

# Qnect for Autodesk® Revit® v1.4 FAQ

October 17, 2024

## Help and Support

Q: Who do I contact to get support or ask questions?

A: To create a ticket for support, contact Qnect at [support@qnect.com](mailto:support@qnect.com)

Support may ask for a log file of your latest runs. You can find that file at this location:

C:\Users\Qnect\AppData\Local\Qnect\QnectRevit\logs

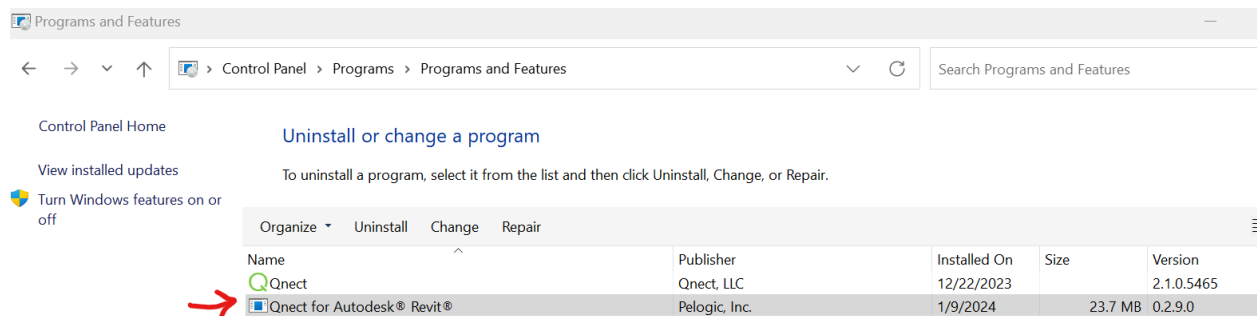
## Installation

Q: What versions of Revit does the application work with?

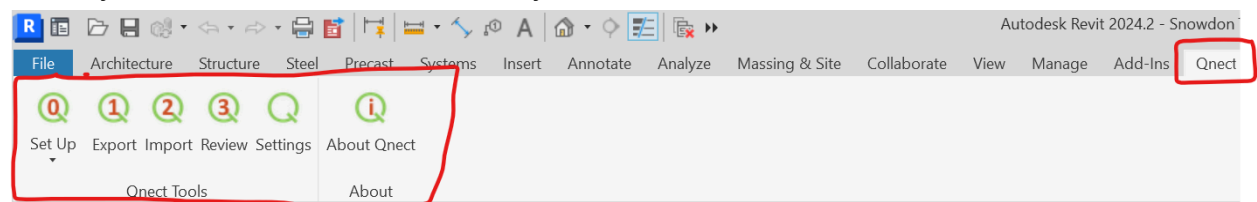
A: Revit 2025, 2024, 2023, 2022

Q: I don't see Revit installed on my toolbar.

A: check if uninstalled within Programs. If not, check with your IT



A: Verify the Qnect ribbon is installed in your Revit menu



A: Uninstall and reinstall as Local User

## Autodesk Login:

If logged in as an Autodesk user, it will automatically login with user's Autodesk account. Here are steps

## Sign in



Email

mgustafson@qnect.com


**Next**

New to Autodesk? [Create account](#)

## Authorize application



Qnect for Revit is requesting permission for the following information:

 mgustafsonS9WKR

**Identity**  
View your profile info

**Allow**

Don't want to authorize? [Cancel](#)

User will see dialog that login was successful.

### Modeling Best Practices

Q: Can users set parameters differently for parts when using Revit groups?

A: No. If users create several elements within a group, only parameters are defined for the group as a whole. If users want to define individual parameters like end forces per member, the user must separate the elements from the group.

Q: What framing profiles or objects are supported by the software?

A: W shape beams and columns are the most commonly used framing system and these are supported by Qnect. Support for other framing types and profiles varies. See below for what is supported and not supported. If elements are not supported, they will be ignored in the analysis.

	Framing type*				
Steel Profile**	Beam-Column	Beam-Beam	Vertical Bracing	Horizontal Bracing	Steel embed plate
WF	WF beam to WF col	WF beam to WF beam			WF beam to steel plate
HSS Rect	WF beam to HSS col	WF beam to HSS beam	HSS brace (field weld)		
Single L			Single L brace		
HSS Round C, WT, 2L Pipe					

\*Framing types not supported include miscellaneous steel (kicks, stairs, railings), walls, slabs or foundations

\*\* Non-structural steel profiles are not supported (concrete, timber, etc).

Q: How are users informed if a supporting profile is not recognized?

A: If supporting members are not recognized (concrete columns, walls, footings, etc) then the user will be notified with a NoConnect description (*Could not identify the support member to connect to or the support member is not supported by Qnect.*). To connect a steel beam to a concrete element, the user must model an embed plate in the concrete column or wall or footing, name it EMBED, and then model a beam to it. The software will then detect this as a framing condition and will try to connect it.

Q: What are modeling best practices for checking steel beams framing into embed plates?

A: Embed Name must be called EMBED. Supports framing W-shape beams into embed plates. Embeds can be modeled freely or part of concrete wall or column

Q: Can I import a model from another 3D solution to then run Qnect on?

A: Only if the 3D solution creates system families in Revit. If importing IFC models, they may import as in-place families and therefore cannot be run with this tool.

Q: What profile offset settings can I use to get picked up by Qnect? For example, can I use both Centerline or Origin in my family offsets?

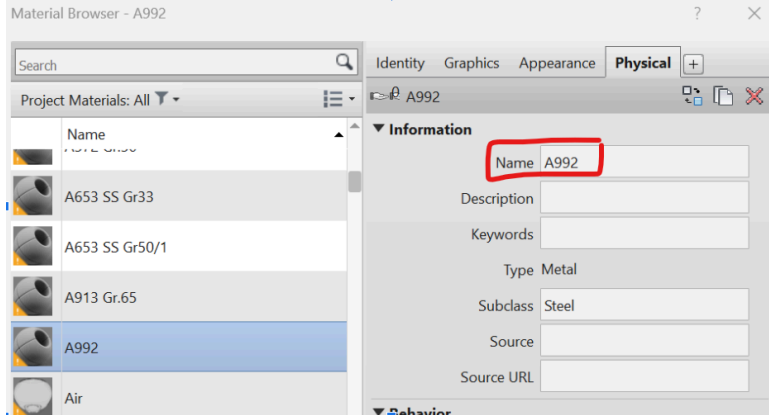
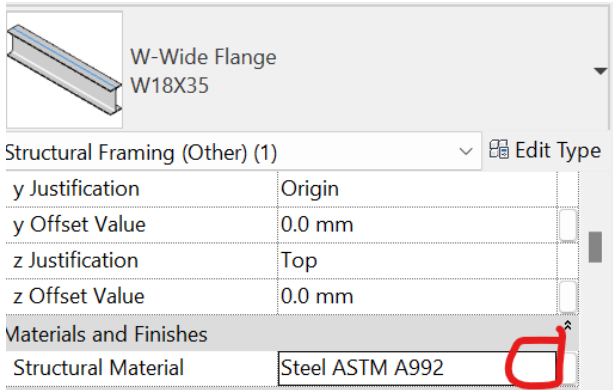
A: Users can use whatever method they are used to. Qnect scans the modeling work points that define the "curve driver" of the geometry and compares it to the actual physical location of the profile. Any differences between the axis and physical geometry are calculated and

stored as offset values. Therefore, the export always sends Position Depth = "Behind", Position Plane to "Middle" and computes Dz, Dy, Dx for each end. This allows Qnect to handle ALL Revit positioning settings by looking at the resulting geometry position rather than attempting to reverse engineer the locations from parameters and dealing with the "origin" problem which varies from firm to firm.

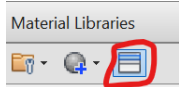
Material Grades

Qnect supports Revit Material grades for most of the out of the box Steel material grades. To add to this list, the user must contact Qnect support to add them to the master database.



















Qnect reads the Physical tab's Info: Name property (see below)














To see what materials Qnect supports, go to Manage > Materials > Material Browser > Asset Browser (icon at bottom).



These Metal Materials are supported by Qnect

	Steel ASTM A242-42	Physical	Metal	Metal: Steel
	Steel ASTM A242-46	Physical	Metal	Metal: Steel
	Steel ASTM A242-50	Physical	Metal	Metal: Steel 
	Steel ASTM A36	Physical	Metal	Metal: Steel
	Steel ASTM A441	Physical	Metal	Metal: Steel
	Steel ASTM A500B-42	Physical	Metal	Metal: Steel
	Steel ASTM A500B-46	Physical	Metal	Metal: Steel
	Steel ASTM A500C-46	Physical	Metal	Metal: Steel
	Steel ASTM A500C-50	Physical	Metal	Metal: Steel
	Steel ASTM A514	Physical	Metal	Metal: Steel 
	Steel ASTM A53 - Grade B	Physical	Metal	Metal: Steel
	Steel ASTM A572	Physical	Metal	Metal: Steel
	Steel ASTM A572-42	Physical	Metal	Metal: Steel
	Steel ASTM A572-50	Physical	Metal	Metal: Steel
	Steel ASTM A572-60	Physical	Metal	Metal: Steel
	Steel ASTM A572-65	Physical	Metal	Metal: Steel

	Steel ASTM A514	Physical	Metal	Metal: Steel
	Steel ASTM A53 - Grade B	Physical	Metal	Metal: Steel
	Steel ASTM A572	Physical	Metal	Metal: Steel
	Steel ASTM A572-42	Physical	Metal	Metal: Steel
	Steel ASTM A572-50	Physical	Metal	Metal: Steel
	Steel ASTM A572-60	Physical	Metal	Metal: Steel
	Steel ASTM A572-65	Physical	Metal	Metal: Steel
	Steel ASTM A913-50	Physical	Metal	Metal: Steel
	Steel ASTM A913-60	Physical	Metal	Metal: Steel
	Steel ASTM A913-70	Physical	Metal	Metal: Steel
	Steel ASTM A992-50	Physical	Metal	Metal: Steel

Note that Qnect also checks if the specified material is valid per AISC Tables 2-4 and 2-5 in the AISC Manual. So if a user specifies A513 steel for WF shape, it will be flagged as a NoConnect reason as "not meeting material specification."

Revit materials NOT supported:

Steel ASTM A760M-420 - corrugated pipe

Steel ASTM A53 - pipe

Steel ASTM A615 - concrete reinforcement

Steel ASTM A992-60 - does not exist in AISC code. Ignored

Steel ASTM A992-70 - does not exist in AISC code. Ignored

S 460

Steel - Carbon

<b>Table 2-5 Applicable ASTM Specifications for Plates and Bars</b>														
Steel Type	ASTM Designation		$F_y$ Min. Yield Stress (ksi)	$F_u$ Tensile Stress <sup>a</sup> (ksi)	Thickness of Plates and Bars, in.									
					to 0.75 incl.	over 0.75 to 1.25	over 1.25 to 1.5	over 1.5 to 2 incl.	over 2 to 2.5 incl.	over 2.5 to 4 incl.	over 4 to 5 incl.	over 5 to 6 incl.	over 6 to 8 incl.	over 8
Carbon	A36		32	58-80										
			36	58-80										
	A529	Gr. 50	50	70-100		b	b	b	b					
Gr. 55		55	70-100		b	b								
High-Strength Low-Alloy	A572	Gr. 42	42	60										
		Gr. 50	50	65										
		Gr. 55	55	70										
		Gr. 60	60	75										
		Gr. 65	65	80										
Corrosion Resistant High-Strength Low-Alloy	A242		42	63										
			46	67										
			50	70										
	A588		42	63										
			46	67										
			50	70										
Quenched and Tempered Alloy	A514 <sup>c</sup>		90	100-130										
			100	110-130										
Quenched and Tempered Low-Alloy	A852 <sup>c</sup>	70	90-110											

= Preferred material specification  
 = Other applicable material specification, the availability of which should be confirmed prior to specification  
 = Material specification does not apply

<sup>a</sup> Minimum unless a range is shown.

<sup>b</sup> Applicable to bars only above 1-in. thickness.

<sup>c</sup> Available as plates only.

**Table 2-4  
Applicable ASTM Specifications  
for Various Structural Shapes**

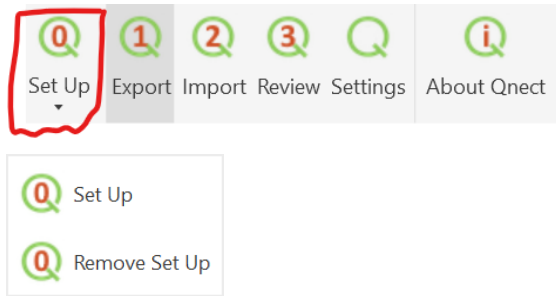
Steel Type	ASTM Designation	F <sub>y</sub> Yield Stress <sup>a</sup> (ksi)	F <sub>u</sub> Tensile Stress <sup>a</sup> (ksi)	Applicable Shape Series											
				W	M	S	HP	C	MC	L	HSS				
											Rect.	Round	Pipe		
Carbon	A36	36	58-80 <sup>b</sup>	■	■	■	■	■	■	■	■	■	■	■	
	A53 Gr. B	35	60											■	
	A501	Gr. B	42	58										■	
			46	58										■	
		Gr. C	46	62										■	■
			50	62									■		
	A501	Gr. A	36	58									■	■	
		Gr. B	50	70									■	■	
	A529 <sup>c</sup>	Gr. 50	50	65-100	■										
		Gr. 55	55	70-100	■										
	A709	36	36	58-80 <sup>b</sup>	■										
	A1043 <sup>d,e</sup>	36	36-52	58	■										
		50	50-65	65	■										
	A1085	Gr. A	50	65										■	
High-Strength Low-Alloy	A572	Gr. 42	42	60	■										
		Gr. 50	50	65	■			■							
		Gr. 55	55	70	■										
		Gr. 60 <sup>f</sup>	60	75	■										
		Gr. 65 <sup>f</sup>	65	80	■										
	A618 <sup>f</sup>	Gr. II <sup>g</sup> , II & I	50 <sup>h</sup>	70 <sup>h</sup>											■
		Gr. III	50	65											■
	A709	50	50	65	■										
		50S	50-65	65	■										
		50W	50	70	■										
	A913	50	50 <sup>h</sup>	65 <sup>h</sup>	■										
		60	60	75	■										
		65	65	80	■										
		70	70	90	■										
	A992	50 <sup>i</sup>	65 <sup>i</sup>	■											
	A1065 <sup>g</sup>	Gr. 50 <sup>j</sup>	50	60										■	

- - Preferred material specification.
- - Other applicable material specification, the availability of which should be confirmed prior to specification.
- - Material specification does not apply.

Footnotes on facing page.

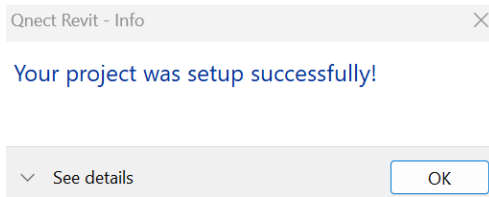


### Button 0: Setup



Q: Why do I need to Set Up project?

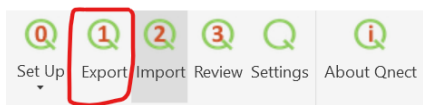
A: The Revit model must be initialized with Revit parameters that are specific to Qnect. These parameters are created to store results returned and also allow users to define end forces that are used to perform the analysis.



Q: What does Remove Set up do?

A: This removes parameter values from previous runs. Users may wish to “purge” or remove the model properties when sharing models with other disciplines. Also users should remove old data if the job preferences are updated on the cloud or if the geometry or forces in the Revit model change.

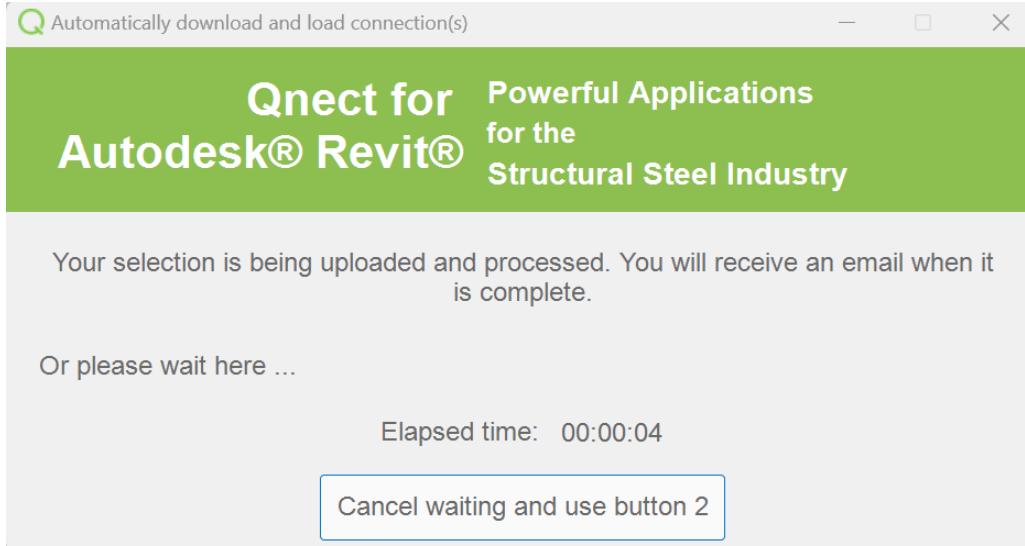
### Button 1: Export



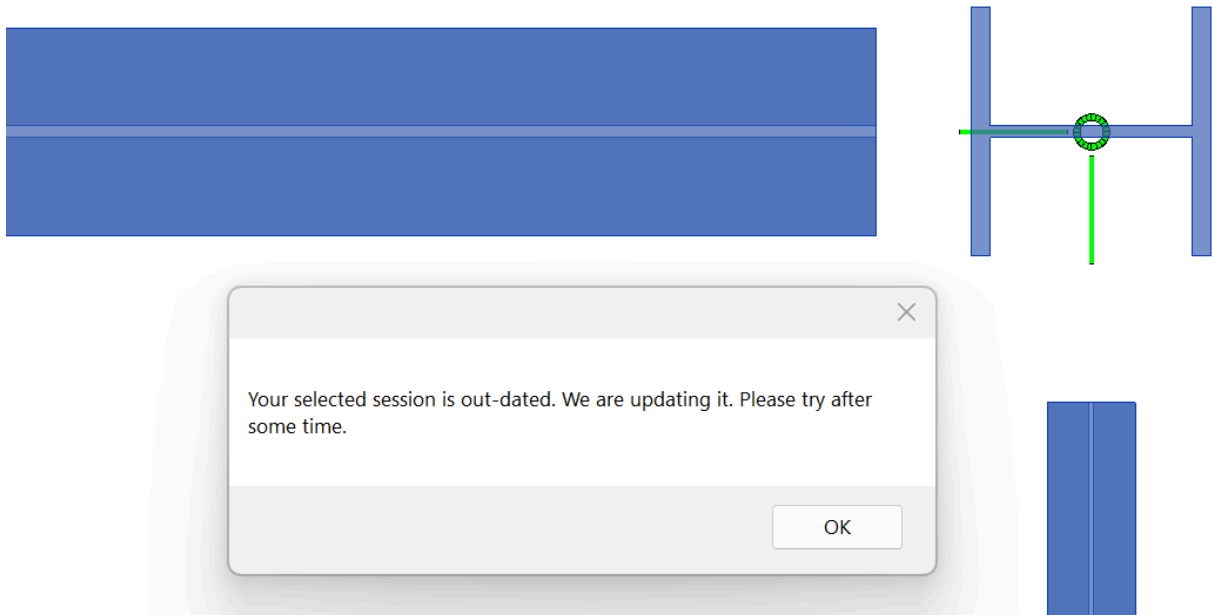
Q: It takes several minutes to export the model. Is this normal?

A: Yes it can. It takes about 1 second per element selected to run the analysis. So for a model with 5000 elements that is over 1 hr. Strategies for the user to manage this include:

- Select Cancel on the dialog (see below) and wait for email confirmation that the analysis is done. Then the user can go to Button 2 to Import results at a later time and then review results within Button 3.
- Select a particular floor or bay to analyze and solve issues for first. As you analyze various regions, they will be remembered and accumulate within the stored results.



Q: When exporting, the user gets an “out of date” dialog message. What does this mean?  
 A: This means the export did not run completely and there will be no Issues returned. The cause of this can be the selected members are too far away from each other to form a connection. The example below shows the beams set back too far from the column.



Q: When I export it completes but no results are provided?  
 A: Note that results do not automatically import and visualize after selecting button 1. For larger models Button 2 must be selected to import results. Then Button 3 is opened to review results. If results are still not showing after selecting button 3, the exported members may be concrete or steel joists or other non-recognized profiles. Also results may still be running on the

cloud. Users should wait for an email confirmation that their run is complete. Try hitting button 2 to return results to see if results then show in button 3.

**Q:** Returns error that profiles selected do not exist or do not form a joint. What action can users take?

**A:** No connection issue is returned since the profile name is not supported in the Qnect database or the framing elements do not intersect geometrically. To add specific profile namings into the Qnect database as Aliases, users should reach out to Qnect Support to add them in. In general concrete elements, built-up steel shapes and other non structural steel elements are not supported so won't return any data. If the user mass selects the model, Qnect will ignore any unknown profiles but will continue to run the session on the valid profiles and return successful or unsuccessful NoConnects.

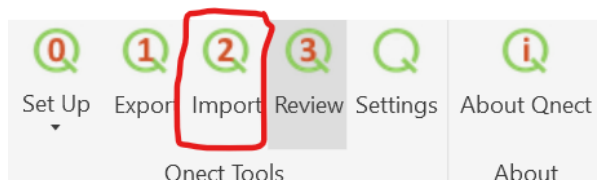
**Q:** How are users informed if a secondary member (beams, vertical bracing) is not recognized?

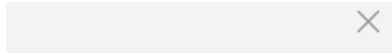
**A:** Users will get a NoConnect issue that says "*Framing condition not yet supported by Qnect.*" This could be for several reasons based on these constraints:

- Revit elements modeled as in-place families (vs system families). This is the case when importing framing from an IFC file.
- Concrete, timber, cold-formed steel or other non-steel secondary profiles in Revit
- Steel secondary profiles we don't yet connect like HSS round, channels, WTs, pipe, built-up steel sections
- Steel profiles in Revit not in our database that need to be mapped. Contact Qnect to update its web database.
- Steel profiles with the incorrect NAME field in Revit. Here are required names:
  - EMBED for embed plates
  - BEAM for beams
  - COLUMN for column
  - PLATE for plate
  - V\* BRACE for vertical bracing

To troubleshoot, it is suggested to diagnose areas of the model in regions like floor by floor or bay by bay. That way the user gets a feel of what is being studied and can expand from there. As the user runs more batches, they will all show up in the Review dialog (button 3) when simply launching it with nothing in the model selected.

### Button 2: Import

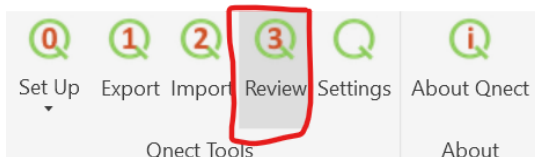




Import operation completed!



### Button 3: Review. Visualizing Issues in Revit



Q: I see previously ran NoConnects that are not being resolved. How do I remove them?

A: Data is stored from all past runs. With every new run, the connection data for each GUID pairings (ie a beam to girder) are replaced with updated data. However, if there is no new data to replace it or the NoConnect goes away then the 'tombstone' will remain. To eliminate these, users must go to Set up > Remove Set Up. This will remove past runs data.

### Using End Forces

Q: What if no forces are used in the model? How does Qnect do design checks without forces?

A: Qnect makes these assumptions

AISC LRFD - 15

UDL factors - % factor of total Uniform Distributed Load

- Composite beam - 0.80. If shear connectors field is non-blank (0 and up)
- Non-composite beam: 0.50. If shear connectors field is empty

UDL forces for shorter spans:

- See Preferences documentation.

Note the default settings can be adjusted by Qnect or by the user if a more advanced version of the software is used.

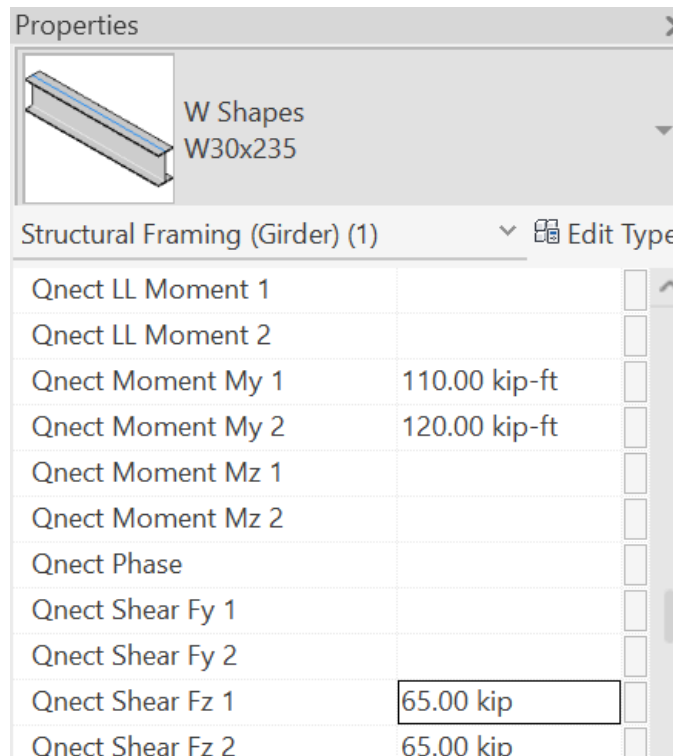
Q: How are forces utilized in Qnect?

A: Users have 3 places where they can specify the end forces to be used by Qnect. This is for shear, moment and axial forces:

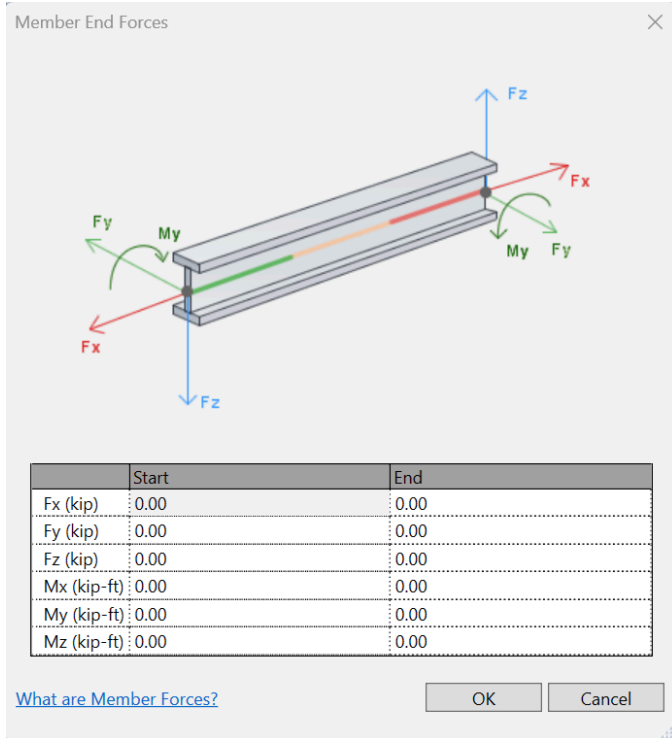
- Fill in Qnect parameters on the beam element itself. There are shear, axial, moment and torsion parameters for each end in the beam element parameters.

- The user can also, if they have an associated analytical model to the beam, enter the member end forces there in the associated analytical stick. The plugin will look for analytical forces in the associated stick and offer to copy them into the Qnect parameters. The user can either enter them in the analytical model themselves, or if their analytical package updates those then our plugin will offer to copy them from there.
- The user can also enter the member end forces there in the associated physical member ends. The plugin gives option to match parameters from physical parameters over into Qnect properties.

The major shear vertical force (Fz), major moment force (My) and axial (tension and compression) are utilized by Qnect. Weak axis shear and moment forces can be filled in but are currently not utilized by Qnect’s connection design engine.



Physical and Analysis Force Fields:



Q: How are load combinations considered in checking the connection design?

A: Currently Qnect does not handle different combinations of forces resulting from different load combinations. There is currently only one set of analytical forces on the analytical member itself.

### Drawing Maps

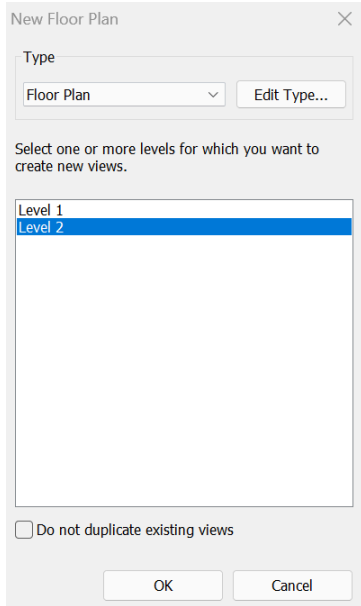
Q: It is taking a while to generate the drawing map. What can I do?

A: The user can't control the drawing settings being created. However you could limit the floor plan range in your Floor Plan settings or set up specific plans for the Qnect maps within Floor Plans so that a smaller plan drawing can be utilized. There is no way to cancel the action once the drawing is generated.

Q: How do I customize the appearance of the drawing maps

A: The drawing templates are reading available plan views the user already has in the Revit project. The user can go to View > Plan Views > Floor Plan to set up the plan views they want to use in Qnect's drawing map pull down. What is in this Floor Plan list will show up in Qnect for Revit. Using this plan, Qnect just applies the NoConnect code and description at the start and end locations of the framing.

In Revit:



In Qnect:

