.

Extend the concrete walls up to the Roof

1 In the Project Browser, under Structural Plans, double-click 02 - Floor.

2 Right-click one of the concrete walls in the northeast stairwell, and click Select All Instances.

All the concrete walls are highlighted.

3 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

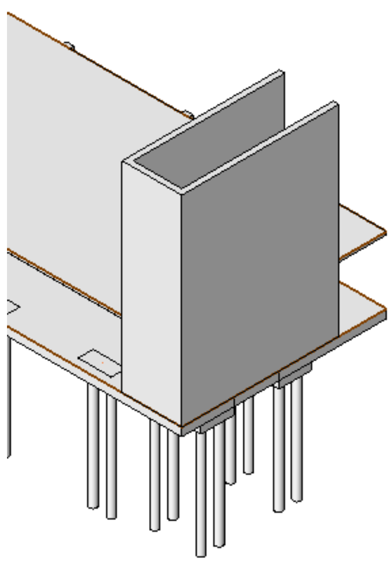
4 In the Instance Properties dialog, under Constraints:

- For Top Constraint, select Up to level: Roof.
- Click OK.

5 Press *Esc*.

6 In the Project Browser, under 3D Views, double-click 3D.

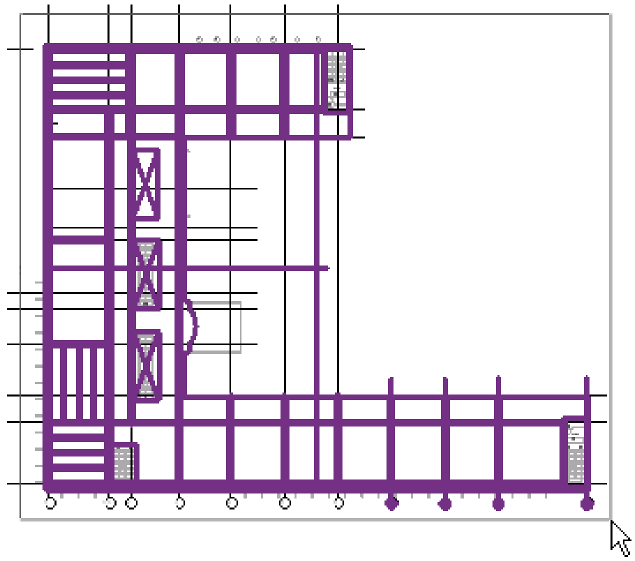
The concrete walls extend to the roof level.



Extend all structural elements to the roof

7 In the Project Browser, under Structural Plans, double-click 02 - Floor.

8 In the drawing area, click above the northwest corner of the structure and draw a pick box around the entire floor plan.



NOTE Do not highlight elements from the entry way.

9 Click Filter panel ➤ Filter.

10 In the Filter dialog:

- Clear Shaft Openings.
- Clear Walls.
- Click OK.

Only the selected items remain highlighted in the drawing area.

11 Click Clipboard panel ➤ Copy.

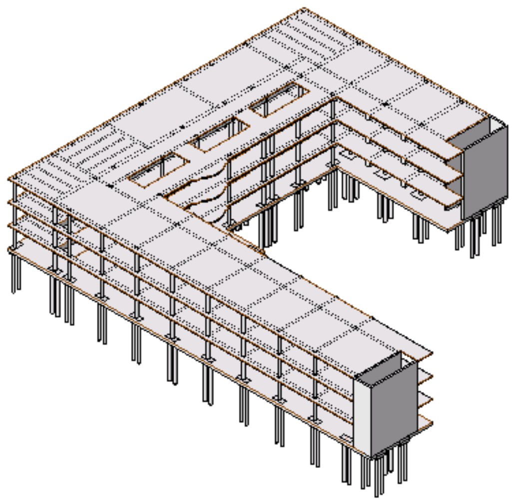
12 Click Clipboard panel ➤ Paste Aligned drop-down ➤ Select Views.

13 In the Select Views dialog:

- Select Structural Plan: 03-Floor.
- While pressing *Ctrl*, select Structural Plan: Roof.
- Click OK.

14 In the Project Browser, under 3D Views, double-click 3D.

All the structural elements now extend to the roof level.

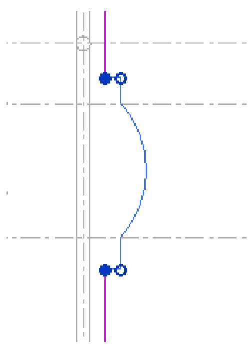


Edit the level 3 slab

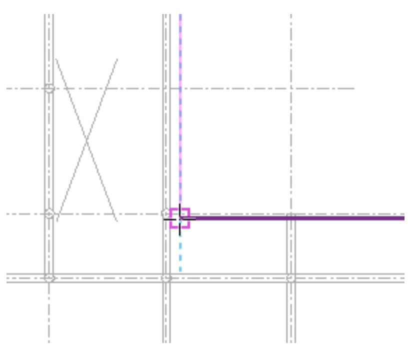
15 In the Project Browser, under Structural Plans, double-click 03- Floor.

16 Edit the slab boundary as follows:

- Zoom in on the balcony ledge between grids F and G.
- Select the slab.
- Click Edit panel ➤ Edit Boundary.
- Select the 5 sketch lines that form the balcony ledge, including the line that extends to grid line H.



- Press *Delete*.
- Click and extend the slab sketch line to connect with the existing line on grid line H to complete the sketch loop.

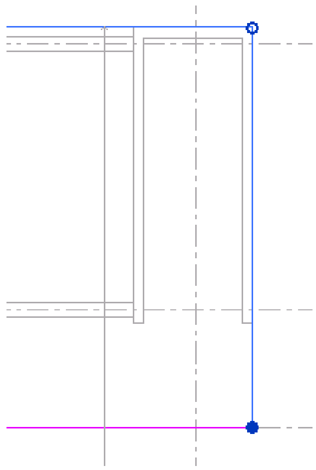


NOTE Do not duplicate sketch lines when editing the slab. If you receive a warning, check to make sure you have not sketched a line twice.

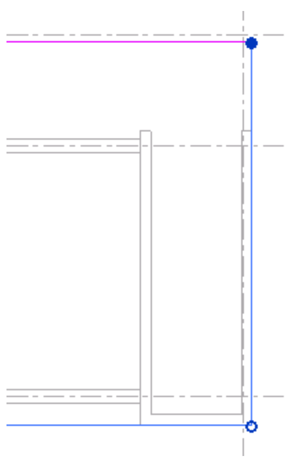
Edit the Roof slab

17 Edit the Roof slab as follows:

- In the Project Browser, under Structural Plans, double-click Roof.
- Using the same method as described in the previous step, modify the slab boundary by deleting the balcony ledge.
- Zoom in to the northeast stairs.
- Modify the corner of the slab as shown.



- Zoom in to the southeast stairs.
- Modify the corner of the slab as shown.



- Click Floor panel ➤ Finish Floor.

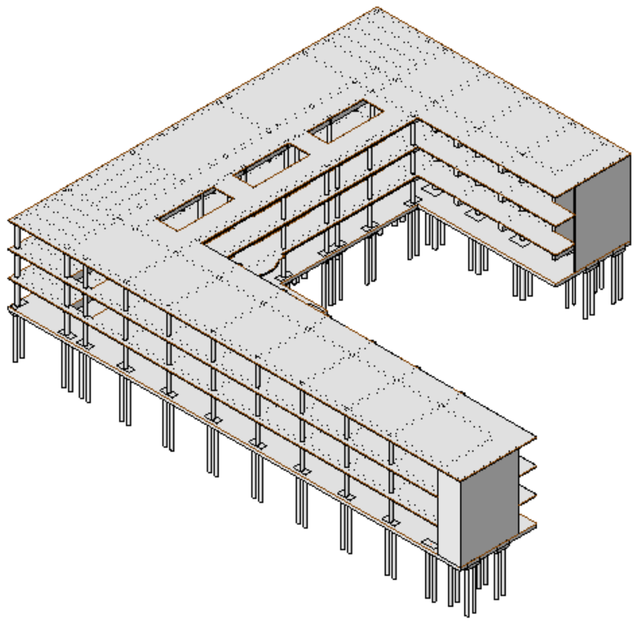
NOTE In the Revit dialog, click Yes to attach the walls to the bottom of the roof level.

18 Press *Esc*.

19 In the Project Browser, under 3D Views, double-click 3D.

20 Enter **ZF** (Zoom to Fit).

The structural elements now extend to the roof level.



21 Close the file with or without saving it.

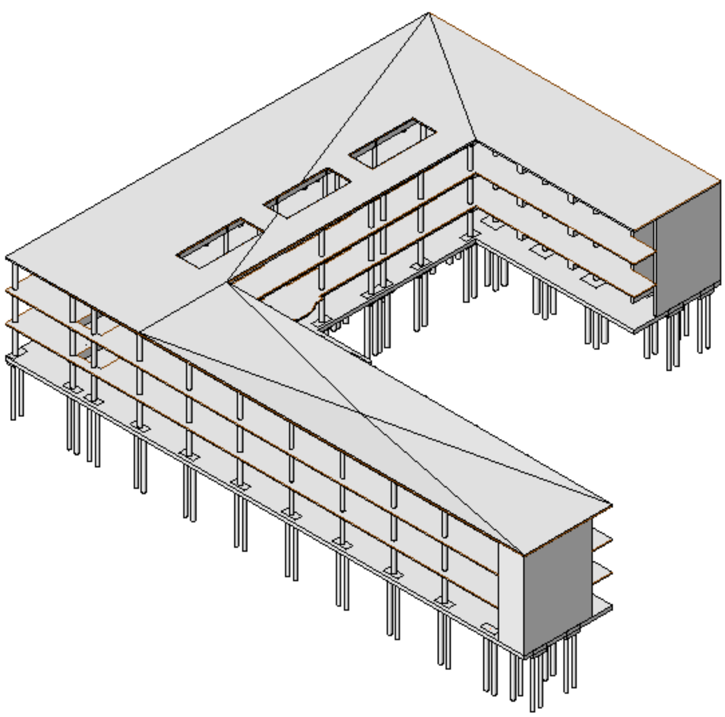
In the next exercise, a new training file is supplied.

Adding a Shape-Modified Slab

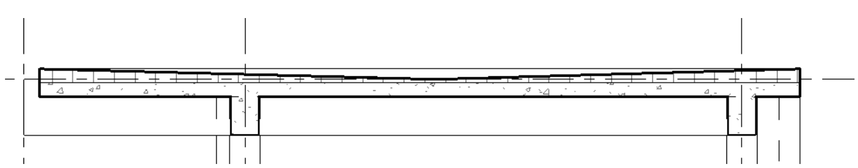
16

In this lesson, you add a shape-modified slab to the roof of the structure. You learn to:

- Add a slope to the roof slab to allow for roof drainage.

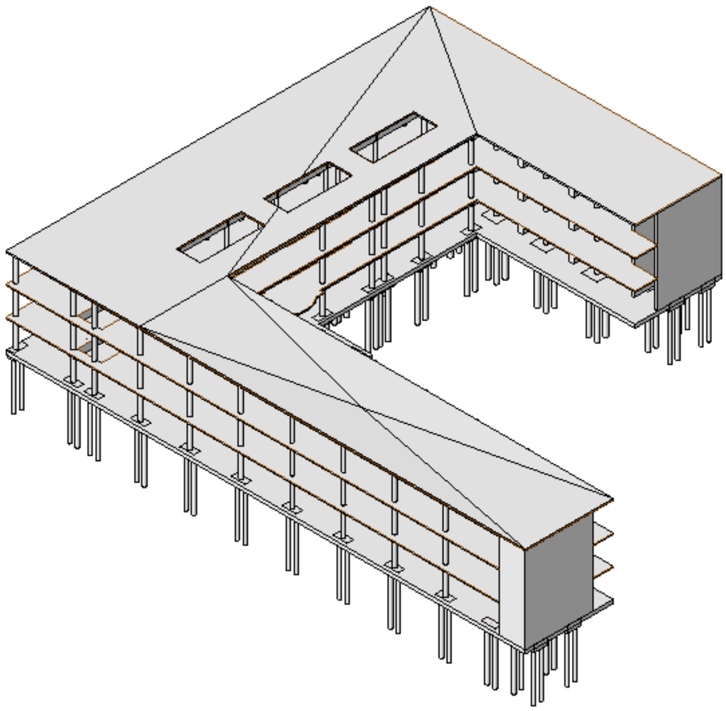


- Add a new insulation layer above the concrete slab.

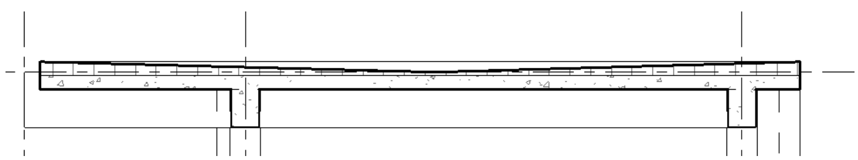


Modifying a Flat Slab

In this exercise, you add a slope to the roof slab of the structure to allow for roof drainage conditions. For this exercise, you only modify one section of the roof.



Also, you add a new insulation layer. You modify the slab construction to more accurately reflect the actual construction of the roof and the way the slope will be accommodated in the insulation of the roof.



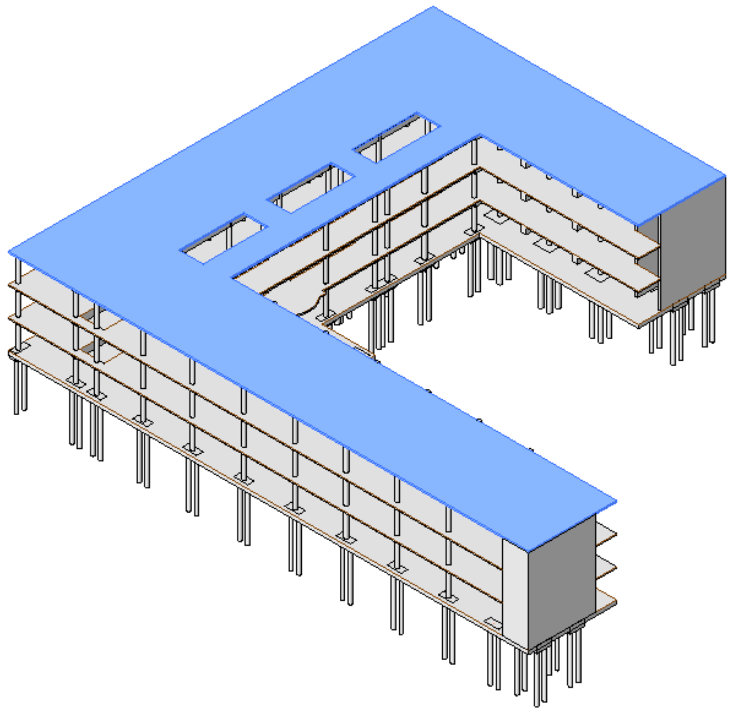
Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_Slope_Slab_i.rvt.

Draw ridge lines

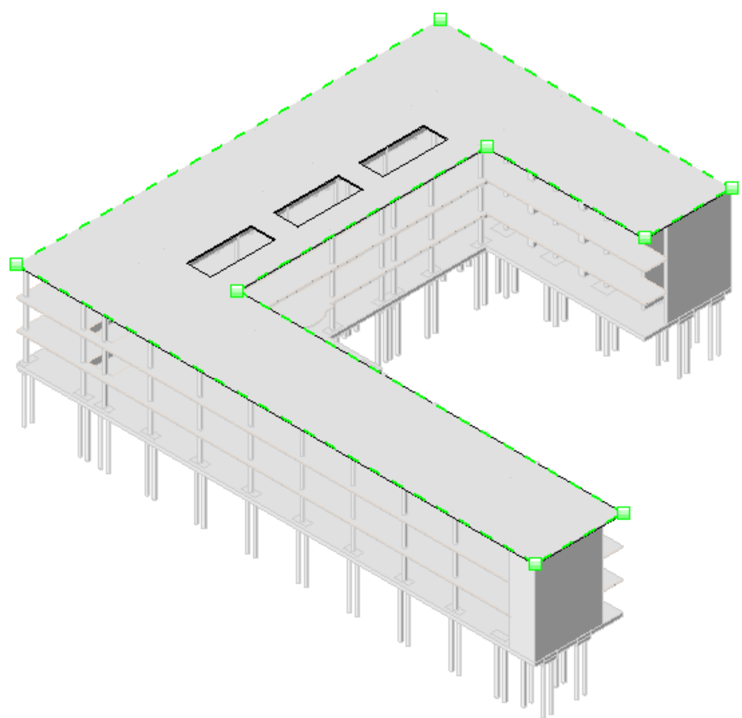
1 In the Project Browser, under 3D Views, double-click 3D.

2 Select the slab.



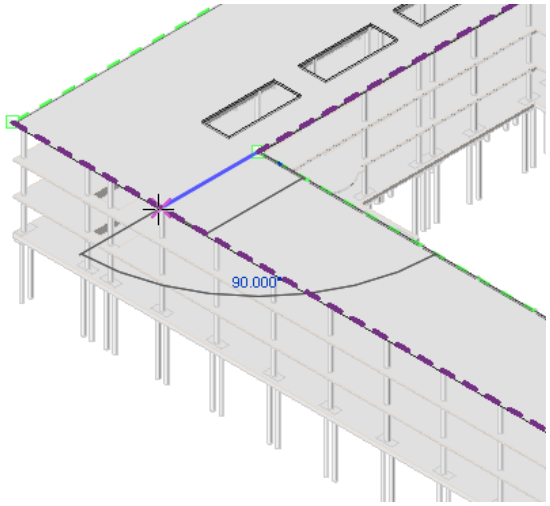
3 Click Modify Floors tab ➤ Shape Editing panel ➤ Add Split Line.

Ridge lines can be added to the slab. In this exercise, you work on a specific area of the slab. You can add additional ridge lines to a slab to accommodate conditions, as required.



4 Draw a ridge line to separate the longer wing of the building, as shown.

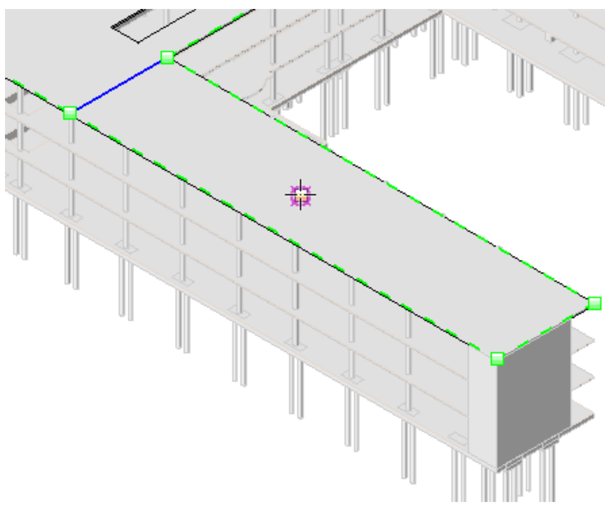
In addition to ridge lines, you can also place points on the slab surface. Then, you can use both ridge lines and points to modify the shape of the roof slab.



Place points

5 Click Modify Floors tab ➤ Shape Editing panel ➤ Add Point.

6 Click to add a point to the middle of the section of roof over the longer wing to represent the roof drain location. (Exact placement is not critical for this exercise.)

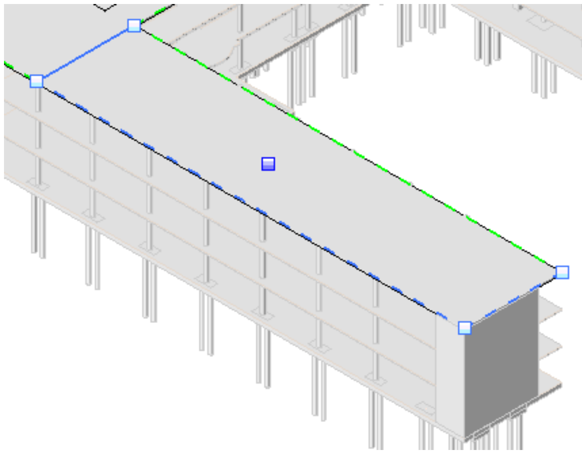


Modify roof edge elevation

7 Click Modify Floors tab ➤ Shape Editing panel ➤ Modify Sub-Elements.

Now, you can alter the elevations of edges, ridges, and points to create a shaped slab. In this case, you raise the edges around a drainage point from the original roof position.

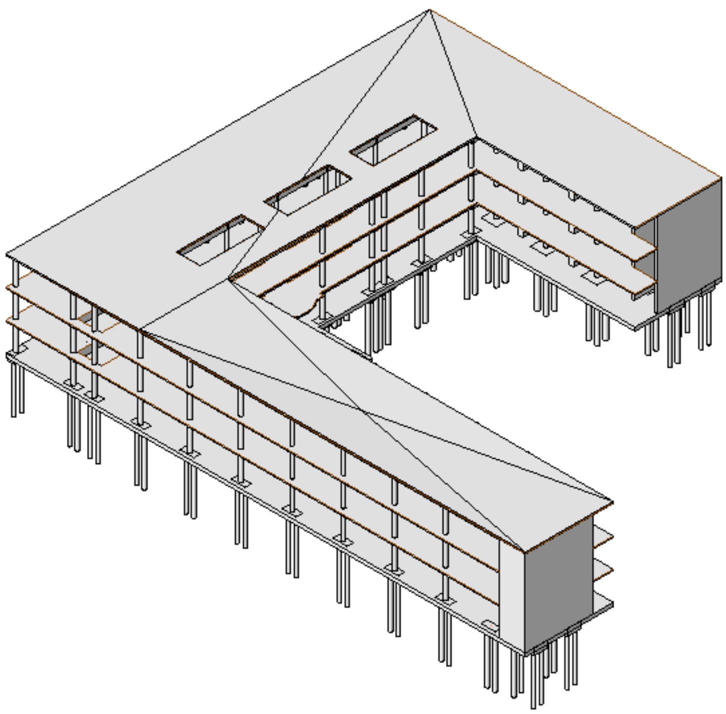
8 Place the cursor over the longer wing roof edge, press *Tab* until Chain of walls or lines displays in the Status Bar, and click to select the chain of roof edges.



9 On the Options Bar, for Elevation, type **6**", and press *Enter*.

10 Press *Esc* twice.

The roof slab now displays with edges representing the sloping faces leading down to the roof drain. You can use the same method to modify the remainder of the roof to accommodate the drainage conditions.



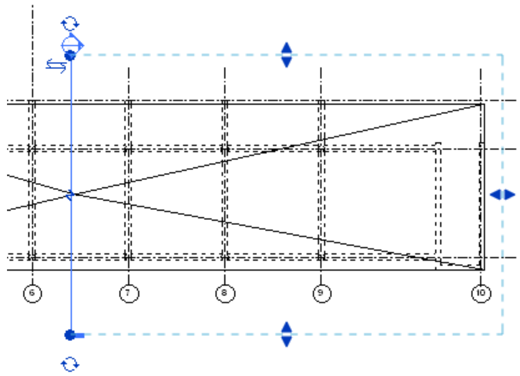
Create a section

11 In the Project Browser, under Structural Plans, double-click Roof.

12 Click View tab ➤ Create panel ➤ Section.

You create a section through the building near the low point on the sloped section of the roof.

13 Draw a vertical section between grid lines 6 and 7 through the longer wing of the building. (Exact placement is not critical.)



14 Press *Esc*.

Open the section view

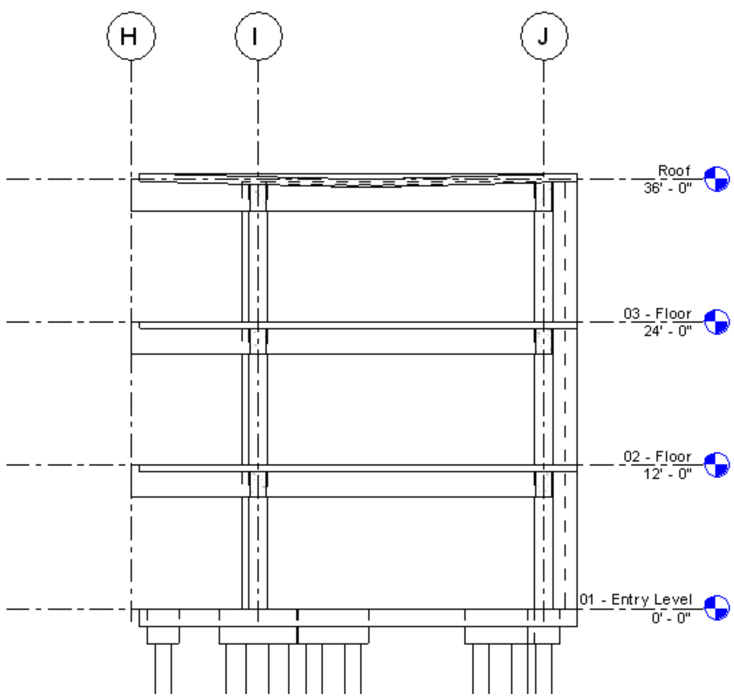
15 Double-click the section head (the circle at the top of the section line) to open the section view.

Change the section view visibility

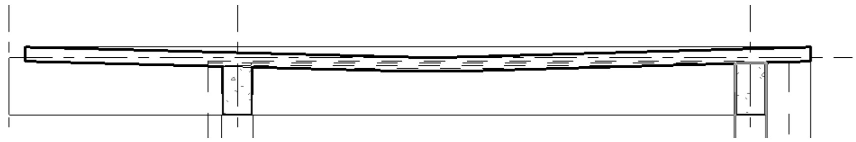
16 Click View tab ➤ Graphics Panel ➤ Visibility/Graphics.

17 On the Visibility/Graphic Overrides dialog:

- Select the Revit Links tab.
- Under Visibility, clear Technical_School-current.rvt.
- Click Apply, and then click OK.

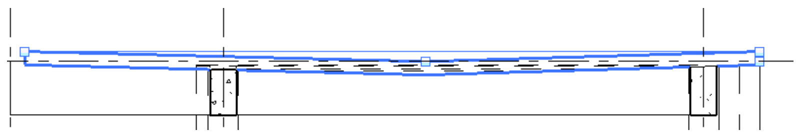


You can see in the section view, the slab is warped. The design specifies a slab that is flat along the bottom edge and that you create the slope using tapered insulation. You modify the slab construction to reflect this design.



Modify slab structure

18 Select the roof.



Because the roof type you need is a system family, you cannot load it into the project. You must create it. To create the roof type, you select the existing roof family, duplicate it, and modify the properties as needed.

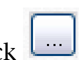
19 Click Element panel ➤ Element Properties drop-down ➤ Type Properties.

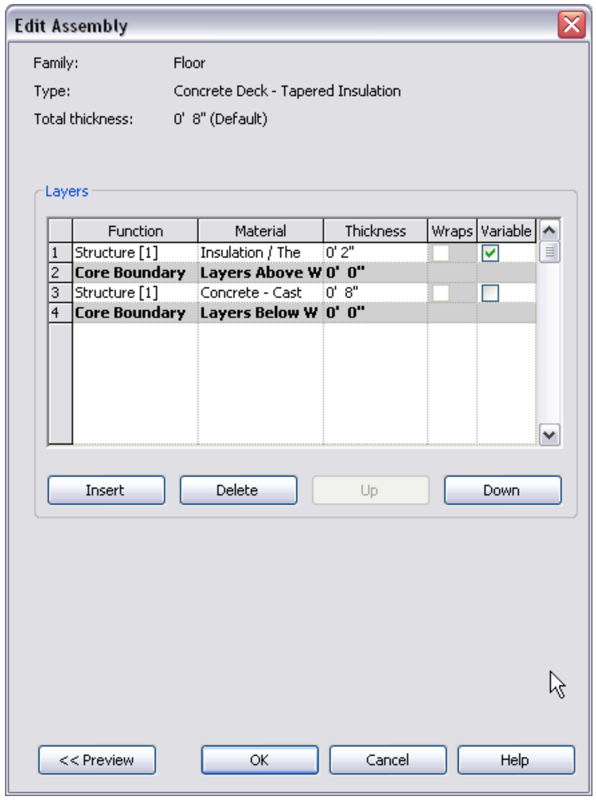
20 In the Type Properties dialog, click Duplicate.

21 In the Name field, type **Concrete Deck - Tapered Insulation**, and click OK.

22 In the Type Properties dialog, for Structure, click Edit.

23 Add an insulation layer and change the thickness and material properties:

- In the Edit Assembly dialog, select Layer 1 (Core Boundary), and click Insert.
  A new layer is added above Layer 2. This is the new insulation layer that will be tapered.
- With Layer 1 selected, click the Materials field, and click  .
- In the Materials dialog, select Insulation / Thermal Barriers - Rigid insulation, and click OK.
- In the Edit Assembly dialog, for Layer 1 (Insulation / Thermal), click Thickness, type **0' 2"**, and select Variable.
  Selecting this option allows the insulation to slope while the other layers of the roof remain horizontal.

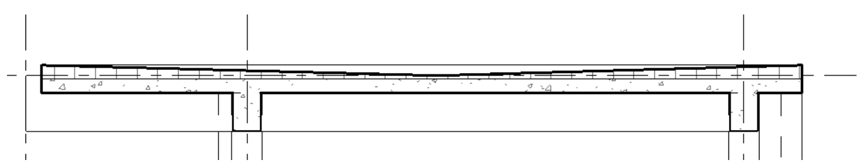


■ Click OK twice.

Change the view level detail

24 On the View Control Bar, click Detail Level ➤ Fine.

25 Zoom in to see more clearly how the insulation layer is sloping.



26 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Reinforcement Modeling

In this tutorial, you use a recommended workflow to model reinforcement in your project.

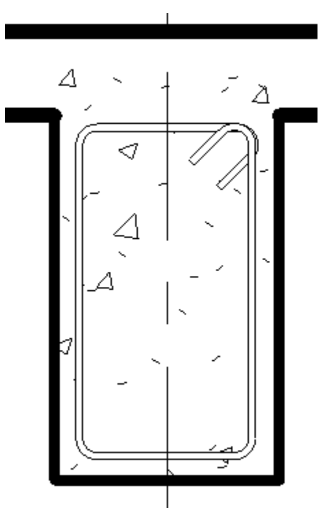
Adding Reinforcement in a Beam

17

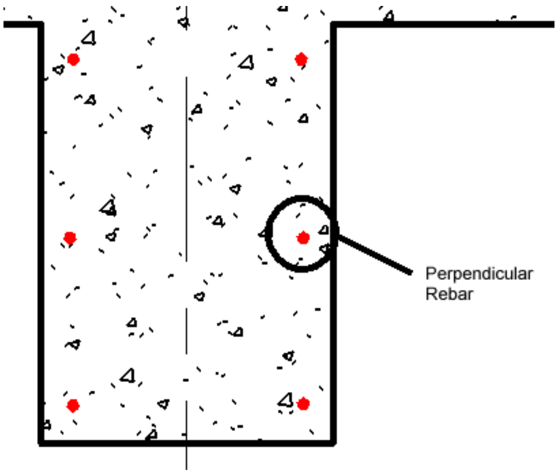
In this lesson, you create steel reinforcing bars (rebar) in a concrete beam, using the reinforcement tools and the Rebar Shape Browser available in Revit Structure. Rebar is recognized as a modelling element, much like a beam or column, and contains properties that can be customized and duplicated, based on the placement within the project. Rebar also includes a cover setting that is defined by a series of instance parameters that control the internal offset of the rebar from the faces of the rebar host (beam, wall, or column). When a rebar shape is placed within a host, it will expand to the cover settings specified. You learn to:

- Place rebar parallel to a work plane in a beam cross-section.
- Place rebar perpendicular to a work plane in a beam cross-section.
- Set the rebar cover.
- Create and view a rebar set.

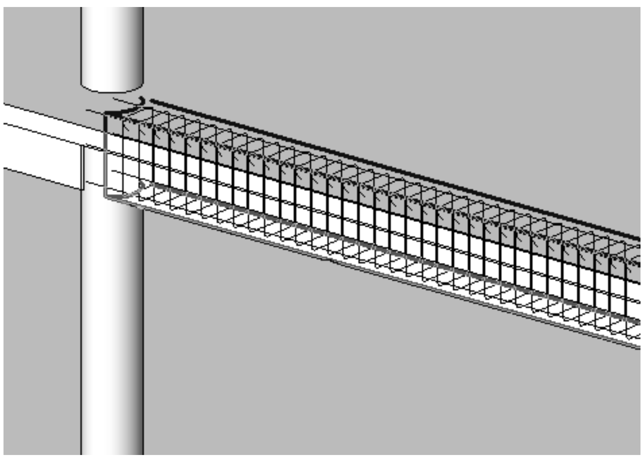
Rebar placed parallel to a work plane in a beam cross-section



Rebar placed perpendicular to a work plane in a beam cross-section



Rebar set in 3D



Placing Rebar (Parallel)

In this exercise, you place rebar in a cross-section view. The rebar is placed parallel to the exterior face of the concrete beam.

Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_REINF_Beam_01_Parallel_i.rvt.

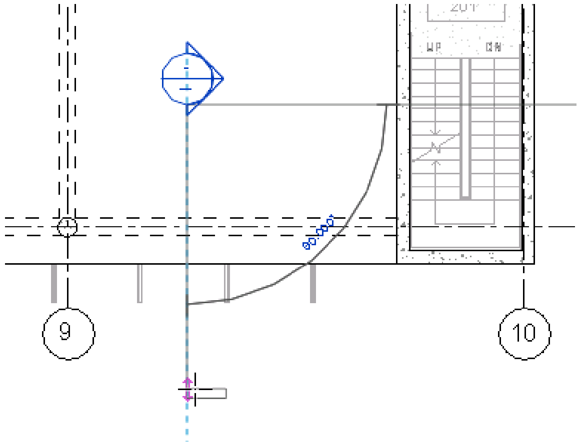
Create a section view

1 In the Project Browser, under Structural Plans, double-click 02 Floor.

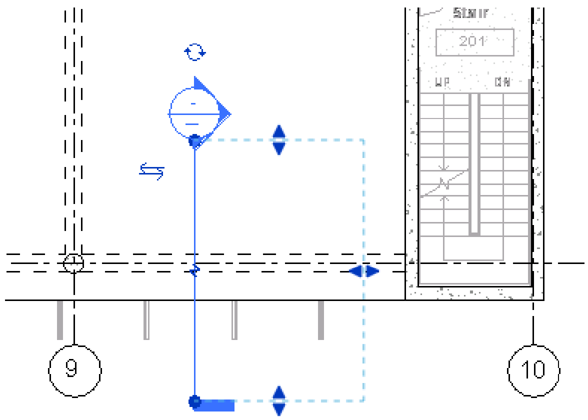
2 Zoom in to the southeast corner of the structure.

3 Click View tab ➤ Create panel ➤ Section.

4 Click inside the wall between grid lines 9 and 10, move the cursor down, and click outside the south wall to place the section.



5 Drag the right control until it is positioned to the left of the stairs, approximately as shown.



You adjust the extents of the section to include only what you want to show in the section.

6 Press *Esc*.

Change the section view visibility

7 Double-click the section head to open the section view.

8 On the View Control Bar:

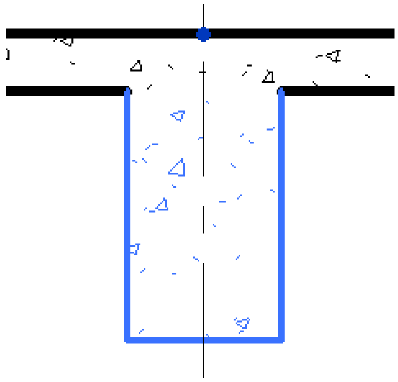
- Click Scale control, and select 1/4" = 1'-0".
- Click Detail Level ➤ Fine.
- Click  (Hide Crop Region).

9 Zoom in to the beam on level 02 - Floor.

Open the rebar shape browser

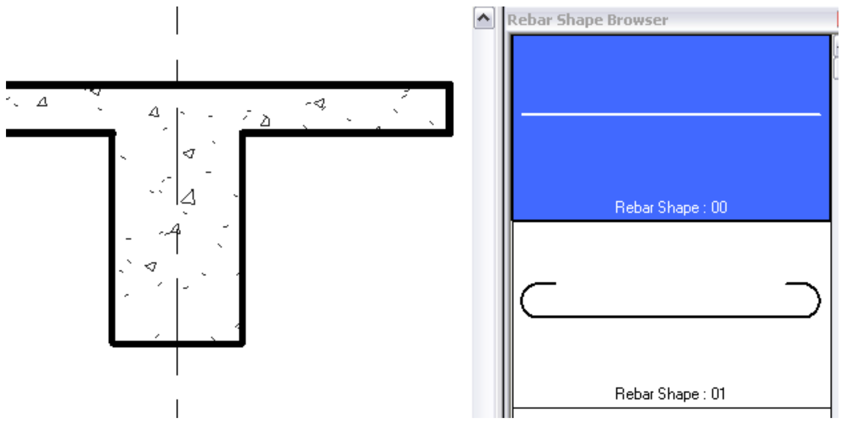
Revit Structure provides a library of rebar shapes that define the layout of the rebar type instance in the project. The shapes are displayed in a Rebar Shape Browser that displays when you select the rebar tool.

10 Select the concrete beam.



11 Click Reinforcement panel ➤ Rebar drop-down ➤ Place Rebar Parallel to Work Plane.

The Rebar Shape Browser opens, positioned to the right of the drawing area. The active rebar shape is highlighted, and corresponds to the shape specified on the Options Bar.



NOTE Open or close the Rebar Shape Browser by clicking ... on the Options Bar. The Rebar Shape Library is included in the template file that is located in the Rebar Shapes folder in the ImperialMetric Library.

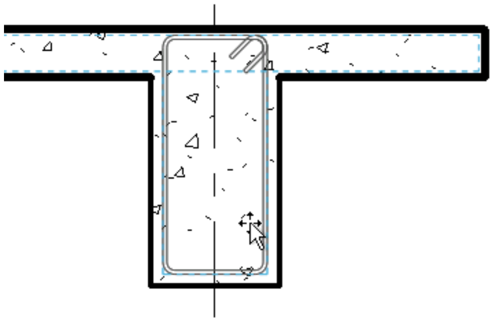
Select rebar size and shape

12 Click Element panel ➤ Change Element Type drop-down ➤ Rebar Bar : #4.

13 In the Rebar Shape Browser, select Rebar Shape : T1.

Place rebar parallel to the beam face

14 Move the cursor over the beam, and press the *Spacebar* to change the location of the rebar until it is positioned at the upper-right corner of the beam as shown.



15 Click to place the rebar.

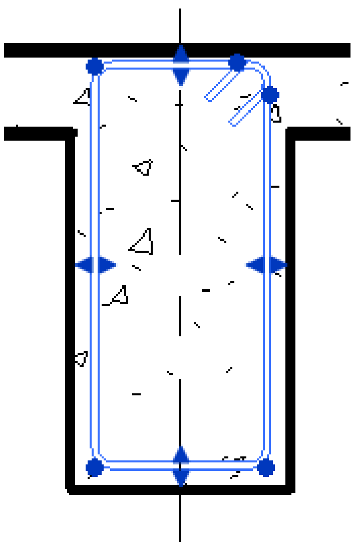
16 Press *Esc*.

Modify the rebar shape

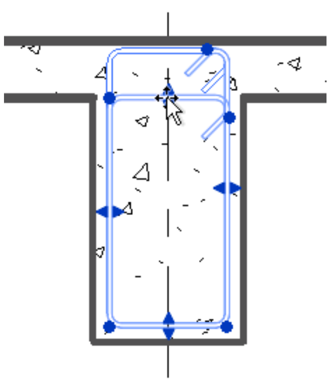
Revit Structure joins the concrete slab to the beam automatically. However, for this exercise, the rebar shape is modified so that the rebar is contained within the concrete beam.

17 Select the rebar.

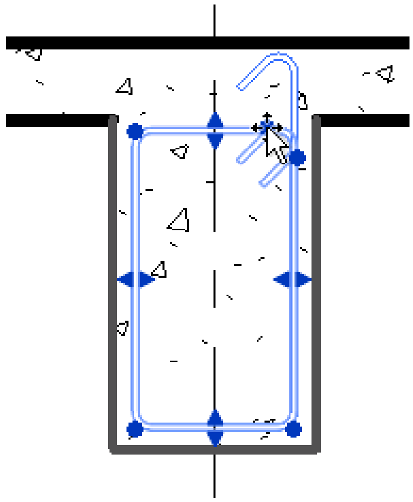
Using the triangle controls, you can pull the shape only in the direction of the control. The dot controls are multi-directional.



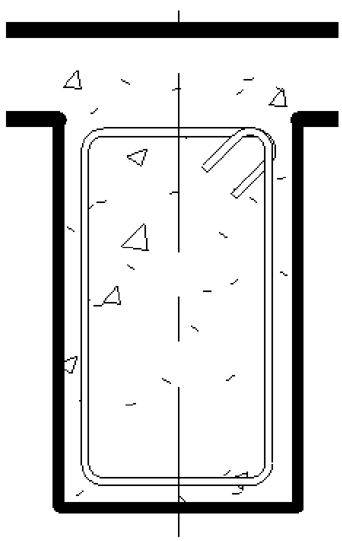
18 Click the top triangle control, and drag the rebar within the boundaries of the beam, as shown.



19 Using the same method, modify the shape of the remaining rebar.



20 Press *Esc*.



21 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Placing Rebar (Perpendicular)

In this exercise, you place rebar in a cross-section of a beam in the longitudinal direction. The rebar is placed perpendicular to the concrete beam cross-section.

Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_REINF_Beam_02_Perpendicular_i.rvt.

Open the rebar shape browser

1 In the Project Browser, under Sections (Building Section), double-click Section 2.

2 Zoom in to the beam on level 02 - Floor.

3 Select the concrete beam.

Revit Structure provides a library of rebar shapes that define the layout of the rebar type instance in the project. The shapes are displayed in a Rebar Shape Browser that displays when you select the rebar tool.

4 Click Reinforcement panel ➤ Rebar drop-down ➤ Place Rebar Perpendicular to Work Plane.

The Rebar Shape Browser opens, positioned to the right of the drawing area. The active rebar shape is highlighted. It corresponds to the shape specified on the Options Bar.

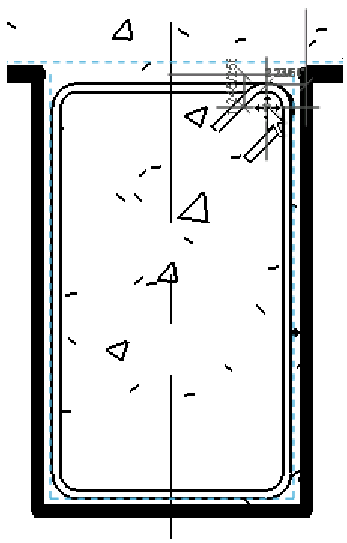
NOTE Open or close the Rebar Shape Browser by clicking ... on the Options Bar. The Rebar Shape Library is included in the template file that is located in the Rebar Shapes folder in the ImperialMetric Library.

Select the rebar shape

5 In the Rebar Shape Browser, select Rebar Shape: 00.

Place rebar perpendicular to the beam face

6 Click to place the rebar in the upper-right corner of the section view, approximately as shown.



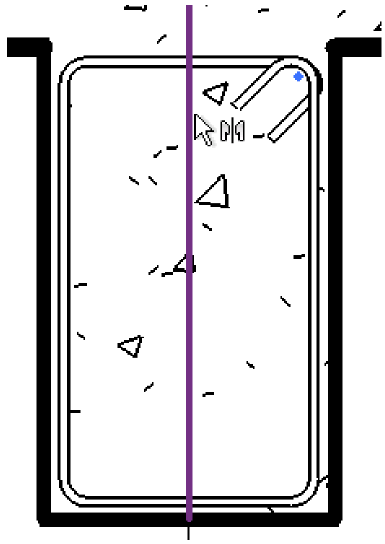
7 Press *Esc*.

Mirror the single rebar

8 Select the single rebar.

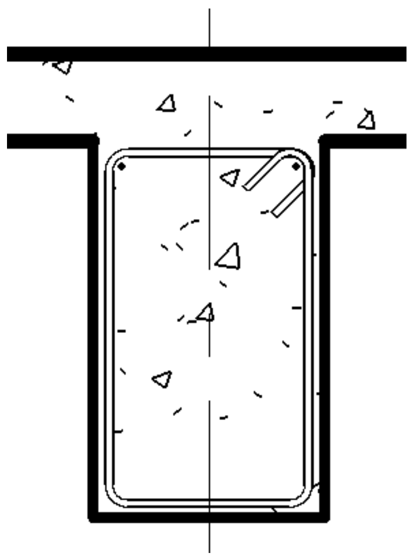
9 Click Modify panel ➤ Mirror drop-down ➤ Pick Mirror Axis.

10 Select grid line J for the axis of the (mirror) reflection as shown.



11 Click to place the rebar.

12 Press *Esc*.



Copy the rebar

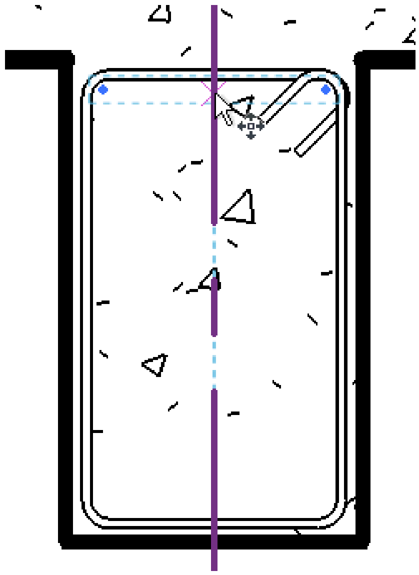
After the rebar is placed within the section view, you can easily copy it to the center and bottom locations of the beam.

13 Select the first rebar, and while pressing *Ctrl*, select the second rebar.

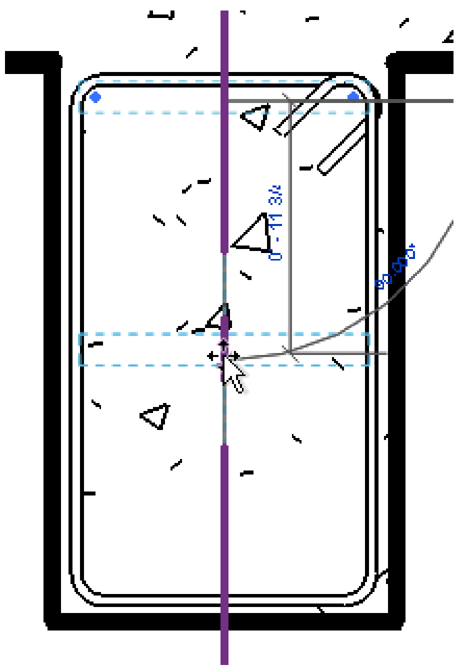
14 Click Modify panel ➤ Copy.

15 On the Options bar, click Multiple.

16 Select the highlighted rebar to define the start point.



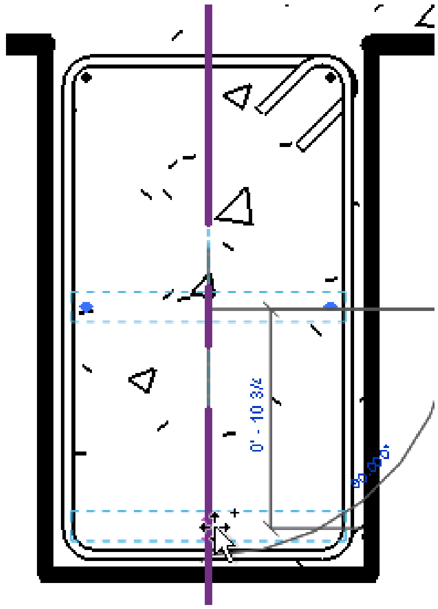
17 Move the cursor to approximately the midpoint of the beam.



18 Click to place the rebar.

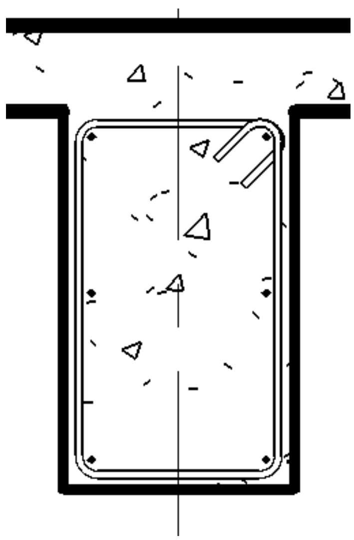
Copy the rebar to the bottom of the beam

19 Move the cursor to the bottom of the beam.



20 Click to place the rebar.

21 Press *Esc*.



22 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Setting the Rebar Cover

In Revit Structure, rebar includes a cover setting that is defined by a series of instance parameters that control the internal offset of the rebar from the faces of the rebar host (beam, wall, or column). When you place a rebar shape within a host, it expands to the cover settings specified.

In this exercise, you set rebar cover based on the bar size and by picking the face of the beam, you then create new rebar cover settings, and then apply the settings to the reinforcement placed in the concrete beam.

Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_REINF_Beam_03_Rebar_Cover_i.rvt.

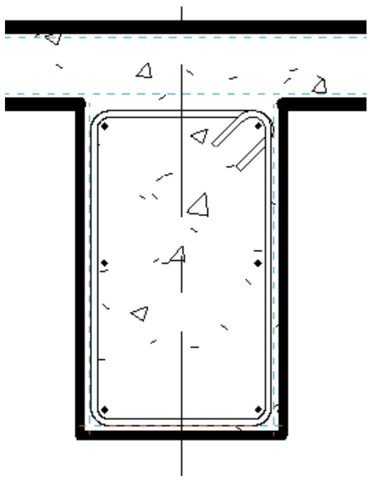
Set rebar cover based on bar size

1 In the Project Browser, under Sections (Building Section), double-click Section 2.

2 Zoom in to the beam on level 02 - Floor.

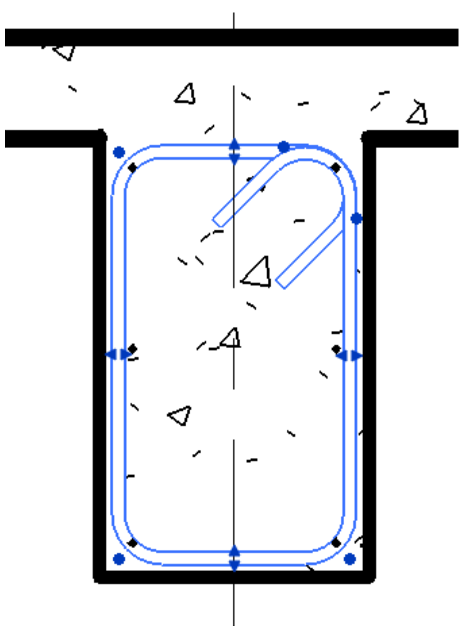
3 Click Home tab ➤ Reinforcement panel ➤ Cover.

The default rebar cover settings display in the section view as dotted lines.



4 Select the rebar, and click Element panel ➤ Change Element Type drop-down ➤ Rebar Bar: #6.

The rebar cover is automatically changed based on the new bar size.



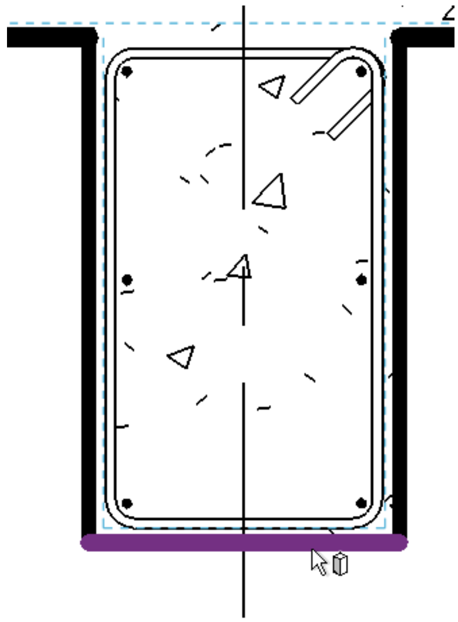
5 On the Quick Access Toolbar, click (Undo).

Set rebar cover by face

6 Click Reinforcement panel ➤ Cover.

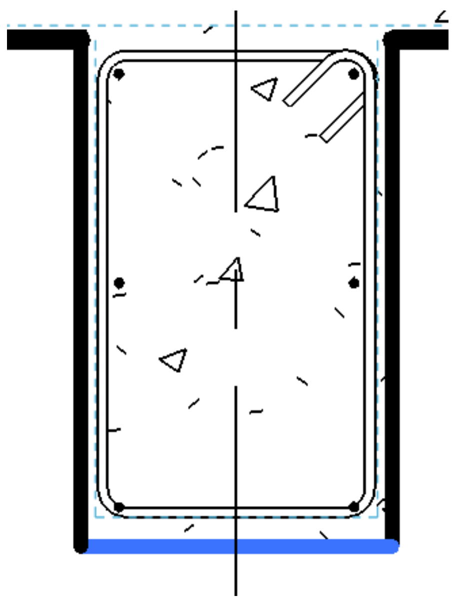
7 On the Options bar, click (Pick Faces).

8 Select the bottom face of the beam.



9 On the Options bar, for Cover Settings, select Interior (Framing, Columns).

The rebar cover changes based on the new cover setting.



10 On the Quick Access Toolbar, click (Undo).

11 Press *Esc*.

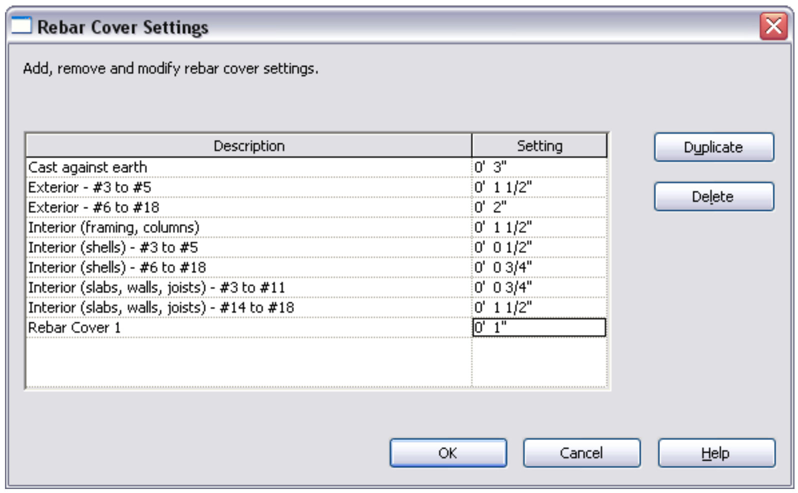
Create a new rebar cover setting

12 On the Options Bar, click ... (Edit Cover Settings).

The dialog provides a list of cover settings recommended for specific instances. For this exercise, the size you need is not listed, so you create a new cover setting.

13 In the Rebar Cover Settings dialog:

- Click Add.
- Under Rebar Cover 1, for Setting, type **1**".
- Click OK.



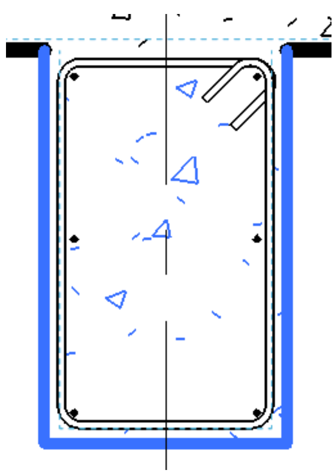
Apply the new rebar cover

14 On the Options Bar, click (Pick Elements).

15 Select the beam.

16 On the Options Bar, for Cover Settings, select Rebar Cover 1 <0' 1">.

The rebar cover settings change automatically in the section view.



17 Press *Esc*.

Select a default setting

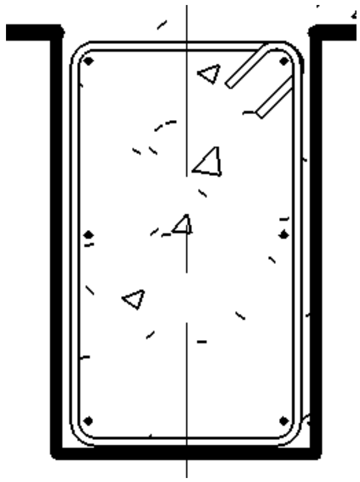
Now, you select a default setting, based on the size of the rebar that was placed in the beam. In this example, the rebar size is #4.

18 Click Home tab ➤ Reinforcement panel ➤ Cover.

19 Select the beam.

20 On the Options Bar, for Cover Settings, select Interior (shells) - #3 to #5 <0' - 0' 1/2">.

21 Press *Esc*.



22 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Creating and Viewing a Rebar Set

In this exercise, you create a rebar set within the concrete beam, and then view it in 3D.

You use rebar sets functionality, to turn a single rebar element into a linear set. You can create rebar sets perpendicular to the sketch plane of the rebar, and also define the number of bars and bar spacing.

Training File

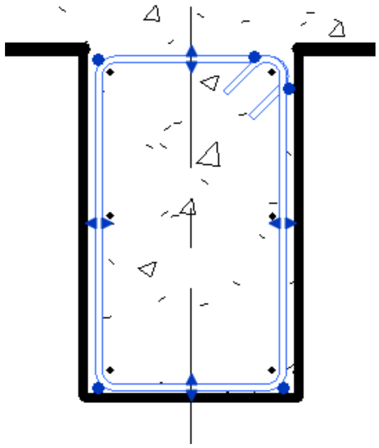
- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_REINF_Beam_04_Rebar_Set_i.rvt.

Create the rebar set

1 In the Project Browser, under Sections (Building Section), double-click Section 2.

2 Zoom in to the beam on level 02 - Floor.

3 Select the parallel rebar placed in the beam as shown.



4 On the Options Bar, for Layout, select Maximum Spacing, and for Spacing, type **6**".

5 Click Modify tab ➤ Selection panel ➤ Modify.

Select all rebar

6 Draw a pick box around the beam and structural floor.

7 Click Filter panel ➤ Filter.

8 In the Filter dialog, click Check None.

9 Under Category, select Structural Rebar, click Apply, and then click OK.

All the rebar in the section view is now selected.

Change the rebar visibility in 3D

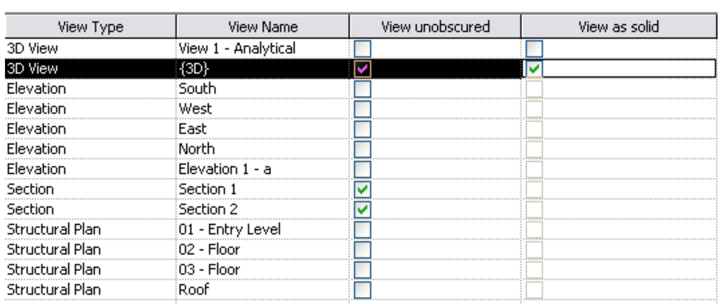
10 Click Modify Structural Rebar tab ➤ Element panel ➤ Element Properties drop-down ➤ Instance Properties.

11 In the Instance Properties dialog, under Graphics, for View Visibility States, click Edit.

12 In the Rebar Element View Visibility States dialog:

- Under View Name, select 3D, click both View unobscured and View as solid.
  The rebar displays in the drawing area regardless of any obstructions from the slab or structural walls.

- Click OK.



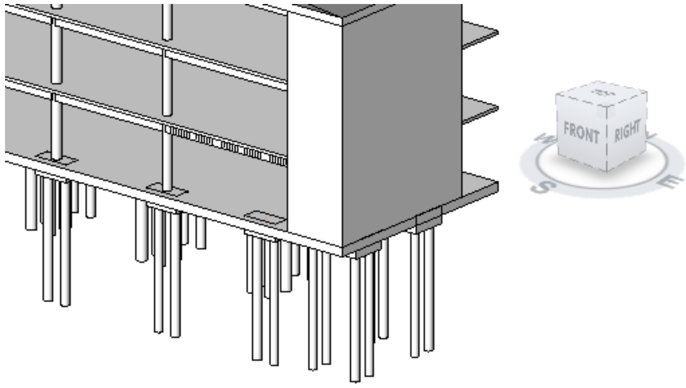
View Type	View Name	View unobscured	View as solid
3D View	View 1 - Analytical	☐	☐
3D View	{3D}	☑	☑
Elevation	South	☐	☐
Elevation	West	☐	☐
Elevation	East	☐	☐
Elevation	North	☐	☐
Elevation	Elevation 1 - a	☐	☐
Section	Section 1	☑	☐
Section	Section 2	☑	☐
Structural Plan	01 - Entry Level	☐	☐
Structural Plan	02 - Floor	☐	☐
Structural Plan	03 - Floor	☐	☐
Structural Plan	Roof	☐	☐

13 In the Instance Properties dialog, click OK.

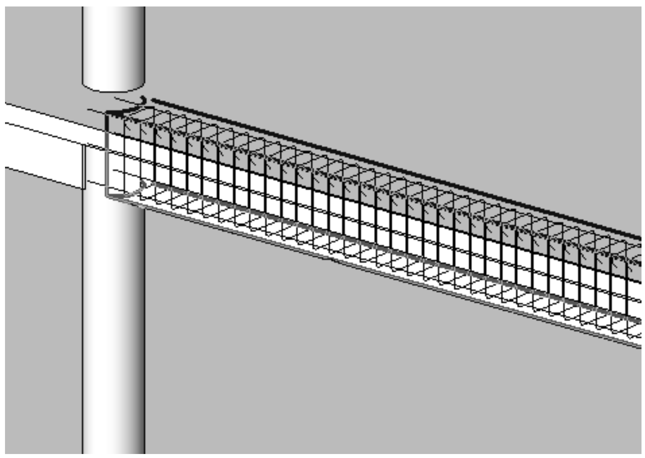
14 Press *Esc*.

15 In the Project Browser, click 3D Views ➤ 3D.

16 Use the ViewCube to rotate the model so the south-facing side of the structure is visible.



17 Zoom in to the beam located on the southeast corner of the structure to view the rebar set.



18 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

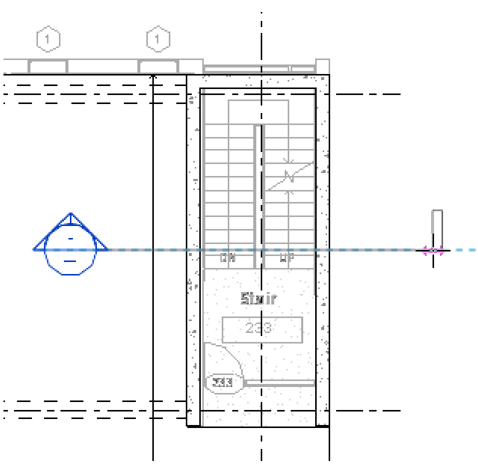
Adding Area Reinforcement

18

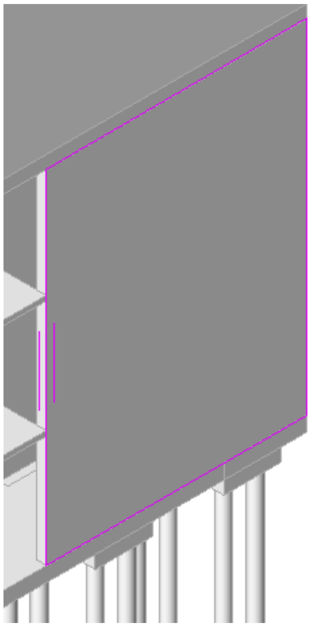
In this lesson, you add area reinforcement in the concrete wall of the northeast stairs. You use the Sketch Area reinforcement tool to place large amounts of evenly spaced reinforcing bars (rebar), and to create up to 4 layers of rebar in the host (such as a slab or wall). You learn to:

- Create a section view for placing the area reinforcement.
- Sketch the area reinforcement.
- Modify the rebar properties to remove the major bars, change the number of rebar, and change the bar type.

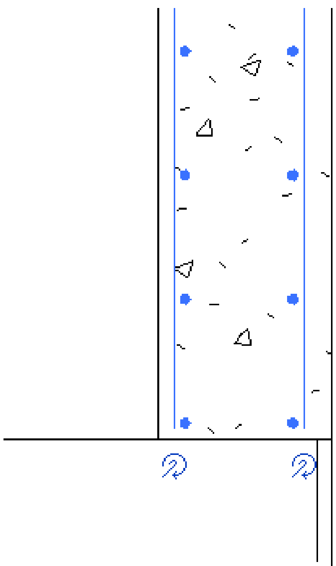
Create a section view



Sketch the area reinforcement

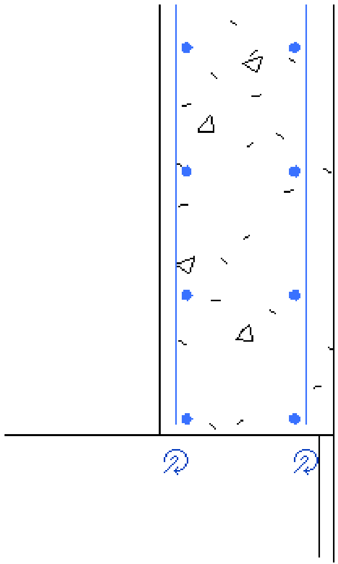


Modify area reinforcement properties



Area Reinforcement in a Structural Wall

In this exercise, you use create a section view for placing the area reinforcement in the structural wall of the northeast stairs. You then modify the rebar properties to remove the major bars, change the number of rebar, and change the bar type.



Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_REINF_Area_i.rvt.

Create a section view

1 In the Project Browser, under Structural Plans, double-click 02 Floor.

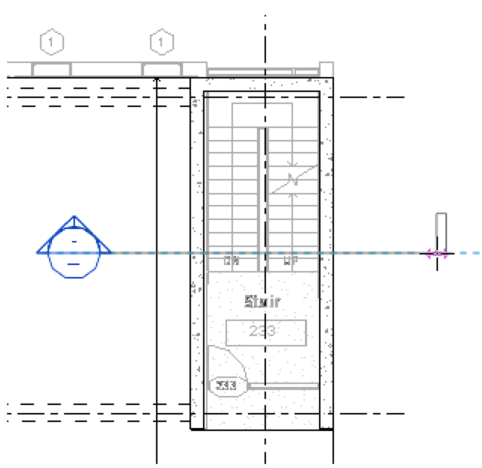
NOTE The line weights in your file may differ from the examples shown. Some of the rebar line weights were increased in the illustrations for training purposes.

---

2 Zoom in to the northeast corner of the structure.

3 Click View tab ➤ Create panel ➤ Section.

4 Add a section line that cuts through both walls of the stairs on the northeast corner of the structure.



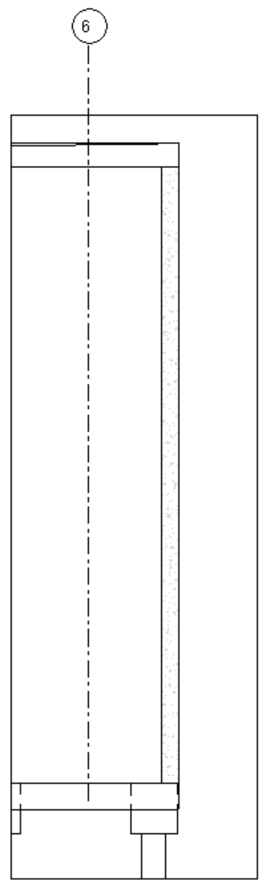
5 Press *Esc*.

Change the section view visibility

6 Double-click the section bubble to open the section view.

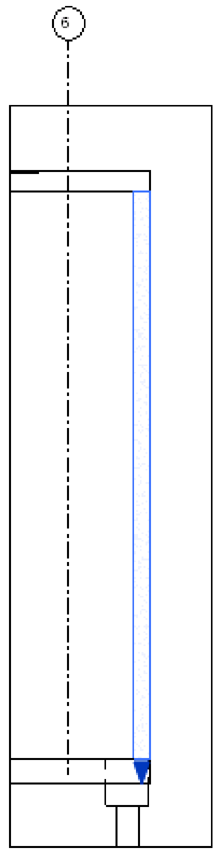
7 On the View Control Bar, select 1/4" = 1' - 0" for Scale.

8 In the section view, select the crop view, and drag the controls to resize the view so that only the concrete wall is visible.



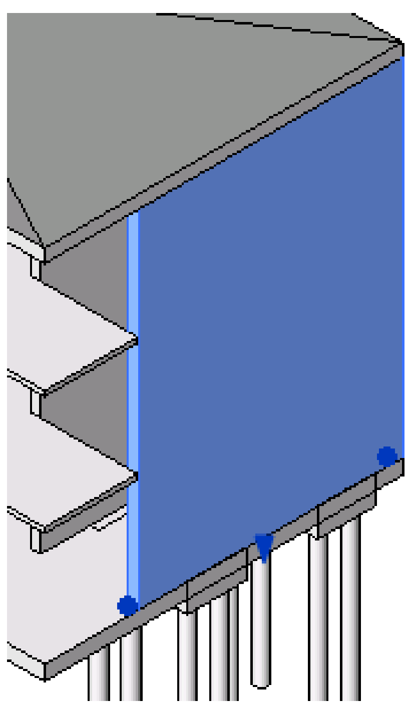
Sketch the area reinforcement

9 Select the foundation wall.



10 In the Project Browser, under 3D Views, double-click 3D.

11 Use the ViewCube to rotate the model so the concrete wall on the northeast corner of the structure is visible. The foundation wall will be highlighted, as shown.

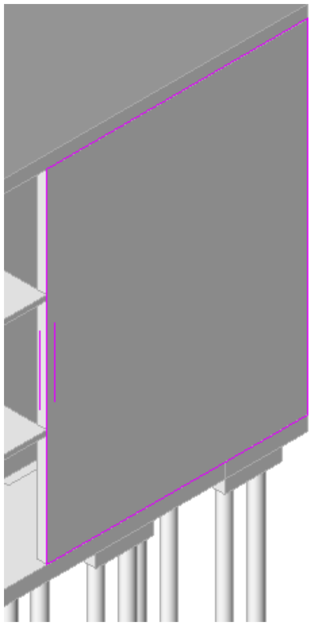


12 Click Home tab ➤ Reinforcement panel ➤ Area.

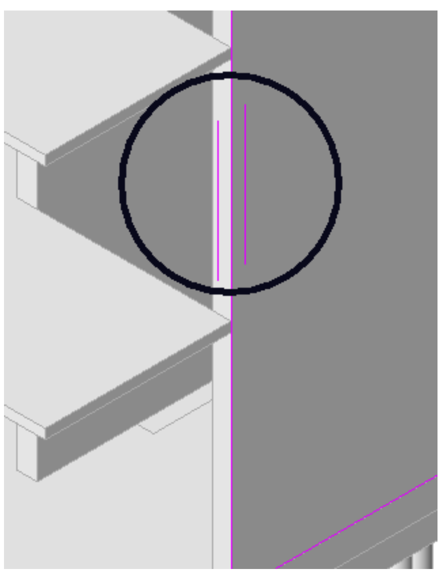
13 Select the wall.

You are now in sketch mode.

14 Click Draw panel ➤ (Line), and trace the outline of the foundation wall, as shown.

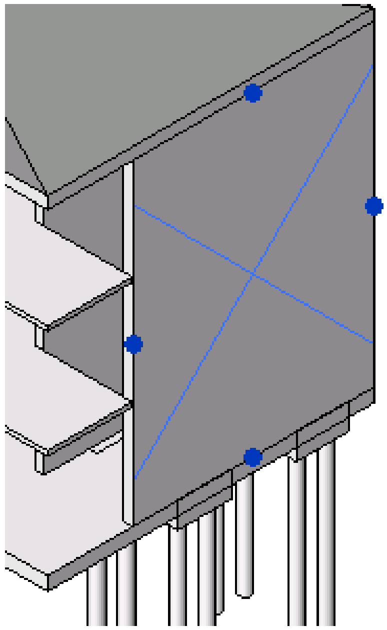


NOTE The 2 short lines adjacent to the vertical line of the outline represent the rebar major direction. Bars will be placed parallel to the major direction near both wall faces. Minor bars will be placed perpendicular (inside position) to the major bars (outside position), adjacent to the major bars. To change the major direction, on the Design Bar, click Major Direction Edge, and select one of the horizontal lines of the outline.



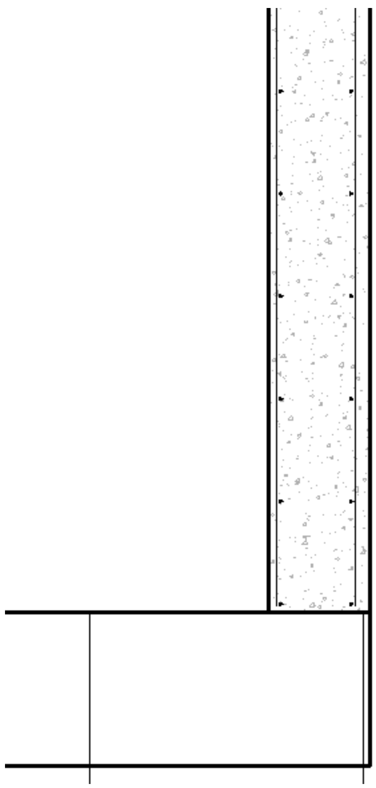
15 Click Area Reinforcement panel ➤ Finish Area.

The area reinforcement is automatically applied to the selected foundation wall, and is indicated on the 3D view with an "X" as shown.



16 Press *Esc*.

17 In the Project Browser, under Sections (Building Section), double-click Section 3.

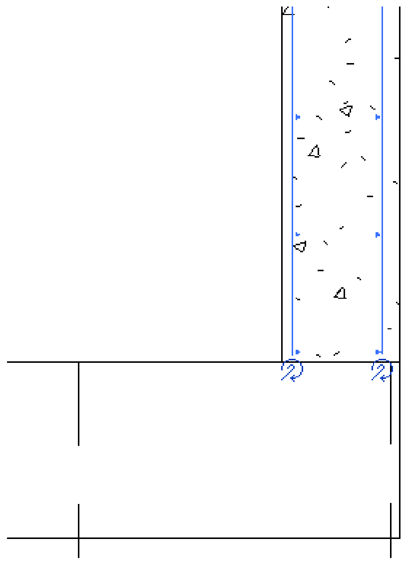


The area reinforcement for the foundation wall displays.

Remove the major bars

The area reinforcement sketch includes two layers of rebar placed perpendicular to each adjacent face (interior and exterior faces of the wall). In this case, the rebar that displays in the vertical direction is referred to as the “major” bars which are always placed closer to the element face. The rebar that displays in the horizontal direction is referred to as the “minor” bars. In the following steps, you will change the element properties of the “major” rebar placed in both layers in the wall.

18 Select the area reinforcement.

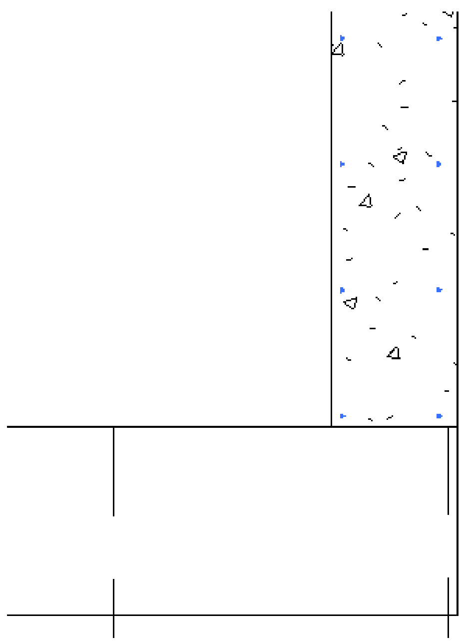


The graphical controls you use to toggle the hook orientation display at the base of the reinforcement.

19 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

20 In the Instance Properties dialog, under Layers, clear the values for Exterior Major Direction and Interior Major Direction, and click OK.

The rebar for the major span direction of the foundation wall is deleted from the section view, and the minor bars move out to the clear cover setting for the wall.



Reselect Interior Major Direction and Exterior Major Direction, and click OK.

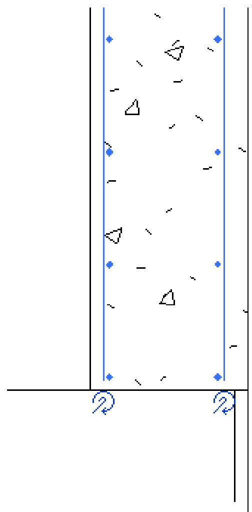
Change the number of rebar for the exterior and interior layers

21 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

22 In the Instance Properties dialog:

- Under Construction, for Layout Rule, select Fixed Number.
- Under Layers, For Exterior Major Number of Lines, type **50**.
- For Exterior Minor Number of Lines, type **50**.
- For Interior Major Number of Lines, type **50**.
- For Interior Minor Number of Lines, type **50**.
- Click OK.

Additional rebar is added to the section view.



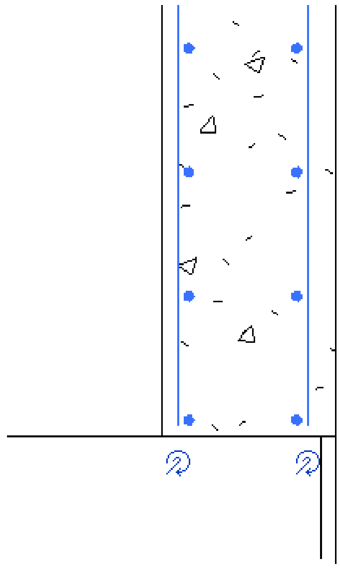
23 Press *Esc*.

Change bar type

24 Select the Area Reinforcement.

25 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

26 In the Instance Properties dialog, under Layers, change all Bar Types to #6, and click OK.



27 Press *Esc* twice.

28 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

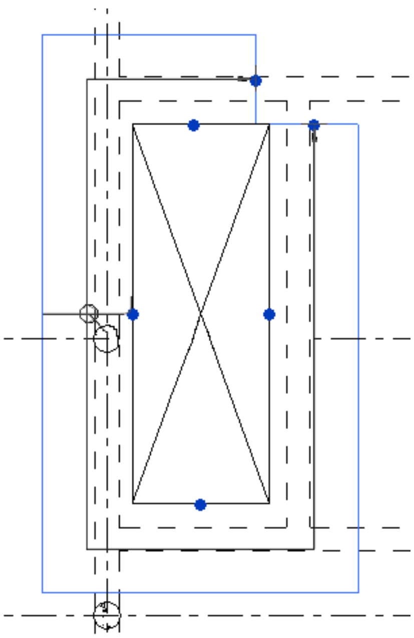
Adding Path Reinforcement

19

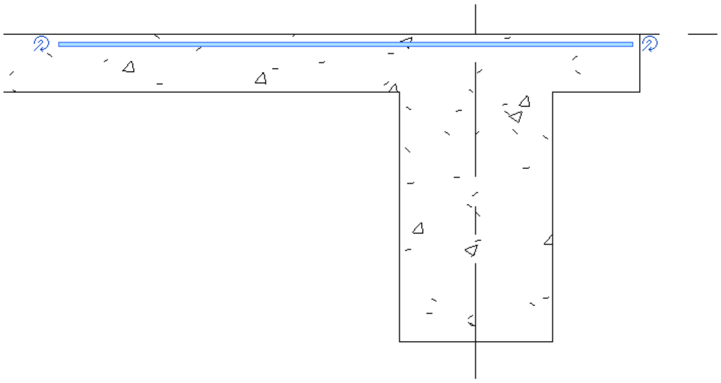
In this lesson, you add path reinforcement around the shaft openings on level 2 of the structure. You learn to:

- Sketch the path reinforcement.
- Create a section view.
- Change the section view visibility.
- Change the rebar type and hook style.

Sketch path reinforcement



Change rebar hook style



Path Reinforcement in a Slab

In this exercise, you model path reinforcement in a concrete slab. You will use the Sketch Path Reinforcement tool to lay out rebar along the outside perimeter of the shaft opening. The rebar will have the same length, but will not be parallel to each other, and will be placed perpendicular to the boundary you specify.

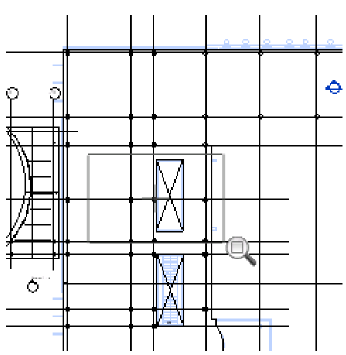
Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_REINF_Path_i.rvt.

Sketch the path reinforcement

1 In the Project Browser, under Structural Plans, double-click 02 - Floor.

2 Zoom in to the top shaft opening, located on the north side of the structure.



3 Click Home tab ➤ Reinforcement panel ➤ Path.

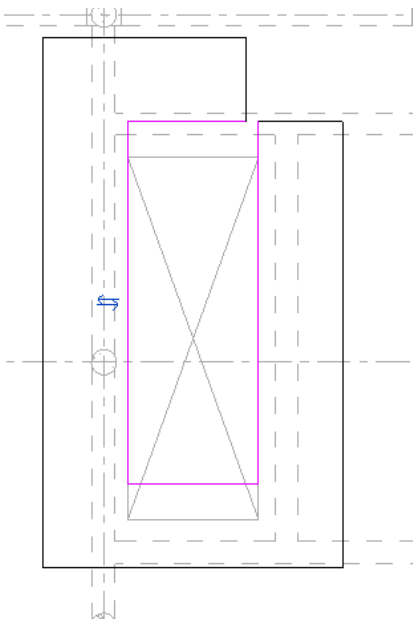
4 Select the slab.

You are now in sketch mode.

5 Click Draw panel ➤  (Line).

6 On the Options bar, for Offset, type **1' 0"**

7 Draw a path reinforcement box around the perimeter of the shaft opening, approximately as shown.

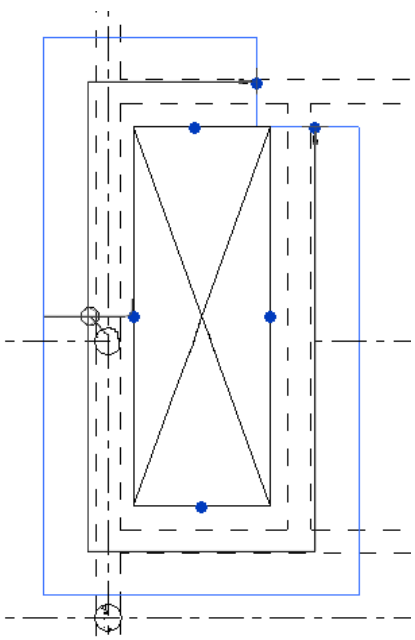


NOTE The lines you sketch for the path reinforcement box cannot intersect and must not form a closed loop.

---

8 Click Path Reinforcement panel ➤ Finish Path.

The path reinforcement for the shaft opening displays.



9 Press *Esc*.

Change the rebar type

10 Select the Path Reinforcement.

The shape handles display as blue dots. You can use these handles to change the shape of the path reinforcement, if necessary.

11 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

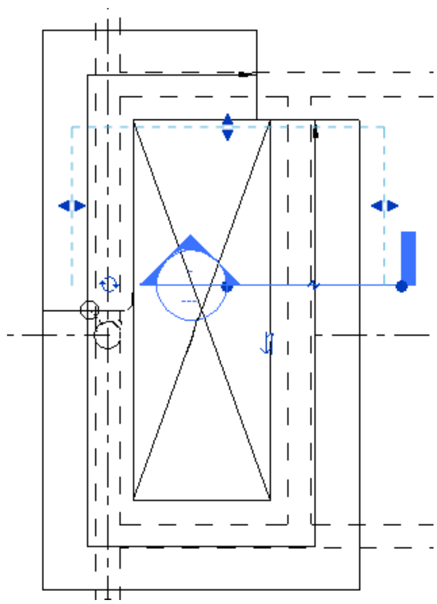
12 In the Instance Properties dialog, under Layers, for Primary Bar - Type, select #6, and click OK.

13 Press *Esc*.

Create a section view

14 Click View tab ➤ Create panel ➤ Section.

15 Add a section line that cuts through the slab opening, as shown.



16 Press *Esc*.

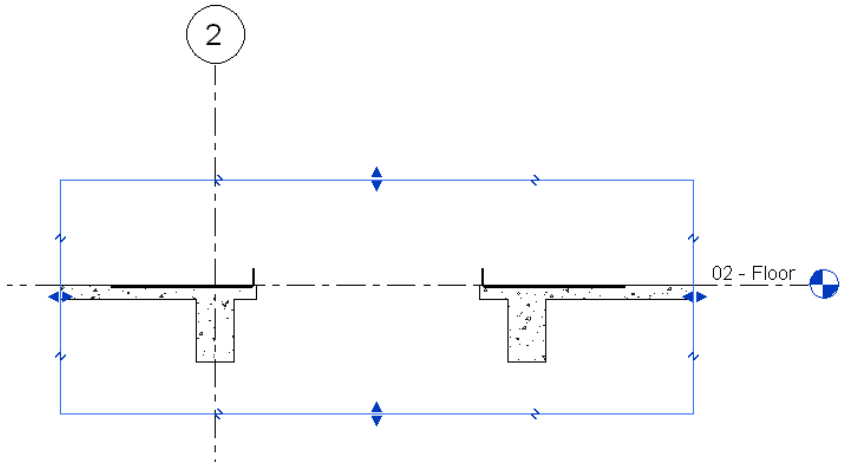
17 Double-click the section head to open the section view.

Change the section view visibility

18 On the View Control Bar:

- For Scale, select 1/4" = 1' - 0".
- For Detail Level, select Fine.
- For Model Graphics Style, select Shading w/Edges.

19 In the section view, select the crop view, and drag the controls to resize the view so that only the edge of the opening in the slab and the path reinforcement on Level 2 is visible.



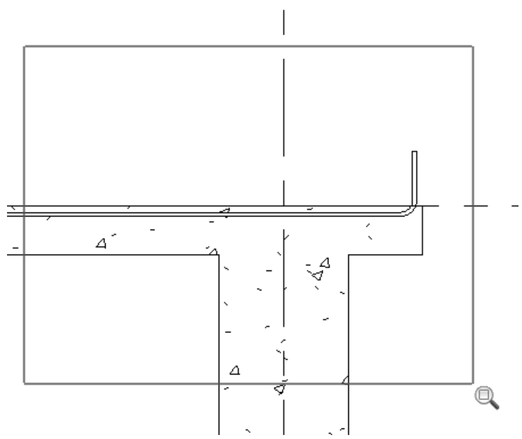
NOTE The line weights in your file may differ from the examples shown. Some of the rebar line weights were increased in the illustrations for training purposes.

20 Press *Esc*.

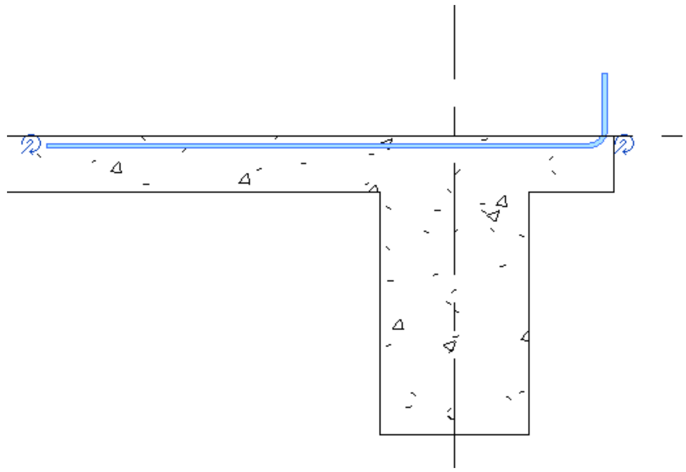
Change the rebar hook

Rebar hooks have a matrix relationship between the hook angle and the rebar size. After you place the rebar, you can use graphical controls to modify the rebar orientation.

21 Zoom in around the edge of the slab opening.

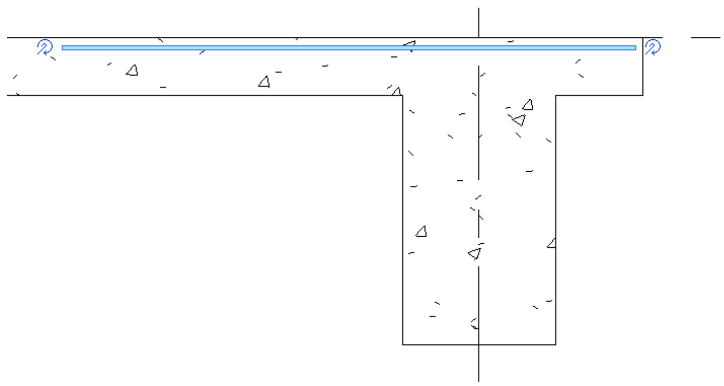


22 Select the rebar.



The toggle hook orientation icon appears.

23 Click (Toggle Hook Orientation) to change the rebar hook type as shown.



24 Press *Esc*.

25 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

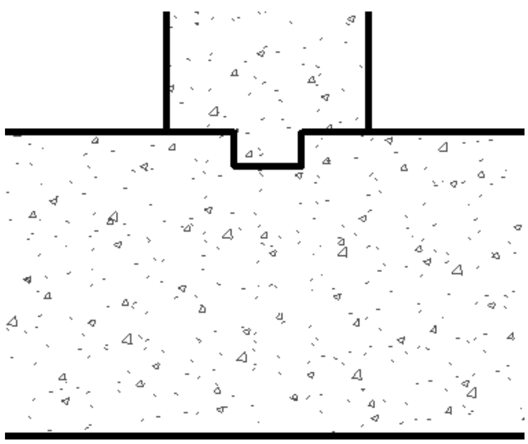
Sketch Reinforcement

20

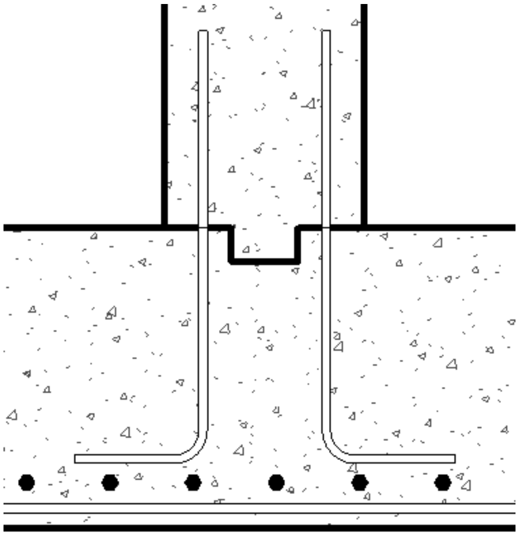
In this lesson, you sketch reinforcement in the slab foundation of the structure to better demonstrate the drawing capabilities of Revit Structure. You learn to:

- Add a wall key at the base of the concrete wall.
- Sketch reinforcement in the concrete slab.

Sketch a wall key

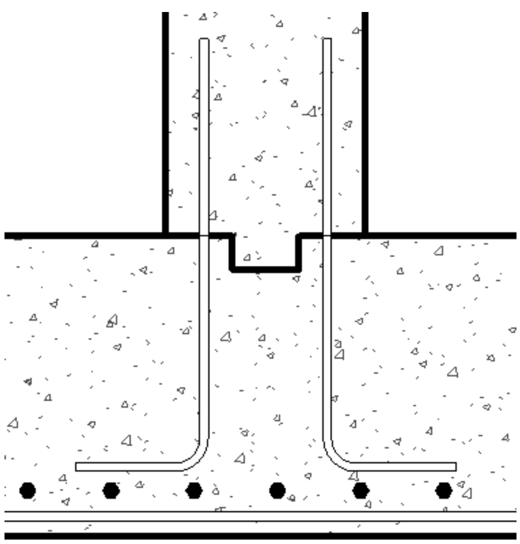


Sketch reinforcement in the slab



Sketch Reinforcement in a Slab

In this exercise, you use the drawing tools provided with Revit Structure to add wall keys to the footing and sketch reinforcement in the slab foundation. The reinforcement will include both parallel and perpendicular rebar that extend from the concrete slab into the structural wall.



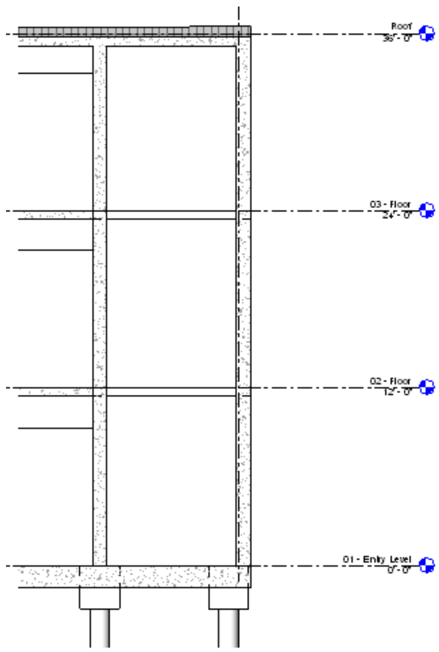
Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_MDL_REINF_Sketch_i.rvt.

Open the section view

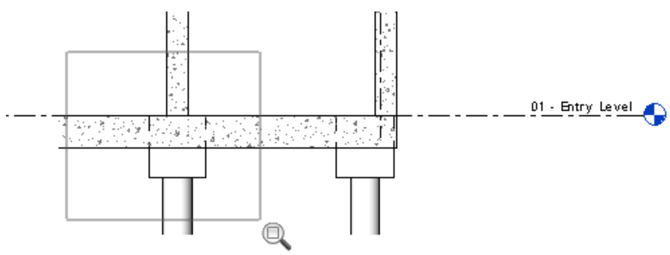
1 In the Project Browser, under Sections (Building Section), double-click Section 5.

NOTE The line weights in your file may differ from the examples shown. Some of the rebar line weights were increased in the illustrations for training purposes.



Add wall keys at the bottom of wall

2 Zoom in around the base of the concrete wall.



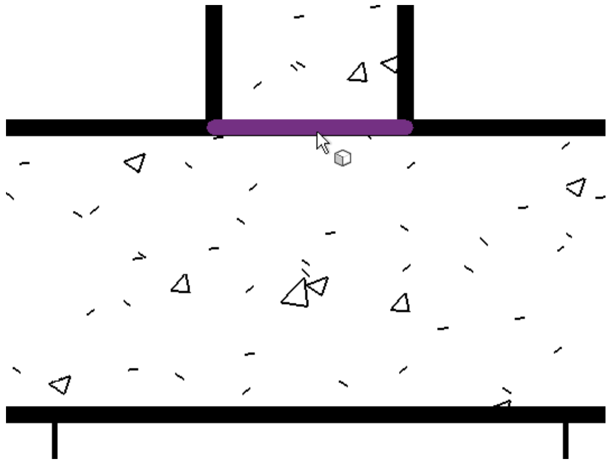
NOTE For training purposes, Model Graphics Style is set to Shading w/Edges (on the View Control Bar).

---

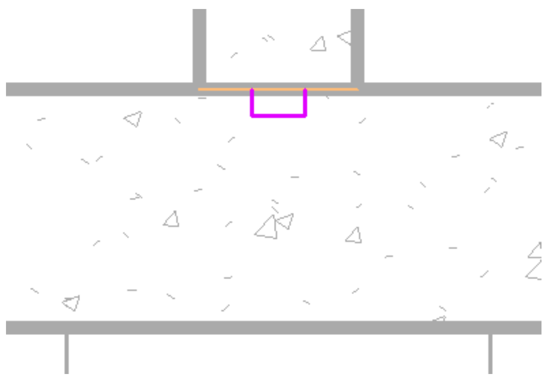
3 Click ➤ Modify tab ➤ Edit Linework panel ➤ Cut Profile.

4 On the Options Bar, select Boundary between faces.

5 Select the boundary between the wall and footing.

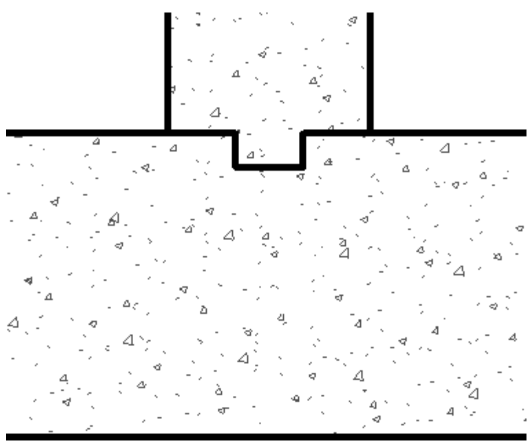


6 Click Draw panel ➤ (Line), on the Options Bar, select Chain, and sketch 3 lines approximately as shown.



7 Click Cut Profile panel ➤ Finish Cut Profile.

8 Press *Esc*.



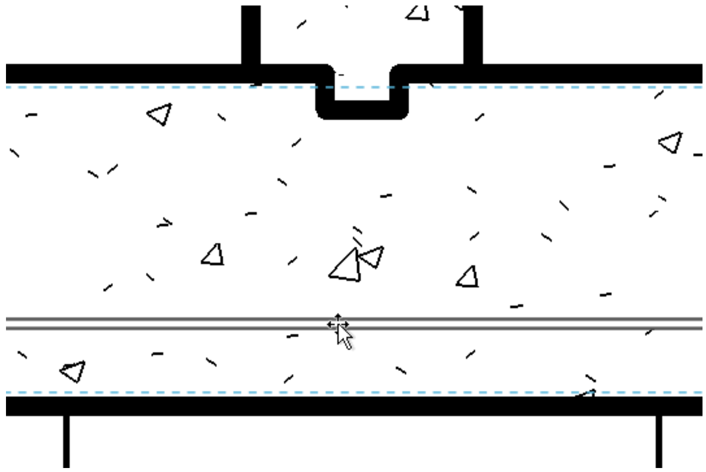
Place rebar parallel to the slab

9 Click Home tab ➤ Reinforcement panel ➤ Rebar drop-down ➤ Place Rebar Parallel to Work Plane.

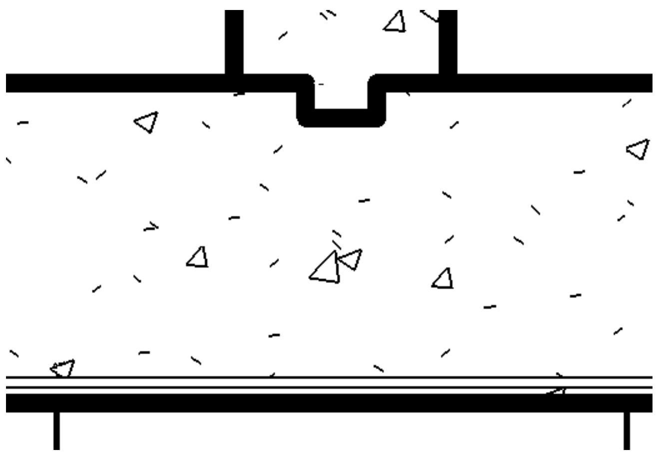
10 On the Options Bar:

- Select Rebar Shape : 00.

After you select a rebar type, the default rebar cover displays as a blue dotted line as you move the cursor over each element.



11 Click to place the parallel rebar in the slab as shown.



12 Press *Esc*.

Sketch rebar parallel to the slab

13 Click Reinforcement panel ➤ Rebar drop-down ➤ Place Rebar Parallel to Work Plane.

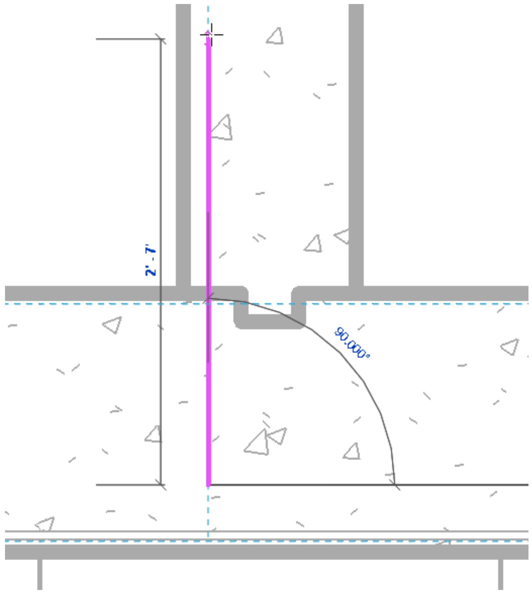
14 Click Sketch panel ➤ Sketch Rebar.

15 Select the concrete slab as the host element.

You are now in sketch mode.

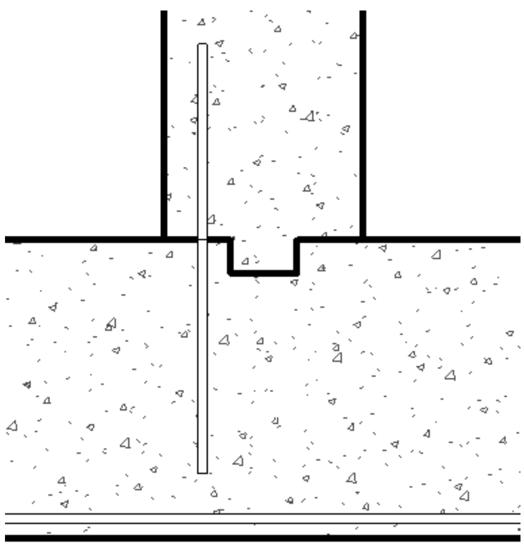
16 Sketch a rebar by clicking inside the slab first and then moving the cursor into the wall, as shown.

NOTE Be sure to sketch from the slab to the wall.



17 Click Rebar Sketch panel ➤ Finish Rebar.

NOTE Rebar contributes only to the estimated reinforcement volume of its host (the slab). Therefore, this rebar will not be included in rebar estimates of the structural wall.



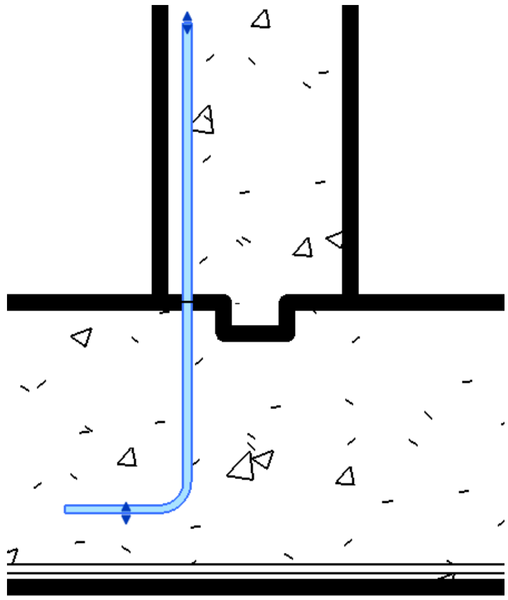
18 Click Selection panel ➤ Modify.

Add hook

19 Right-click the rebar you added in the previous steps, and click Element Properties.

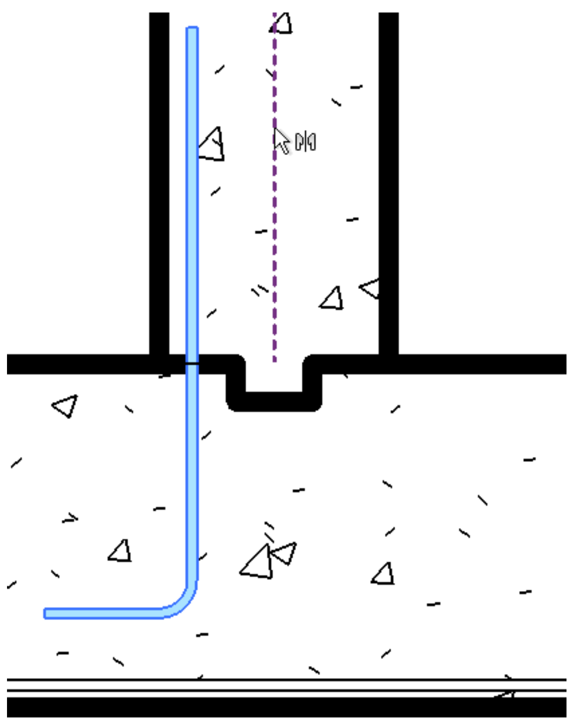
20 In the Instance Properties dialog, under Construction, for Hook At Start, select Standard - 90 deg.

21 Click OK.

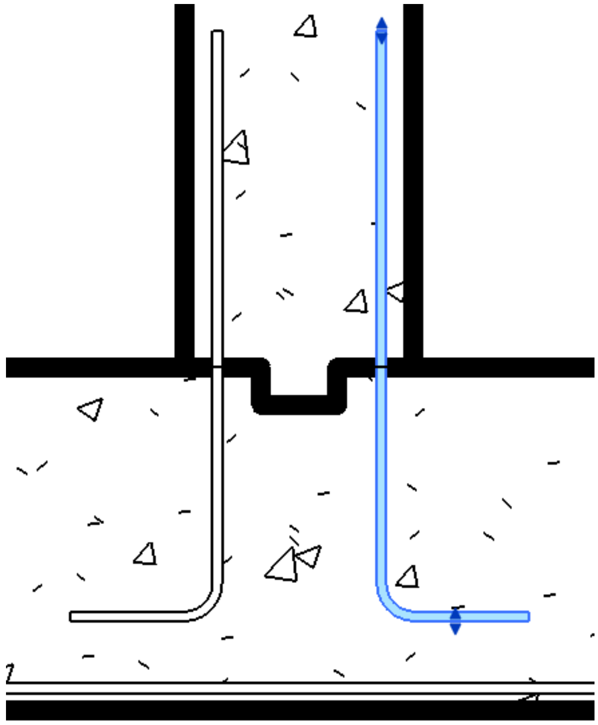


Mirror the rebar

22 Click Modify panel ➤ Mirror, and select the center reference plane of the structural wall.



23 Click to place the rebar.



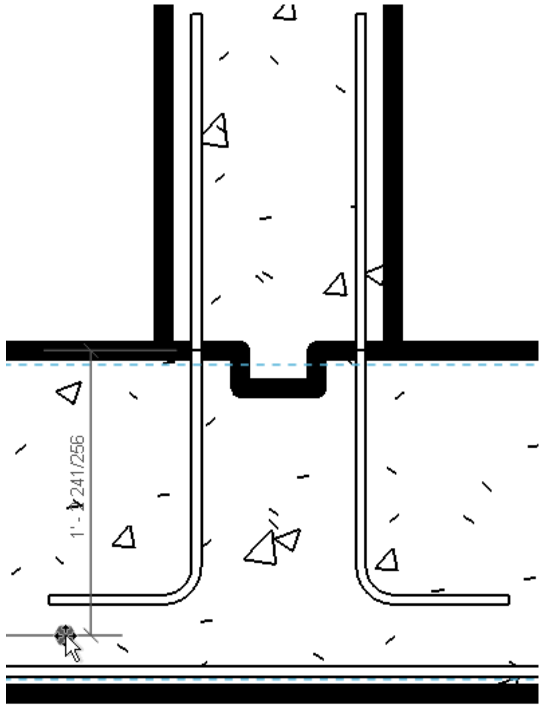
24 Press *Esc*.

Place rebar perpendicular to the slab

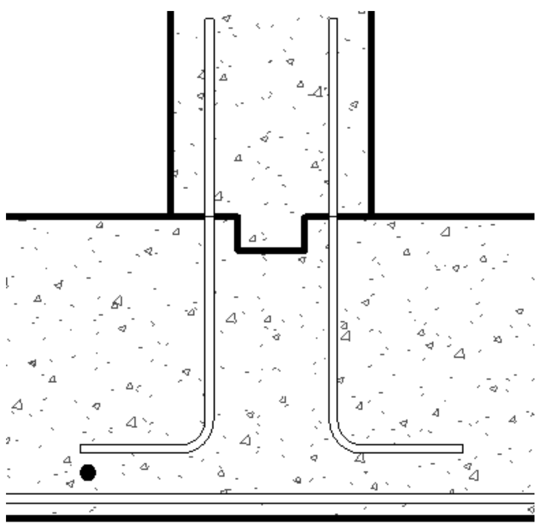
25 Click Home tab ➤ Reinforcement panel ➤ Rebar drop-down ➤ Place Rebar Perpendicular to Work Plane.

26 Click Element panel ➤ Change Element Type drop-down ➤ Rebar Bar: #8.

27 Select the end of the hooked bar, and place the pointer between the hooked and straight bar, approximately as shown.



28 Click to place the first rebar as shown.



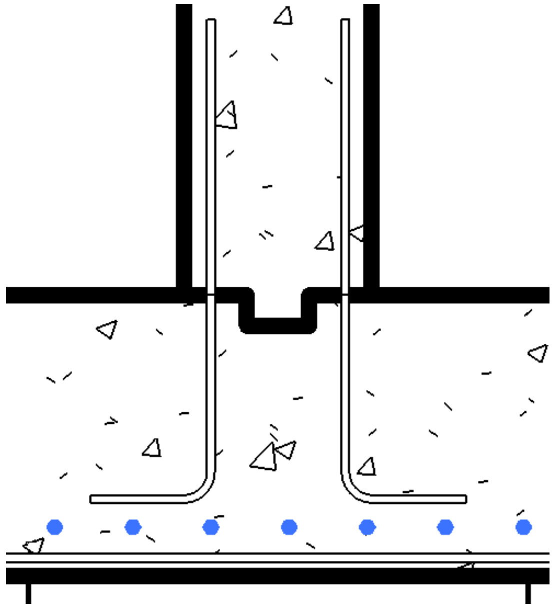
29 Click Selection panel ➤ Modify.

Place rebar set

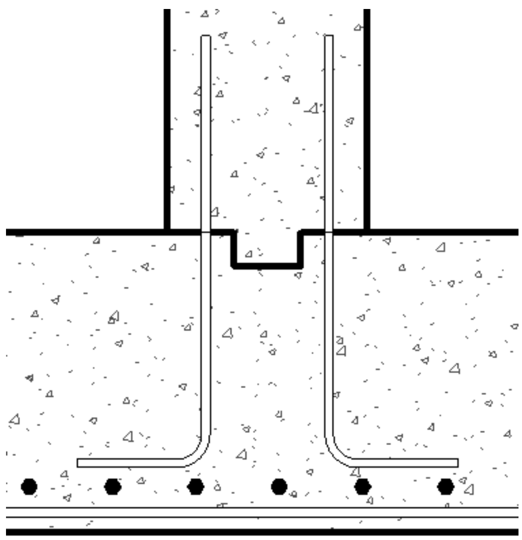
30 Select the single rebar.

31 On the Options Bar:

- For Layout, select Minimum Clear Spacing.
- For Spacing, type **4**".



32 Press *Esc* twice.



33 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Documenting a Project

In this tutorial, you learn how to use the tools in Revit Structure 2010 to document a building information model project.

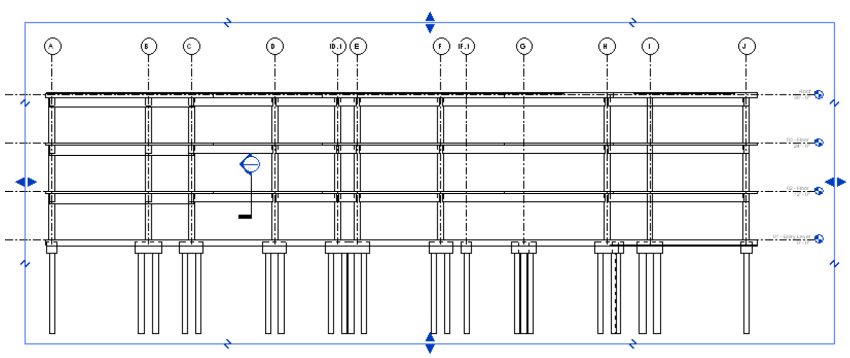
Adding Views and Sheets to a Project

21

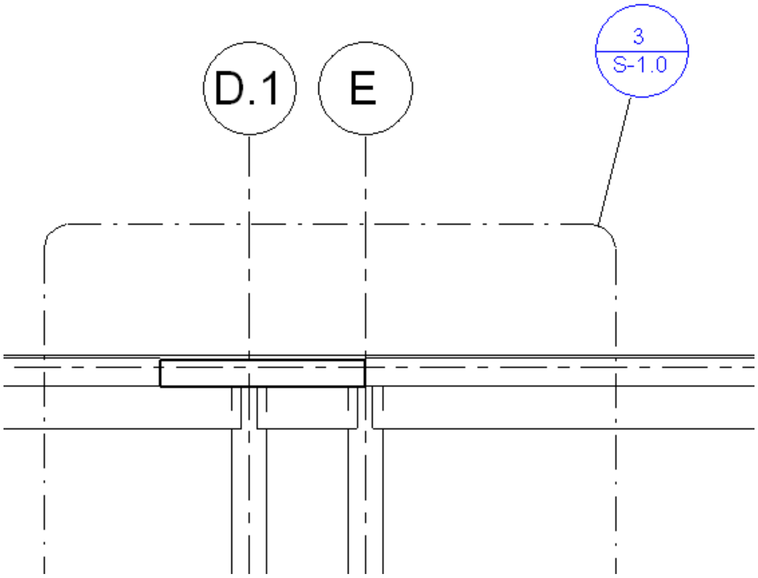
In this lesson, you learn how to create views from your projects and then place the views on documentation sheets. You learn to:

- Create sheets composing the document set.
- Create a section and callout view.
- Place views onto the created sheets.

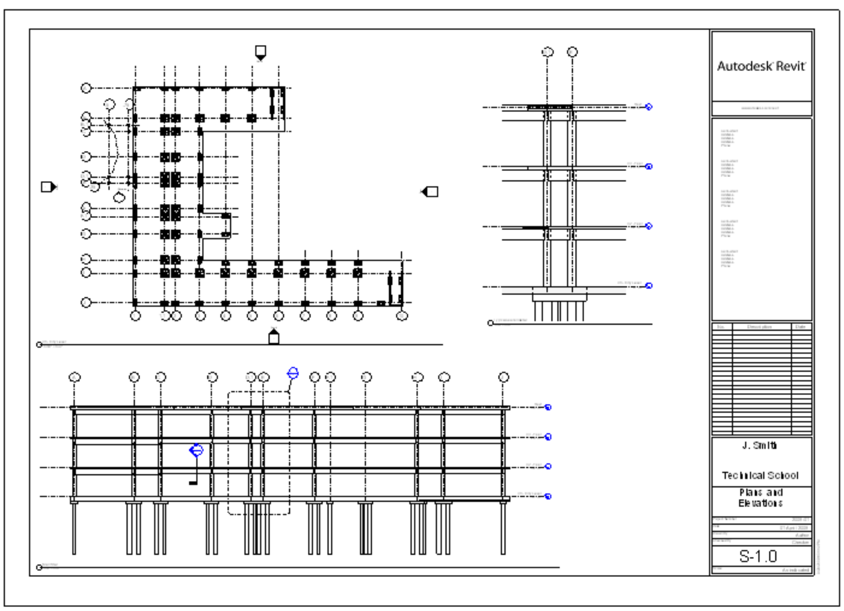
Section view



Callout view

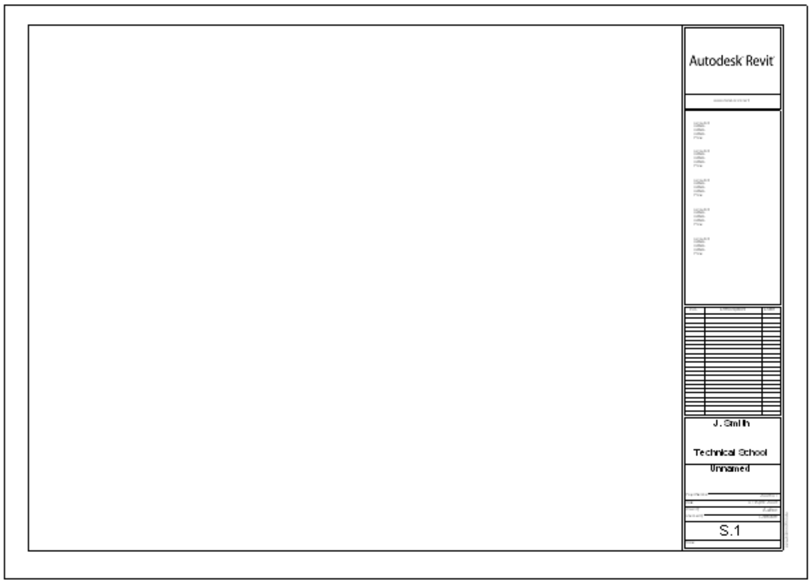


Sheet with views



Creating Sheets

In this exercise, you add documentation sheets to the project. After you create the sheets, you add project information to customize the title block.



Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_AVS_01_Create_Sheets_i.rvt.

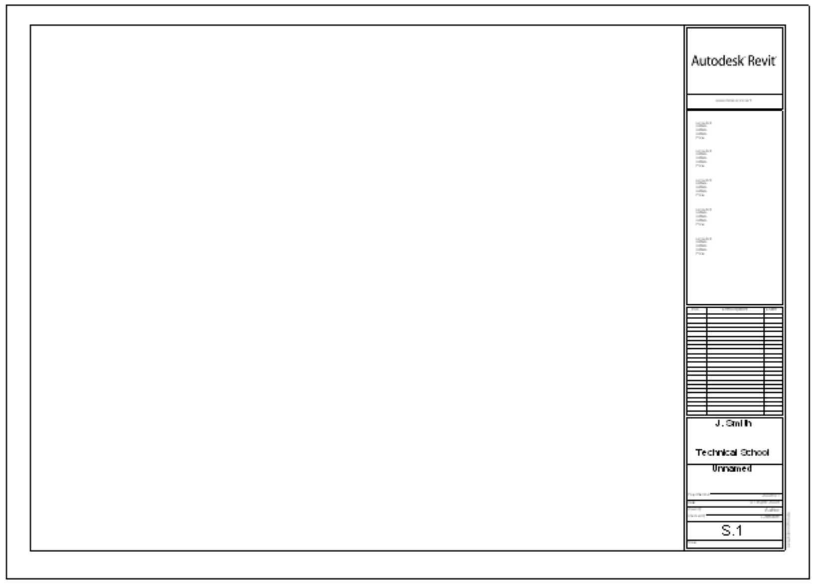
Enter project information for the title block

1. Click Manage tab ➤ Project Settings panel ➤ Project Information.
2. In the Instance Properties dialog, click Edit for Project Address.
3. In the Edit Text dialog, type the following address: **123 Main Street, Anytown, MA 12345**.
4. Click OK.
5. Specify the remaining instance parameters:
   - For Project Issue Date, type **01 April 2009**.
   - For Project Status, type **Initial Draft**.
   - For Client Name, type **J. Smith**.
   - For Project Name, type **Technical School**.
   - For Project Number, type **2009-01**.
6. Click OK.

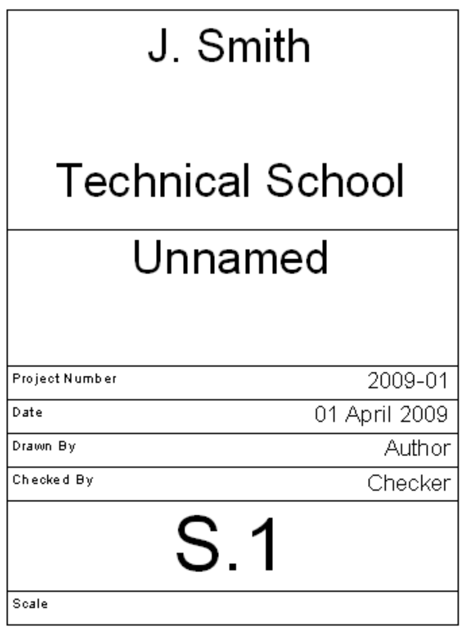
Create a sheet

7. Click View tab ➤ Sheet Composition panel ➤ New Sheet.
8. In the Select a Title block dialog, select E1 30 x 42 Horizontal: E1 30x42 Horizontal, and click OK.

   A title block and drawing borders display on the drawing sheet.

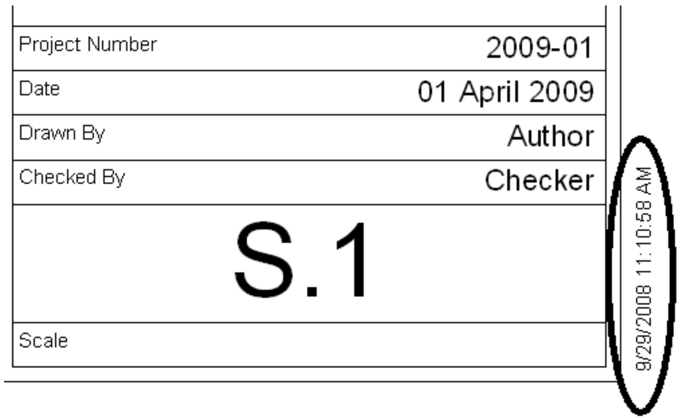


The title block that you selected is a family that has already been loaded into the project. The text fields in the title block family contain labels that automatically display the corresponding project information that you entered.



NOTE The project date and time displays in the lower-right corner of the sheet view and automatically updates every time you save the project file.

Project date and time



9 In the Project Browser, expand Sheets (all).

The new sheet is displayed in the Project Browser with the name S.1 - Unnamed.

10 Press *Esc*.

Change the sheet name and number in the title block

11 Select the title block.

12 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

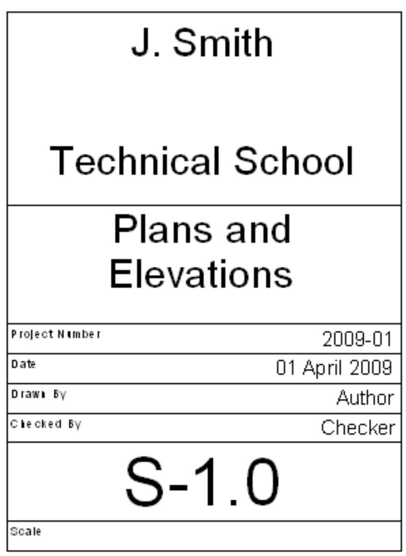
13 In the Instance Properties dialog:

- Under Identity Data, for Sheet name, type **Plans and Elevations**.
- Under Identity Data, for Sheet number, type **S-1.0**.

14 Click OK.

15 Press *Esc*.

The sheet name and number display in the title block and in the Project Browser.

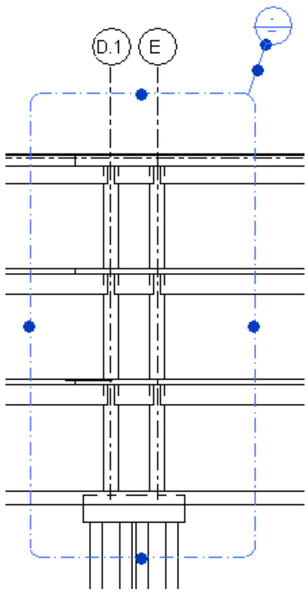


16 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Creating Section and Callout Views

In this exercise, you create a section view through Level 2 of the structure. Also, you create a view from a callout of the concrete column/structural slab.



Training File

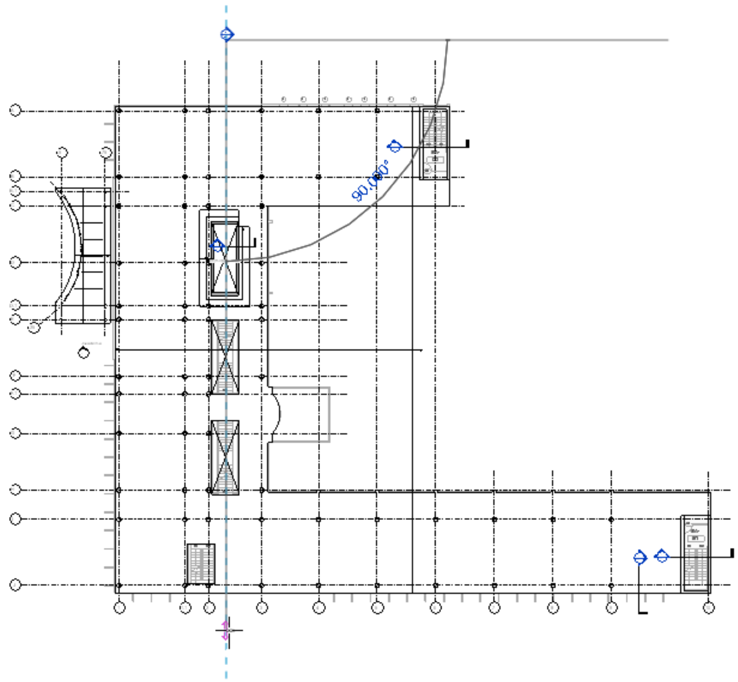
- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_AVS_02_Create_Views_i.rvt.

Create a section view of Level 2

1 In the Project Browser under Structural Plans, double-click 02 Floor.

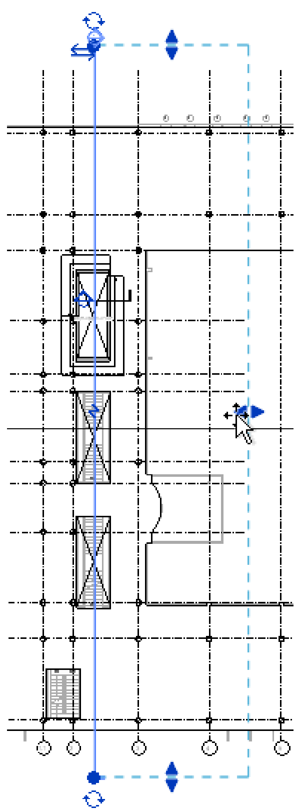
2 Click View tab ➤ Create panel ➤ Section.

3 Click above the floor plan, and draw a vertical section between grid lines 2 and 3, cutting through the stairs as shown.



4 Click and drag the right control to the left until it is just to the right of the small floor extension, as shown

You adjust the depth of the section to include only what you want to show in the section.

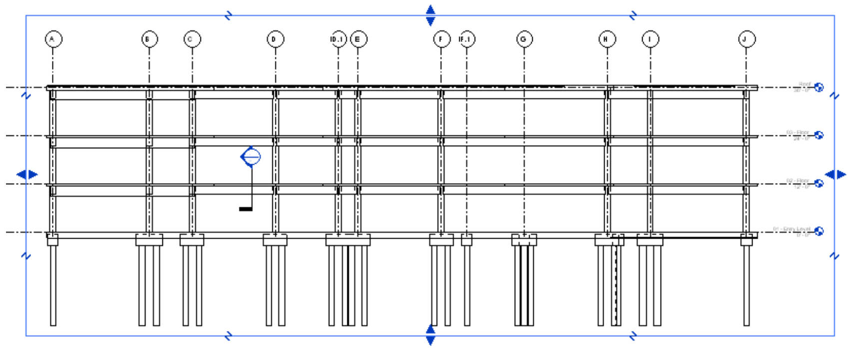


5 Press *Esc*.

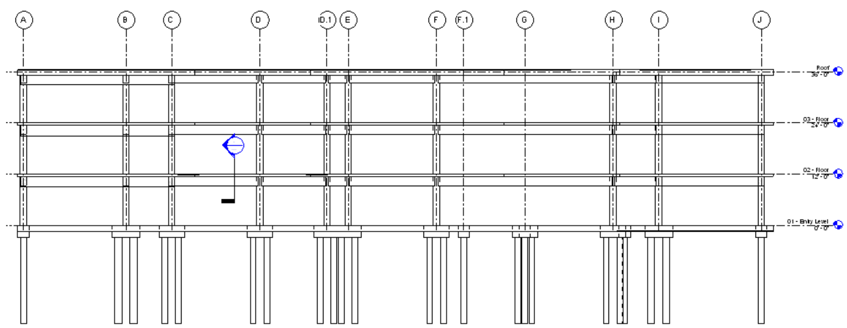
Modify element visibility in the view

6 Double-click the section head to open the section view.

7 Adjust the crop region boundaries by dragging the controls to include what you want to show.



8 On the View Control Bar, click  (Hide Crop Region).



Rename the view

9 In the Project Browser, under Sections (Building Section), right-click Section 1, and click Rename.

10 In the Rename dialog, type **Main Stair**, and click OK.

Create a callout view of the structural walls

11 In the Project Browser, under Sections (Building Sections), double-click Main Stair.

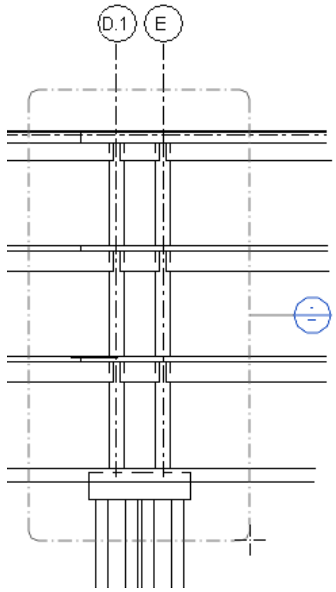
12 Click View tab ➤ Create panel ➤ Callout.

13 Click Element panel ➤ Change Element Type drop-down ➤ Detail.

The type of callout determines where the view being created will be placed in the Project Browser.

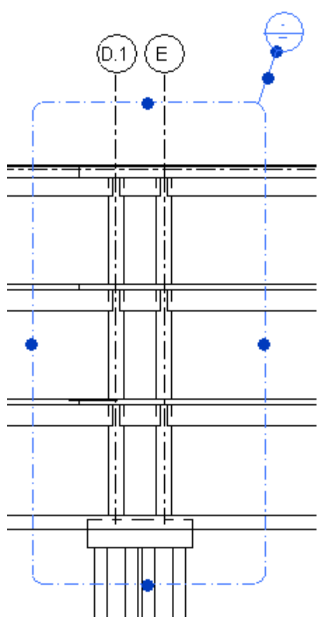
14 On the Options Bar, for Scale, select 1/4" = 1'-0".

15 Starting in the upper-left corner, between grid lines D and D.1, draw a callout box around the columns of Level 2, as shown.



16 Adjust the callout head position:

- Select the callout.
- Click and hold the Drag Head control and move the callout head to the upper-right corner of the callout, as shown.



17 Press *Esc*.

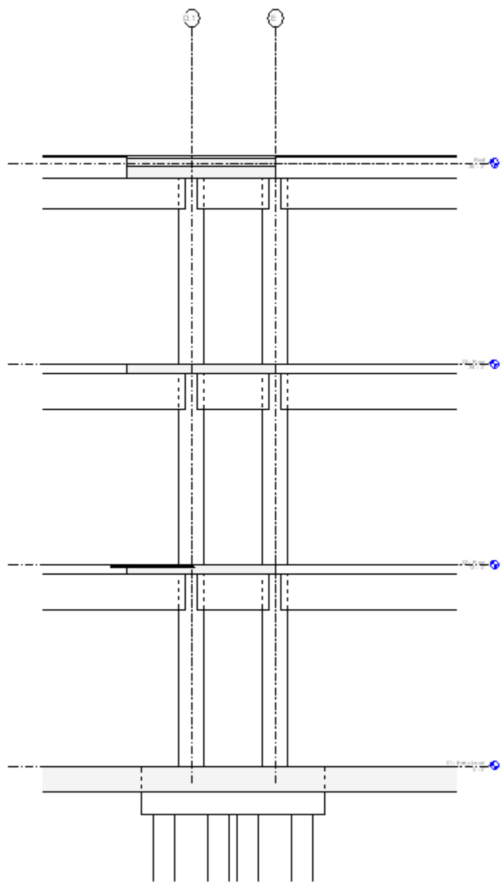
Open the callout view

18 Double-click the view reference (callout head) to open the callout view.

19 On the View Control Bar:

- Select Detail Level: Fine.

■ Click (Hide Crop Region).



Rename the callout view

20 In the Project Browser, under Detail Views (Detail), right-click Detail 0, and click Rename.

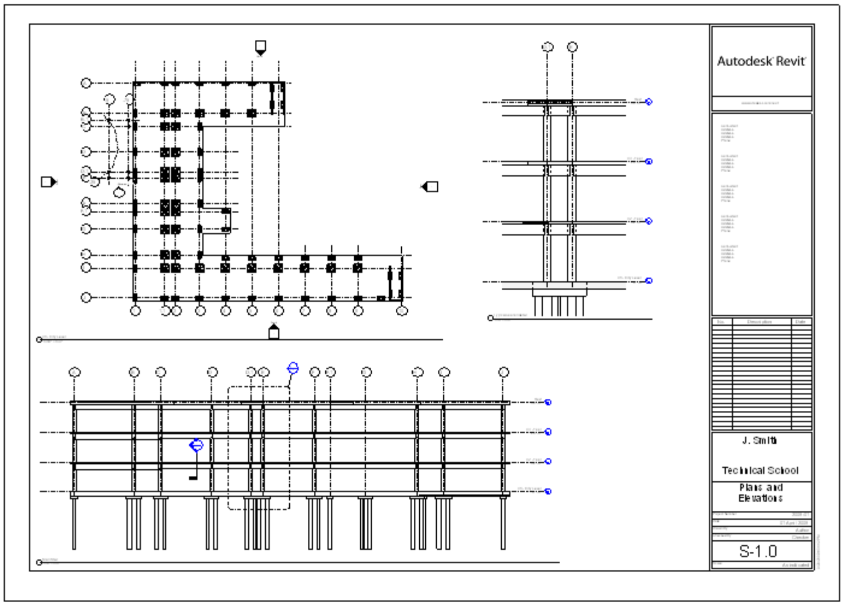
21 In the Rename dialog, type **Typical Column Detail**, and click OK.

22 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Placing Views

In this exercise, you add views to the sheets you created in a previous exercise. To place a view on a sheet, you drag it from the Project Browser and position it on the sheet. You can easily move views from one sheet to another. Views are updated automatically as you modify the project.



Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_AVS_03_Place_Views_i.rvt.

Add a plan view to the sheet

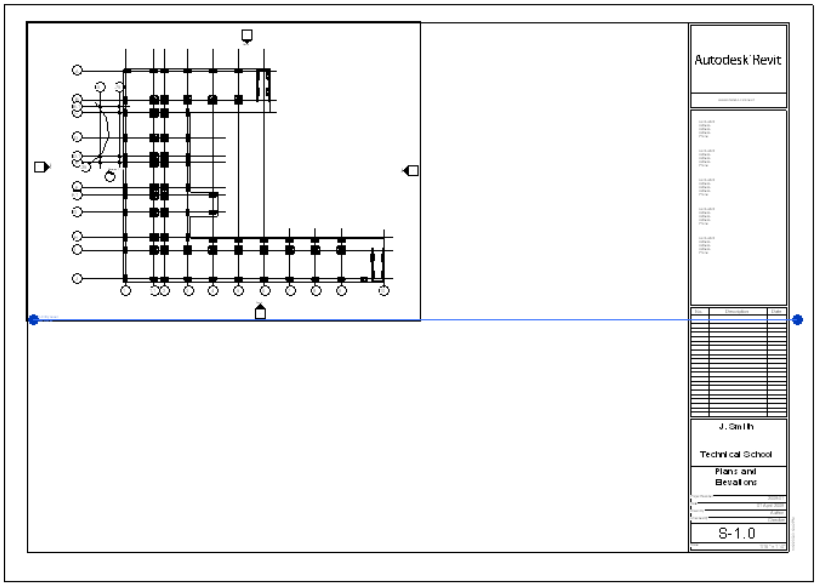
1. In the Project Browser, under Sheets (all), double-click S-1.0 - Plans and Elevations.
2. Click View tab ➤ Sheet Composition ➤ View.
3. In the Views dialog, select Structural Plan: 01 - Entry Level, and click Add View to Sheet.
4. Move the cursor to the center of the sheet, and click to place the view.

Change the view scale

5. With the view selected, right-click, and select Element Properties.
6. In the Element Properties dialog, under Graphics, verify 1/16" = 1'- 0" for View Scale, and click OK.

   The scale of the view on the sheet changes. You can open the Structural Plan: Level 2, right-click, and click View Properties, to see that the scale plan view is now 1/16" = 1'- 0".

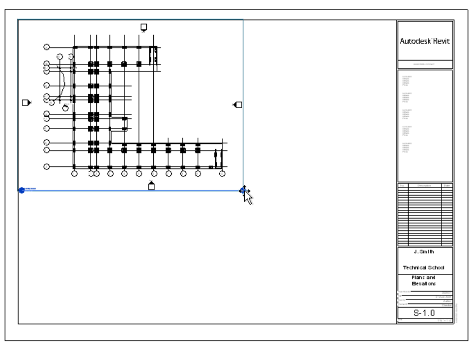
7. Drag the view to the upper-left corner of the sheet.



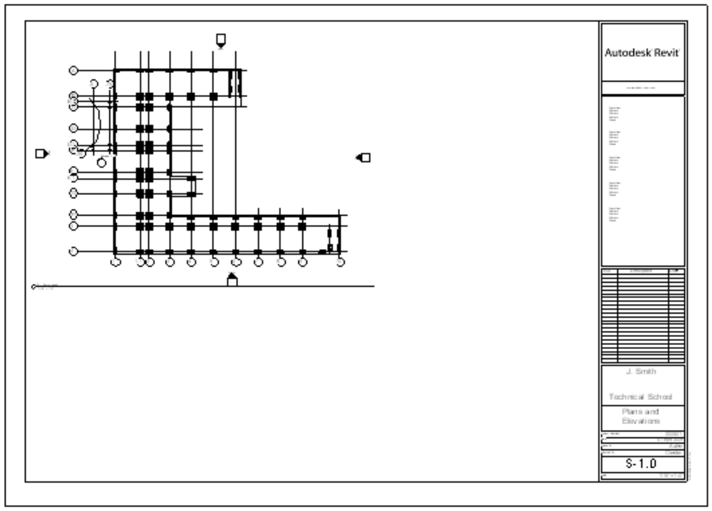
Position the view title

8 Adjust the length of the view title line (located under the view), by first clicking the view, and then dragging the right end control until it fits under the view.

NOTE If necessary, you can select the view title separately and move it to a new position.



9 Press *Esc*.



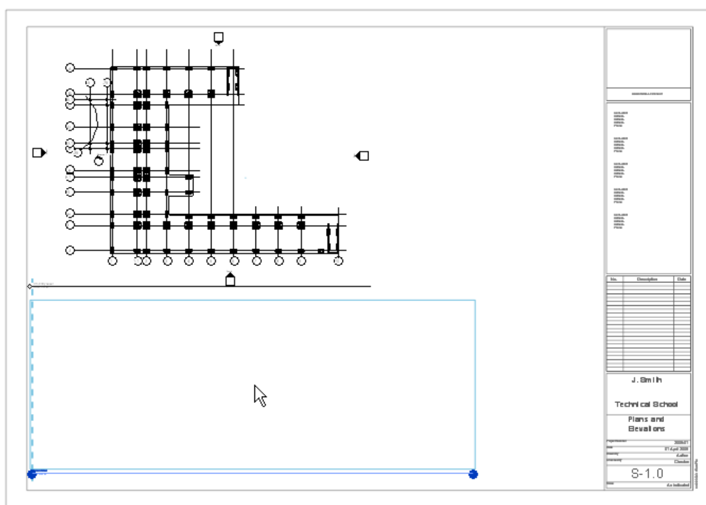
Add an elevation to the sheet

10 Click View tab ➤ Sheet Composition ➤ View.

11 In the Views dialog, select Section: Main Stair, and click Add View to Sheet.

12 Click on the lower half of the sheet to place the view.

Notice that when you place the Main Stair view below the plan view, guide lines display to help you position the views so the view titles are aligned.



13 Press *Esc*.

Add a callout view to the sheet

14 Click View tab ➤ Sheet Composition ➤ View.

15 In the Views dialog, select Detail View: Typical Column Detail, and click Add View to Sheet.

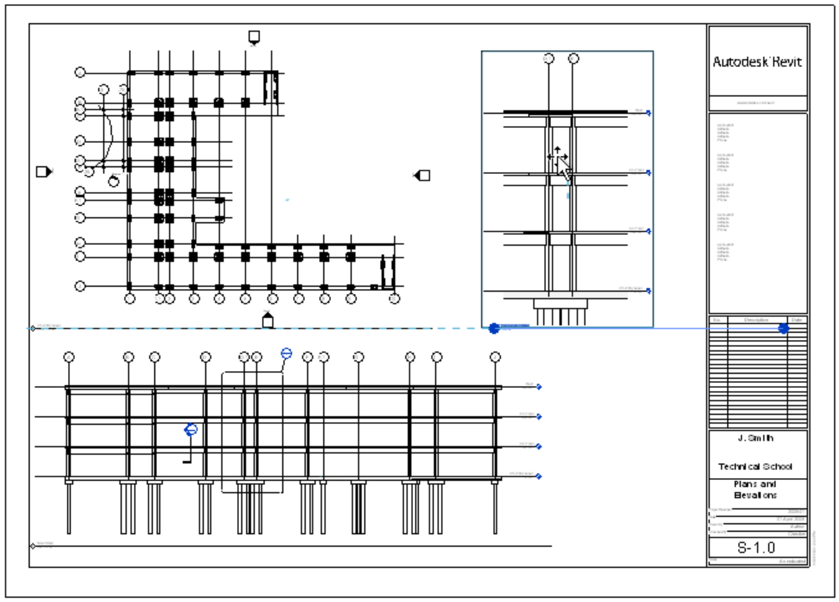
16 Click on the upper-right corner of the sheet to place the view.

Change the callout view scale

17 With the view selected, right-click, and click Element Properties.

18 In the Element Properties dialog, under Graphics, specify 1/4" = 1'- 0" for View Scale, and click OK.

19 Drag the view to the upper-right corner of the sheet.



20 Press *Esc*.

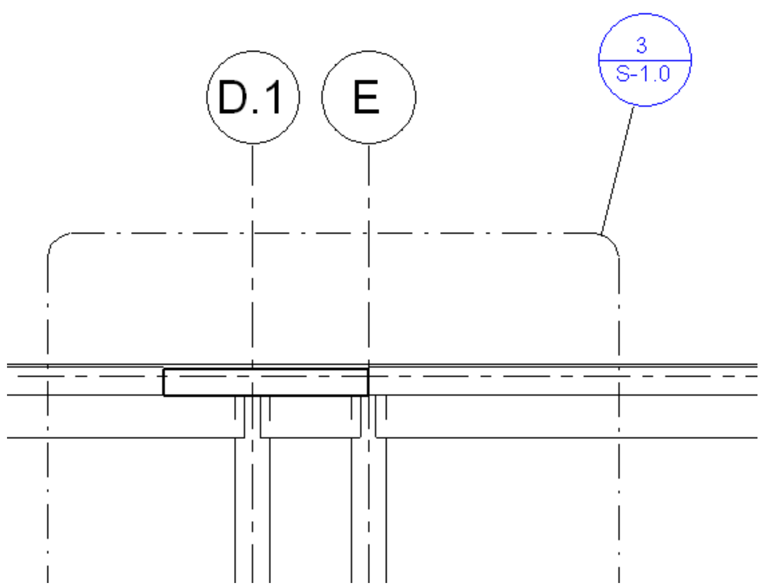
21 Adjust the length of the title line by selecting the view, and dragging the right end control until it fits under the view.

NOTE If necessary, you can select the view title separately and move it to a new position.

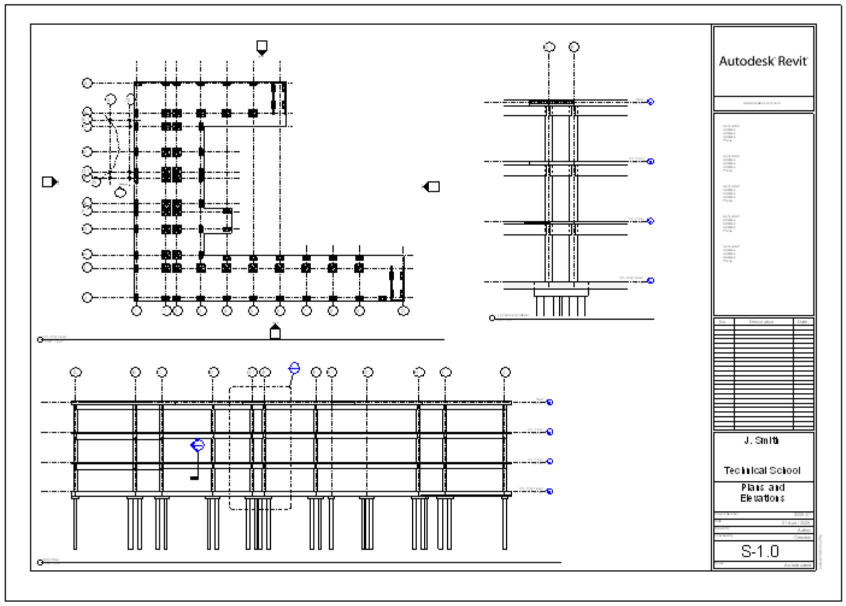
---

22 Zoom in to the callout on the Main Stair view.

The callout head displays the correct sheet and view number references.



23 Enter **ZF** (Zoom to Fit).



24 Close the file with or without saving it.

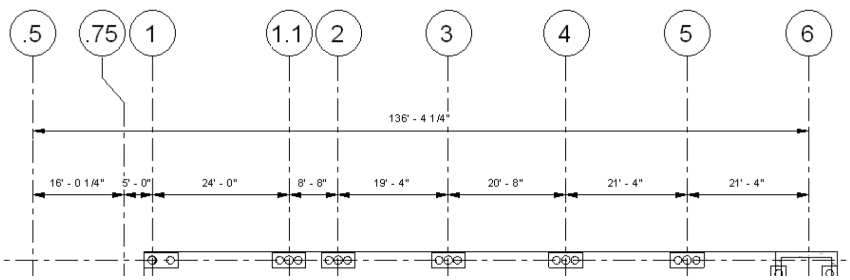
In the next exercise, a new training file is supplied.

Annotating and Dimensioning

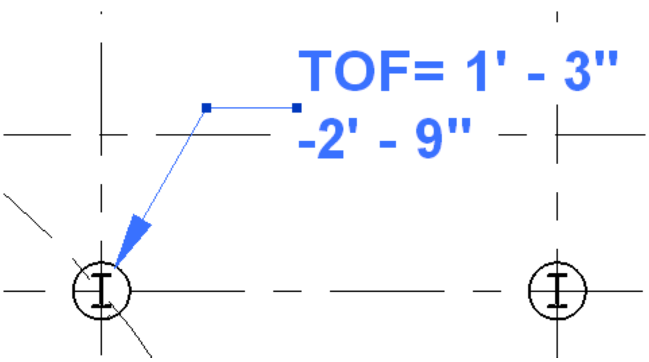
22

In this lesson, you create permanent dimensions to control and document the structure. You learn to:

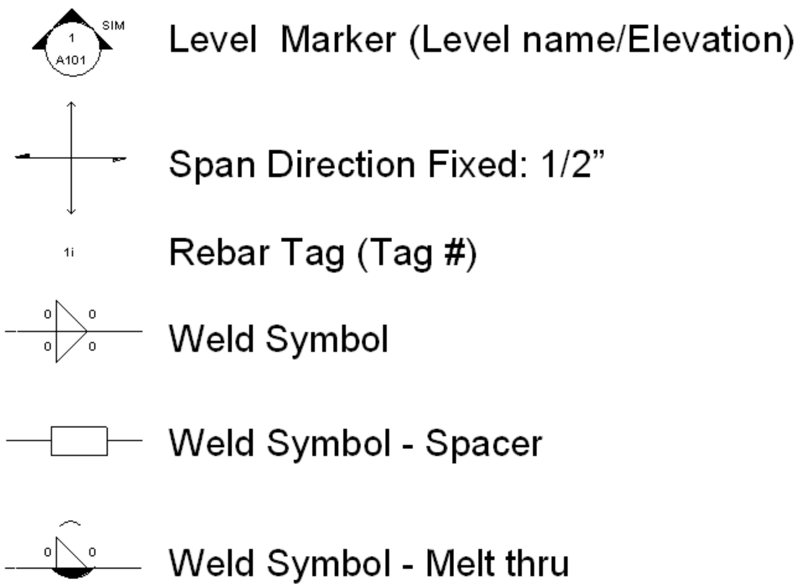
- Create dimensions by clicking points in the model.



- Create a spot dimension for the shape-modified slab and the footings on the entrance.

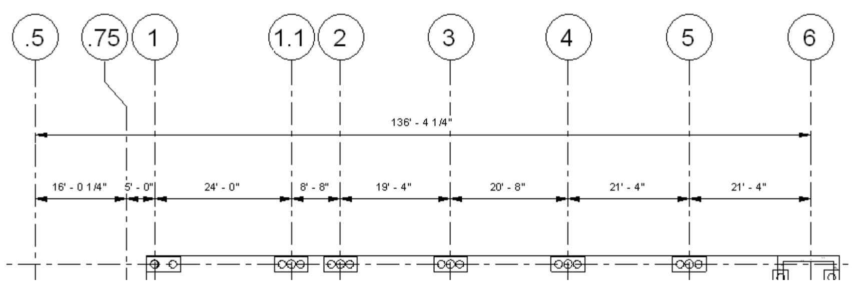


- Create an annotation legend view to display typical structural components used in construction documentation.



Creating Dimensions

In this exercise, you dimension the grid lines on the north side of the structure. You place these dimensions in the view by clicking the appropriate points.

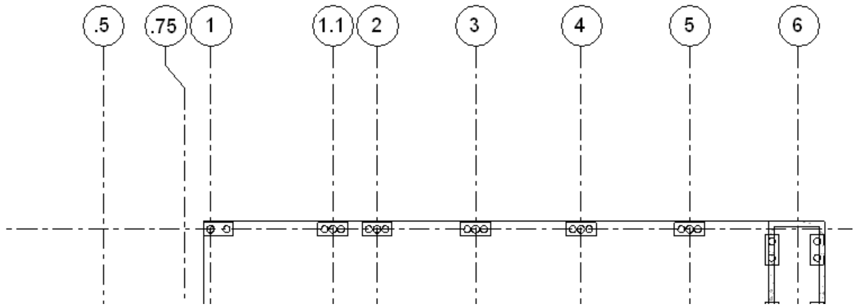


Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_AD_01_Create_Dimensions_i.rvt.

Create a permanent dimension

1 In the Project Browser, under Structural Plans, double-click 01 - Entry Level, and zoom in to the north side of the structure.

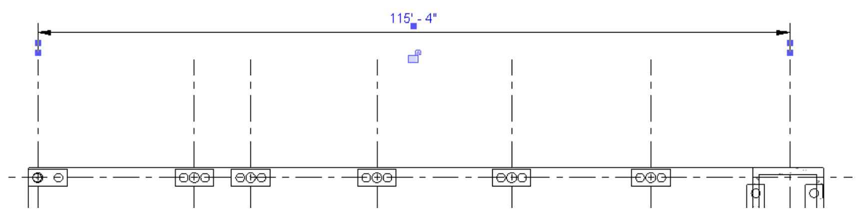


2 Click Annotate tab ➤ Dimension panel ➤ Aligned.

You place an overall dimension between grid lines .5 and 6.

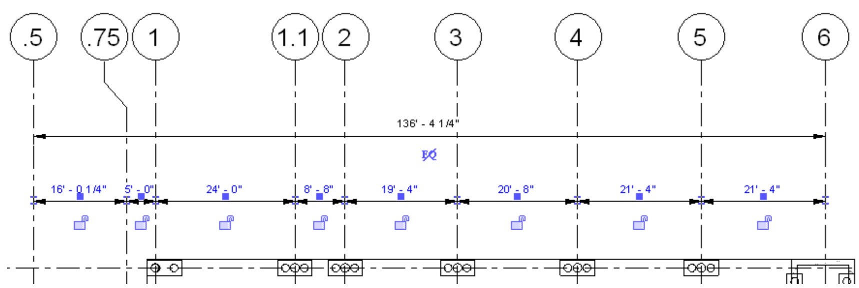
3 To place the dimension:

- Click grid line .5.
- Click grid line 6.
- Move the cursor to the right of grid line 6.
- Click to place the dimension.

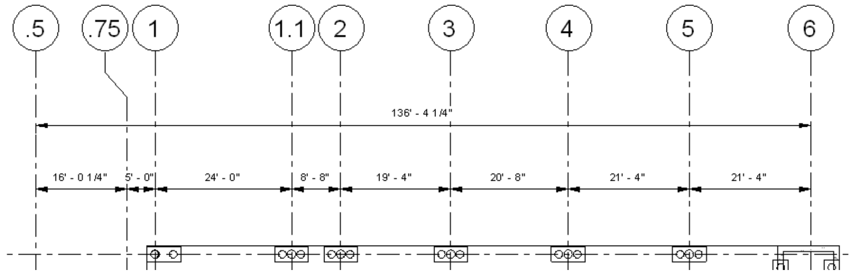


4 Using the same method, place a string of dimensions by clicking each grid line, from left to right, as shown.

The dimension string snaps into position in relation to the first dimension string placed.



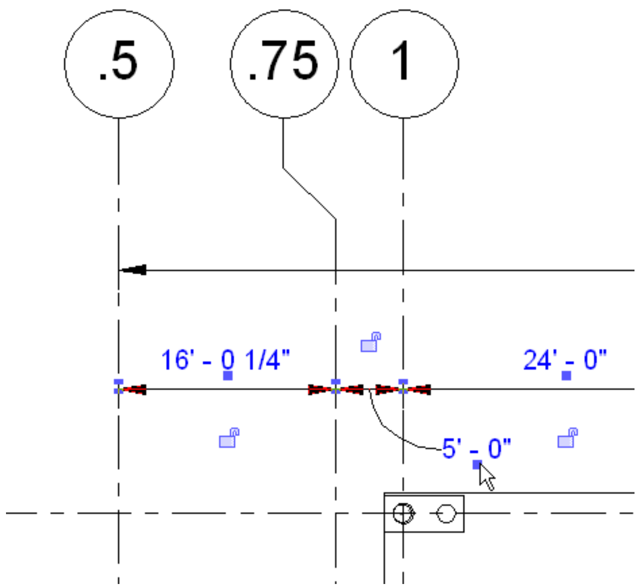
5 Click Selection panel ➤ Modify.



Modify the position of dimension text

6 Select the dimension string.

7 Select the drag text control for the second dimension text from the left, and drag it down and to the right to make it easier to read.



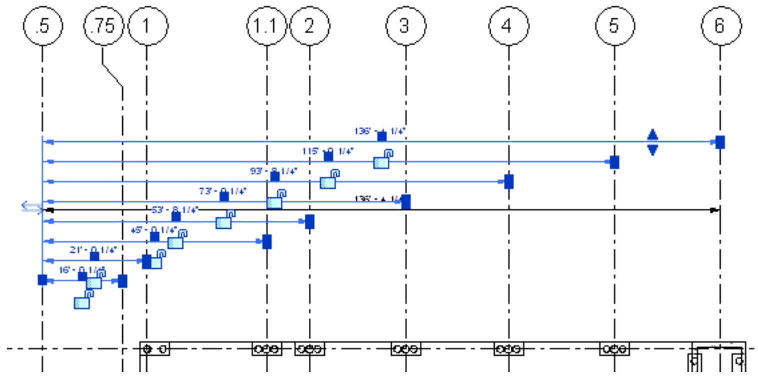
8 Press *Esc*.

Change the dimension type

9 Select the dimensions placed on the north side of the structure.

10 Click Element panel ➤ Change Element Type drop-down ➤ Linear Dimension Style : Arrow - Baseline - 3/32" Arial.

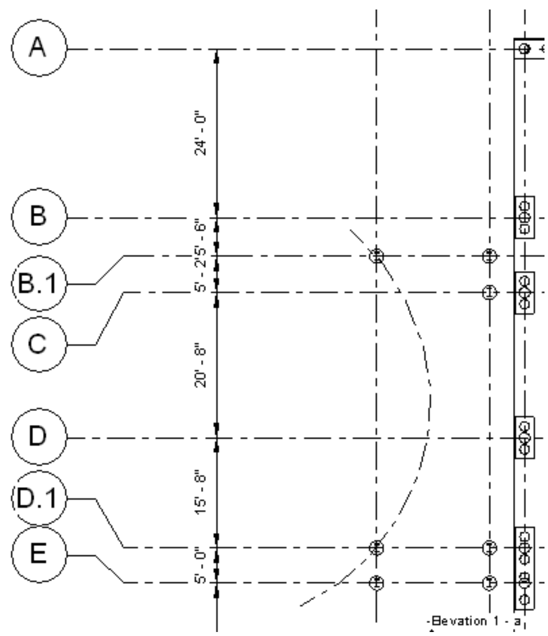
The multiple dimensions display the distance from the same baseline (grid .5).



11 Press *Esc* twice.

Dimension the horizontal grids

12 Using the same method, place a string of dimensions on the horizontal grids by clicking each grid line, starting at grid A, as shown.



The dimension string snaps into position in relation to the first dimension string placed.

Create a radial dimension

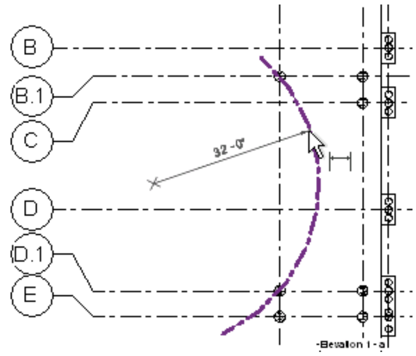
13 Zoom in to the entry way.

14 Click Dimension panel ➤ Radial.

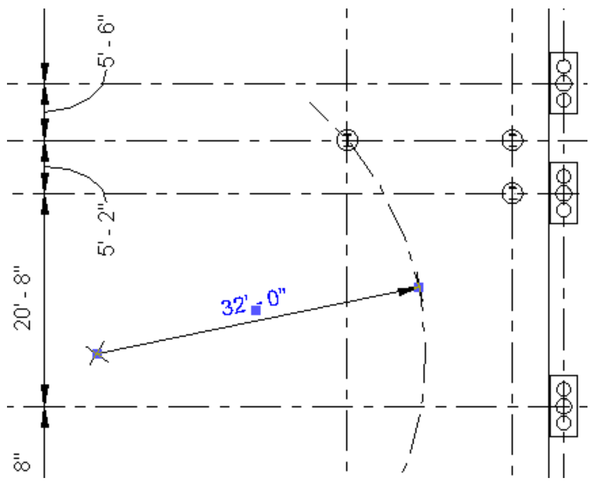
15 On the Options Bar, for Place Dimensions, select Wall centerlines.

16 Dimension the arc:

- Select the arc to display the radial dimension.



- Move the cursor outside the arc, and click to place the dimension.



17 Press *Esc.*

Add a dimension to the walls in the stairwell

18 Zoom in to the northeast stairwell.

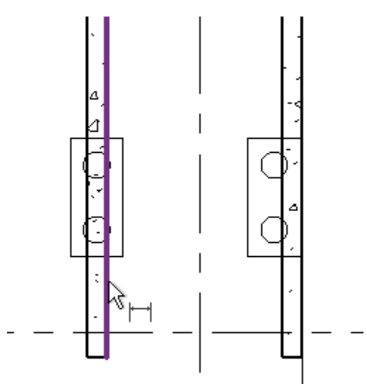
19 Click Dimension panel ➤ Aligned.

20 On the Options Bar:

- For Place Dimensions, select Wall faces.
- For Pick, select Individual References.

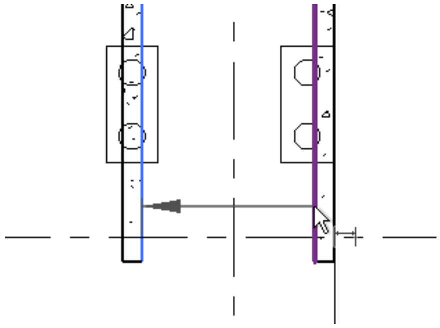
You change the options so you can select the faces of the concrete walls individually.

21 In the Drawing area, press *Tab* to highlight the inside face of the retaining wall as shown.

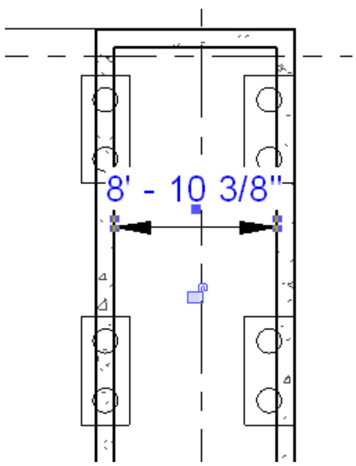


22 In the opening for the stair, click to place the first arrow of the dimension.

23 Click the inside face of the concrete wall on the opposite side of the stair opening.



24 Click outside the wall to place the dimension.



25 Press *Esc* twice.

Add text to the dimension value

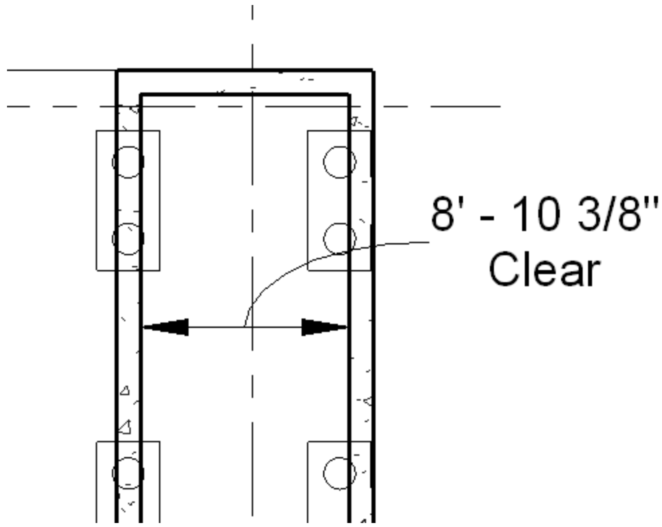
26 Double-click the dimension text.

Using the options in the Dimension Text dialog, you can replace or append dimensions values with text. The values do not affect the model geometry.

27 In the Dimension Text dialog, under Text Fields, for Below, type **Clear**, and click OK.

28 Select the drag text control, and drag the text outside of the stairs.

29 Press *ESC*.

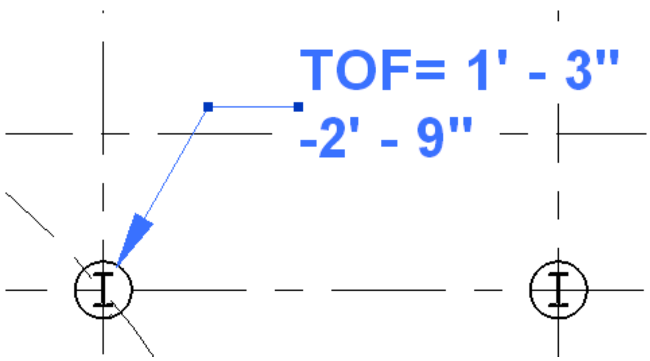


30 Close the file with or without saving it.

In the next tutorial, a new training file is supplied.

Creating Spot Dimensions

In this exercise, you add spot dimensions to the sloped roof of the structure and the footings on the entrance. The dimensions display the height from ground level.



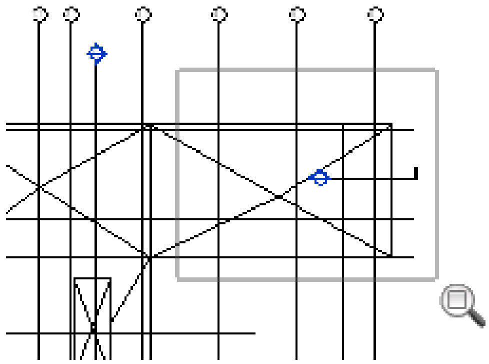
Training File

- Click [R icon] ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_AD_02_Spot_Dimensions_i.rvt.

Add spot dimensions to the tapered roof

1 In the Project Browser, expand Structural Plans, and double-click Roof.

2 Zoom in to the slope slab on the northeast corner of the structure.



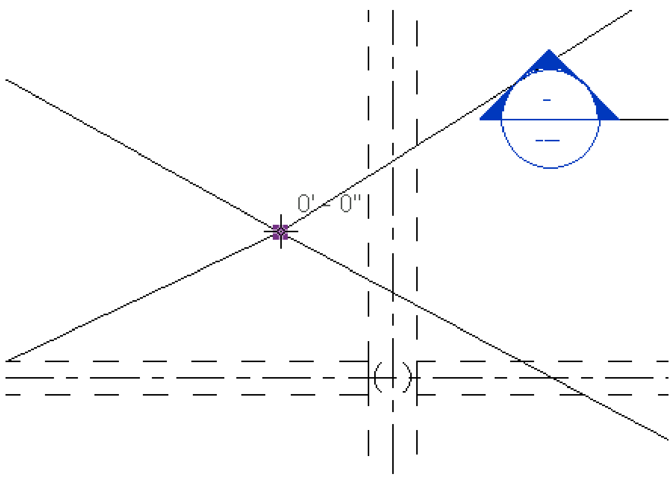
3 Click Annotate tab ➤ Dimension panel ➤ Spot Elevation.

4 Click Element panel ➤ Change Element Type drop-down ➤ Spot Elevations: No Symbol (Relative).

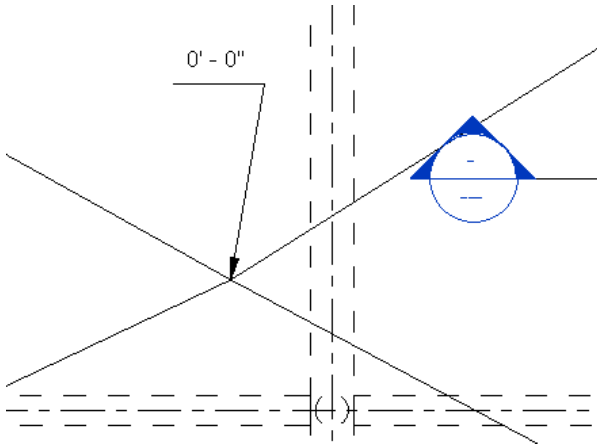
5 On the Options Bar, select Leader, and for Relative Base, select Current Level.

6 Place the spot dimension as follows:

- Move the cursor along the edge of the tapered roof.
  The value of the spot elevation displays as you move the cursor.
- Click the center of the tapered insulation to create the first leader point, as shown.



- Move the cursor up and to the right.
- Click to create the second leader point.
- Move the cursor to the left.
- Click to place the spot dimension.

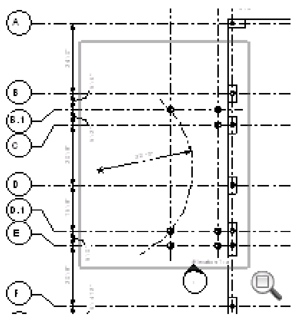


- Click Selection panel ➤ Modify.

Add a spot dimension to the entrance footing

7 In the Project Browser, expand Structural Plans, and double-click 01 - Entry Level.

8 Zoom in to the footings of the entry way.



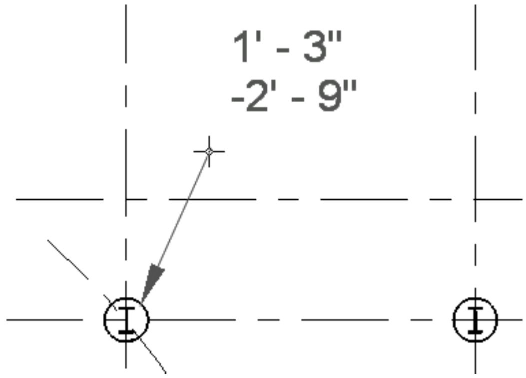
9 Click Dimension panel ➤ Spot Elevation.

10 On the Options Bar:

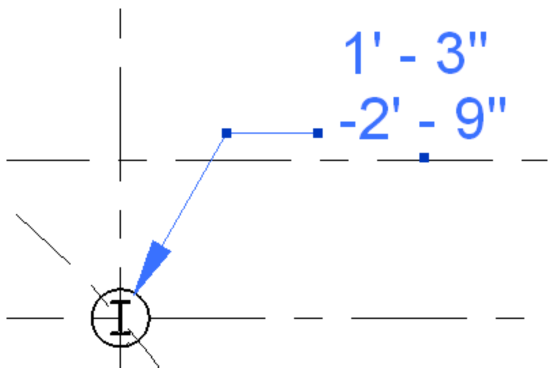
- Click Leader.
- For Relative Base, select Current Level.
- For Display Elevations, select Top & Bottom Elevations.

11 Place the spot elevation as follows:

- Click the upper-right corner of the footing.
- Move the cursor up and to the right.
- Click to create the second leader point.



- Move the cursor to the right.
- Click to create the third leader point.
- Press *Esc* twice.
- Select the dimension drag control (blue square) and place it to the right of the leader. The elevation at the top and bottom of the footing displays.



- Press *Esc* twice.

Change the spot dimension properties

12 Select the spot dimension.

13 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

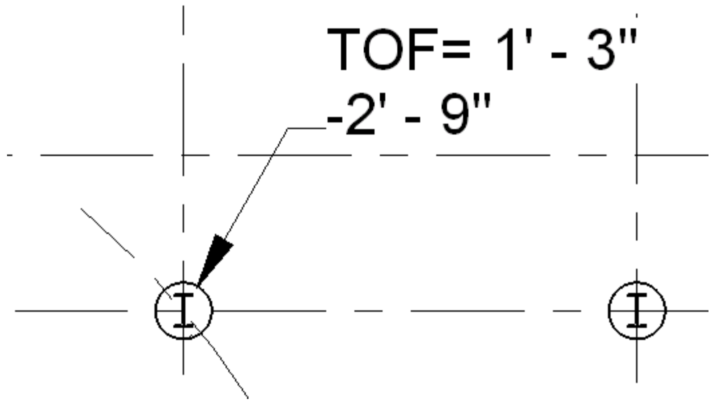
14 In the Instance Properties dialog:

- Under Text, for Single/Upper Value Prefix, type **TOF=**.

Parameter	Value
Constraints	
Relative Base	Current Level
Graphics	
Leader Shoulder	☑
Leader	☑
Text	
Display Elevations	Top & Bottom Elevations
Single/Upper Value	0' 0"
Single/Upper Value Prefix	TOF=
Single/Uppper Value Suffix	
Lower Value	-4' 0"
Lower Value Prefix	
Lower Value Suffix	

- Click OK.

15 Press *Esc*.



Change the text orientation

16 Select the spot dimension.

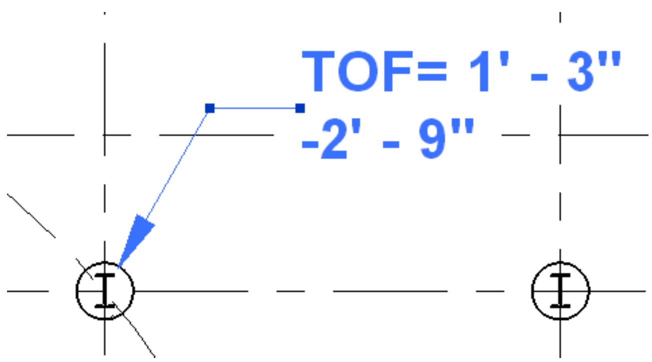
17 Click Element panel ➤ Element Properties drop-down ➤ Type Properties.

18 In the Type Properties dialog, under Text:

- Select Bold.
- For Text Location, select In-line with Leader.
- Click Apply, and then click OK.

19 In the Element Properties dialog, click OK.

20 Click and drag the dimension.



21 Press *Esc* twice.

22 Close the file with or without saving it.

In the next lesson, a new training file is provided.

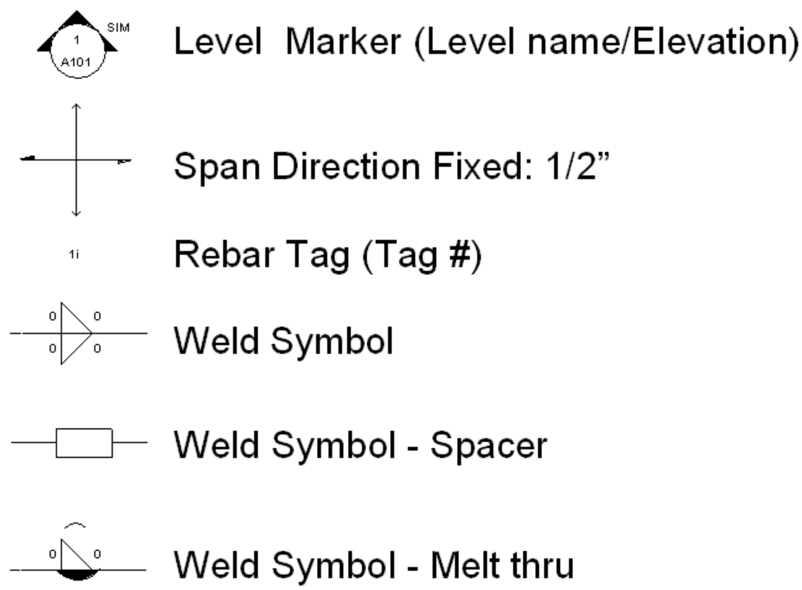
Creating an Annotation Legend

In this exercise, you create a symbol legend annotation for the project.

Legends provide a way to display a list of the various structural components and annotations used in a project. The two most common types of construction document legends are annotation legends and building component legends.

Annotation legends consist of components, such as section markers, that are paired with text that identifies them. On construction documents, annotation legends are often referred to as symbol legends.

Building component legends list and identify components, such as weld symbols and rebar tags. On construction documents, component legends are often called schedules (beam schedule, concrete schedule, and so on).



Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_AD_03_Symbols_Legend_i.rvt.

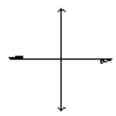
Create a legend view

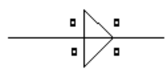
1. Click View tab ➤ Create panel ➤ Legends drop-down ➤ Legend.
2. In the New Legend View dialog, for Name, type **Symbol Legend**, and click OK.

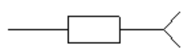
Add symbols to the legend

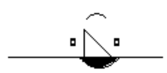
3. In the Project Browser, under Legends, double-click Symbol Legend to open the blank legend view.
4. Click Annotate tab ➤ Symbol panel ➤ (Symbol).
5. Add the following symbols to the legend view by clicking Element panel ➤ Change Element Type drop-down, and placing them in the drawing area, as shown.
   - Section Head - Filled
   - Span Direction Fixed: One Way Slab - 1/2”
   - Rebar Tag : Type
   - Weld Symbol: Both
   - Weld Symbol - Spacer: Spacer

■ Weld Symbol-w-Preparation: Melt Thru









6 Click Selection panel ➤ Modify.

Create a new text type

7 Click Annotate tab ➤ Text panel ➤ Text.

The text size for the symbol legend is not available. You create a text type with the necessary size by duplicating an existing text type and then modifying the type properties.

8 Click Element panel ➤ Change Element Type drop-down ➤ Text: 1/4" Arial text.

9 Click Element panel ➤ Element Properties drop-down ➤ Type Properties.

10 In the Type Properties dialog, click Duplicate.

11 In the Name dialog, type **1/8" text** for Name, and click OK.

12 In the Type Properties dialog, under Text, type **1/8"** for Text Size, and click OK.

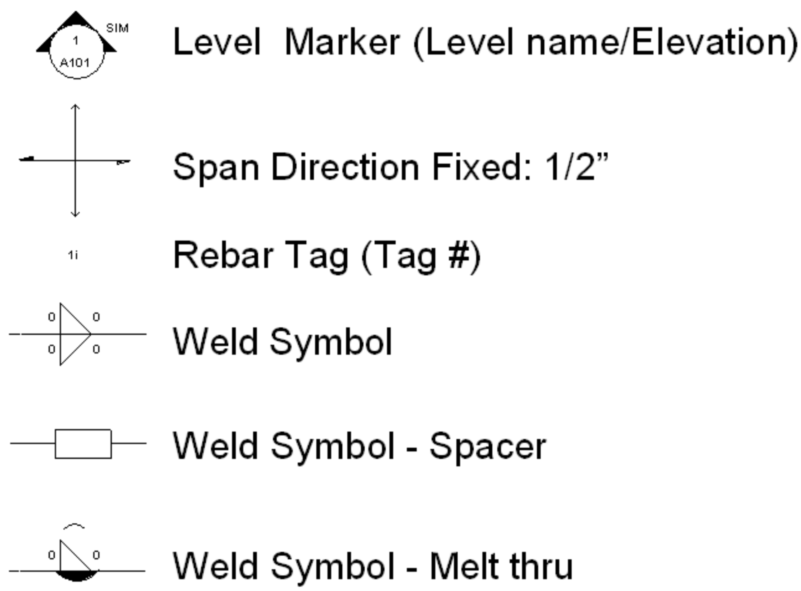
Add text to the legend

13 Click to the right of the first symbol to specify the text start point.

14 Type **Level Marker (Level name/Elevation)** for the text note.

15 Working from the top down, type the following text for the remaining symbols in the legend.

- Span Direction Fixed: 1/2”
- Rebar Tag (Tag#)
- Weld Symbol
- Weld Symbol - Spacer
- Weld Symbol - Melt thru



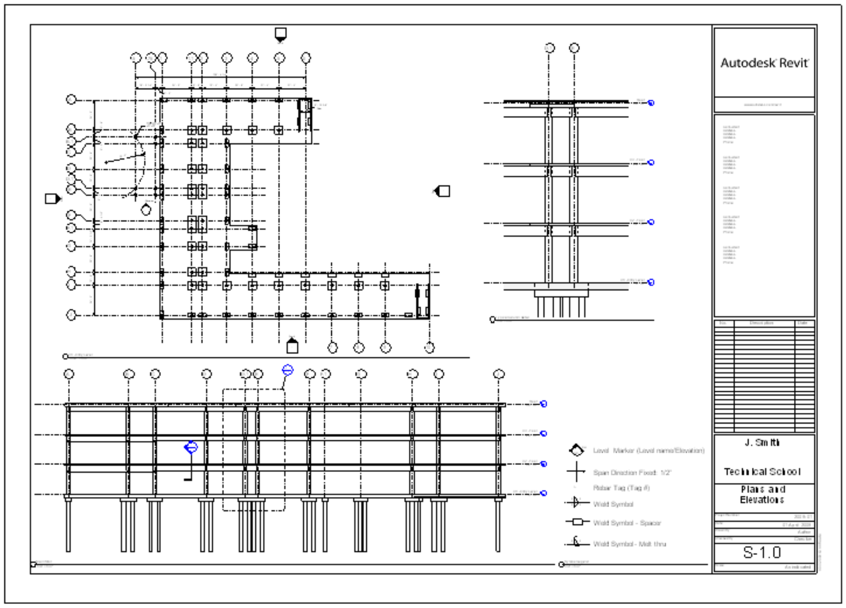
16 Click Selection panel ➤ Modify.

Place the symbol legend on a sheet

17 In the Project Browser, expand Sheets (all), and double-click S-1.0 - Plans and Elevations.

18 In the Project Browser, click Symbol Legend, drag it to the lower-right corner of the sheet, and click to place it.

19 Click Selection panel ➤ Modify.



The symbol legend is added to the sheet.

20 Close the file with or without saving it.

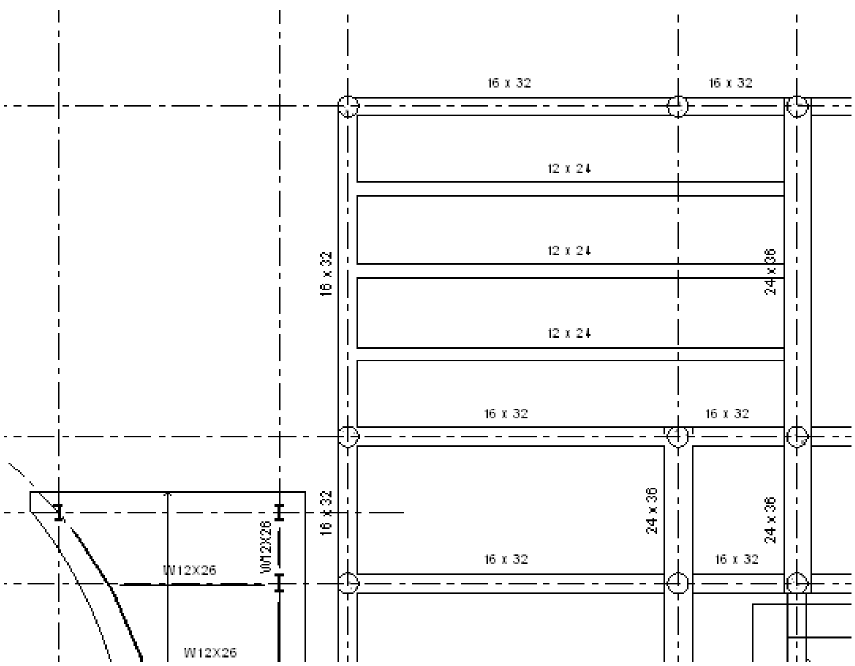
In the next tutorial, a new training file is supplied.

Tagging Objects

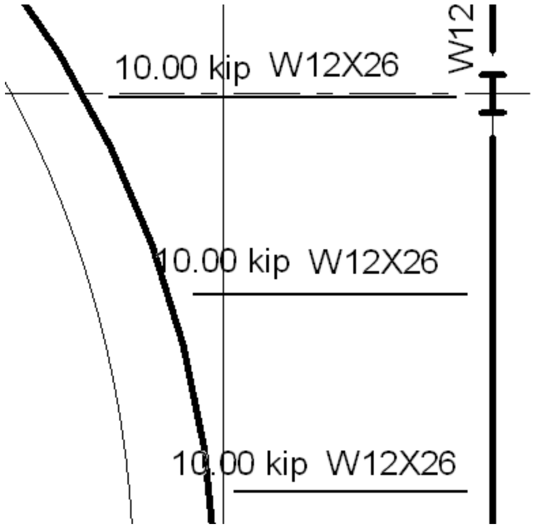
23

In this lesson, you use the annotation tools available in Revit Structure to tag beams in the structural plan view. A tag is a useful annotation for identifying elements in a project. You can modify the appearance of each tag before or after they are placed. You learn to:

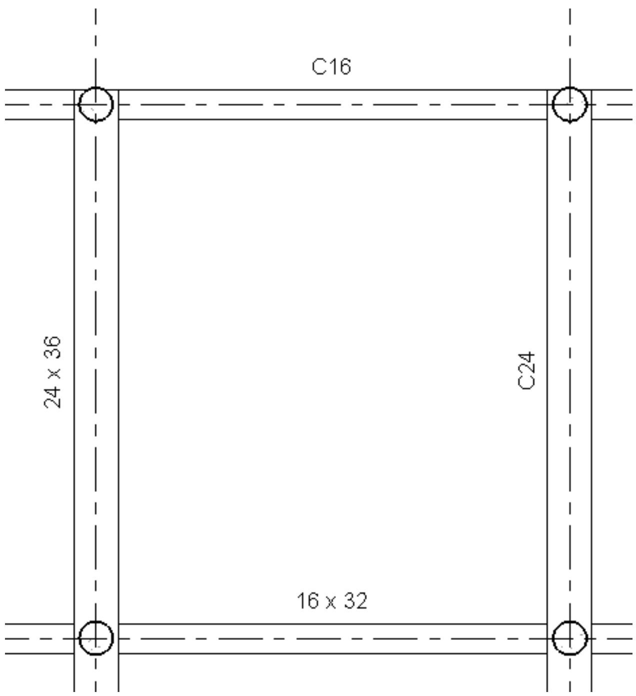
- Tag all beams in a plan view.



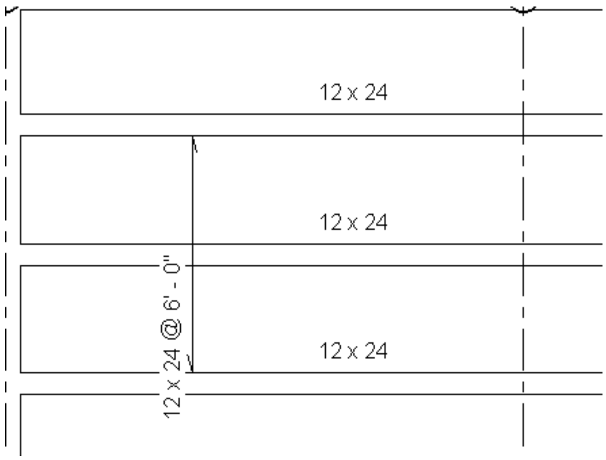
- Add a reaction value to the steel beam tags on the entry way frame.



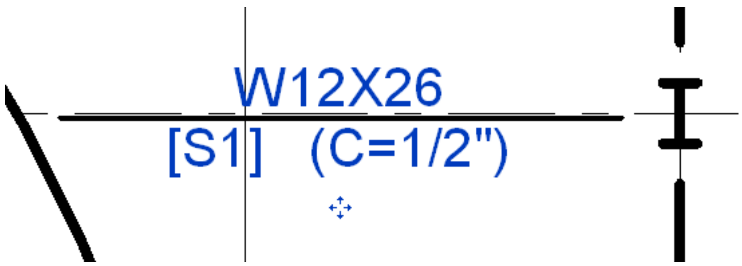
- Create a custom type mark.



- Tag a concrete beam system.



- Create a custom tag that includes the camber offset dimension for the steel beams of the entry way frame.



Tagging Beams

In this exercise, you learn how to use the beam annotation tool to tag all beams in a plan view, and add an end reaction annotation to the steel beams of the entry way roof frame. You also learn how to create a custom type mark to use in place of an existing beam tag, and how to tag a beam system.

Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_TAGS_01_Beam_Tags_i.rvt.

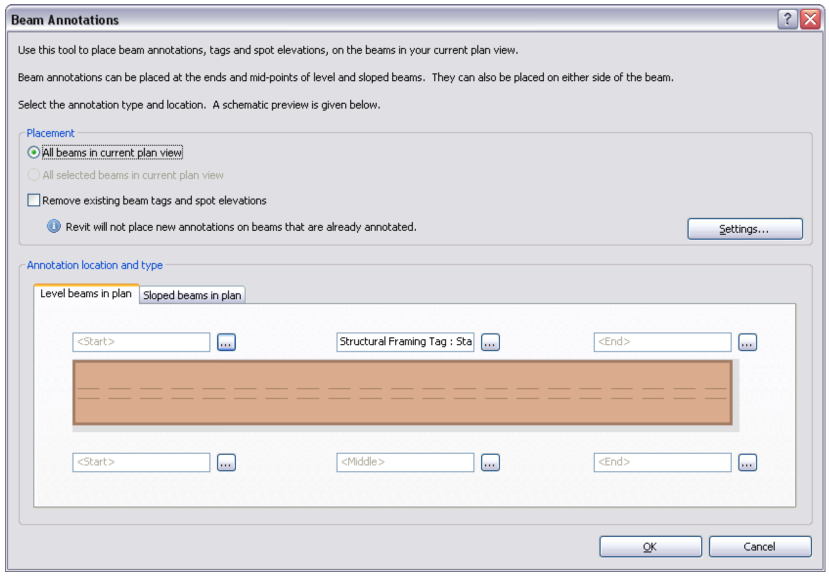
Tag all beams in plan view

1 In the Project Browser, expand Structural Plans, and double-click 02 Floor.

NOTE For this exercise, the slab is temporarily hidden, and the architectural drawing visibility is turned off so the beam tags are visible.

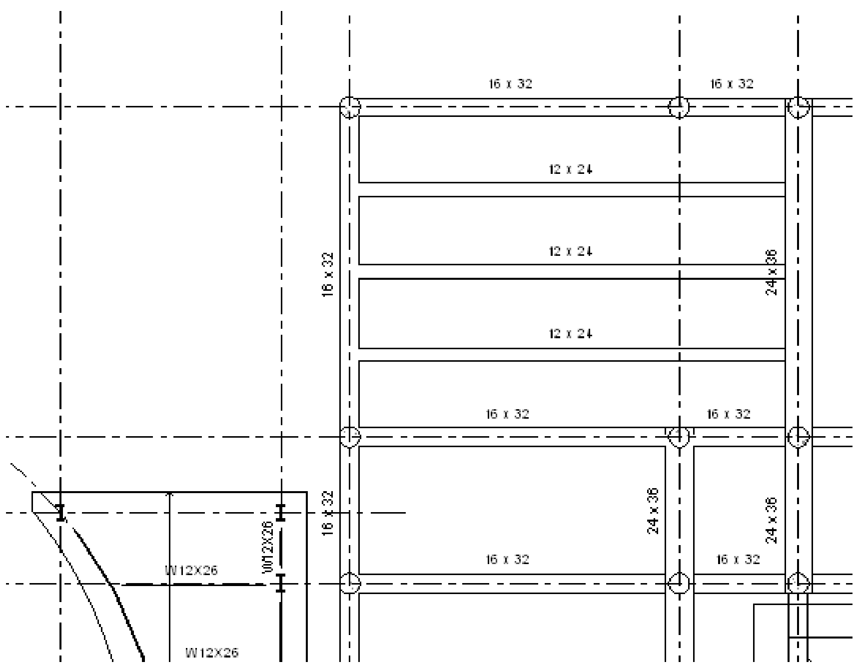
2 Click Annotate tab ➤ Tag panel ➤ Beam Annotations.

The Beam Annotations dialog opens.



3 In the Beam Annotations dialog:

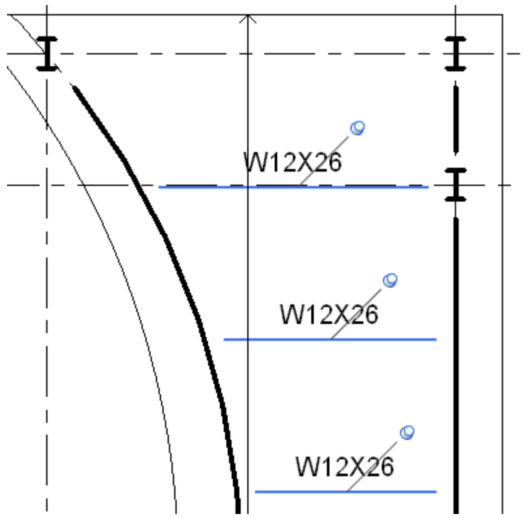
- Under Placement, select All beams in current plan view.
- Click OK.
  In the Revit dialog, click Yes to turn on the structural framing tag visibility for the plan view.



A beam tag is added to the top-middle location on all beams within the plan view.

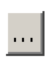
Add an end-reaction value to the steel beams of the entry way

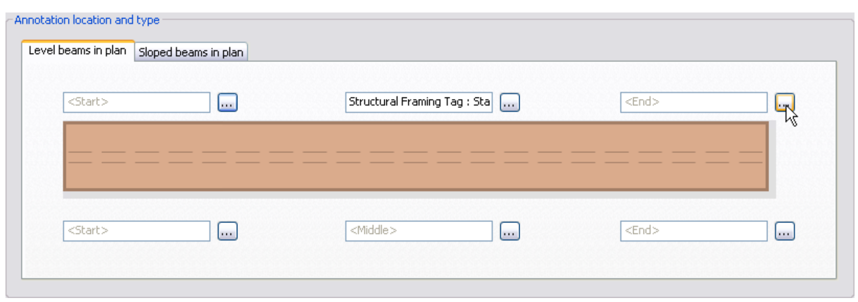
4 Select multiple beams located in the steel frame of the entry way.



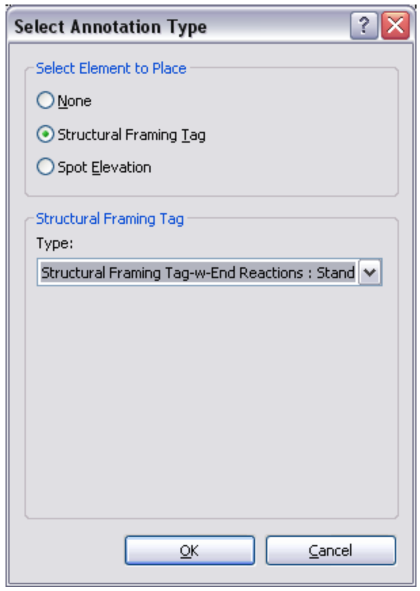
5 Click Annotate tab ➤ Tag panel ➤ Beam Annotations.

6 In the Beam Annotations dialog:

- Under Placement, select All selected beams in current plan view, and click Remove existing beam tags and spot elevations.
- Click Settings.
- In the Placement Settings dialog, for Horizontal End Offset, type **1**", and click OK. This dimension sets the annotation offset distance measured from the beam start location, as indicated in the dialog.
- Under Annotation location and type, click the Level beams in plan tab.
- Click , to specify the top-end annotation type.



- In the Select Annotation Type dialog, under Select Element to Place, click Structural Framing Tag.
- Under Structural Framing Tag, select Structural Framing Tag-w-End Reactions : Standard, and click OK.



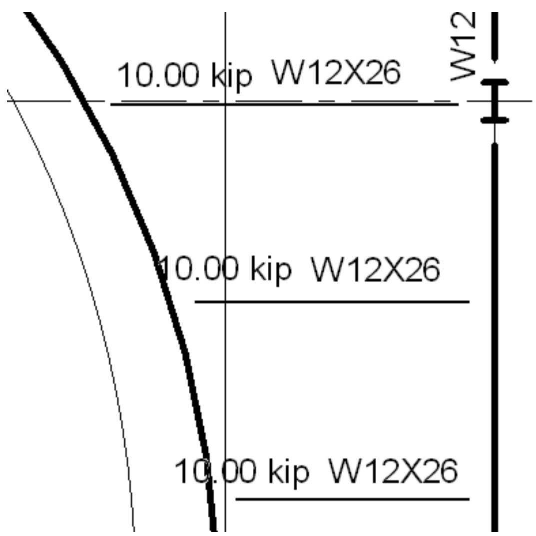
- In the Beam Annotations dialog, click OK.

Enter the reaction force

7 Right-click one of the beams, and select Element Properties.

8 In the Instance Properties dialog, under Analysis Results, type **10.0 kip** for End Reaction - Live, and click OK.

The end reaction value is added to the selected beams.

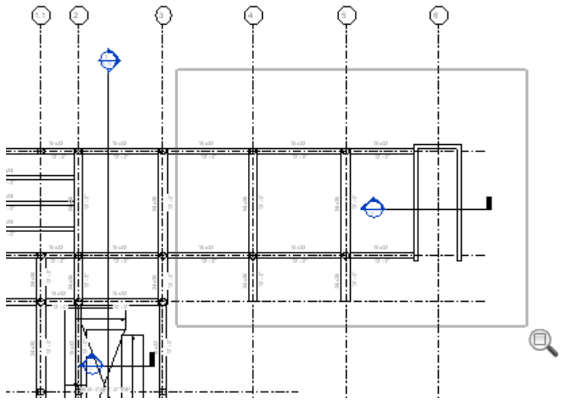


9 Press *Esc*.

Create a custom type mark tag

10 Type **ZF** (Zoom to Fit).

11 Zoom in to the beams on the northeast corner of the structure.



12 Click one of the 16 x 32 concrete beams, and click Element panel ➤ Element Properties drop-down ➤ Type Properties.

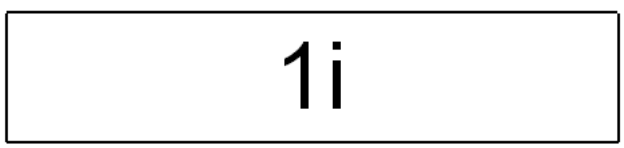
13 In the Type Properties dialog, under Type Mark, type **C16**, click Apply, and then click OK.

14 Using the same method, add type mark **C24** to one of the 24 x 36 concrete beams.

Add a beam type mark by editing the beam tag family

15 Select one of the 16 x 32 beam tags, and click Family panel ➤ Edit Family.

When asked whether to open the Structural Framing Tag for editing, click Yes. You are now in the Family Editor, and the structural beam tag label for the selected beam type displays in the drawing area.

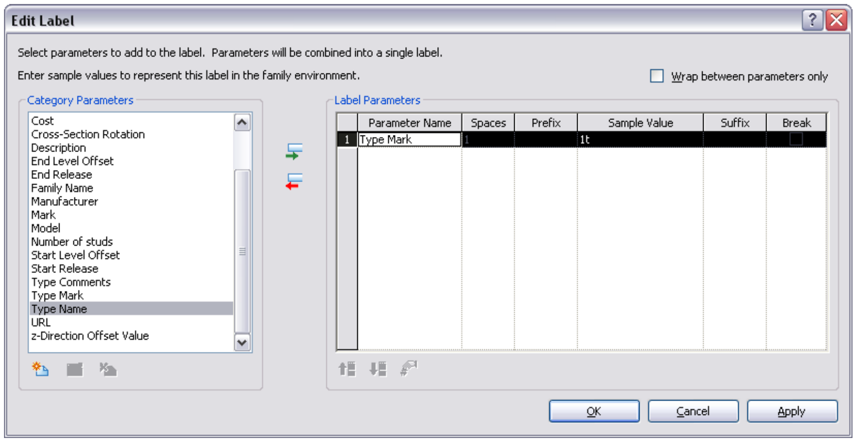


16 Select the structural beam tag label and click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

17 In the Instance Properties dialog, under Label, for Value, click Edit.

18 In the Edit Label dialog:

- Under Category Parameters, select Type Mark, click  (Add parameter(s) to label).
- Under Label Parameters, select Type Name, click  (Remove parameter from label).



19 Click OK twice.

Save the new beam tag with type mark

20 Click  ➤ Save As ➤ Family.

21 In the Save As dialog:

- Select a folder location.
- Type **Structural Framing Tag - by Type Mark** for File name.
- Click Save.

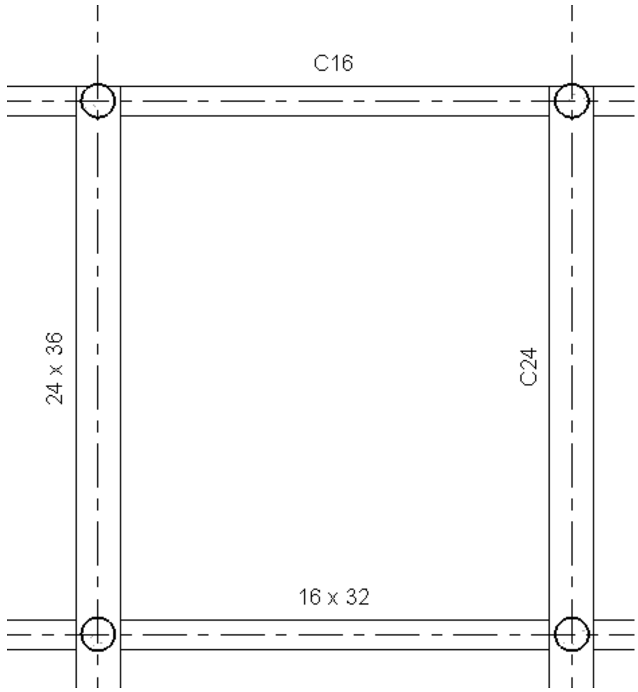
22 Click Family Editor panel ➤ Load into Project.

Apply the new type mark tag

23 In the plan view, click the 16 x 32 concrete beam tag, and, while pressing *Ctrl*, select the 24 x 36 concrete beam tag.

24 Click Element panel ➤ Change Element Type drop-down ➤ Structural Framing Tag - by Type Mark: Standard.

The beam tags for the selected beams displays the new type mark.

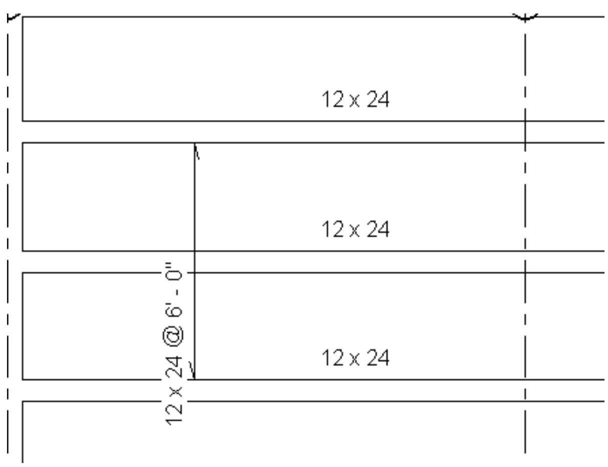


Tag the beam system

25 Zoom in to the beam system on the northwest side of the structure.

26 Click Annotate tab ➤ Symbol panel ➤ Beam.

27 Click the beam system to place the tag.



28 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Creating a Custom Beam Tag

In this exercise, you learn how to create a custom beam tag that includes additional beam parameters, including stud size and camber size. You can use these parameters when organizing beam information for a framing schedule.

Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_TAGS_02_Custom_Beam_Tags_i.rvt.

Edit the beam properties to include the new parameters

1 In the Project Browser, expand Structural Plans, and double-click 02 - Floor.

2 Zoom in to the steel fame of the entry way on the west side of the structure.

3 Click one of the W12X26 beams, press *Ctrl*, and select the remaining beams of the same type.

NOTE For purposes of this exercise, do not select beams that include an end reaction value (10.00 kip).

---

4 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

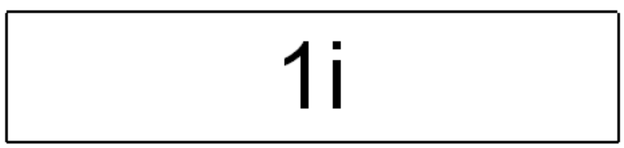
5 In the Instance Properties dialog:

- Under Structural, for Camber Size, type **C=1/2"**.
- Under Structural, for Number of studs, type **S1**.
- Click OK.

Open the beam tag family

6 Right-click one of the W12X26 beam tags, and click Edit Family.

When asked whether to open the Structural Framing Tag for editing, click Yes. You are now in the Family Editor, and the structural beam tag label for the selected beam type displays in the drawing area.



7 Click the structural beam tag label, and click Label panel ➤ Edit Label.

Add multiple categories to the beam label

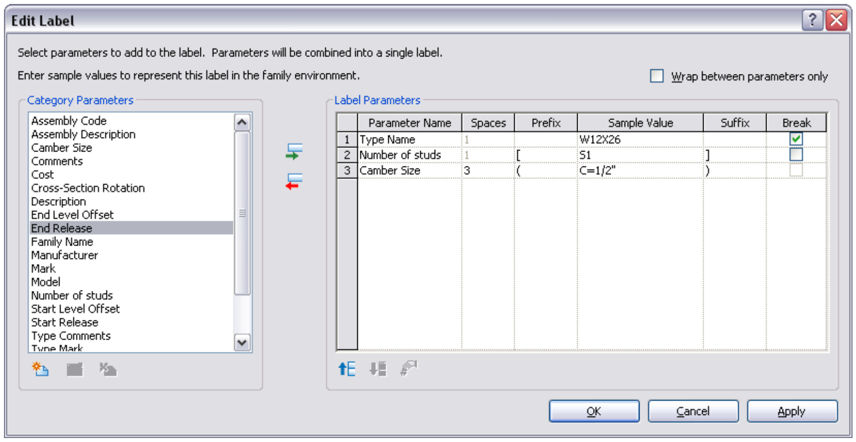
8 In the Edit Label dialog, add the following categories:

- Under Category Parameters, select Number of studs, and click (Add parameter). Number of studs is added to the label as parameter 2.
- Under Category Parameters, select Camber Size, and click . Camber Size is added to the label as parameter 3.

9 In the Edit Label dialog, add the following Label Parameters:

- Select parameter 1, for Sample Value, type **W12X26**, and select Break.
- Select parameter 2, for Prefix, type **[** (open bracket), for Sample Value, type **S1**, and for Suffix, type **]** (close bracket).

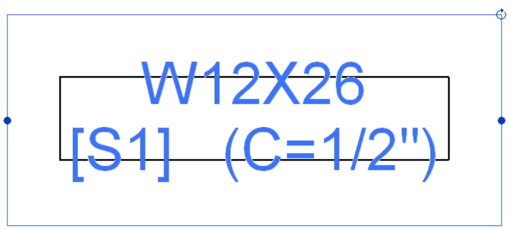
■ Select parameter 3, for Spaces, select 3, for Prefix, type **(** (left parenthesis), for Sample Value, type **C=1/2"**, and for Suffix, type **)** (right parenthesis).



10 In the Edit Label dialog, click Apply, and then click OK.

Align the beam tag

11 Click the drag control for the label until the text is positioned on two lines, as shown.



12 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

13 In the Instance Properties dialog, under Graphics, for Vertical Align, select Top, and click OK.

14 Type **VV** (Visibility/Graphic Overrides).

15 In the Visibility/Graphic Overrides dialog:

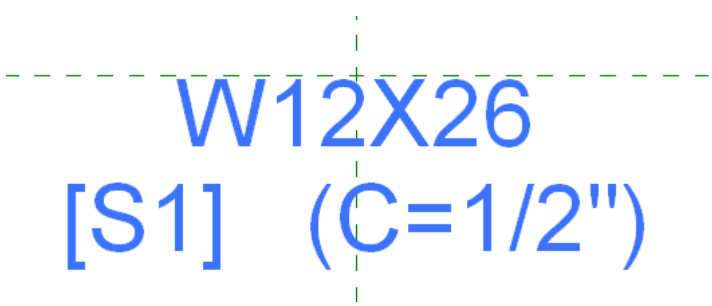
■ Click the Annotation Categories tab.

■ Under Visibility, select Reference Planes.

■ Click OK.

16 Select the beam label text.

17 Press the *Up Arrow* or *Down Arrow* to move the text until the beam type is positioned just below the horizontal reference plane, as shown.



18 Press *Esc*.

Save the new beam tag with type mark

19 Click ➤ Save As ➤ Family.

20 In the Save As dialog:

- Select a folder location.
- Type **Structural Framing Tag - with Camber** for File name.
- Click Save.

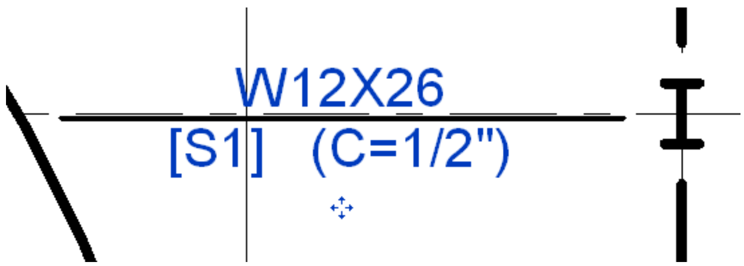
Load the new beam tag into the project

21 Click Family Editor panel ➤ Load into Project.

22 Select a W12X26 beam tag.

23 Click Element panel ➤ Change Element Type drop-down ➤ Structural Framing Tag - with Camber : Standard.

The beam tag displays the camber setting and stud size.



24 Press *Esc* twice.

25 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Scheduling

24

In this lesson, you create a custom schedule of structural framing elements in your project. You learn to:

- Create a structural framing schedule.

Structural Framing Schedule

Mark	REBAR	Comments	Reference Level	Family and Type
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32

- Add structural elements to the standard schedule template, using shared parameters.

Structural Framing Schedule

Mark	b	h	REBAR	Comments	Reference Level
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor

- Create a type schedule.

	Concrete Beam Schedule					
	SIZE					Structural
MARK	W	D	REBAR	REMARKS	Reference Level	Usage
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
Girder: 90						
Joist						
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
Joist: 16						
Other						
	24"	36"			02 - Floor	Other
	24"	36"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
Other: 8						

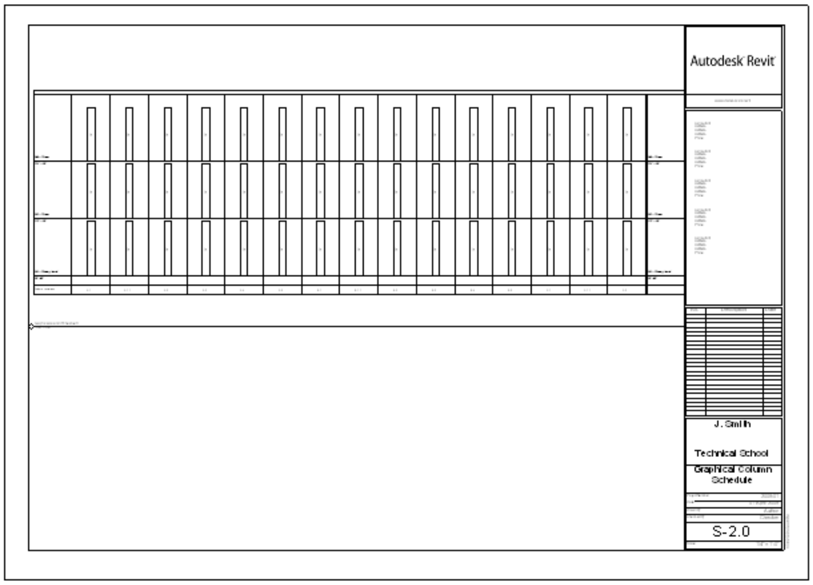
■ Customize the type schedule.

	Concrete Beam Schedule					
	SIZE		REBAR			Structural
MARK	W	D	Bottom Bars	Top Bars	REMARKS	Usage
B1	24"	36"	2-#7A 1-#7B	2-#5C		
B2	16"	32"	2-#6A 1-#6B	2-#5C		Girder
B3	12"	24"	2-#5A 1-#5B	2-#5C		Joist

■ Create an instance schedule.

	Concrete Beam Schedule								
	SIZE		REBAR			Structural			Total Cost
MARK	W	D	Bottom Bars	Top Bars	REMARKS	Usage	Volume	Cost	
B1	24"	36"	2-#7A 1-#7B	2-#5C		Girder	3931.15 CF	3.00	$11793.45
B2	16"	32"	2-#6A 1-#6B	2-#5C		Girder	10372.74 CF	1.50	$15559.12
B3	12"	24"	2-#5A 1-#5B	2-#5C		Joist	1080.42 CF	4.00	$4321.67

■ Create a graphical column schedule.



Creating a Structural Framing Schedule

In this exercise, you learn how to create a structural framing concrete beam schedule.

Structural Framing Schedule				
Mark	REBAR	Comments	Reference Level	Family and Type
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32

Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_SCH_01_Framing_Schedule_i.rvt.

Create the schedule

1 Click View tab ➤ Create panel ➤ Schedules drop-down ➤ Schedule/Quantities.

2 In the New Schedule dialog, under Category, select Structural Framing, and click OK.

Choose which fields to include in the beam schedule

3 In the Schedule Properties dialog, click the Fields tab.

4 Under Available fields, select Mark, and click Add.

The Mark field is moved under Scheduled fields.

5 Using the same process, add the following fields to the schedule:

- Comments
- Reference Level
- Family and Type

NOTE The width and depth parameters will be added to the schedule in the next exercise, Creating Shared Parameters.

---

6 In the Schedule Properties dialog, click Add Parameter.

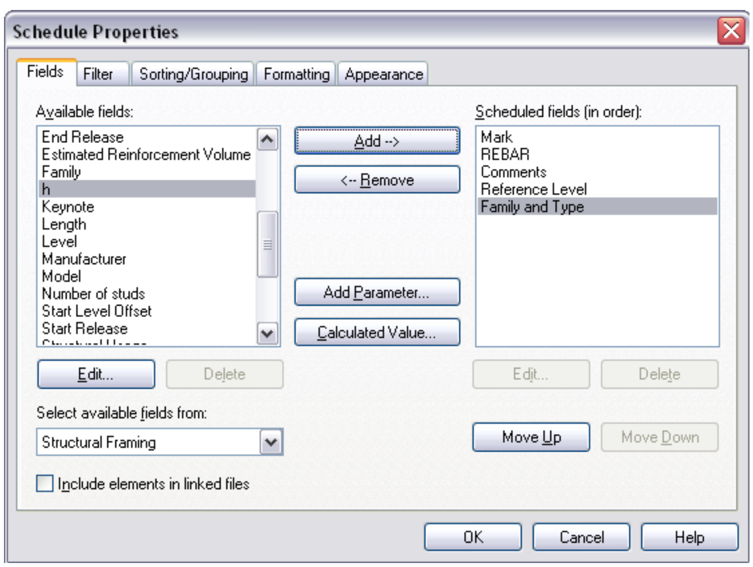
7 In the Parameter Properties dialog:

- Under Parameter Data, type **REBAR** for Name.
- Under Type of Parameter, select Text.
- Click OK.

NOTE Rebar is now a project parameter that can also be found in the properties of all structural framing components, including beams.

---

8 Under Scheduled fields, order the fields, as shown, by selecting them and clicking Move Up or Move Down.



9 In the Schedule Properties dialog, click OK.

A new schedule displays all structural framing elements in the project.

Structural Framing Schedule

Mark	REBAR	Comments	Reference Level	Family and Type
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32
			02 - Floor	Concrete-Rectangular Beam: 16 x 32

10 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Creating Shared Parameters

In this exercise you learn how to use shared parameters to define additional elements that are usually not included in the beam schedule when it is created within the project template. You can add shared parameters to any family, regardless of category. They are defined and stored in an external file, ensuring consistency across families and projects. Their values may also be aggregated and reported within Revit Structure multi-category schedules.

For example, you can use shared parameters when you need to add width and depth information in a concrete beam schedule. This requires assigning the existing width and depth parameters originally defined as family parameters to shared parameters in the beam family. The following exercise demonstrates the solution for this situation as well as the process for setting up shared parameters and adding them to a family.

Structural Framing Schedule

Mark	b	h	REBAR	Comments	Reference Level
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor

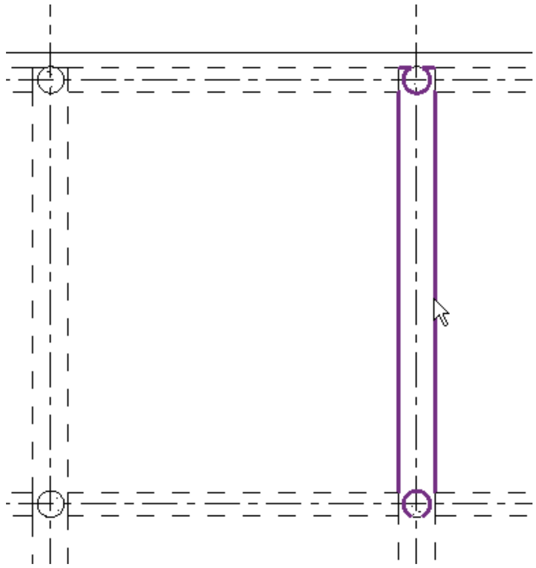
Training File

- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_SCH_02_Shared_Parameters_i.rvt.

Editing the family parameters

1 In the Project Browser, under Structural Plans, double-click 02 - Floor.

2 Select one of the concrete beams.

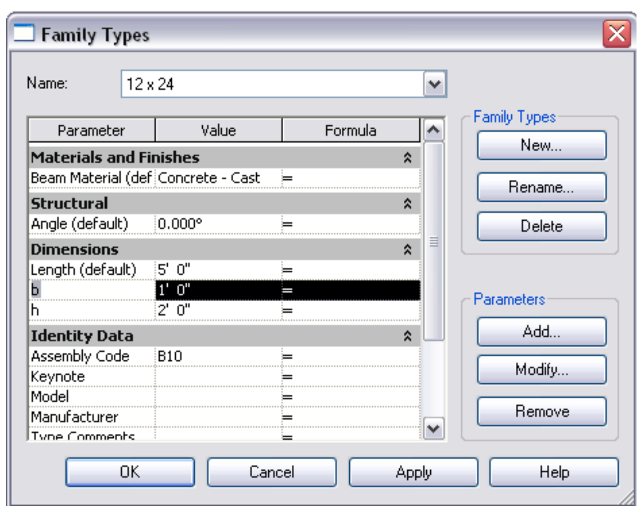


3 Click Family panel ➤ Edit Family.

When asked whether to open the beam for editing, click Yes. You are now in the Family Editor, and the selected beam family displays in the drawing area.

4 Click Family Properties panel ➤ Types.

5 In the Family Types dialog, under Dimensions, select the b parameter, and under Parameters, click Modify.



6 In the Parameter Properties dialog, select Shared Parameter, and click Select.

In the Revit dialog, click Yes to specify a shared parameter file.

7 In the Edit Shared Parameters dialog, click Create.

8 In the Save As dialog, specify a location for the file, and for name, type **Project Shared Parameters**. Click Save.

9 In the Edit Shared Parameters dialog, under Groups, click New.

10 In the New Parameter Group dialog, for name, type **Dimensions**, and click OK.

11 In the Edit Shared Parameters dialog, under Parameters, click New.

12 In the Parameter Properties dialog:

- Under Name, type **b**.
- Under Type of Parameter, select Length.
- Click OK.

13 In the Edit Shared Parameters dialog, under Parameters, click New.

14 In the Parameter Properties dialog:

- Under Name, type **h**.
- Under Type of Parameter, select Length.
- Click OK.

15 In the Edit Shared Parameters dialog, click OK.

16 In the Shared Parameter dialog, select the b parameter, and click OK.

17 In the Parameter Properties dialog, click OK.

18 In the Family Types dialog, under Dimensions, select the h parameter, and under Parameters, click Modify.

19 In the Parameter Properties dialog, select Shared Parameter, and click Select.

20 In the Shared Parameter dialog, select the h parameter, and click OK.

21 In the Parameter Properties dialog, click OK.

22 In the Family Types dialog, click OK.

NOTE The b and h parameters, which were originally family parameters, are now shared parameters. They will appear in the structural framing schedule field after they are reloaded into the project file.

---

23 Click Family Editor panel ➤ Load into Project.

24 In the Revit dialog, select Overwrite the existing family.

Remove Family and Type parameter

25 In the Project Browser, under Schedules/Quantities, right-click Structural Framing Schedule, and select Properties.

26 In the Instance Properties dialog, under Other, for Fields, click Edit.

27 In the Schedule Properties dialog, under Scheduled fields (in order), select Family and Type, and click Remove.

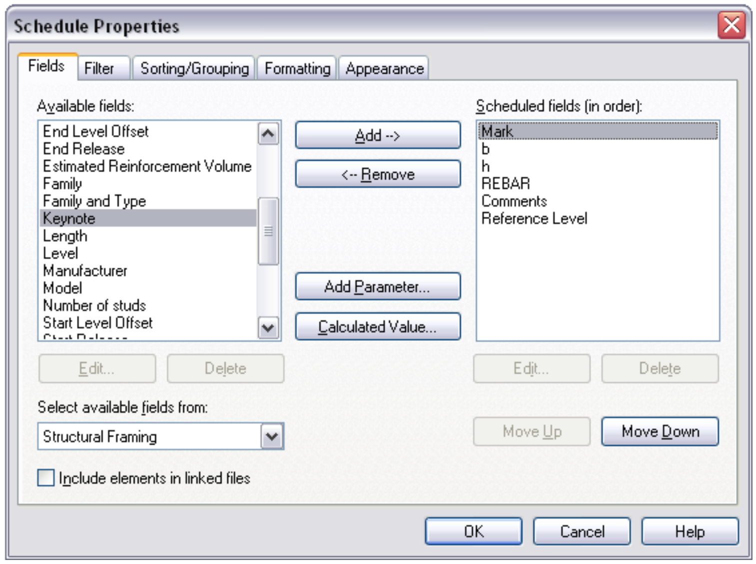
The Family and Type field is removed from the Scheduled fields column.

Add depth and width parameters to the beam schedule

28 Under Available Fields, select b (Width) and h (Depth), and click Add.

The b and h fields move to the Scheduled fields list.

29 Under Scheduled fields, order the fields, as shown, by selecting them and clicking Move Up or Move Down.



30 Click OK twice.

31 In the Project Browser, under Schedules/Quantities, double-click Structural Framing Schedule.

Structural Framing Schedule					
Mark	b	h	REBAR	Comments	Reference Level
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor

32 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Creating a Type Schedule

In this exercise, you learn how to create a type schedule.

Concrete Beam Schedule						
	SIZE					Structural
MARK	W	D	REBAR	REMARKS	Reference Level	Usage
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
Girder: 90						
Joist						
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
Joist: 16						
Other						
	24"	36"			02 - Floor	Other
	24"	36"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
Other: 8						

Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_SCH_03_Type_Schedule_i.rvt.

Modify the table elements

1 In the Project Browser, under Schedules/Quantities, double-click Structural Framing Schedule.

2 Modify the schedule headings as follows:

- Select Mark, and type **MARK**.
- Select b (Width), and type **W**.
- Select h (Depth), and type **D**.
- Select Comments, and type **REMARKS**.
- Select the title, and type **Concrete Beam Schedule**.

NOTE The schedule name has changed in the Project Browser.

3 Select headings W and D.

4 Click Schedule panel ➤ Headers: Group.

A new blank cell is created above columns W and D.

5 Click the new cell, and type **SIZE**.

Concrete Beam Schedule					
	SIZE				
MARK	W	D	REBAR	REMARKS	Reference Level

Select a filter

6 Right-click the open area next to the schedule, and select View Properties.

7 In the Instance Properties dialog, under Other, for Filter, click Edit.

8 In the Schedule Properties dialog:

- Click the Filter tab.
- Select Reference Level for Filter by.
- Select 02 - Floor.
- Click OK.

9 In the Instance Properties dialog, click OK.

Because the schedule is filtered, it displays the structural framing elements only of Level 2.

Concrete Beam Schedule					
	SIZE				
MARK	W	D	REBAR	REMARKS	Reference Level
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor
	1' - 4"	2' - 8"			02 - Floor

Format units

10 Right-click the open area next to the schedule, and click View Properties.

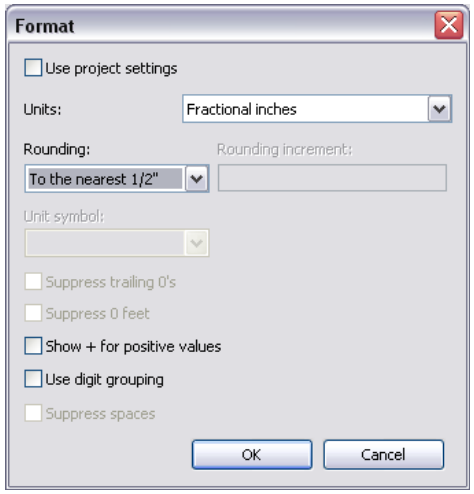
11 In the Instance Properties dialog, under Formatting, click Edit for Value.

12 In the Schedule Properties dialog, click the Formatting tab.

13 Under Fields, click b, and click Field Format.

14 In the Format dialog:

- Clear Use project settings.
- Under Units, select Fractional inches.
- Under Rounding, select To the nearest 1/2".



- Click OK.

15 Using the same method, format the units for field h.

16 In the Schedule Properties dialog, click OK.

17 In the Instance Properties dialog, click OK.

Columns W and D now display fractional inches.

Concrete Beam Schedule					
	SIZE				
MARK	W	D	REBAR	REMARKS	Reference Level
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor
	16"	32"			02 - Floor

Add structural usage parameter to the beam schedule

18 In the Project Browser, under Schedules/Quantities, right-click the Concrete Beam Schedule, and select Properties.

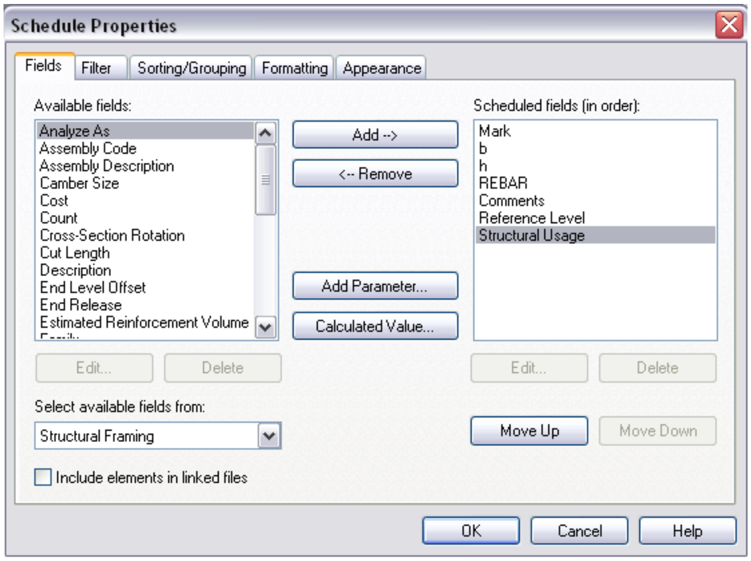
19 In the Instance Properties dialog, under Other, for Fields, click Edit.

20 In the Schedule Properties dialog, click the Fields tab.

21 Under Available Fields, select Structural Usage, and click Add.

The Structural usage moves to the Scheduled fields list.

22 Under Scheduled fields, order the fields, as shown, by selecting them and clicking Move Up or Move Down.



23 In the Schedule Properties dialog, click OK.

24 In the Instance Properties dialog, click OK.

The schedule now displays the structural usage of each item.

Concrete Beam Schedule						
	SIZE					Structural
MARK	W	D	REBAR	REMARKS	Reference Level	Usage
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Other
	24"	36"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other

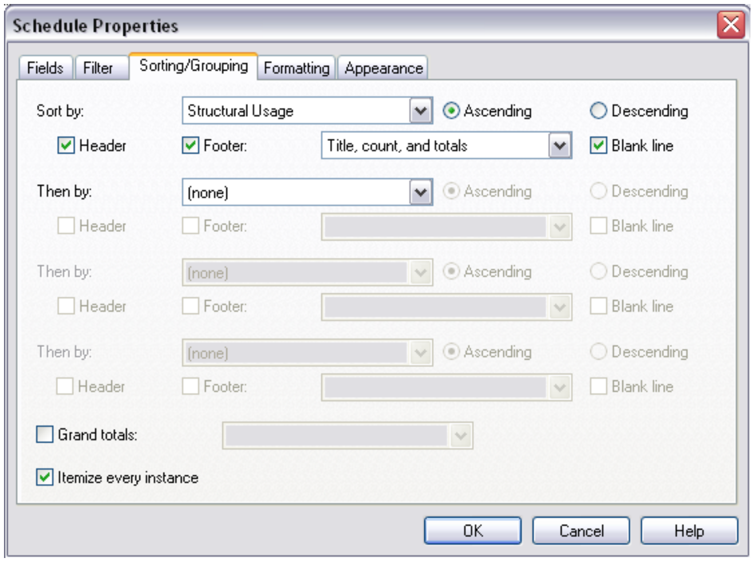
Sort by Structural Usage

25 Right-click the open area next to the schedule, and select View Properties.

26 In the Instance Properties dialog, under Sorting/Grouping, click Edit for Value.

27 In the Schedule Properties dialog, click the Sorting/Grouping tab:

- Under Sort by, select Structural Usage.
- Select Header.
- Select Footer.
- Select Blank Line.
- Click OK.



28 In the Instance Properties dialog, click OK.

The schedule updates and now provides both a header and footer for each type, sorted by structural usage.

Concrete Beam Schedule

	SIZE					
MARK	W	D	REBAR	REMARKS	Reference Level	Structural Usage
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
	24"	36"			02 - Floor	Girder
Girder: 90						
Joist						
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
	12"	24"			02 - Floor	Joist
Joist: 16						
Other						
	24"	36"			02 - Floor	Other
	24"	36"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
	16"	32"			02 - Floor	Other
Other: 8						

29 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Customizing the Type Schedule

In this exercise, you first add mark information to identify each beam type, you then sort the schedule by mark, hide specific columns, and finally, you modify the existing rebar information.

Concrete Beam Schedule						
	SIZE		REBAR			Structural
MARK	W	D	Bottom Bars	Top Bars	REMARKS	Usage
B1	24"	36"	2-#7A 1-#7B	2-#5C		
B2	16"	32"	2-#6A 1-#6B	2-#5C		Girder
B3	12"	24"	2-#5A 1-#5B	2-#5C		Joist

Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_SCH_04_Customize_Schedule_i.rvt.

Add mark data for different beam types

1. In the Project Browser, under Structural Plans, double-click 02 - Floor.
2. Right-click any beam labeled C24, and click Select All Instances.
3. Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.
4. In the Instance Properties dialog:
   - Under Identity Data, type **B1** for Mark Value.
   - Click OK.

   NOTE Click OK to close the Revit warning regarding elements having duplicate Mark values.

5. In the Project Browser, under Schedules/Quantities, double-click Concrete Beam Schedule.

   The schedule updates and includes new mark information.

Concrete Beam Schedule						
	SIZE					Structural
MARK	W	D	REBAR	REMARKS	Reference Level	Usage
B1	24"	36"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
B1	24"	36"			02 - Floor	Girder
B1	24"	36"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
	16"	32"			02 - Floor	Girder
B1	24"	36"			02 - Floor	Girder

6. In the Project Browser, under Structural Plans, double-click 02 - Floor.
7. Right-click any beam labeled C16, and click Select All Instances.
8. Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.
9. In the Instance Properties dialog:
   - Under Identity Data, type **B2** for Mark Value.
   - Click OK.

NOTE Click OK to close the Revit warning regarding elements having duplicate Mark values.

10 Right-click any beam labeled C12, and click Select All Instances.

11 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

12 In the Instance Properties dialog:

- Under Identity Data, type **B3** for Mark Value.
- Click OK.

NOTE Click OK to close the Revit warning regarding elements having duplicate mark values.

13 In the Project Browser, under Schedules/Quantities, double-click Concrete Beam Schedule.

The schedule updates and now includes the new mark information.

Concrete Beam Schedule

MARK	SIZE		REBAR	REMARKS	Reference Level	Structural Usage
	W	D				
B2	16"	32"			02 - Floor	Girder
B2	16"	32"			02 - Floor	Girder
B2	16"	32"			02 - Floor	Girder
B1	24"	36"			02 - Floor	Girder
B1	24"	36"			02 - Floor	Girder
B1	24"	36"			02 - Floor	Girder
B1	24"	36"			02 - Floor	Girder
B3	12"	24"			02 - Floor	Joist
B3	12"	24"			02 - Floor	Joist
B3	12"	24"			02 - Floor	Joist
B3	12"	24"			02 - Floor	Joist
B3	12"	24"			02 - Floor	Joist
B3	12"	24"			02 - Floor	Joist
B3	12"	24"			02 - Floor	Joist
B3	12"	24"			02 - Floor	Joist
B3	12"	24"			02 - Floor	Joist

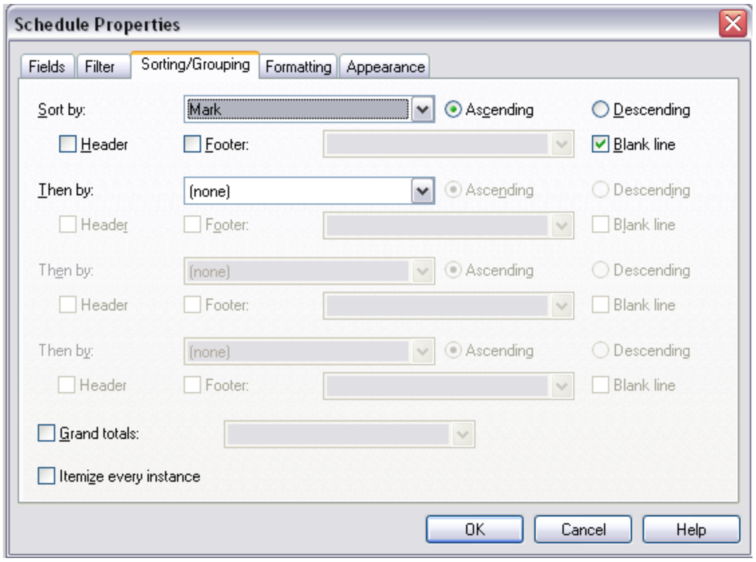
Sort and filter the schedule by mark value

14 Right-click the open area next to the schedule, and select View Properties.

15 In the Element Properties dialog, under Sorting/Grouping, click Edit for Value.

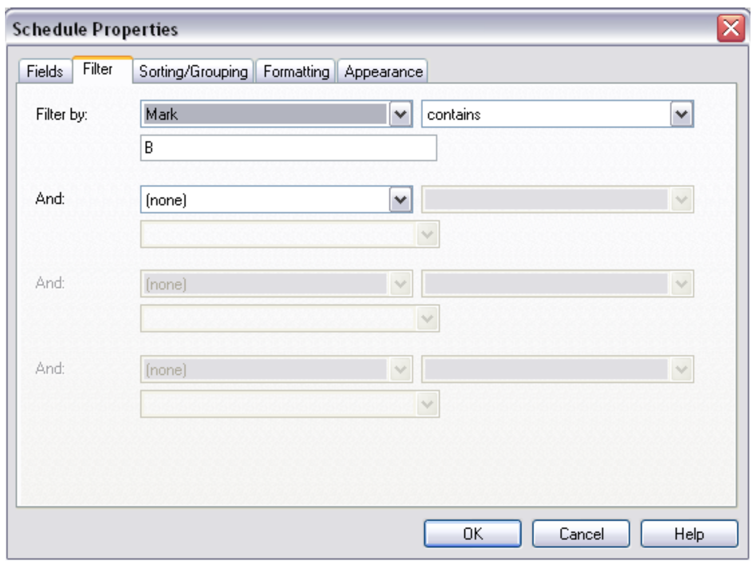
16 In the Schedule Properties dialog:

- Clear Footer, Header, Itemize every instance, and Grand totals.
- Under Sort by, select Mark.



17 In the Schedules Properties dialog, click the Filter tab:

- Under Filter by, select Mark, then select contains, and type **B**.



Hide/unhide columns

18 In the Schedule Properties dialog:

- Click the Formatting tab.
- Under Fields, select Reference Level.
- Under Field Formatting, select Hidden field.
- Click OK.

19 In the Instance Properties dialog, click OK.

The schedule is no longer itemized and does not show each beam. It now groups all the beams of the same mark in a single row. Also, the Reference Level column is now hidden.

Concrete Beam Schedule					
	SIZE				Structural
MARK	W	D	REBAR	REMARKS	Usage
B1	24"	36"			
B2	16"	32"			
B3	12"	24"			Joist

NOTE To show all hidden columns, right-click the open area next to the schedule, and select Unhide All Columns. You can also hide a column by right-clicking the desired column, and selecting Hide Column(s) (from the drop-down menu).

Rename existing rebar parameter

20 Right-click the open area next to the schedule, and click View Properties.

21 In the Instance Properties dialog, under Formatting, click Edit for Value.

22 In the Schedule Properties dialog:

- Under Fields, select REBAR.
- Under Heading, type **Bottom Bars**.
- Click OK.

23 In the Instance Properties dialog, click OK.

The Rebar field is renamed.

Concrete Beam Schedule					
	SIZE		Bottom		Structural
MARK	W	D	Bars	REMARKS	Usage
B1	24"	36"			
B2	16"	32"			Girder
B3	12"	24"			Joist

Add new rebar parameter

24 Right-click the open area next to the schedule, and click View Properties.

25 In the Instance Properties dialog, under Fields, click Edit for Value.

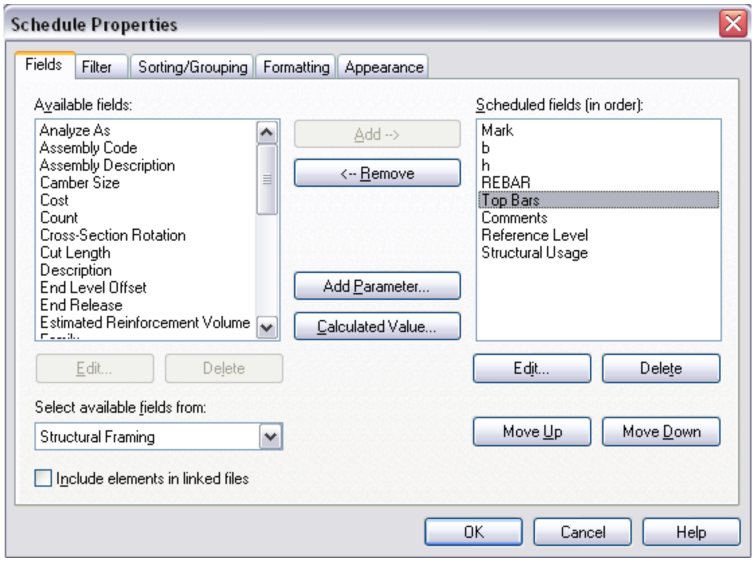
26 In the Schedule Properties dialog, click Add Parameter.

27 In the Parameter Properties dialog:

- Under Parameter Data, type **Top Bars** for Name.
- Under Type of Parameter, select Text.
- Click OK.

This parameter provides text information that is added to all beams within the model. Rebar will not be added to the model with the data entered in this dialog.

28 Under Scheduled fields, order the fields, as shown, by selecting them and clicking Move Up or Move Down.



29 In the Schedule Properties dialog, click OK.

30 In the Instance Properties dialog, click OK.

The new rebar field is added.

Concrete Beam Schedule						
	SIZE					Structural
MARK	W	D	Bottom Bars	Top Bars	REMARKS	Usage
B1	24"	36"				
B2	16"	32"				Girder
B3	12"	24"				Joist

Group rebar columns

31 Select both the Bottom Bars and Top Bars headings.

32 Click Schedule panel ➤ Headers: Group.

A new blank cell is created above the columns.

33 Click the new cell, and type **REBAR**.

Concrete Beam Schedule						
	SIZE		REBAR			Structural
MARK	W	D	Bottom Bars	Top Bars	REMARKS	Usage
B1	24"	36"				
B2	16"	32"				Girder
B3	12"	24"				Joist

Enter rebar data

34 Enter the following rebar set information for each rebar instance in the schedule:

- For Mark B1, under Bottom Bars, type **2-#7A 1-#7B**, and under Top Bars, type **2-#5 C**.
- For Mark B2, under Bottom Bars, type **2-#6A 1-#6B**, and under Top Bars, select 2-#5C.
- For Mark B3, under Bottom Bars, type **2-#5A 1-#5B**, and under Top Bars, select 2-#5C.

The schedule now includes the joist information.

Concrete Beam Schedule						
	SIZE		REBAR			Structural
MARK	W	D	Bottom Bars	Top Bars	REMARKS	Usage
B1	24"	36"	2-#7A 1-#7B	2-#5C		
B2	16"	32"	2-#6A 1-#6B	2-#5C		Girder
B3	12"	24"	2-#5A 1-#5B	2-#5C		Joist

35 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Creating an Instance Schedule

In this exercise, you create a formula to calculate the unit cost for each beam instance, as well as the total cost of all structural items for level 2 of the structure.

Concrete Beam Schedule									
	SIZE		REBAR			Structural			Total Cost
MARK	W	D	Bottom Bars	Top Bars	REMARKS	Usage	Volume	Cost	
B1	24"	36"	2-#7A 1-#7B	2-#5C		Girder	3931.15 CF	3.00	$11793.45
B2	16"	32"	2-#6A 1-#6B	2-#5C		Girder	10372.74 CF	1.50	$15559.12
B3	12"	24"	2-#5A 1-#5B	2-#5C		Joist	1080.42 CF	4.00	$4321.67

Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_SCH_05_Instance_Schedule_i.rvt.

Create the formula

1 In the Project Browser, under Schedules/Quantities, double-click Concrete Beam Schedule.

2 Right-click the open area next to the schedule, and click View Properties.

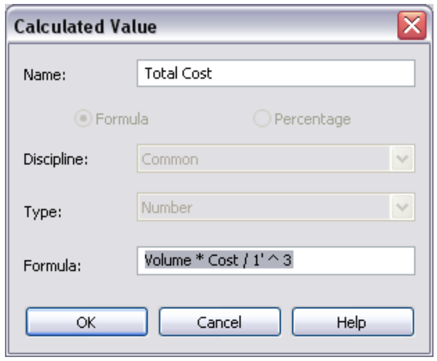
3 In the Instance Properties dialog, under the Fields parameter, click Edit for Value.

4 In the Schedule Properties dialog:

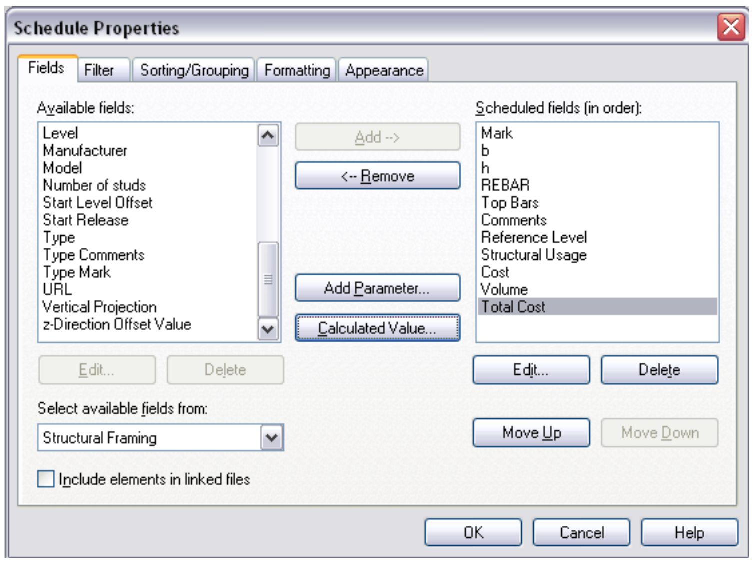
- Under Available Fields, select Volume, and click Add.
- Under Available Fields, select Cost, and click Add.
- Click Calculated Value.

5 In the Calculated Value dialog:

- For Name, type **Total Cost**.
- Select Number for Type.
- For Formula, type **Volume*Cost/1'^3**.
  Note that the formula will calculate the total cost, based on a volume unit of 1 cubic yard.



■ Click OK.



Calculate the beam volume

6 In the Schedule Properties dialog:

■ Click the Formatting tab.

■ Under Fields, select Volume.

■ Under Field Formatting, select Calculate totals.

■ Click OK.

7 In the Instance Properties dialog, click OK.

Concrete Beam Schedule									
	SIZE		REBAR						
MARK	W	D	Bottom Bars	Top Bars	REMARKS	Structural Usage	Volume	Cost	Total Cost
B1	24"	36"	2-#7A 1-#7B	2-#5C		Girder	3931.15 CF		0
B2	16"	32"	2-#6A 1-#6B	2-#5C		Girder	10372.74 CF		0
B3	12"	24"	2-#5A 1-#5B	2-#5C		Joist	1080.42 CF		0

Calculate total cost

8 Right-click the open area next to the schedule, and click View Properties.

9 In the Instance Properties dialog, under Formatting, click Edit for Value.

10 In the Schedule Properties dialog:

- Click the Formatting tab.
- Under Fields, select Total Cost.
- Under Field Formatting, select Calculate totals.
- Click OK.

11 In the Instance Properties dialog, click OK.

The schedule now includes the sum for Total Cost.

Enter a cost value

12 Under Mark B1, type **3** for Cost, and press *Enter*.

Because cost is a type parameter, the value will be applied to all elements of the same type. Then, the schedule calculates the total cost for all beams of the same type.

Concrete Beam Schedule									
	SIZE		REBAR			Structural			
MARK	W	D	Bottom Bars	Top Bars	REMARKS	Usage	Volume	Cost	Total Cost
B1	24"	36"	2-#7A 1-#7B	2-#5C		Girder	3931.15 CF	3.00	11793.45353
B2	16"	32"	2-#6A 1-#6B	2-#5C		Girder	10372.74 CF		0
B3	12"	24"	2-#5A 1-#5B	2-#5C		Joist	1080.42 CF		0

NOTE The cost value represents a random value chosen for demonstration purposes only. Also notice that the Total Cost parameter does not have a unit value assigned.

13 Enter the following values in the schedule:

- Under Mark B2, type **1.5** for Cost.
- Under Mark B3, type **4.0** for Cost.

14 Press *Enter*.

Concrete Beam Schedule									
	SIZE		REBAR			Structural			
MARK	W	D	Bottom Bars	Top Bars	REMARKS	Usage	Volume	Cost	Total Cost
B1	24"	36"	2-#7A 1-#7B	2-#5C		Girder	3931.15 CF	3.00	11793.45353
B2	16"	32"	2-#6A 1-#6B	2-#5C		Girder	10372.74 CF	1.50	15559.11515
B3	12"	24"	2-#5A 1-#5B	2-#5C		Joist	1080.42 CF	4.00	4321.666667

Format total cost to include currency value

15 Right-click the open area next to the schedule, and click View Properties.

16 In the Instance Properties dialog, under Formatting, click Edit for Value.

17 In the Schedule Properties dialog:

- Click the Formatting tab.
- Under Fields, select Total Cost.
- Under Field formatting, click Field Format.
- In the Format dialog, clear Use default settings.
- Under Units, select Currency.
- Under Rounding, select 2 decimal places.
- Under Unit Symbol, select $.

- Click OK.

18 In the Schedule Properties dialog, click OK.

19 In the Instance Properties dialog, click OK.

The Total Cost column displays a currency value in dollars.

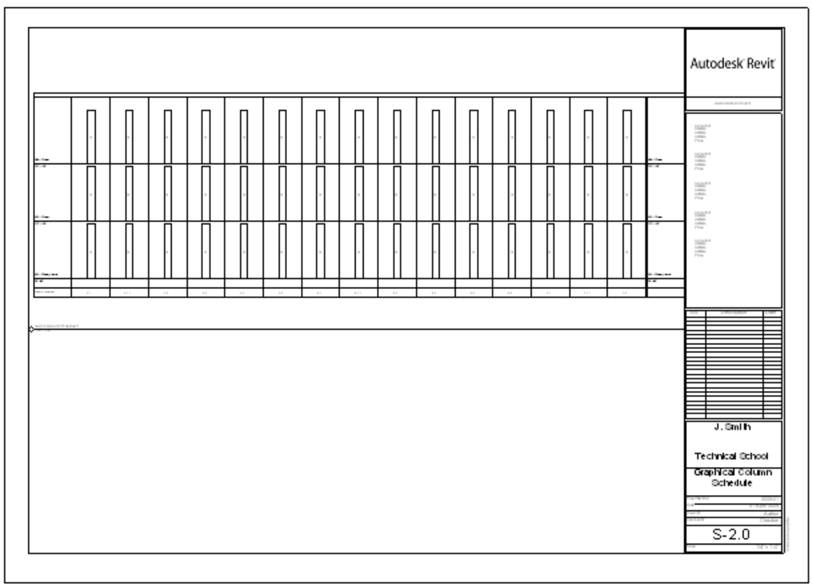
Concrete Beam Schedule									
	SIZE		REBAR						
MARK	W	D	Bottom Bars	Top Bars	REMARKS	Structural Usage	Volume	Cost	Total Cost
B1	24"	36"	2-#7A 1-#7B	2-#5C		Girder	3931.15 CF	3.00	$11793.45
B2	16"	32"	2-#6A 1-#6B	2-#5C		Girder	10372.74 CF	1.50	$15559.12
B3	12"	24"	2-#5A 1-#5B	2-#5C		Joist	1080.42 CF	4.00	$4321.67

20 Close the file with or without saving it

In the next exercise, a new training file is supplied.

Create the Graphical Column Schedule

In this exercise, you learn how to create a graphical column schedule.



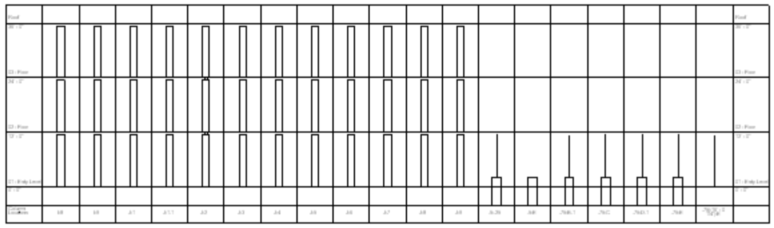
Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_SCH_06_Graphical_Column_i.rvt.

Create the schedule

1 Click View tab ➤ Create panel ➤ Schedules drop-down ➤ Graphical Column Schedule.

The schedule is created automatically.



2 Right-click the column schedule, and select View Properties.

3 In the Instance Properties dialog, under Other, click Edit for Hidden Levels.

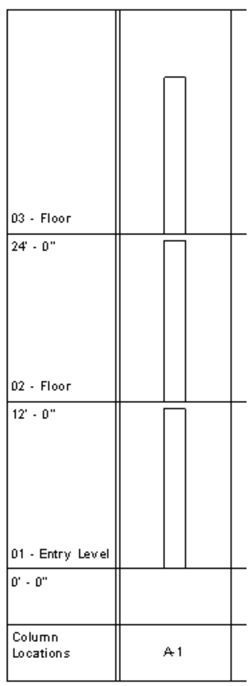
4 In the Levels Hidden dialog, select Roof, and click OK.

NOTE The roof level will not appear on the graphical column schedule.

5 In the Instance Properties dialog, click OK.

6 Zoom in to the schedule.

Notice that the top level of the schedule is 03 - Floor.



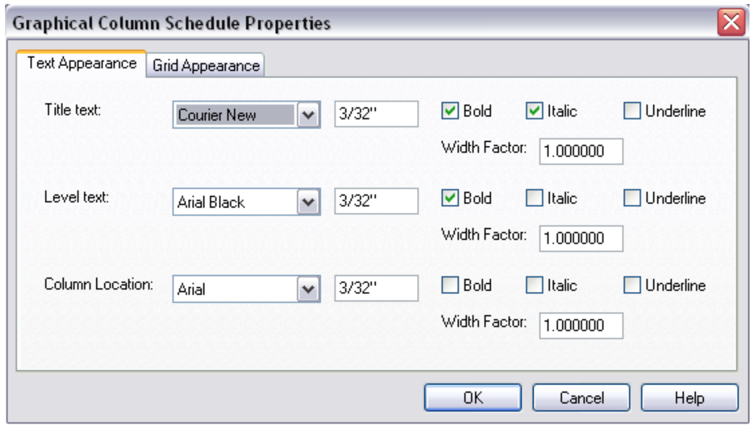
Customize the schedule appearance

7 Right-click the schedule, and select View Properties.

8 In the Instance Properties dialog, under Text Appearance, click Edit for Value.

9 In the Graphical Column Schedule Properties dialog:

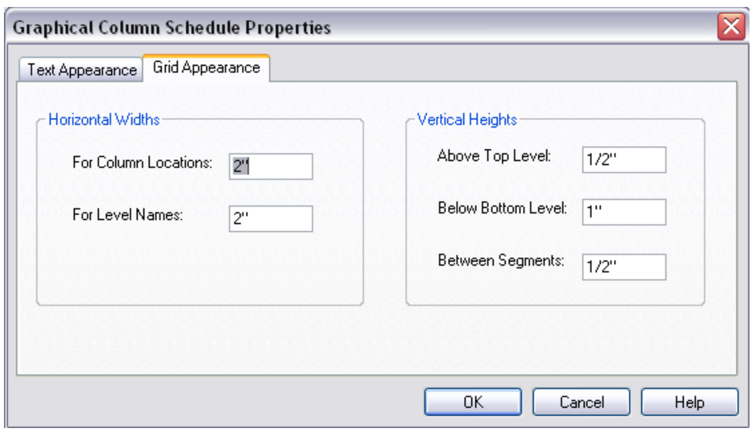
- For Title text, select Courier New, and then select Bold and Italic.
- For Level text, select Arial Black, and then select Bold.



10 Click the Grid Appearance tab.

11 Under Horizontal Widths:

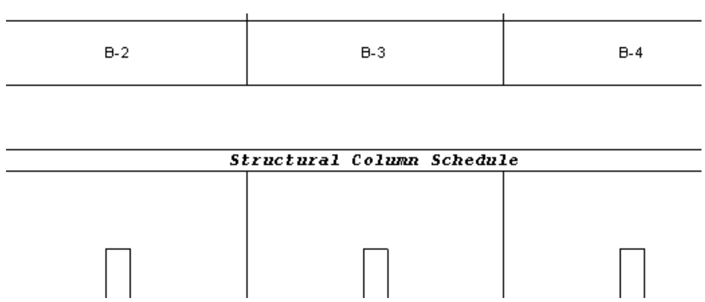
- Type **2**" for Column Locations,
- Type **2**" for Level Names.



12 Click OK.

13 In the Instance Properties dialog, under Identity Data, for Title, type **Structural Column Schedule**.

14 Click OK.

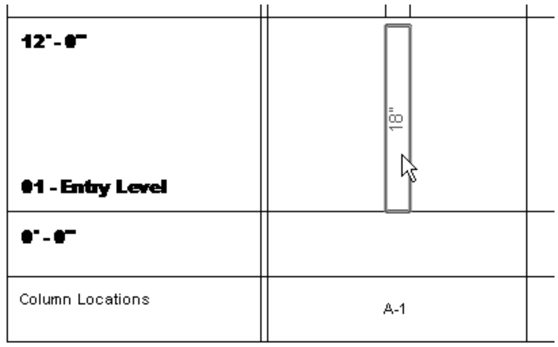


Tag the columns

15 Click Annotate tab ➤ Tag panel ➤ Tag drop-down ➤ By Category.

16 On the Options bar, click Vertical, and clear Leader.

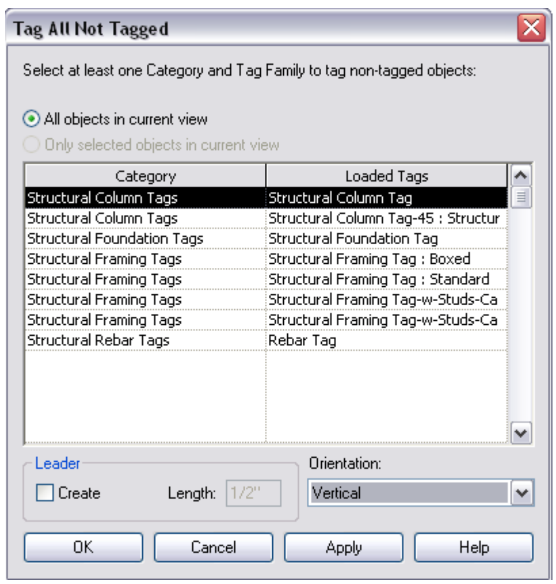
17 Click any column on the entry level to place the tag.



18 Click Tag panel ➤ Tag All.

19 In the Tag All Not Tagged dialog:

- Under Category, select Structural Column Tags.
- Under Orientation, select Vertical.
- Click Apply.
- Click OK.



Split the column schedule

20 Right-click the schedule, and select View Properties.

21 In the Instance Properties dialog:

- Under View Scale, select 1/4" = 1' - 0".
- Under Column Locations per Segment, type **15**.
- Click OK.

The schedule is split into multiple segments with 15 column locations per segment.

Click OK in the Revit dialog indicating some of the columns exceed the segment's upper/lower bounds.

22 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Creating Multiple Sheets for the Graphical Column Schedule

In this exercise, you create multiple sheets for the graphical column schedule.

Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_SCH_07_GCS_Sheets_i.rvt.

Create a sheet

1 Click View tab ➤ Sheet Composition panel ➤ New Sheet.

2 In the Select a Titleblock dialog, select E1 30 x 42 Horizontal: E1 30x42 Horizontal, and click OK.

A titleblock and drawing borders display on the drawing sheet.

3 In the Project Browser, expand Sheets (all).

The new sheet displays in the Project Browser.

Change the sheet name and number

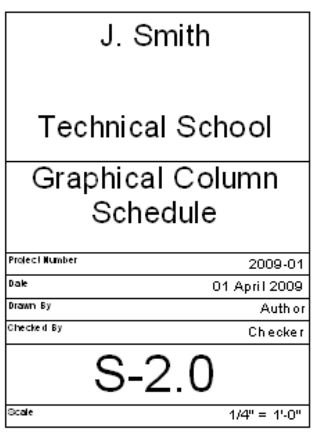
4 Select the titleblock.

5 Click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

6 In the Instance Properties dialog:

- Under Identity Data ➤ Sheet Name, type **Graphical Column Schedule**.
- Under Sheet Number, type **S-2.0**.
- Click OK.

The sheet name and number display in the titleblock and in the Project Browser.



Add the column schedule to the sheet

7 Click Sheet Composition panel ➤ View.

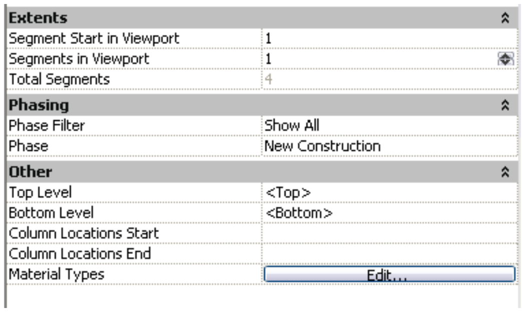
8 In the Views dialog, select Graphical Column Schedule, and click Add View to Sheet.

9 Move the cursor to the center of the sheet, and click to place the view.

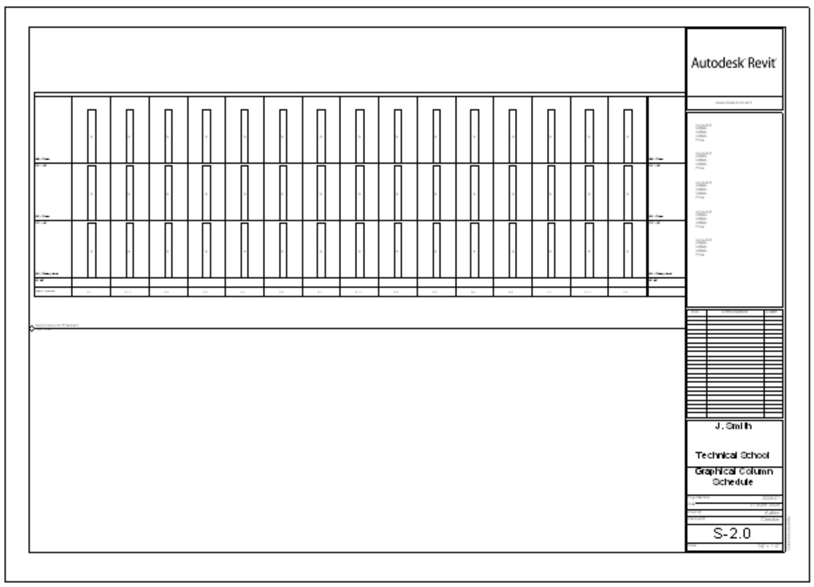
Close the Revit Warning dialog.

10 Select the schedule, and click Element panel ➤ Element Properties dialog ➤ Instance Properties.

11 In the Instance Properties dialog, under Extents, type **1** for Segments in Viewport, and click OK.



Place the first segment on sheet S-2.0.



Add additional sheets

12 Click View tab ➤ Sheet Composition panel ➤ New Sheet.

13 In the Select a Titleblock dialog, select E1 30 x 42 Horizontal: E1 30x42 Horizontal, and click OK.

14 Select the title block, and click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

15 In the Instance Properties dialog:

- Under Identity Data ➤ Sheet Name, select Graphical Column Schedule.
- Under Sheet Number, type **S-2.1**.
- Click OK.

16 Using the same method, add 3 additional sheets; select Graphical Column Schedule for name, and type **S-2.2**, **S-2.3**, and **S-2.4** for sheet numbers.

Place remaining views

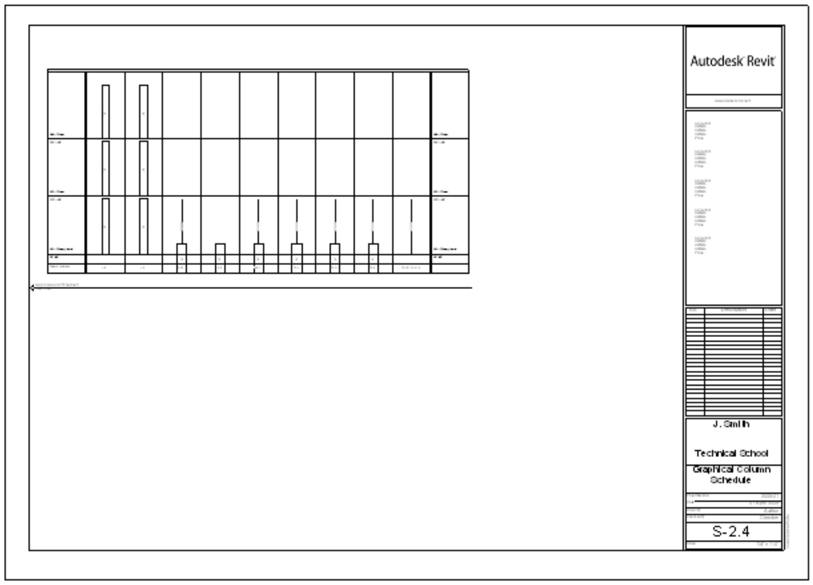
17 On the Project Browser, expand Sheets (all), and double-click Sheet S-2.1.

18 On the Project Browser, under Graphical Column Schedule, click Graphical Column Schedule 1, and drag it to the sheet.

19 Move the cursor to the center of the sheet, and click to place the view.

20 Position the view, as necessary.

21 Place the remaining schedule views on the remaining sheets.



22 Close the file with or without saving it

In the next exercise, a new training file is supplied.

Detailing

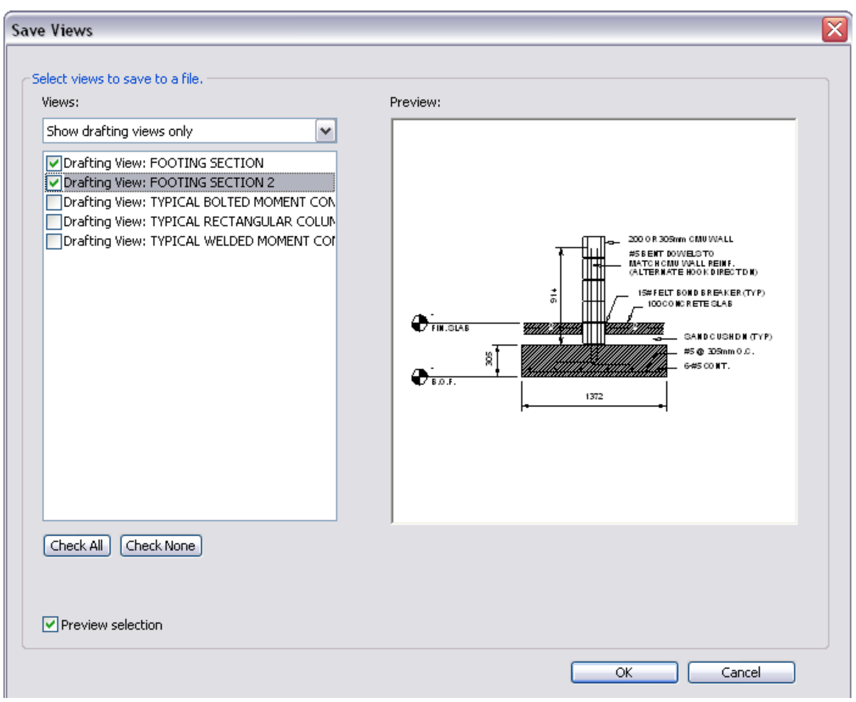
25

In Revit Structure, details are either based upon the geometry of the structure as a detail view, or referenced as a drafting view, with parametric tags that automatically track and display detail view and drawing sheet placement. In this lesson, you learn to:

- Create a drafting view detail library by importing both detail views and sheets from an existing project.
- Import detail views from the detail library into your project.

Creating a Drafting View Detail Library

In this exercise, you begin by opening a project that contains typical drafting view details. Instead of having to draw these details for every project, you will learn how to save both the individual views and complete detail sheets as a new detail library.



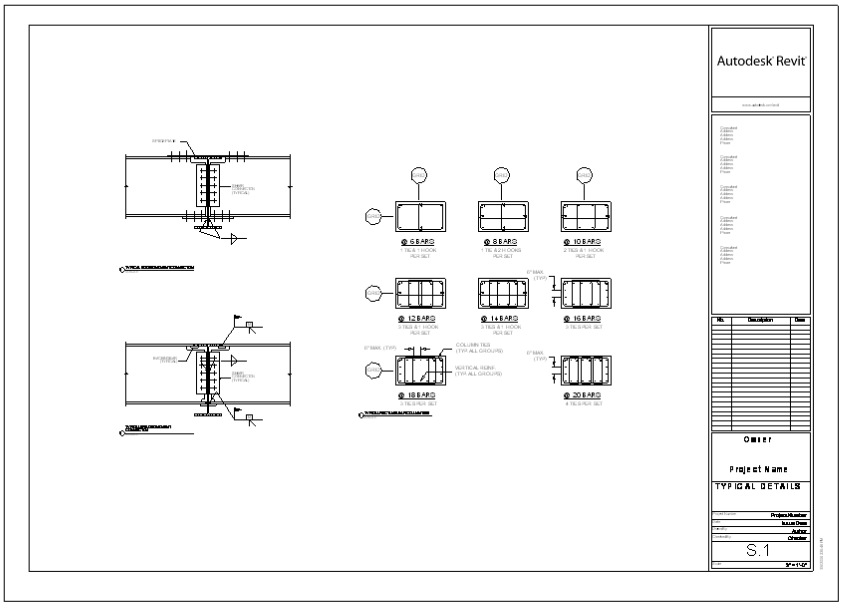
Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_DET_01_Typical_Drafting_Views_i.rvt.

View the typical details sheet in the project

1 In the Project Browser, expand Sheets (all), and double-click S.1 - TYPICAL DETAILS.

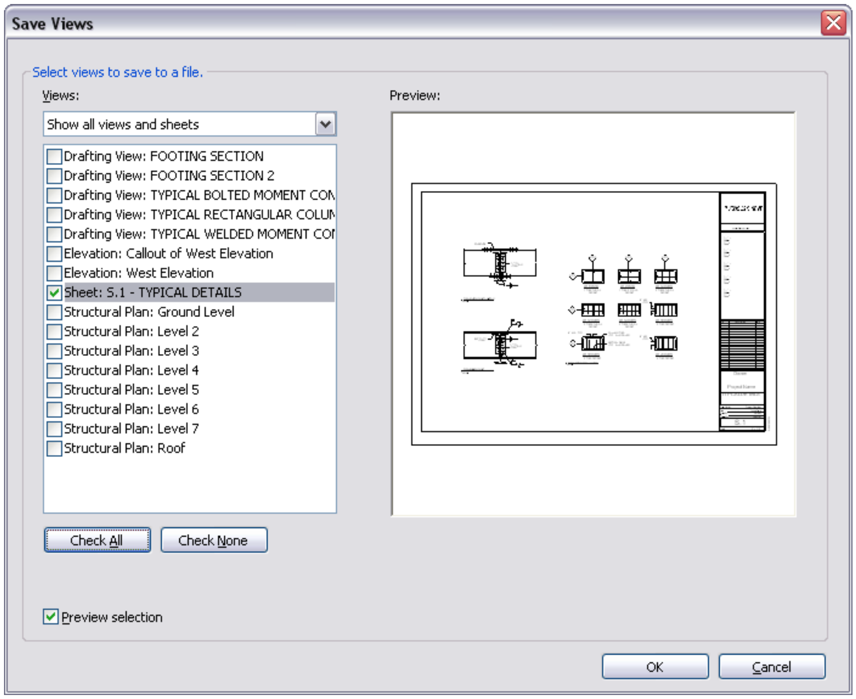
Notice there are 3 typical detail views on this sheet.



Save multiple views to the detail library

2 Click ➤ Save As ➤ Library ➤ View.

3 In the Save Views dialog, select Sheet: S.1 - TYPICAL DETAILS, and click OK.



4 In the Save As dialog:

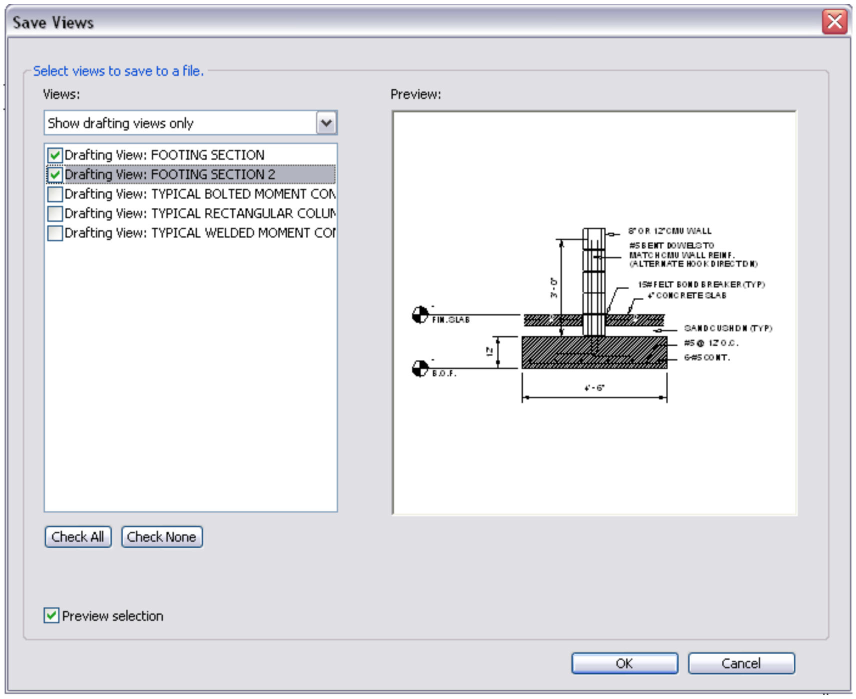
- Locate a common drive that can be accessed by all team members.
- Click ➤ (Create new folder).
- For folder name, type **Detail Library**.
- Open the new folder.
- Under File name, type **TYPICAL COLUMN BEAM AND GIRDER DETAILS**.
- Click Save.

Save individual views to the library

5 Click ➤ Save As ➤ Library ➤ View.

6 In the Save Views dialog:

- In the list of views, clear Drawing Sheet: S.1 - TYPICAL DETAILS.
- Under Views, select Show drafting views only.
  Only the drafting views are displayed.
- Select both Footing Section views.
- Click OK.



7 In the Save As dialog:

- Under Save in, navigate to the Detail Library folder previously created in this exercise.
- For File name, type **FOOTING SECTION**.
- Click Save.

NOTE When you save your file, Revit Structure will save views that contain 3D objects; however, when you insert these views into a new project, only the 2D elements will be inserted.

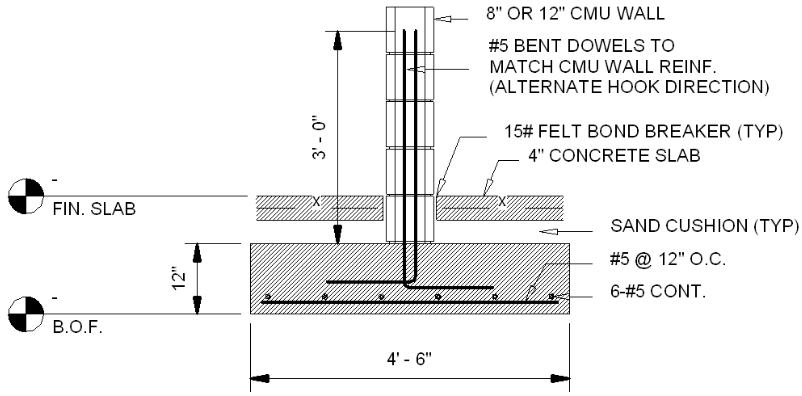
---

8 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Importing Details from the Library

In this exercise, you learn how to import details from the newly created detail library.



Training File

- Click  ➤ Open ➤ Project.

- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_DET_02_Import_Details_i.rvt.

Insert drawing sheet from the detail library

1 Click Insert tab ➤ Import panel ➤ Insert from File drop-down ➤ Insert Views from File.

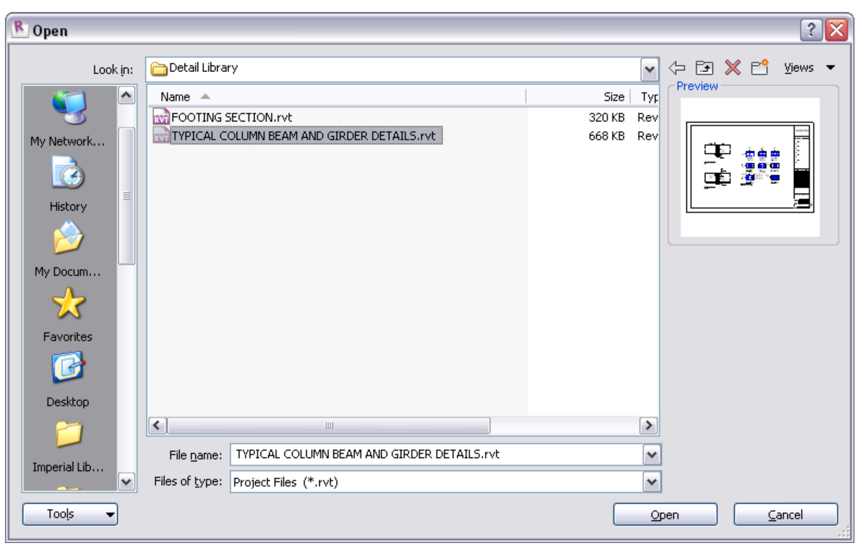
2 In the Open dialog:

- For Look in, click Training Files\Imperial\Detail Library.

> **NOTE** For training purposes , the detail views that are required for this exercise have already been placed in this folder. You need to save typical detail views in a location that is accessible by all members of your team.

- Select the file TYPICAL COLUMN BEAM AND GIRDER DETAILS.rvt.

- Click Open.

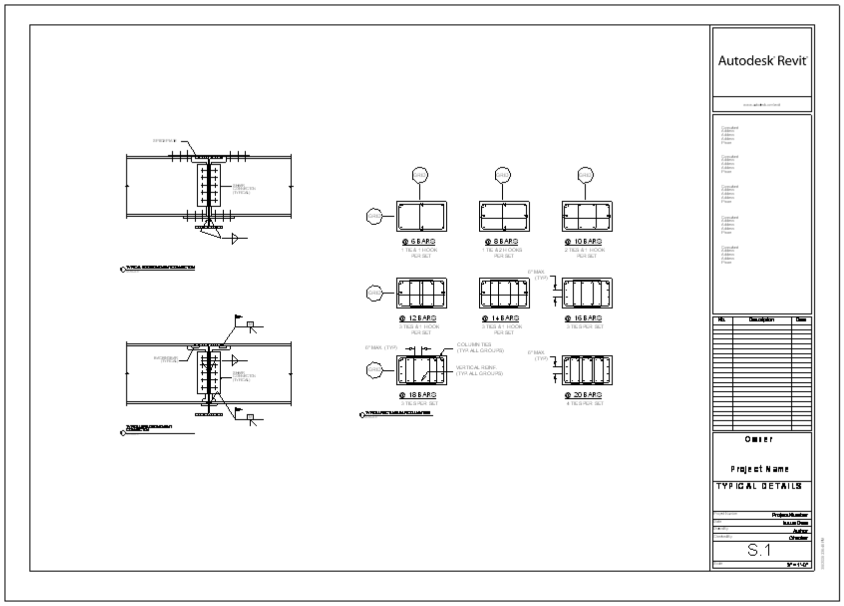


3 In the Insert Views dialog:

- Select Sheet: S.1 - TYPICAL DETAILS.
- Click OK.

A duplicate types dialog displays because you are importing element types that already exist in the project. Click OK to close the Revit dialog, and also close the associated Revit Warning.

4 In the Project Browser, expand Sheets (all), and double-click S.1 - TYPICAL DETAILS.



Rename the sheet

5 Select the title block.

6 When the title block highlights, click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

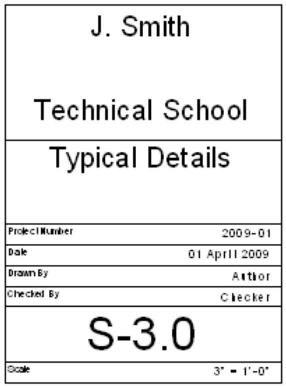
7 In the Instance Properties dialog:

- Under Identity Data, for Sheet name, type **Typical Details**.
- Under Identity Data, for Sheet number, type **S-3.0**.

8 Click OK.

9 Press *Esc*.

The sheet name and number display in the title block and in the Project Browser.

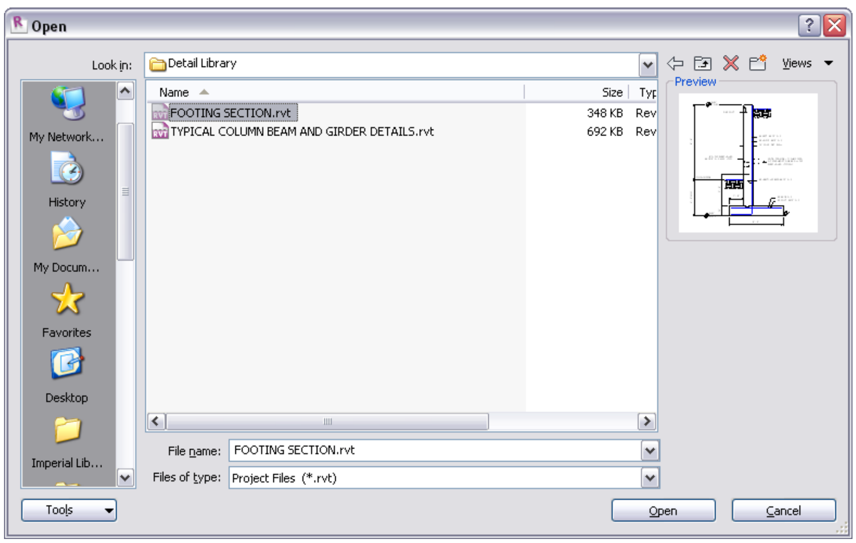


Insert separate footing sections from the library

10 Click Insert tab ➤ Import panel ➤ Insert from File drop-down ➤ Insert Views from File.

11 In the Open dialog:

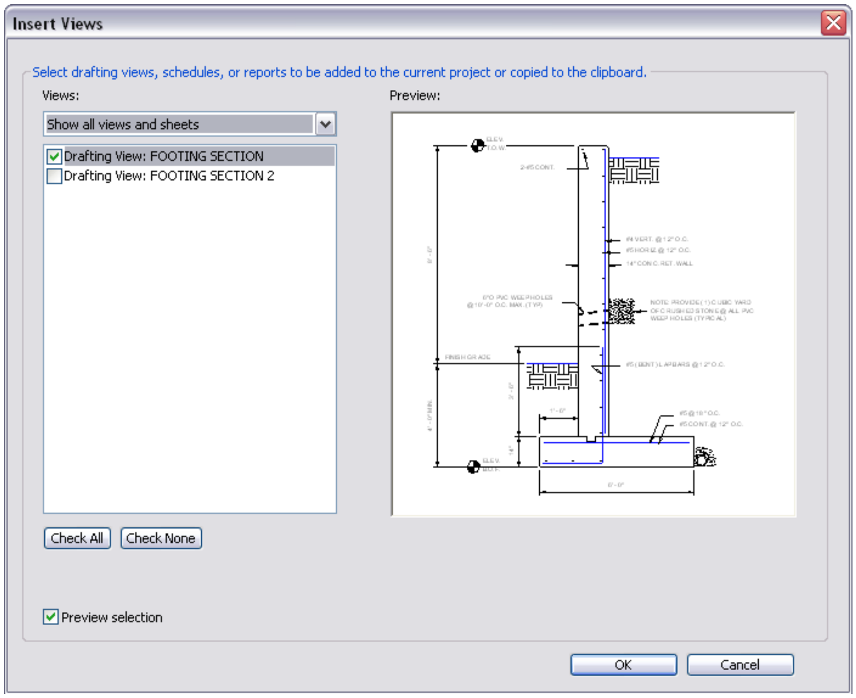
- For Look in, click Training Files\Imperial\Detail Library.
- Select the file FOOTING SECTION.rvt.
- Click Open.



12 In the Insert Views dialog:

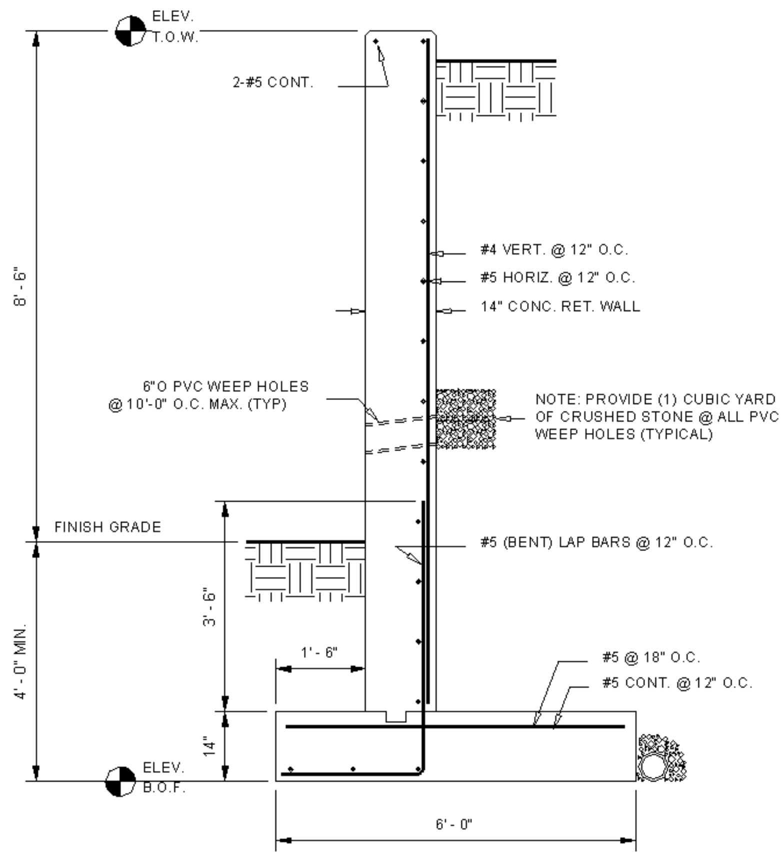
- Under Views, select Drafting View: FOOTING SECTION.

■ Click OK.



A duplicate types dialog displays because you are importing element types that already exist in the project. Click OK to close the Revit dialog, and also close the associated Revit Warning.

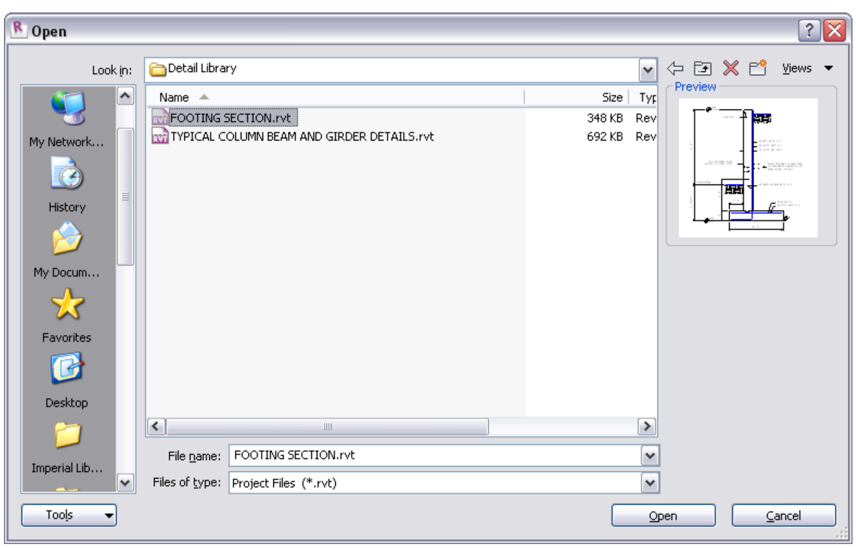
The following view displays.



13 Click Import panel ➤ Insert from File drop-down ➤ Insert Views from File.

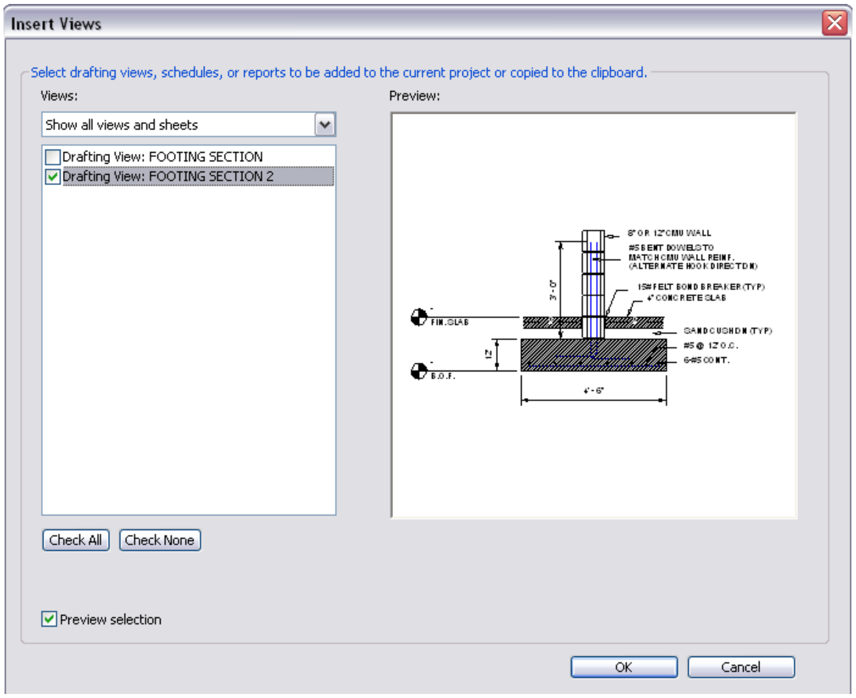
14 In the Open dialog:

- For Look in, click Training Files\Imperial\Detail Library.
- Select the file Footing Sections.rvt.
- Click Open.



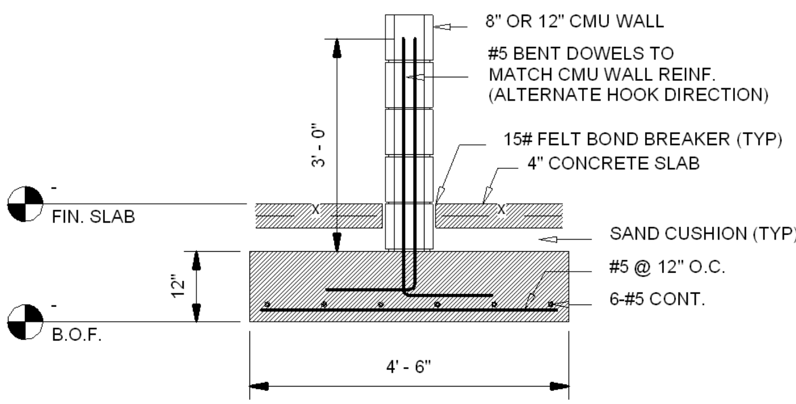
15 In the Insert Views dialog:

- Under Views, select Drafting View: Footing Section 2.
- Click OK.



A duplicate types dialog displays because you are importing element types that already exist in the project. Click OK to close the Revit dialog, and also close the associated Revit Warning.

The following view displays.



16 In the Project Browser, expand Drafting Views (Detail); the imported details are listed.

17 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

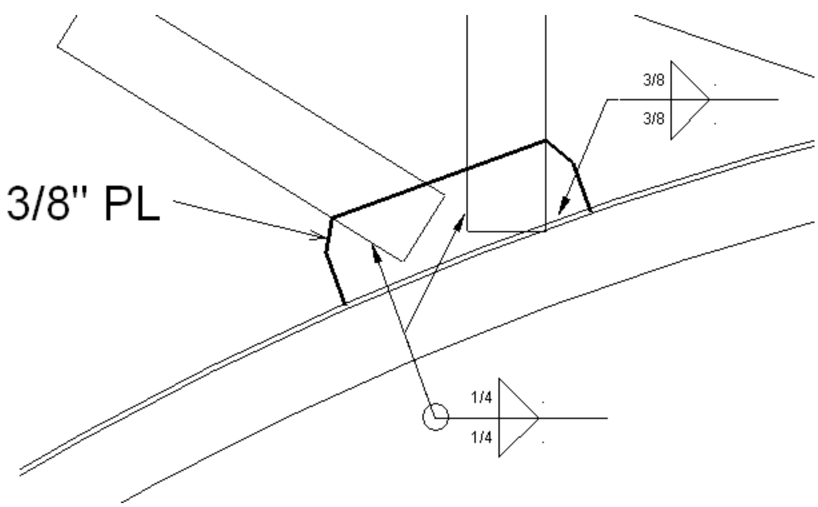
26
Creating Steel Details

In this lesson, you learn how to use Revit Structure 2010 to create model-based steel details from the structural model, and how to create a typical drafting detail, using the tools provided. Detail drawings describe how particular pieces of the structure go together. You typically create them in the middle to later portion of the design process after the general building shape and structural elements have been decided upon. You learn to:

- Create a welded brace detail.
- Create a bolted angle detail.
- Create a typical drafting view detail for a deck span transition.

Creating a Welded Brace Detail

In this exercise, you create a drafting detail for a welded brace plate to support the individual webs of the entry way truss.



Training File

- Click ➤ Open ➤ Project.

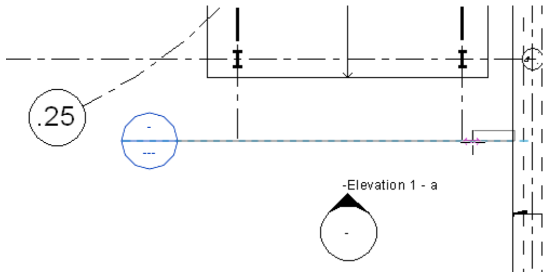
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_DET_STEEL_01_Brace_Detail_i.rvt.

Create a section view

1 In the Project Browser, under Structural plans, double-click 02 Floor.

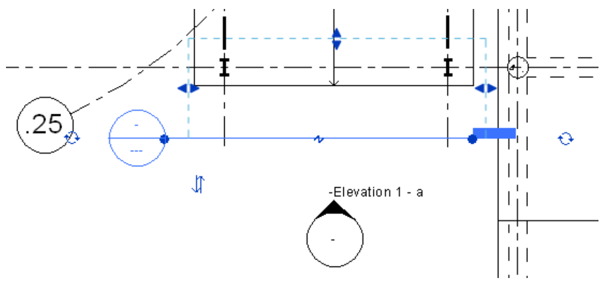
2 Zoom in to the entry way, and click View tab ➤ Create panel ➤ Section.

3 Click to the left of grid line .5, and move the cursor to the right of grid line .75 to place the section.



4 Click and drag the top control down until it is just above grid line E.

You adjust the depth of the section to include only what you want to show in the section.



5 Press *Esc*.

Set the section view properties and boundaries

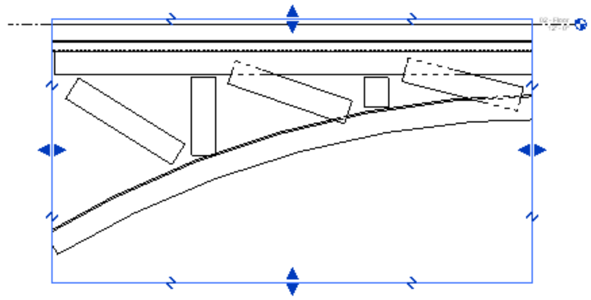
6 In the Project Browser, expand Detail Views (Detail), right-click Detail 0, and select Properties.

7 In the Instance Properties dialog:

- Under Graphics, for View scale, select 1 1/2" = 1' - 0".
- For Detail Level, select Fine.
- For Model Graphics Style, select Hidden Line.
- Under Identity Data, for View Name, type **Welded Brace Detail**.
- Click OK.

8 Open the Welded Brace Detail.

9 Adjust the crop region boundaries, as shown.



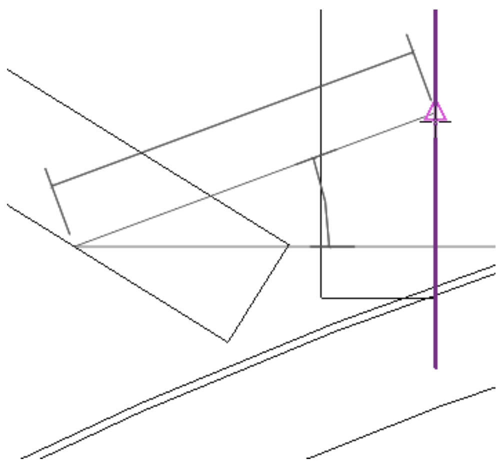
10 Press *Esc*.

Sketch the brace plate

11 Click Annotate tab ➤ Detail panel ➤ Detail Lines.

12 Click Draw panel ➤  (Line).

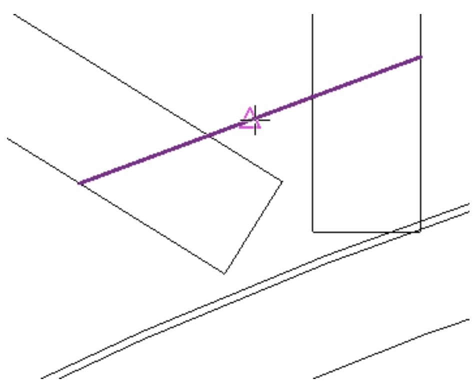
13 Place the cursor on the left diagonal web and move the cursor to the right to sketch the top line of the brace.



14 Press *Esc*.

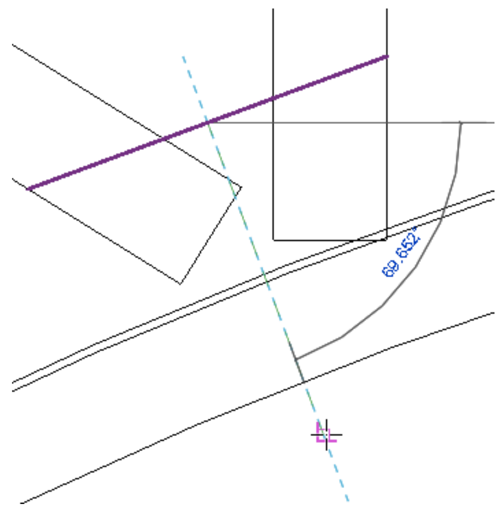
15 Click Home tab ➤ Work Plane panel ➤ Ref Plane drop-down ➤ Draw Reference Plane.

16 Move the cursor over the top line until a triangle displays.



The triangle indicates the center point of the detail line.

17 Sketch the reference plane, as shown.

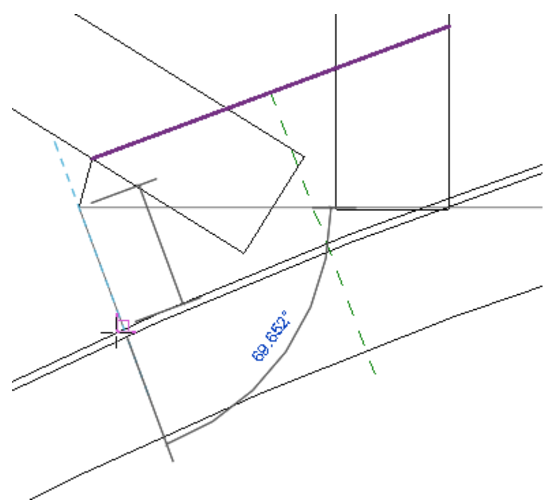


18 Press *Esc*.

19 Click Annotate tab ➤ Detail panel ➤ Detail Lines.

20 Click Draw panel ➤ (Line).

21 Sketch 2 detail lines that are offset from the top line, as shown.



22 Press *Esc*.

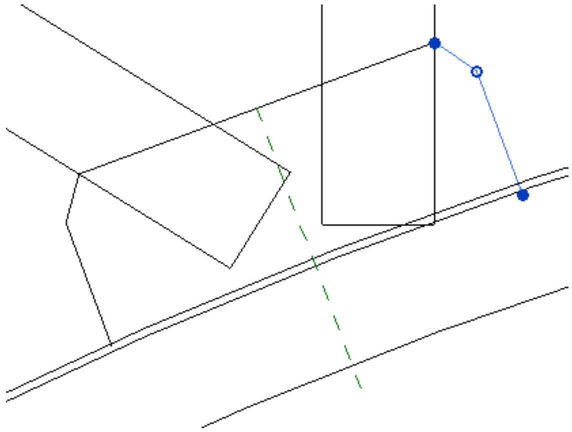
Mirror the detail lines

23 Select the 2 lines that were just created.

24 Click Modify panel ➤ Mirror.

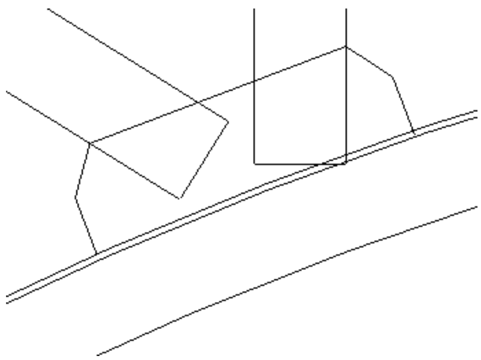
25 Click the reference plane as the axis of reflection.

The lines mirror each other, forming the right edge of the plate.



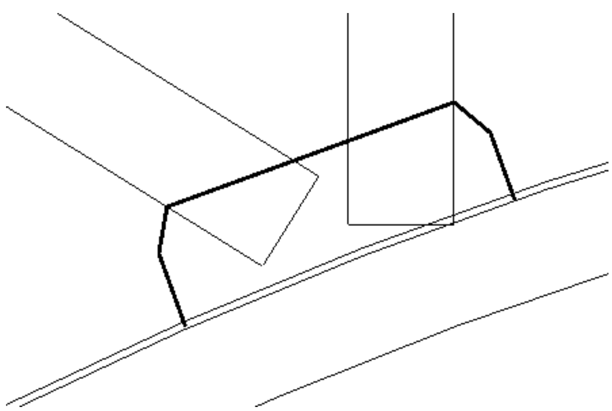
26 Press *Esc*.

27 Select the reference plane, and press *Delete*.



Change the line weight

28 Select the detail lines, and click Element panel ➤ Thin Lines drop-down ➤ Medium Lines.



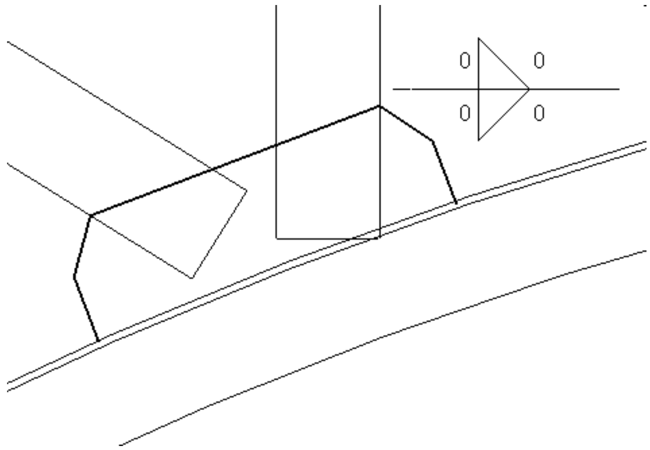
29 Press *Esc*.

Add weld symbols

30 Click Annotate tab ➤ Symbol panel ➤ (Symbol).

31 Click Element panel ➤ Change Element Type drop-down ➤ Weld Symbol ➤ Both.

32 Drag the symbol to the view, approximately as shown, and click to place it.



The weld symbol has 4 numerical parameters that are currently set to 0.

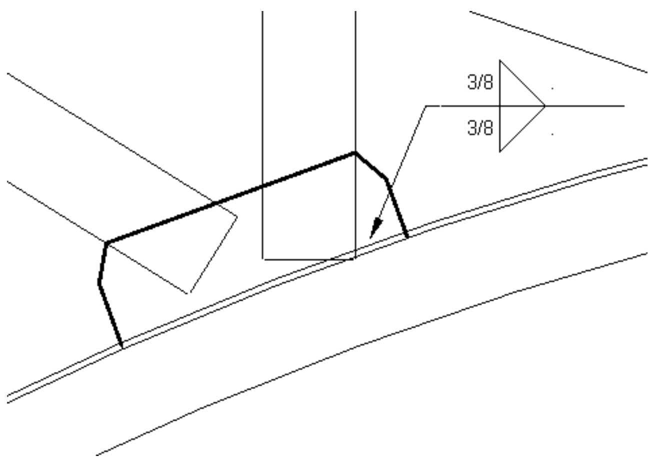
33 Click Selection panel ➤ Modify.

34 Select the weld symbol, click the left top weld annotation, and type **3/8**. Repeat for the left bottom value.

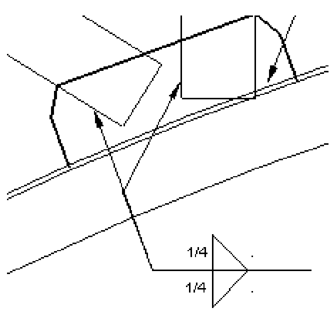
35 Click the right top weld annotation, and type a period (.). Repeat for the right bottom value.

36 Select the weld symbol, and click Leader panel ➤ Add.

37 Drag the leader arrowhead to the position as shown.



38 Using the same method, add a second weld symbol to the opposite side of the plate, as shown.

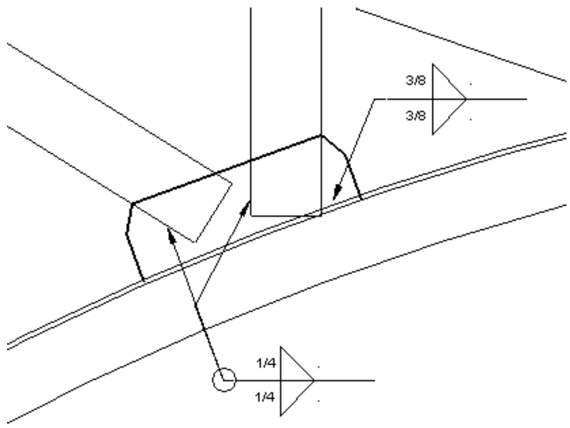


39 Press *Esc*.

40 Right-click the new weld symbol, and click Element Properties.

41 In the Instance Properties dialog:

- Under Structural, select Weld All Around.
- Click OK.

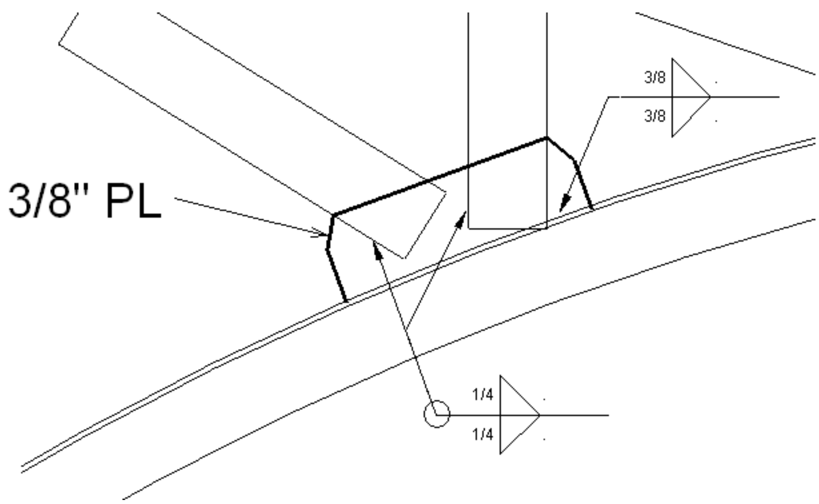


Add text to the plate

42 Click Annotate tab ➤ Text panel ➤ Text.

43 Click Leader panel ➤ One Segment.

44 Draw a text leader to the plate, and type **3/8" PL** for text.



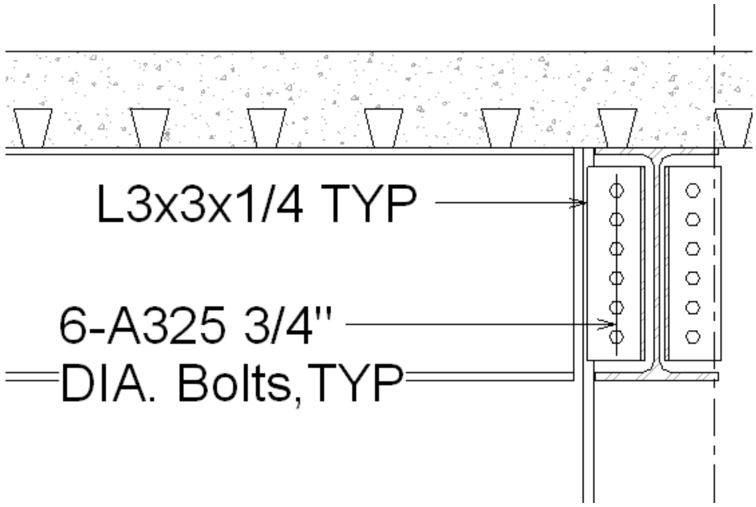
45 Press *Esc*.

46 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Creating a Bolted Angle Detail

In this exercise, you create a detail view for a typical bolted angle plate.



Training File

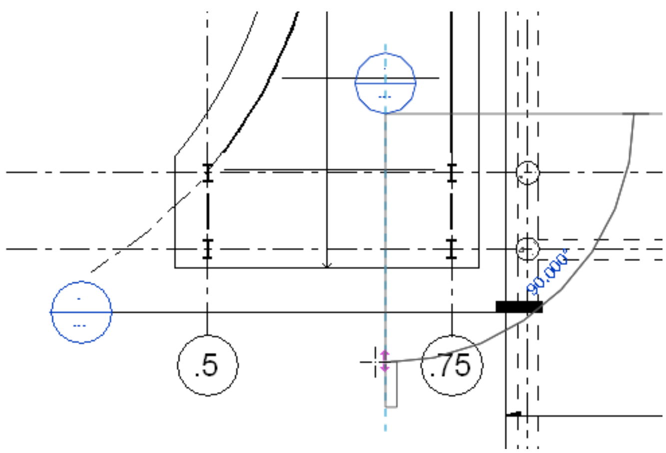
- Click  ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_DET_STEEL_02_Bolted_Angle_Detail_i.rvt.

Create a section view

1 In the Project Browser, under Structural plans, double-click 02 Floor.

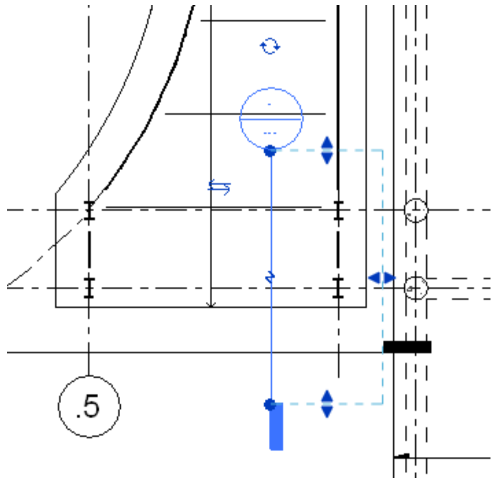
2 Zoom in to the entry way.

3 Click View tab ➤ Create panel ➤ Section, click to the left of grid line .75, and move the cursor down to place the section.



4 Click and drag the right control to the left until it is just outside the structural slab.

You adjust the depth of the section to include only what you want to show in the section.



5 Press *Esc*.

Set the section view properties

6 In the Project Browser, expand Detail Views (Detail), right-click Detail 0, and select Properties.

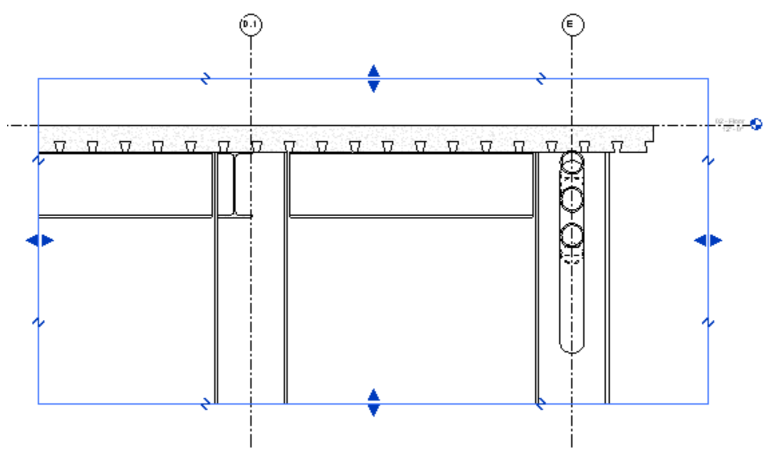
7 In the Instance Properties dialog:

- Under Graphics, for View scale, select 1 1/2" = 1' - 0".
- For Detail Level, select Fine.
- For Model Graphics Style, select Hidden Line.
- Under Identity Data, for View Name, type **Bolted Angle Detail**.
- Click OK.

Adjust the section view crop boundaries

8 Open the Bolted Angle Detail.

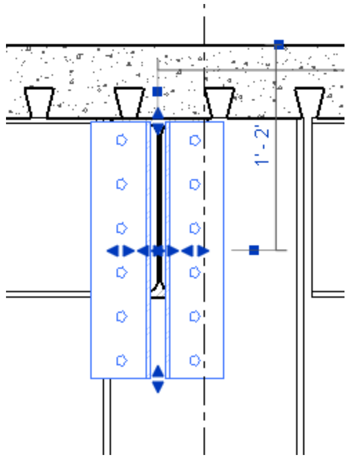
9 Adjust the crop region boundaries, as shown.



Add detail items to the section view

10 In the Project Browser, click Families ➤ Detail Items ➤ L-Angle-Bolted Connection Section, select L4x4x5/16.

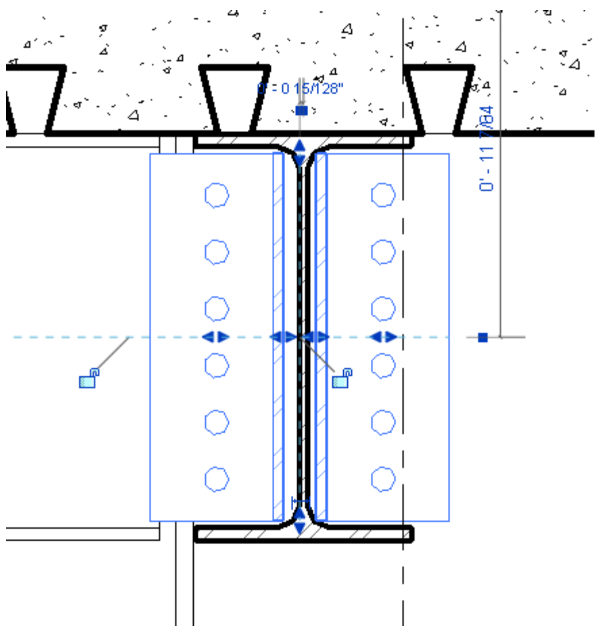
11 Drag the family to the grid line in the view as shown.



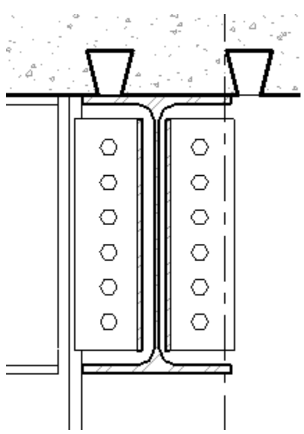
The connector is too large for the beam. You will resize it in the next steps.

12 Press *Esc*.

13 Select the connector, and click the bottom shape handle to resize it, approximately as shown.



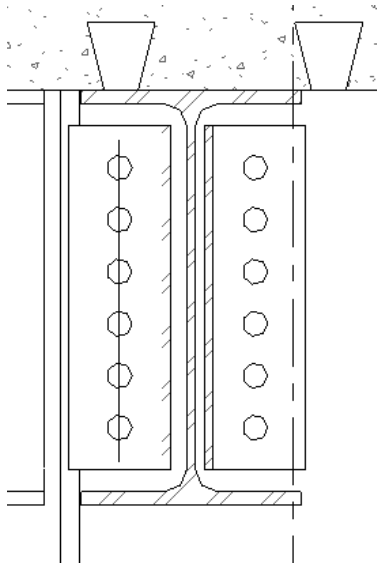
14 Press *Esc* twice.



Add annotations

15 Click Annotate tab ➤ Detail panel ➤ Detail Lines.

16 Sketch a vertical line through the center of the left set of bolts, as shown.



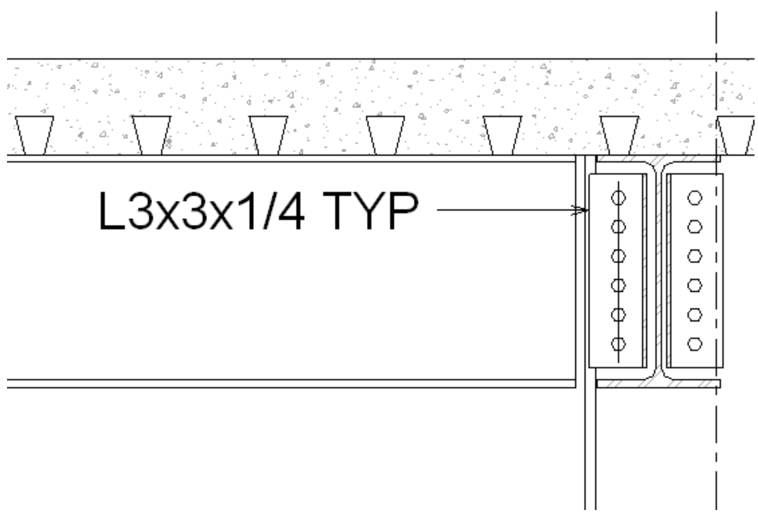
17 Press *Esc* twice.

18 Click Text panel ➤ Text.

19 Click Leader panel ➤ One Segment.

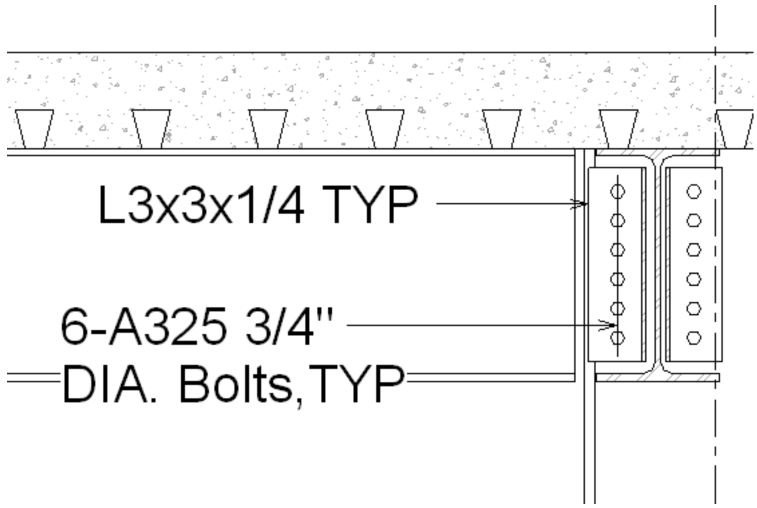
20 Click the vertical line (placed over the left bolts), and sketch a single leader.

21 Click to enter the text note as shown.



22 Click Selection panel ➤ Modify.

23 Using the same method, create a second annotation as shown.

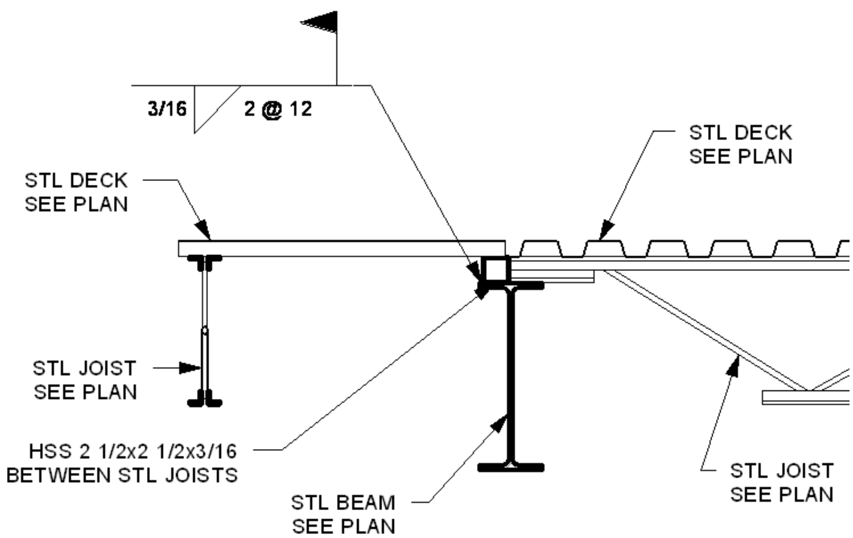


24 Close the file with or without saving it.

In the next exercise, a new training file is supplied.

Creating a Deck Span Transition Detail

In this exercise, you create a drafting view detail for a typical deck span transition. The detail is created as a new project, using the tools provided in Revit Structure.



Training File

- Click ➤ Open ➤ Project.
- In the left pane of the Open dialog, click Training Files, and open Imperial\RST_DOC_DET_STEEL_03_Deck_Span_Transition_Detail_i.rvt.

Create the drafting view

1 Click View tab ➤ Create panel ➤ Drafting View.

2 In the New Drafting View dialog:

- Under Name, type **Typical Detail - Deck Span Transition**.
- Under Scale, select 1 1/2" = 1' - 0".
- Click OK.

3 In the Project Browser, expand Drafting Views, and double-click Typical Detail - Deck Span Transition.

Place a steel beam

4 Click Annotate tab ➤ Detail panel ➤ Component drop-down ➤ Detail Component.

5 Click Element panel ➤ Change Element Type drop-down ➤ AISC Wide Flange Shapes - Section: W18x35.

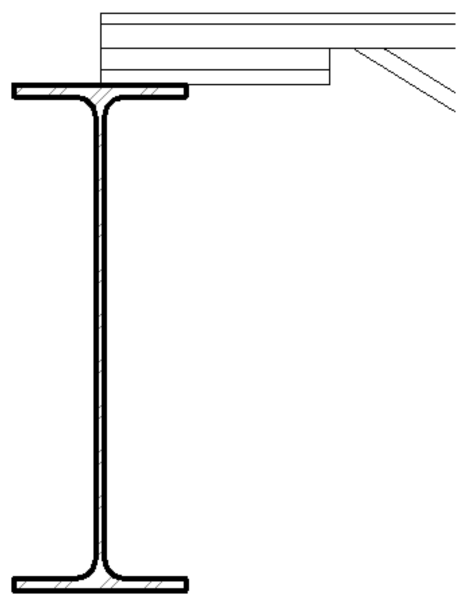
6 Click the drawing area to place the component.

Place a bar joist

7 Click Element panel ➤ Change Element Type drop-down ➤ K-Series Bar Joist - Side: 14K3.

8 Position the joist on the right side of the beam.

Align the bottom of the joist seat with the top of the beam, as shown.



9 Click Selection panel ➤ Modify.

Place a repeating component

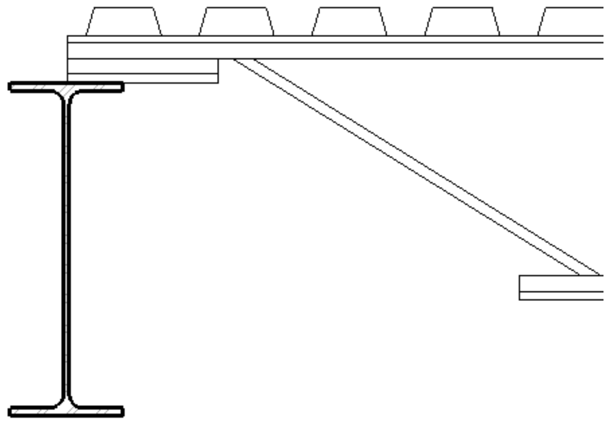
10 Click Detail panel ➤ Component drop-down ➤ Repeating Detail.

The metal deck profile is selected automatically.

11 Click the end of the joist to place the metal deck component.

12 Move the pointer to the right to begin placing the deck.

13 Press the *Spacebar* to rotate the orientation of the deck, and continue to move the pointer to the right until the deck is drawn, as shown.



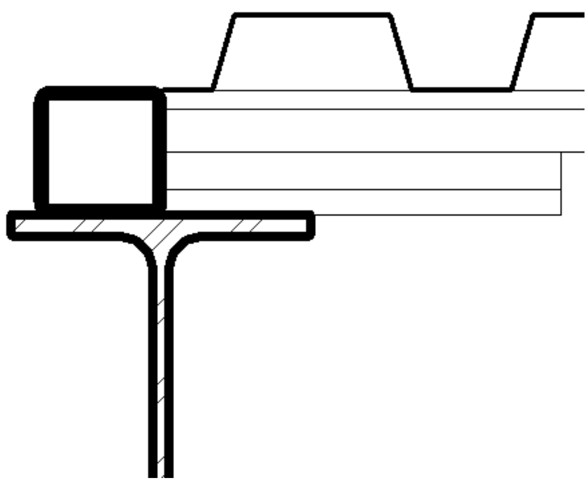
14 Click Selection panel ➤ Modify.

Place a hollow tube section

15 Click Detail panel ➤ Component drop-down ➤ Detail Component.

16 Click Element panel ➤ Change Element Type drop-down ➤ AISC Tube Shapes - Section: HSS2-1/2x2-1/2x1/8.

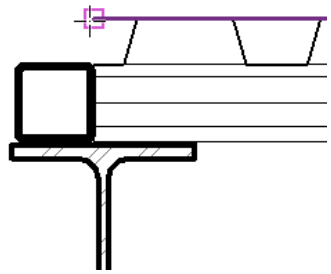
17 Place the tube on the top flange of the steel beam directly next to the end of the joist seat, as shown.



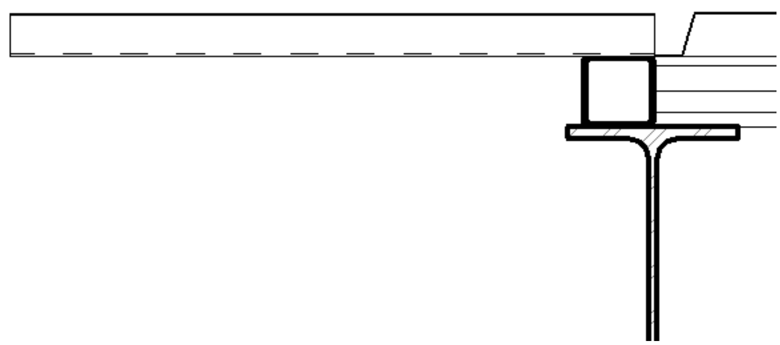
Place roof decking

18 Click Element panel ➤ Change Element Type drop-down ➤ Roof Decking-Side: 1.5 WR 22.

19 Click the top plane of the existing metal deck and the right edge of the metal tube to establish the end point for placing the roof decking.



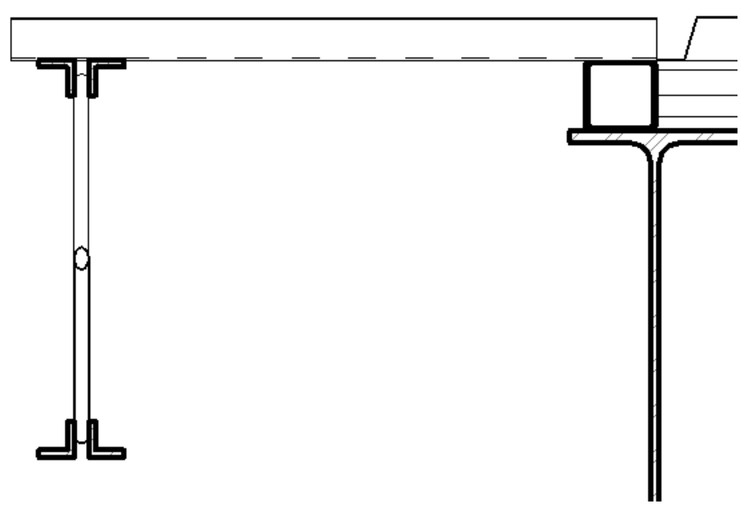
20 Drag the deck component to the left, approximately as shown



Place a bar joist

21 Click Element panel ➤ Change Element Type drop-down ➤ K-Series Bar Joist-Section: 14K3.

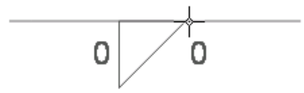
22 Place the joist to the left of the beam, and align the top of the joist with the bottom of the deck, as shown.

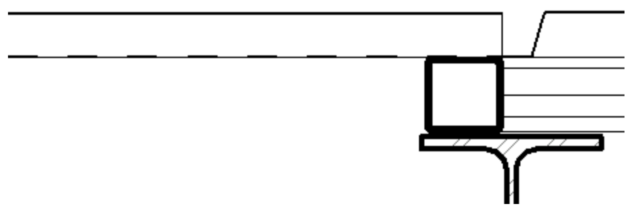


23 Click Selection panel ➤ Modify.

Add weld symbols

24 In the Project Browser, expand Families ➤ Annotation Symbols ➤ Weld Symbol ➤ Bottom, drag it to the view, as shown, and click to place it.





25 Press *Esc* twice.

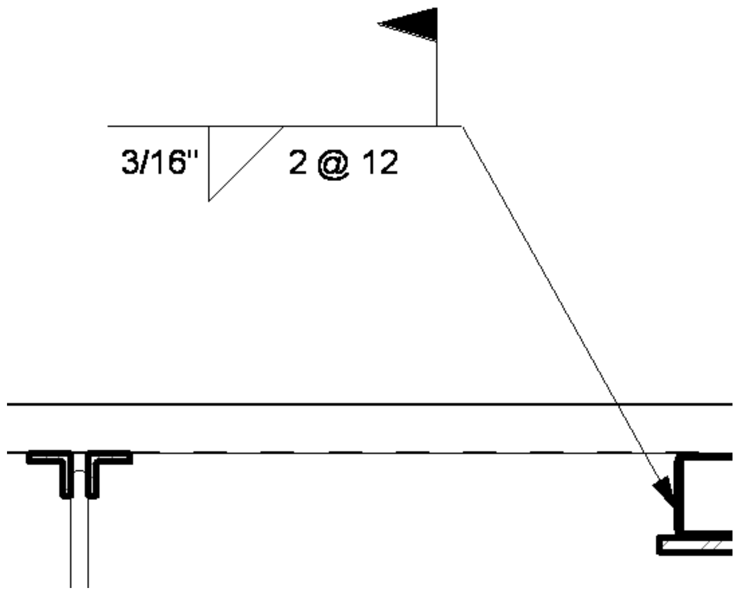
26 Select the weld symbol, and click Element panel ➤ Element Properties drop-down ➤ Instance Properties.

27 In the Instance Properties dialog:

- Under Structural, select Field Weld.
- Type **3/16**" for Bottom Weld Size.
- Type **2 @ 12** for Bottom Weld Length.
- Under Other, clear Symbol Left.
- Click OK.

28 Click Leader panel ➤ Leader: Add.

29 Drag the leader arrowhead to the position as shown.



30 Press *Esc* twice.

Add annotations

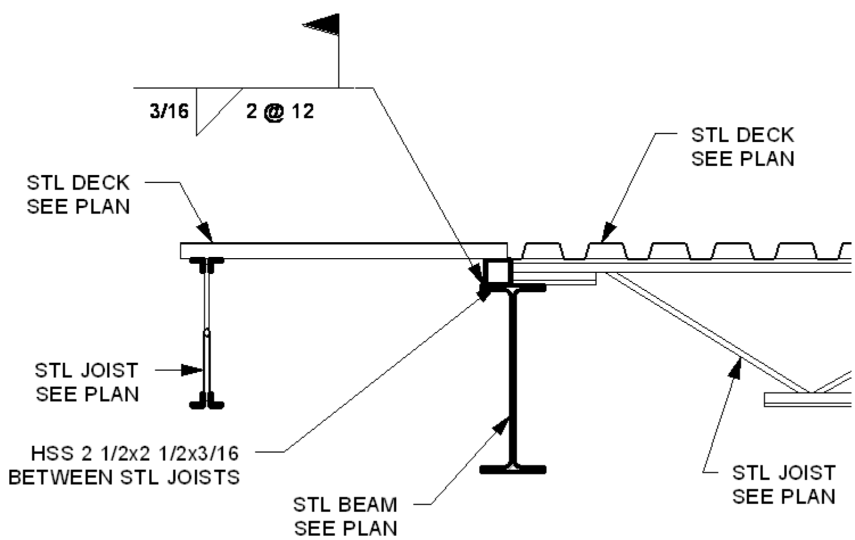
31 Click Text panel ➤ Text.

32 Click Leader panel ➤ One Segment.

33 Draw a text leader to the steel deck and type **STL DECK SEE PLAN** for text.

34 Click Selection panel ➤ Modify.

35 Using the same method, add the remaining text and leader to the detail view, as shown.



36 Close the file with or without saving it.

In the next exercise, a new training file is supplied.