



Integrated Concrete Design in Revit

Ken Marsh – Marsh API LLC

Code SE6880

Learning Objectives

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

- Discover the key technologies that enable integrated structural engineering
- Understand how to use Structural Analysis in Autodesk 360 software to drive downstream design
- Learn how to utilize Results Explorer in Revit software to view and understand analysis results data
- Discover 1 or more examples of integrated design applications built on Revit software

About the Speaker

After 7 years as a structural engineer, Ken Marsh joined Autodesk, Inc., as a quality assurance analyst working on the Revit software product line. Ken has recently started his own firm, which is dedicated to advancing Building Information Modeling (BIM)-based structural engineering through the Revit software API add-ons. Ken is also the author of Robot Structural Analysis Professional 2015—Essentials, and he loves to discuss Autodesk technology as it relates to the architecture, engineering, and construction industry.

ken@marshapi.com

www.marshapi.com

Copyright © 2014 by Marsh API, LLC, Somerville, Massachusetts

All rights reserved. Use of this publication (this “Work”) is subject to these terms. Except as permitted under the Copyright Act of 1976, as amended, and the right to store and retrieve one copy of this Work, you may not decompile, disassemble, reverse engineer, reproduce, modify, create derivative works based upon, transmit, distribute, disseminate, sell, publish or sublicense this Work or any part without Publisher’s prior written consent. You may use this Work for your own commercial and personal use; any other use of this Work is strictly prohibited. Your right to use this Work may be terminated if you fail to comply with these terms. No part of this Work may be reproduced, distributed, or transmitted in any form or by any means, including photocopying, recording, or other electronic or mechanical methods, without the prior written permission of Publisher. Written requests for permission should be addressed to:

Marsh API
179 Albion Street • Unit 2
Somerville, MA 02144
RobotEssentials@marshapi.com

THIS WORK SHALL NOT BE CONSTRUED AS A RENDERING OF ENGINEERING OR OTHER PROFESSIONAL ADVICE AND/OR SERVICES BY PUBLISHER OR AUTHOR. PUBLISHER AND AUTHOR MAKE NO REPRESENTATIONS OR WARRANTIES WITH RESPECT TO THE ACCURACY OR COMPLETENESS OF THE CONTENTS OF THIS WORK. PUBLISHER AND AUTHOR SPECIFICALLY DISCLAIM AND YOU WAIVE ALL REPRESENTATIONS AND WARRANTIES (WHETHER EXPRESS, IMPLIED OR STATUTORY), INCLUDING, WITHOUT LIMITATION, ANY WARRANTY OR CONDITION (A) OF FITNESS FOR A PARTICULAR PURPOSE, NON-INFRINGEMENT, TITLE, SATISFACTORY QUALITY, ACCURACY, OR (B) ARISING FROM ANY COURSE OF DEALING, COURSE OR PERFORMANCE, OR USAGE IN THE INDUSTRY. NO WARRANTY MAY BE CREATED OR EXTENDED BY SALES OR PROMOTIONAL MATERIALS. NEITHER PUBLISHER NOR AUTHOR SHALL BE LIABLE FOR ANY DAMAGES ARISING FROM RELIANCE ON THE ACCURACY OF THIS WORK, ANY “CONSTRUCTION FAILURE” RELATED TO THE USE OF OR RELIANCE ON THIS WORK, OR OTHERWISE FROM THE USE OF THIS WORK. UNDER NO CIRCUMSTANCES SHALL PUBLISHER OR AUTHOR BE LIABLE FOR ANY INDIRECT, INCIDENTAL, SPECIAL, PUNITIVE, CONSEQUENTIAL OR SIMILAR DAMAGES THAT RESULT FROM THE USE OF OR INABILITY TO USE THE WORK, EVEN IF ANY OF THEM HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. THIS LIMITATION OF LIABILITY SHALL APPLY TO ANY CLAIM OR CAUSE WHATSOEVER WHETHER SUCH CLAIM OR CAUSE ARISES IN CONTRACT, TORT, OR OTHERWISE. Reference or citation within this Work to any organization or website does not constitute an endorsement by either Publisher or Author.

Trademarks: Autodesk and Robot are registered trademarks of Autodesk, Inc.

Key Technologies that enable concrete design in Revit

Revit began, of course, as primarily a modeling tool. When development began on a differentiated product for structural engineering, Autodesk began incorporating tools not only for modeling the physical structure but also for providing a workflow for engineers to idealize the structure to feed the structural analysis and design chain. All of these upstream technologies feed the downstream

Revit Analytical Model

The basis for enabling the structural engineering workflow in Revit. All major structural elements in Revit also support a highly manipulate-able analytical idealization of that element. Linear elements for beams and columns, surface elements for floors and walls all feed the finite element analysis of the structure.

Revit works hard to give you a well-connected and analyzable analytical model based on the elements you have created in the physical model but realizing that idealization of a structure is reasonably idiosyncratic and open to interpretation, Revit analytical models have lots of options for adjustment including a dedicated environment for manipulating the analytical model alone. This is a huge benefit to our industry where a significantly split workflow between modeling and engineering staff still exists and probably will for a long time to come.

In addition Revit provides other necessary features for controlling member end releases, defining supports, and specifying loads on the structure.

Revit API

The Revit API is an amazing toolkit which forms the foundation for so much of what we want to accomplish with Revit, in terms of structural engineering that it should get a lot more respect than it probably does. The opening of the Revit database to external manipulation affords us the ability to read the analytical model, update members based on design, create reinforcing programmatically, populate element data parameters, store analysis results, and create advanced solutions drawing on information contained and/or stored in the model.

Originally Revit Structure had an analytical model but provided no capability for structural analysis. In order to facilitate analysis the Revit API offered third party structural analysis software providers the ability to craft link products which could read analytical information from Revit and write back results, design data, and update member sizes based on design performed in those third party softwares.

Revit Results Storage

Directly related to the Revit API but what I will consider a separate enabling technology is the structural analysis result storage system. As of Revit 2014, the Revit API affords the opportunity for analysis softwares to store the full results of structural analysis directly inside Revit. Now Revit is truly becoming a central database for the structural engineering workflow! Having access to finite element analysis results in Revit enables visualization, exploration, and full integration of analysis to Revit. No longer is the result of analysis disconnected from and kept at “arm’s length” from your Revit model.

Bringing analysis results into Revit reconnects the engineer with the analysis in a way that has never before been achievable: One Model which marries both physical model/documentation with analysis results.

The true advantage of results storage in Revit is that we can now drive fully integrated downstream design routines built either as plug-ins on top of Revit or, even as links to third party design software. This really breaks down the barriers to more localized design routines, custom design routines, and just democratized the whole engineering and design workflow: there’s nothing stopping you from creating your own fully integrated design routines in Revit!

It is also worth noting that Revit Results Storage can handle almost any type of data you want to store and visualize. Not only can forces, moments, shears, stresses, etc. be returned, you could also return and store other related design data such as required areas of steel reinforcing, shear stud requirements, numbers of bars, spacing of stirrups, etc.

Analysis Visualization Framework

This is a key piece of technology developed and embedded in Revit somewhere around Revit 2012. The Analysis visualization framework provides the foundation of analysis visualization in Revit by enabling API tools to paint results into Revit, results on both surfaces as well as results on linear elements, vector fields (e.g. reactions), and displaced shapes are supported.

The “results” which can be visualized with the framework are boundless. We can use it to visualize structural analysis results in terms of forces, moments as well as deflections and deflected shapes. We can also use it to visualize almost any type of analysis data which can be painted on a surface, drawn as a diagram along a line/curve or shown as a vector or vector field. For example, heat loss data, current temperature data, proximity data, etc. The API can even be used to drive visualization of data in “real-time” or analysis results which respond to user actions in the model (floor moments resulting from the location of a piece of equipment in the model, or heat transmission to surfaces nearby a heat generating piece of equipment). It’s pretty fun and exciting stuff and it’s barely utilized... yet. Results Explorer is one tool which takes advantage of this technology and is discussed below.

Structural Analysis Toolkit

This is not really a technology but a collection of tools/technology which facilitate several of the things we will need to have a fully integrated workflow: In order to make results storage in Revit available to the user – to create this sense of integration, the structural analysis toolkit provides some key functionality.

Results Manager: Results of any kind can be stored in the model alongside your Revit data. They are stored by the Results Builder technology developed by Autodesk and are represented as result packages in Revit Results Manager. The Results Manager gives you access to see what packages are stored in Revit, which packages are remote cloud based result packages, and the ability to manage them by deleting them, downloading, and launching exploration/visualization of those results.

Results Explorer: Result packages stored in Revit are all available to the Results Explorer tool which enables you to visualize the results using the Analysis Visualization Framework to paint one or more results in a view. As long as results are stored in a results package, the Results Explorer tool can provide access to them for visualization and inspection.

Analyze in Cloud: This is the main integrator that facilitates standalone analysis directly in Revit without a third party analysis software. This analysis is powered by A360 Structural Analysis Cloud Services and is the beginning of fully integrated, full-featured analysis directly inside Revit. We will discuss the capabilities more fully later on in this document.

Autodesk Revit Code Checking Framework

The final piece of the big puzzle for enabling true structural engineering workflows in Revit is the Code Checking Framework. This awesome piece of technology is a set of API functionalities which facilitate assigning design code parameters to analytical elements, setting up general code parameters to guide design, and providing a uniform set of interface tools to allow the user access to results sets, and load cases/combinations to be run in design of materials.

This is a slightly more advanced set of technology but it is not out of reach for reasonably accomplished developers. In particular, the concrete code checking portion of the framework is very well developed providing automated setup of cross sections for design, where forces are readily available and you can generate your own calculations for required and/or provided reinforcing.

The Code Checking Framework isn't only for concrete, you could add design/code check routines for steel members as well. Currently supported element types are analytical beams, analytical columns, analytical floors, analytical walls, analytical foundation slabs. You can provide design/code-checking for any type of material you wish. The framework just makes it much more accessible to get coding on important stuff and takes care of a lot of the background stuff for you.

Autodesk has, in the past few years, begun truly embracing the concept of democratizing software. The Code Checking Framework is a testament to that idea: never before has customization of design routines been nearly as accessible as Excel and never before has this level of tight integration with BIM and structural engineering been possible. It is now completely within the realm of possibilities that you could implement completely custom design routines for structural elements in your own company to your own company standards and in a way that you believe the design code should be interpreted. Not only that but you could conceivably directly hook up any custom Excel tools you have created via the code-checking framework or even just by directly utilizing the Revit API.

The possibilities afforded by the Code-Checking Framework are truly remarkable and I would strongly encourage anyone to investigate these capabilities further to not only accelerate your design workflows but to allow you to make or integrate the tools you may have written yourself and trust.

Rebar Tools

This is kind of obvious in a sense but, I think it's worth mentioning in terms of how much development effort and new technology went into the currently available tools. I suppose it's also worth mentioning that, while these tools are really amazing for what they do, they are valued differently in different parts of the world. One design tool we'll look at later on, relies heavily on these rebar modeling tools which results in some pretty awesome concrete reinforcement modeling in Revit.

If we are to eventually get to a fully integrated overall workflow where Revit serves as the central database the rebar tools are extremely important even if they are not yet part of a standard workflow in some part of the world.

How A360 Structural Analysis drives downstream design

When I was a budding structural engineering intern back in the early 1990s I had the distinct honor of working with Tom Tyson out in Rochester, NY at the then Sear-Brown Company. He shared his PhD research with me thinking that I might be interested and ironically enough, looking back, his PhD research was about a fully integrated analysis, design, and documentation system based on a central database. I truly believe that BIM is the fruit of these types of ideas, Tom was obviously not the only one who had these ideas as they applied to structural engineering but I always think of him when I think about how far we've come in this industry.

In order to support a full workflow in Revit, we have to have analysis integrated in some way. Autodesk made a bold plunge into full analysis integration in Revit by offering the A360 Cloud based structural analysis service. You can, without a third party analysis package, create model in Revit, apply loading, add boundary conditions, configure member end releases, specify load combinations, then send it directly to the cloud for analysis.

Structural analysis results from cloud analysis can be reviewed in the initial cloud report and also downloaded to the local Revit model for exploration and visualization right inside your model. Let's take a look at how this process works:

Analytical Modeling

Beams, columns, floors, walls, and foundation slabs have viable analytical models which can be used for structure idealization. These analytical elements live and die with their physical elements so we no longer have to worry about the analytical model being out of date with our physical model. One key to making this analytical model work is that the analytical model can be manipulated independently of the physical model. Allowing engineers to configure the idealized analytical model while physical modeling and documentation are unaffected, is essential to a combined workflow in one overall data model.

For beams and columns Revit provides parameters you might typically expect to be able to set for analysis such as member number, member end releases, analytical properties derived from the associated physical model as well as parameters which can control the location of the analytical model relative to the physical model. Finer control over the analytical model is available in analytical edit mode.

Analytical Beams (1)		Analytical Columns (1)	
Identity Data		Identity Data	
Member Number	0	Member Number	0
Comments		Comments	
Phasing		Phasing	
Phase Created	New Construction	Phase Created	New Construction
Phase Demolished	None	Phase Demolished	None
Structural Analysis		Structural Analysis	
Start Release	User Defined	Top Release	Fixed
Start Fx	☐	Top Fx	☐
Start Fy	☐	Top Fy	☐
Start Fz	☐	Top Fz	☐
Start Mx	☐	Top Mx	☐
Start My	☒	Top My	☐
Start Mz	☒	Top Mz	☐
End Release	Pinned	Base Release	Fixed
End Fx	☐	Base Fx	☐
End Fy	☐	Base Fy	☐
End Fz	☐	Base Fz	☐
End Mx	☒	Base Mx	☐
End My	☒	Base My	☐
End Mz	☒	Base Mz	☐
Analytical Model		Analytical Model	
Analyze As	Gravity	Analyze As	Gravity
Code Checking	Default3	Code Checking	Default2
Analytical Links	From Column	Analytical Links	☐
Analytical Properties		Analytical Properties	
Family Type	Concrete-Rectangular...	Family Type	Concrete-Rectangular...
Physical Material Asset	Concrete(1)	Physical Material Asset	Concrete(1)
Length	30' 0"	Length	10' 0"
Cross-Section Rotation	0.000°	Cross-Section Rotation	0.000°
Analytical Alignment		Analytical Alignment	
Start Alignment Method	Auto-Detect	Top Alignment Method	Auto-Detect
Start y Projection	Location Line	Top y Projection	Location Line
Start z Projection	Location Line	Top z Projection	Location Line
End Alignment Method	Auto-Detect	Top Extension Method	Auto-Detect
End y Projection	Location Line	Top x Projection	Top Level Reference
End z Projection	Location Line	Base Alignment Method	Auto-Detect
		Base y Projection	Location Line
		Base z Projection	Location Line
		Base Extension Method	Auto-Detect
		Base x Projection	Base Level Reference

Floors, foundation slabs, and walls have slightly different sets of parameters but afford similar control and reporting as beams and columns;

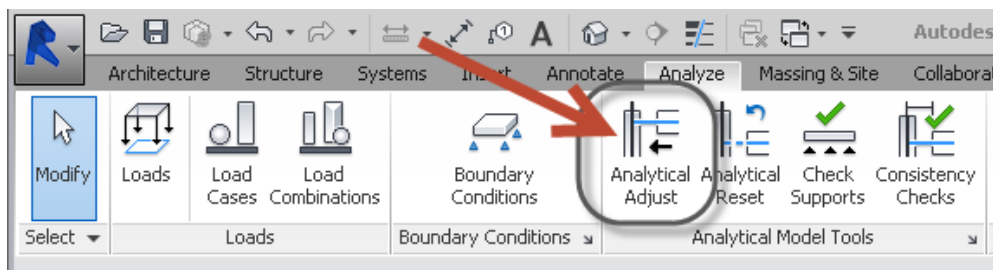
Analytical Floors (1)		Analytical Walls (1)	
Identity Data		Identity Data	
Surface Number	0	Surface Number	0
Comments		Comments	
Phasing		Phasing	
Phase Created	New Construction	Phase Created	New Construction
Phase Demolished	None	Phase Demolished	None
Analytical Model		Analytical Model	
Analyze As	One-Way	Analyze As	Gravity and Lateral
Code Checking	Default1	Code Checking	
Analytical Properties		Analytical Properties	
Family Type	Floor : 6" Concrete	Family Type	Basic Wall : Generic - 8"
Physical Material Asset	LW Concrete 4 ksi	Physical Material Asset	Unassigned
Area	2020.00 SF	Area	165.00 SF
Perimeter	181' 0"	Perimeter	53' 0"
Analytical Alignment		Analytical Alignment	
Alignment Method	Auto-Detect	Alignment Method	Auto-Detect
z Projection	Default	z Projection	Center of Element
		Top Extension Method	Auto-Detect
		Top y Projection	Top Of Wall
		Base Extension Method	Auto-Detect
		Base y Projection	Bottom Of Wall

These sets of parameters form the basis for analytical model definition in Revit.

Analytical Adjust Mode:

Something that perhaps few people are aware of is the powerful analytical edit mode which was introduced in Revit 2012. The edit mode allows you to intuitively manipulate the model with node and element grips to help you create exactly the model you want, aligned the way you want it.

Accessed from the Analyze tab, the Analytical Adjust button launches the analytical adjust mode:



Once in this mode, nodes at the ends of members become visible, selectable, and have grips when selected that enable full custom manipulation of the model. In the case that Revit does not give you what you want, and the parametric adjustment for elements also does not give you quite what you want, this edit mode gives you almost total freedom.

Additional tools in the adjust mode allow you to ignore openings in floors/walls to reduce model complexity and a rigid link tool to allow you to make connections and model eccentricities.

Loads and Boundary Conditions

You can specify load cases, make manual load combinations, apply loads to your model, as well as boundary conditions. These tools together allow you to create a model which is ready for analysis directly in Revit, no need to move to a third party software.

Loading Capabilities:

- Point loads in global or local coordinates both hosted and unhosted
- Line loads in global or local coordinates both hosted and unhosted, projected or not
- Planar loads in global or local coordinates both hosted and unhosted including the option to have load projected. Unhosted surface loads can also be sloped
- All loads contain a load case property so that they may be collected together into load cases

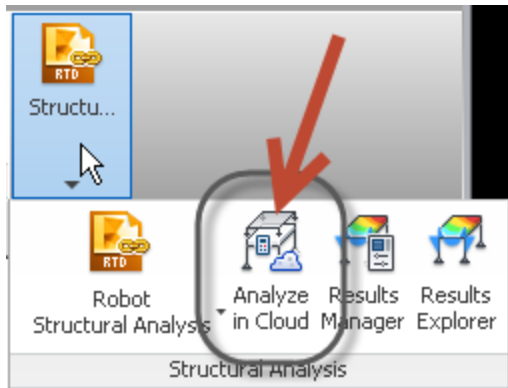
Boundary Condition Capabilities:

- Standard options: Pinned, Roller, fixed
- Six degrees of fixity available for point (nodal) boundary conditions (three translations and three rotations)
- Four degrees of fixity available for linear (edge) boundary conditions (three translations, one rotation)
- Three degrees of fixity for surface (area) boundary conditions. (three translations)
- All boundary conditions may be oriented to the project global coordinate system or the member local coordinate system
- All boundary condition degrees of fixity can be fixed, free, or have a spring constant specified to model elastic supports.
- Boundary conditions have awesome graphics.

Using A360 Analysis

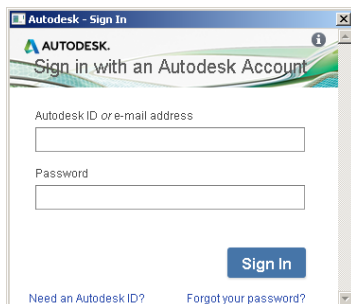
A360 Analysis is a cloud based service which requires cloud credits to utilize. Analyses can be run by anyone with a valid Autodesk Single Sign-On account and available credits. The current capabilities are linear elastic analysis for beams, columns, and shell elements (floors, foundation slabs, and walls)

A360 Analysis is available after the Structural Analysis Toolkit (available from the Revit Exchange AppStore) is installed and functional. You will see a new set of tools collapsed under a panel called “Structural Analysis” on the “Analyze” tab of the ribbon:



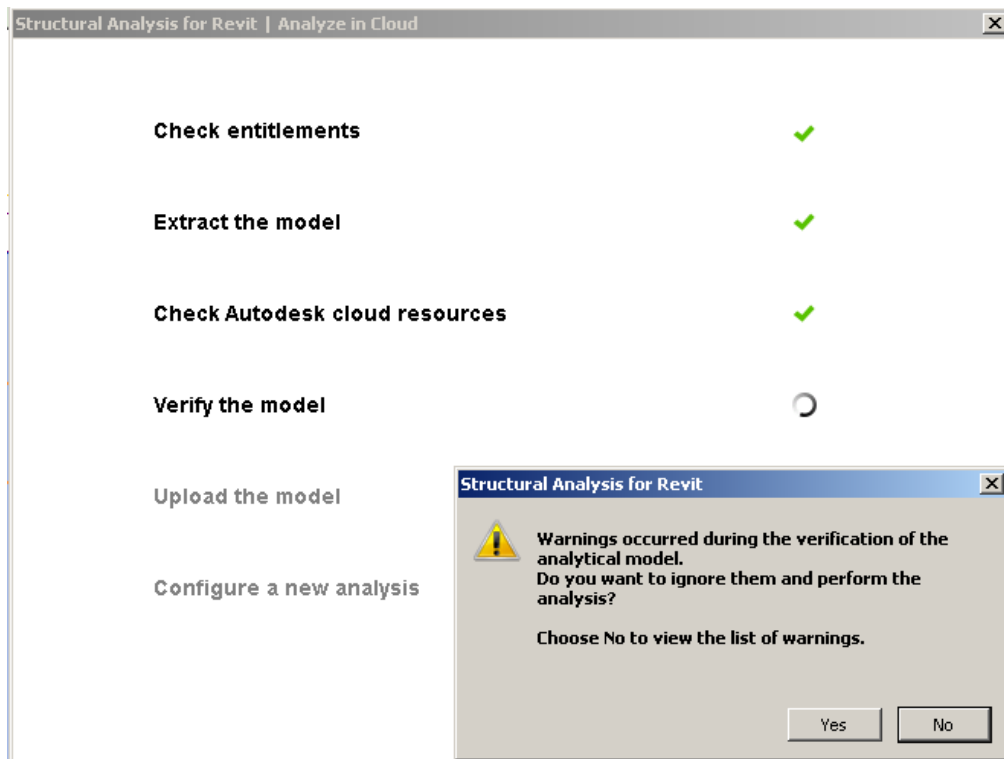
You can analyze a portion of a model or the entire model. Selections sets are a very handy tool for helping to manage sub-models.

Once you press this button, you will be asked to sign in to your Autodesk account. This should be your Single Sign-On account which is associated with available cloud credits.

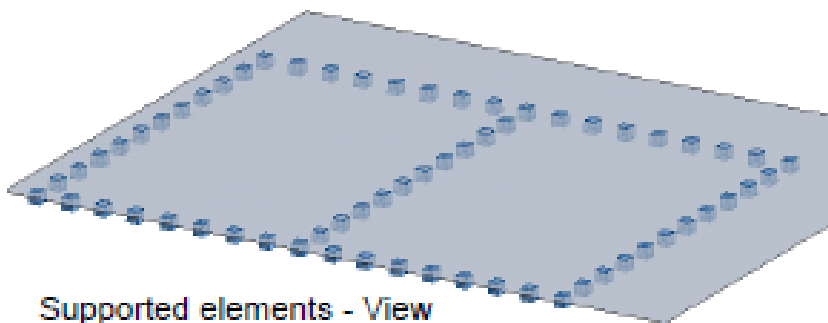


To learn more about cloud credits, follow this [link](#)

Once you sign in, the Analyze in Cloud tool will begin to check the model, your cloud credits, and, if everything is ok, upload it to the cloud servers. In the case that warnings are detected in your model, you will receive a warning similar to this:



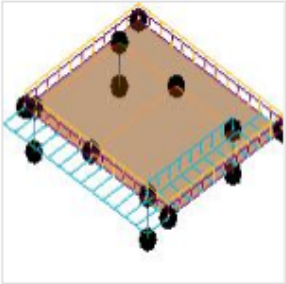
Other things to note: When running only selected elements, Robot will look at co-planar or attached elements to attempt to derive supporting conditions for that model. For instance, when I select only a floor plate without its supporting beams, Robot looks at those elements and attempts to derive supporting conditions from them as shown here:



Notice the outline of the supporting beams visible in the fixed supports which have replaced them. Beams supported by columns will be replaced by fixed supports at the column locations, etc.

Once all verification, checking, and uploading is completed, you will be shown the web-interface to the analysis management. Here you will configure parameters of the analysis, mostly naming

but also you will have control over meshing quality as well as assumptions about self-weight for the structure:



Analysis powered by [Robot Structural Analysis](#) engine

Project:
T-Beams integral with floor_returned results from Robot run

Model:

Static analysis

Advanced analysis

Analysis name

Add comment

Report template

Simple report ▼

Report name

Analysis profile

Normal ▼

Self-weight options

☒ Add to existing load case

DL1 ▼

☐ Ignore

Cloud credits required: 2

Cloud credits available: 72

[Learn more about cloud credits](#)

Start Analysis

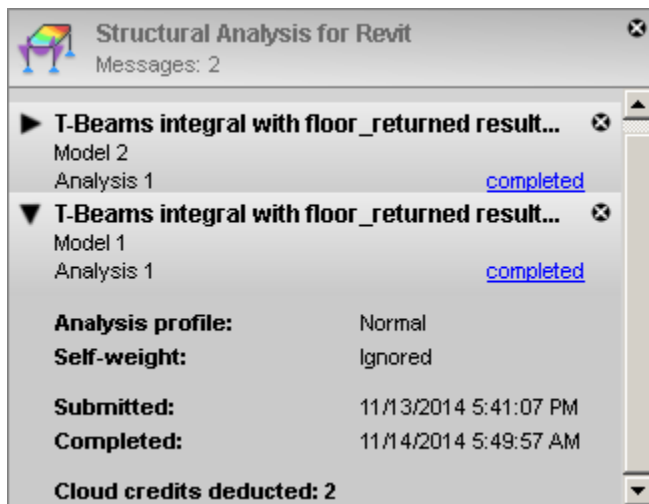
In the Analysis Profile section you can select from a predefined set of mesh refinements: “Draft”, “Normal”, and “Fine”. There are no specifics on actual mesh size for surface elements but the

general idea is that “Draft” increases calculation speed but offers more coarse results whereas “Fine” is a somewhat longer analysis time but finer, more likely to be fully converged, results.

A360 Operation and Reporting

Once you press “Start Analysis” the cloud servers go to work on your model. In theory this should be a lightning fast analysis powered by massive cloud servers; however, the reality is a bit more that the true benefit at this point, in addition to driving the downstream analysis and design of structural elements, the cloud analysis offloads analysis work from your desktop and allows you to continue working with your Revit model while the analysis runs.

In your system tray, you will see the “Structural Analysis for Revit” icon:  This icon allows you to access real-time reporting about the current status of any analyses you are running:



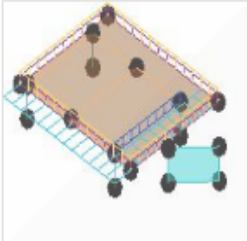
Once completed, you can open the web-portal by clicking on “completed” (you can also access the web-portal while the analysis is running to manage previous analysis runs)


AUTODESK® 360 | Structural Analysis

[Dashboard](#)
[Projects](#)

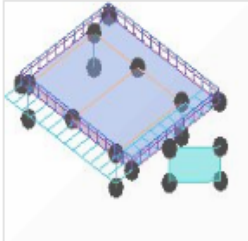
Analyses

Successful [See all](#)



Analysis 1

[Model 2](#)



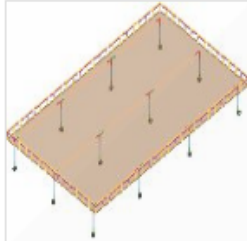
Analysis 1

[Model 1](#)



Analysis 1

[Model 1](#)



Analysis 1

[Model 1](#)

In progress [See all](#)

Currently, there are no analyses in progress.

Failed [See all](#)



InitialAnalysisTbeam

[T-Beams integral with floor](#)

Selecting any of the analysis runs, you can manage the analyses, launch a 3D viewer for the model, delete the analysis, view status/parameters, and also view the analysis report. The report contains common information for a structural analysis: Model setup, load cases/combinations, support conditions, reactions, member forces, and displacements including diagrams and maps of forces on surfaces.

Learning more about A360 Analysis

There are several help files provided by Autodesk to help you learn more about the capabilities and operation of A360 Analysis. I think you'll find that it's a great new capability in Revit especially for early stage analysis and we hope that Autodesk will continue to improve and fill out this functionality with new capabilities, especially for running advanced analyses which could really benefit from the power of the cloud such as time-history type analyses.

Here's a link to the Autodesk help files for A360 Analysis.

<http://help.autodesk.com/view/RVT/2015/ENU/?guid=GUID-7A75586A-129A-4525-8E43-CB087FBD170F>

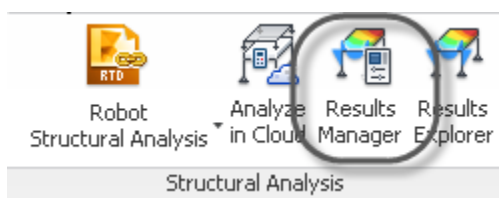
Results Explorer in Revit

As we have discussed, the real key for facilitating engineering workflows inside of Revit is results storage in Revit. Being able to visualize your analytical model performance directly in Revit is a powerful tool and gives you an unprecedented level of confidence that your analysis is truly connected to your model and not at arm's length as it always has been in the past.

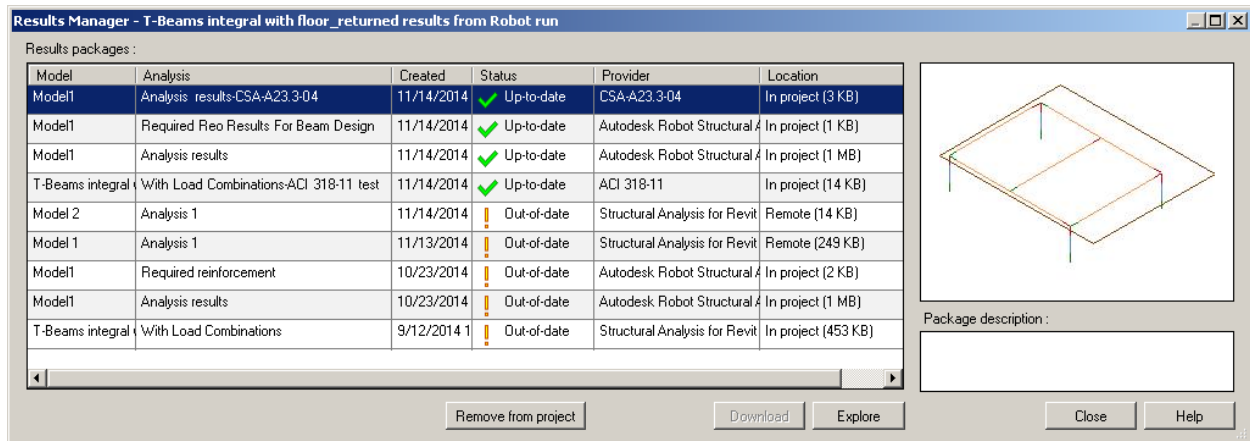
Getting Results Into Revit

Results can be stored by any 3rd party analysis application if they have added that functionality. The Structural Analysis SDK includes functionality specifically for storing and retrieving results packages in/from the Revit model. Autodesk A360 Structural Analysis provides this functionality as well as Robot Structural Analysis.

Downloading Cloud Results: Once a cloud analysis is complete, the results are still on the cloud servers. You will need to download them into your Revit model by using the Results Manager:



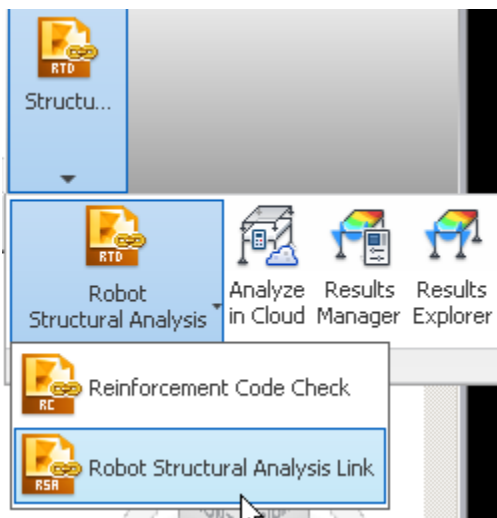
This tool provides visibility into all analysis result packages stored in, or available to the current Revit model:



From here you can clean up old packages by deleting them if you wish, for packages with results currently in the project you can launch “Exploration” from here by using the “Explore” button and for results still in the cloud, you can download them to the project with the “Download” button.

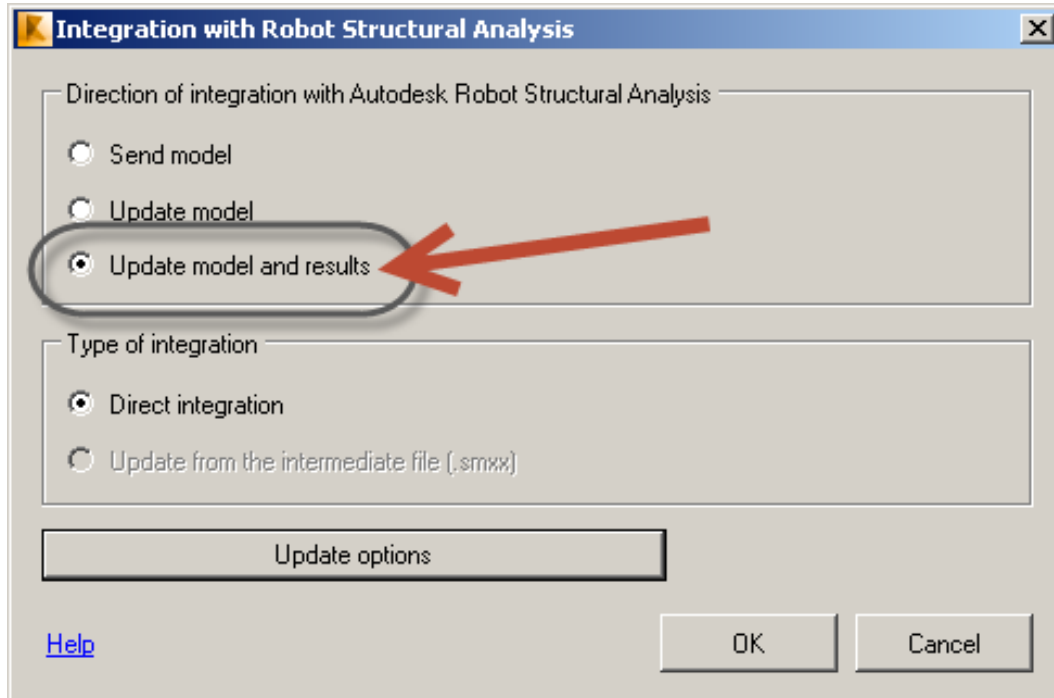
The results will increase the size of your model but not in a way that is bad for Revit performance. The largest factors in Revit performance are number of elements in the model, this data lives alongside the element data and does not tend to impact performance in my experience.

You can also have results returned to Revit from the link with Robot Structural Analysis which you will still need to use for analyses beyond linear static analysis. Robot integration is available also on the “Structural Analysis” panel of the Analyze Tab:



In the “Robot Structural Analysis Link” you have the options to “Update model and results” which will import results for elements analyzed in Robot Structural Analysis back into the Revit

model for both exploration as well as driving downstream analysis, design, and code-checking applications. The key is getting results from analysis into the model, where they come from, is somewhat arbitrary as long as the source is a legitimate structural analysis of course.



Exploring Results

You can either launch Results Explorer from the Structural Analysis panel on the Analyze tab or from the Results Manager with the “Explore” button. Results Explorer will give you the ability to view all results contained in a result package including by load case/combination and results for linear elements as well as maps of calculated quantities on surfaces.

The explorer dialog offers you the options to switch between analysis packages and the currently displayed load case/combination. You can also select which results should be displayed on the model. If you expand the dialog, you will have even more control over the display of the results including the style of result display for the quantity, the units, and the display scale:

Results for Analytical Model | T-Beams integral with floor_ returned results from Robot run

Model: T-Beams integral with floor2

Analysis: With Load Combinations

Load Case: DL + LL + WL2

The selected model is inconsistent with this project.
The current analytical model differs from the model for which the analysis was performed. Results are out of date.

Result	Style	Units	Scale	Max	Min
Forces					
Reaction FX	Vector forces	< kip >	100%	N/A	N/A
Reaction FY	Vector forces	< kip >	100%	N/A	N/A
Reaction FZ	Vector forces	< kip >	100%	N/A	N/A
Moments					
Reaction MX	Vector moments	< kip-ft >	100%	N/A	N/A
Reaction MY	Vector moments	< kip-ft >	100%	N/A	N/A
Reaction MZ	Vector moments	< kip-ft >	100%	N/A	N/A
Displacements					
Displacement UX	Vector displacements	< in >	100%	N/A	N/A
Displacement UY	Vector displacements	< in >	100%	N/A	N/A
Displacement UZ	Vector displacements	< in >	100%	N/A	N/A
Rotation					
Rotation RX	Vector rotation	< rad >	100%	N/A	N/A
Rotation RY	Vector rotation	< rad >	100%	N/A	N/A
Rotation RZ	Vector rotation	< rad >	100%	N/A	N/A
Results for members					
Forces					
Forces Fx	Diagram forces	< kip >	100%	N/A	N/A

☐ Local coordinate system for members

☐ Local coordinate system for surfaces

☐ Global coordinate system

Apply

Close

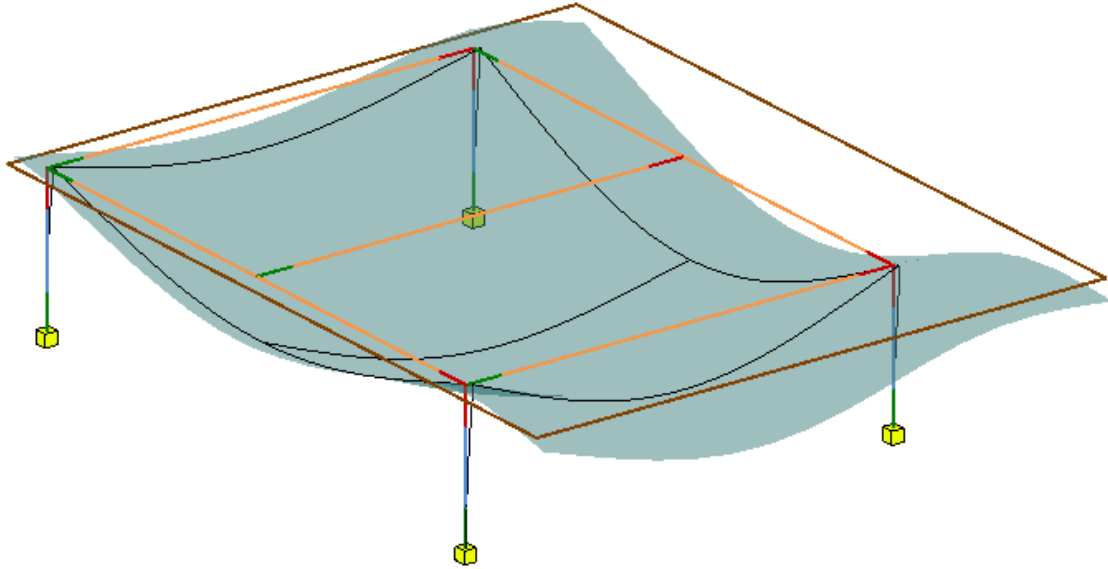
Help

<

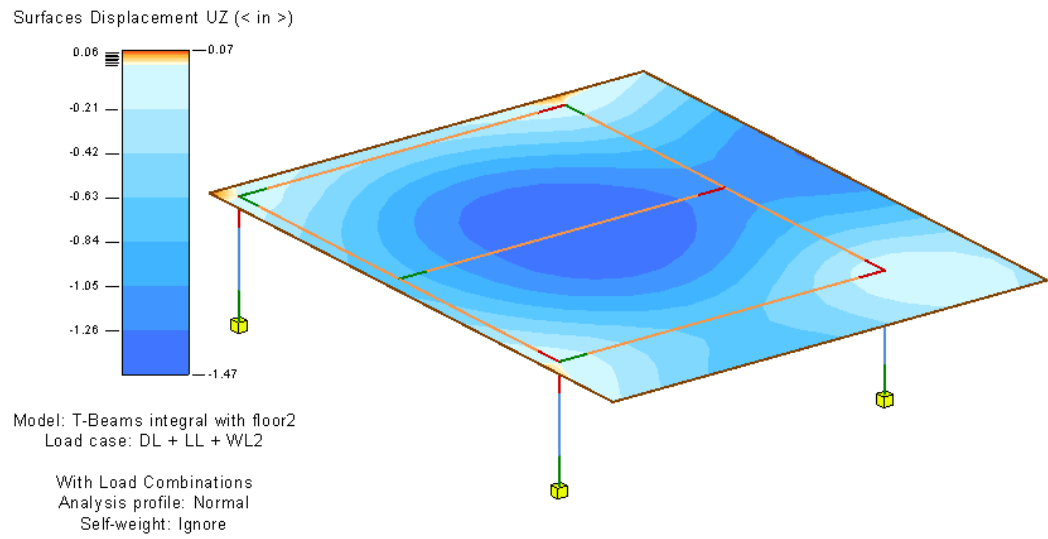
Results are out of date.

Imagine, all that functionality hiding in the Structural Analysis Toolkit! Let's take a look at some of the displays available for this simple model:

Deflected Shape:



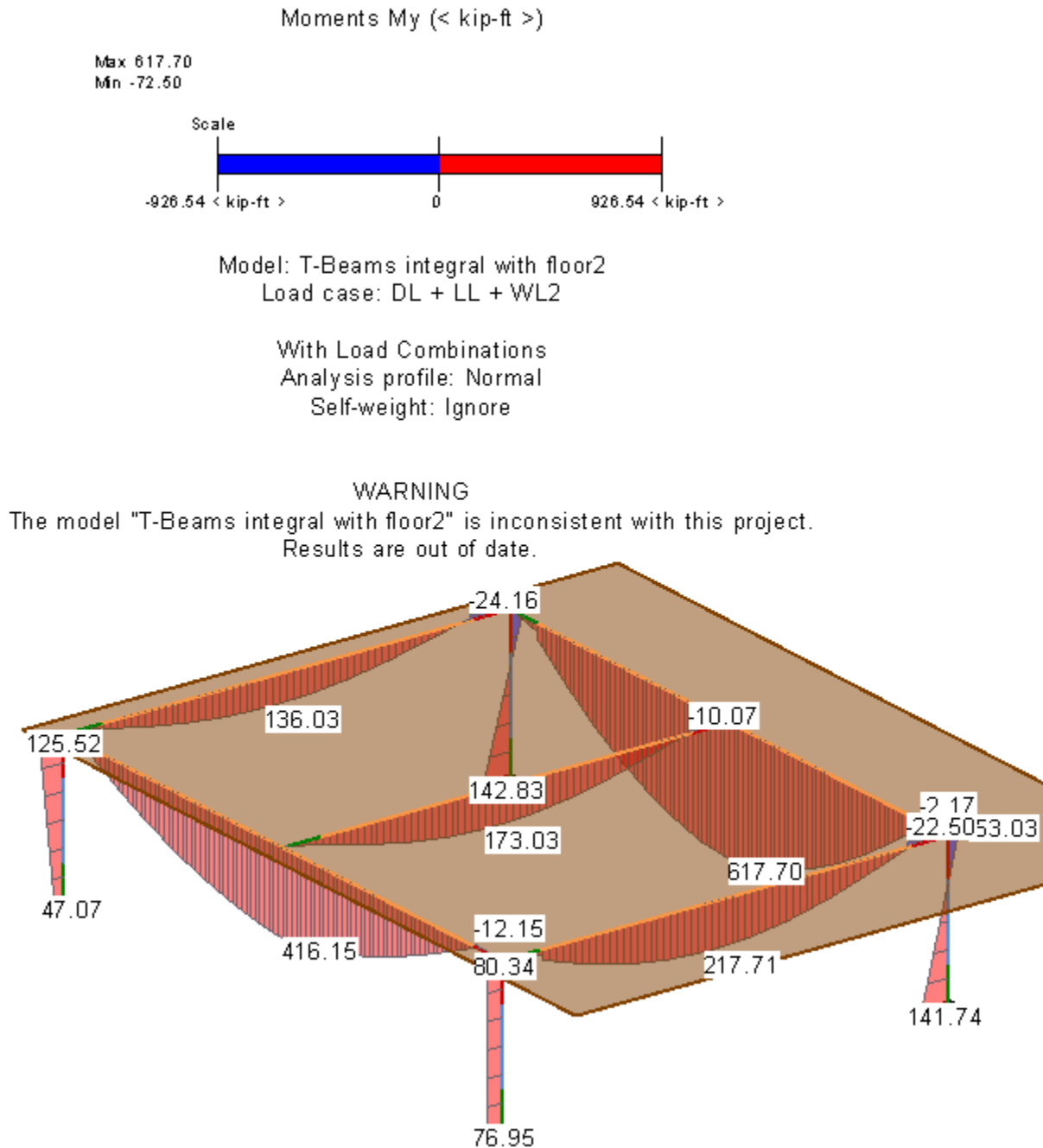
Maps on Surfaces



WARNING

The model "T-Beams integral with floor2" is inconsistent with this project.
Results are out of date.

Diagrams on Linear Elements:



Reaction at Supports:

Reaction MY (< kip-ft >)



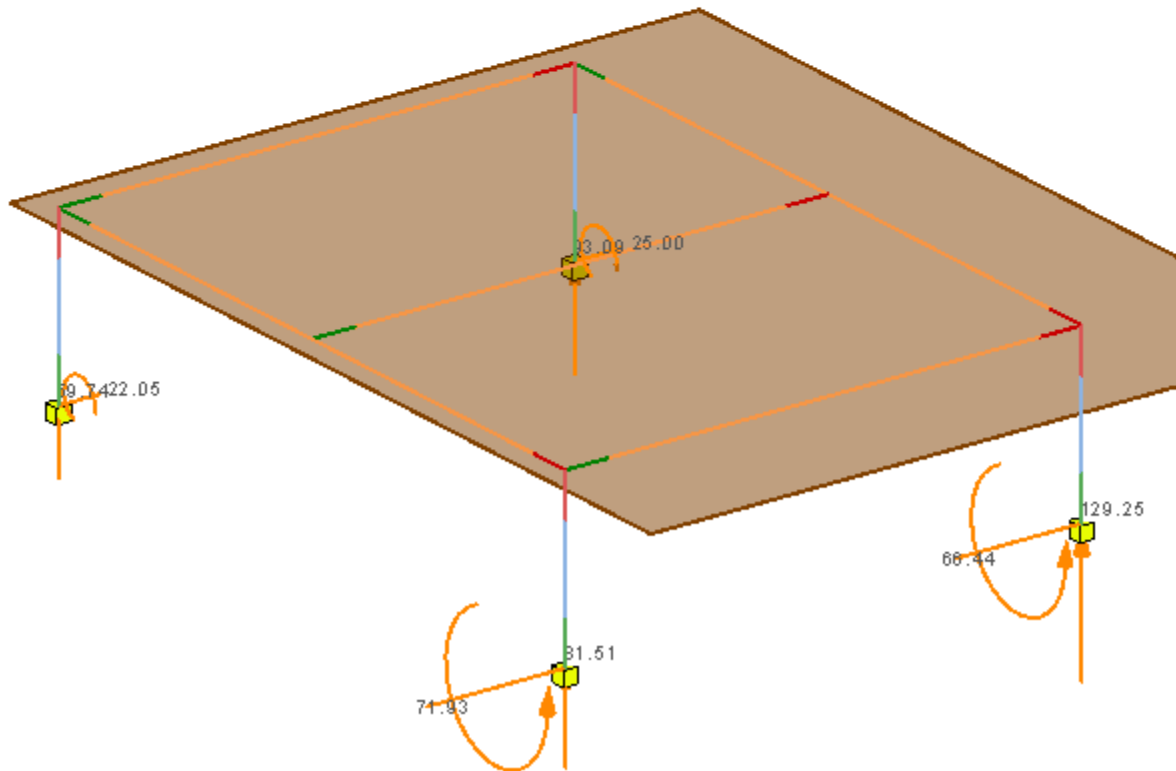
Scale
-0.74 <0kip-ft >

Model: T-Beams integral with floor2
Load case: DL + LL + WL2

With Load Combinations
Analysis profile: Normal
Self-weight: Ignore

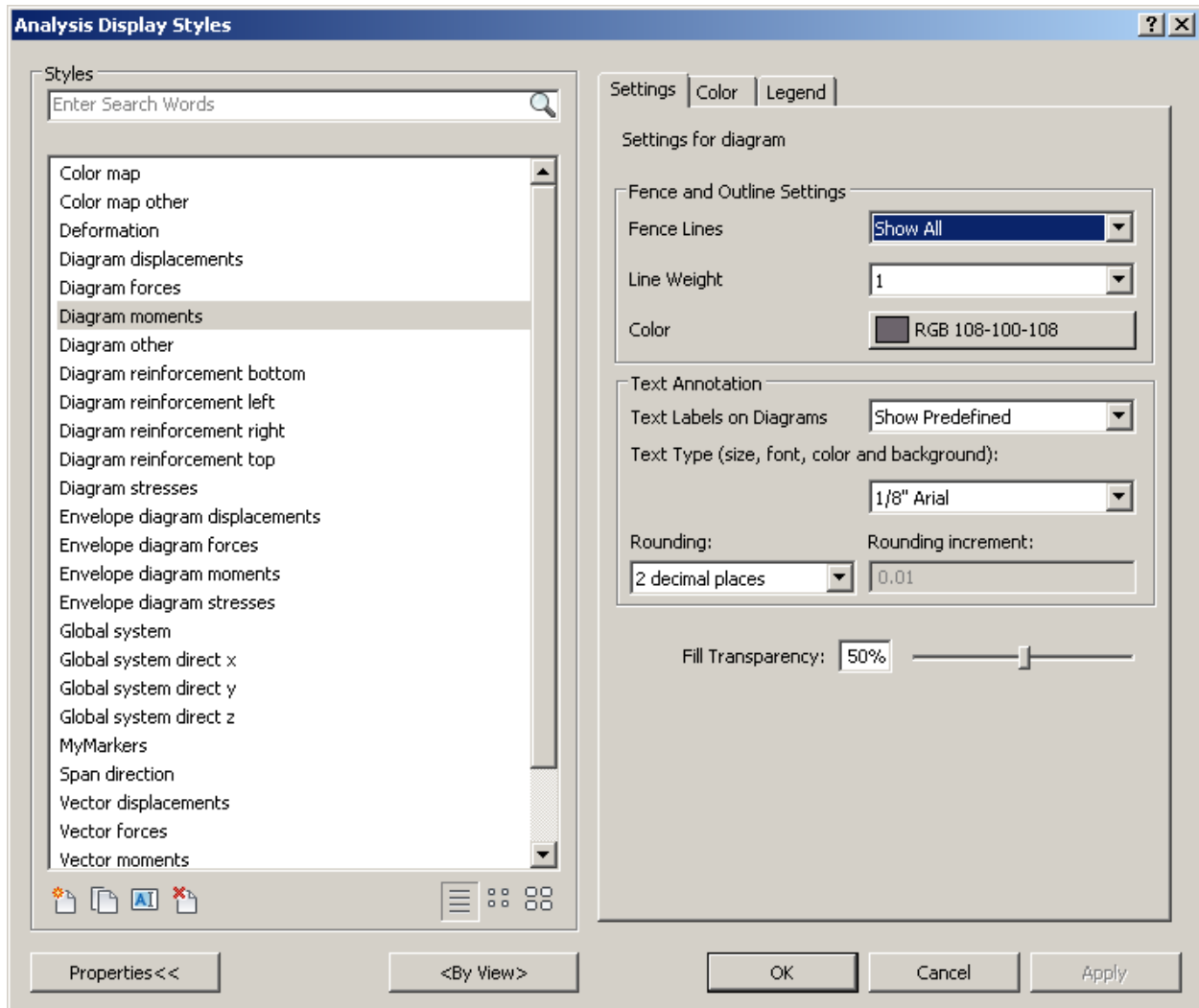
WARNING

The model "T-Beams integral with floor2" is inconsistent with this project.
Results are out of date.

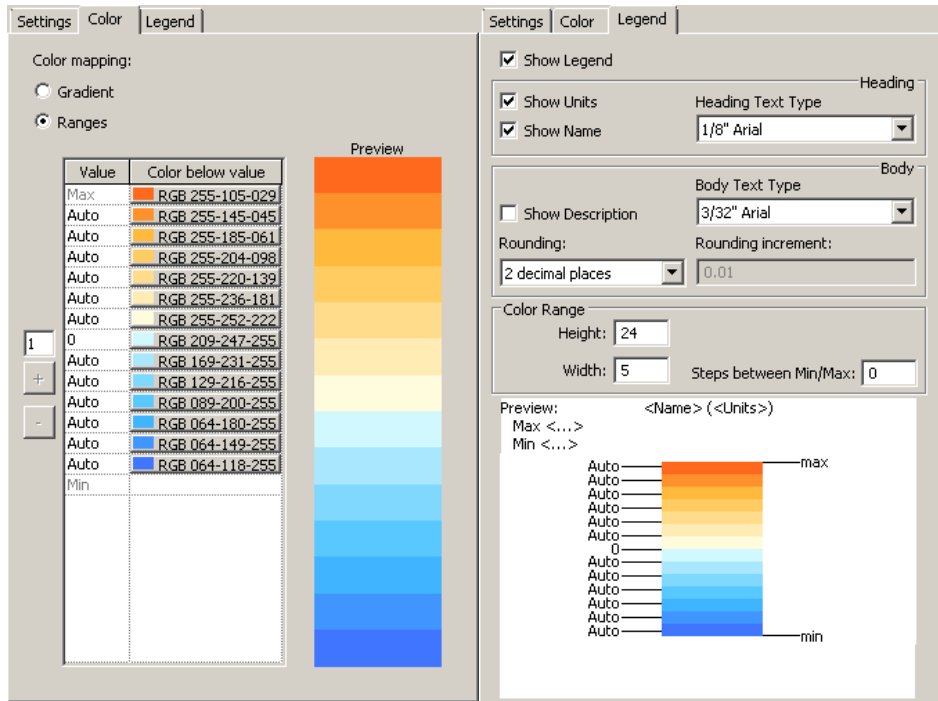


The Results Explorer is a user-friendly interface to display results stored in results packages in Revit. In terms of the display of the results themselves, you have a surprising amount of control over the display of each type of result. If you take a look at the parameters of the view in which you are displaying results you'll notice an "Analysis Display Style" and if you click the ellipsis button you will see all the currently displayed quantities in the view where you can control things like the visibility of the results in the view, the display style associated with the quantity, the units for display, as well as the scale. This is very similar to the information shown in the Results Explorer expanded view.

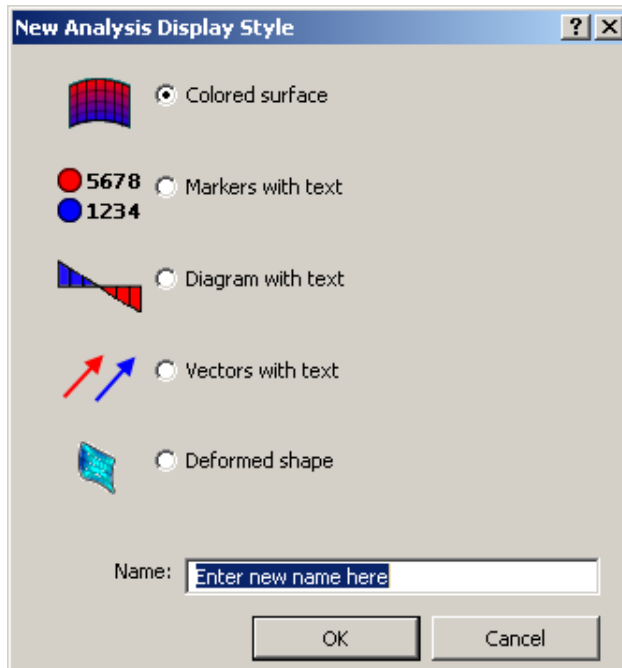
The Analysis Display Style is where the real customization capabilities are and where you might want to tweak things to produce documentation quality graphics for your reports or to store with your model. You can click the ellipsis button in the display style column or you can alternatively access all display styles from the "Manage" tab under "Analysis Display Styles":



Notice that you have control over diagram style, annotation text size/type, transparency of the diagram fill and in the other tabs, you can control colors used in the legend as well as the entire legend and its settings too:



There are different display style categories and you can create your own to use in display. In the Analysis Display Styles dialog, click the new display style button to see what kinds of display styles are available (Note that they will need to be compatible in some way with the data to be displayed... what do “reaction” results look like as color maps??)



These are the display styles available and each has somewhat unique settings available to help you configure the display as you might desire. Take some time to learn more about this and you'll quickly find that this is a powerful visualization tool for structural analysis results.

Examples of integrated concrete design applications for Revit

Now that we have analysis results in Revit and have learned how to manage those results and visualize them, we want to take a look at some examples of applications that can (or will in the future) leverage these results for a fully integrated concrete design workflow in Revit. The first two solutions are geared towards full 3D rebar detailing based on requirements analysis and the last one is the first tool built on the Autodesk Code Checking Framework which intends to address a unique North American concrete design workflow.

Revit Extensions Integration with Required Reinforcing

The first solution is built on the Extensions for Revit Structure. This solution takes advantage of a unique type of results package that can be stored in the Revit model called a “Required Reinforcing” results package. The idea behind this type of results package is to provide for the storage of calculations of required areas of steel reinforcing for members top/bottom/left/right. Robot Structural Analysis Link can store these types of result packages in Revit which you can generate in Robot by configuring and running a required reinforcing analysis on your concrete structural members. When you configure the parameters for updating the Revit model from Robot, you will notice that if you specify “Update model and analysis results” you will be provided with a dialog similar to this which allows you to configure the naming of not only the analysis results but also the current required reinforcing analysis results in Robot:

Update Results

☒ Analysis results package

Model name: Model1

Analysis name: Analysis results

Package description: [Enter additional description for analysis results package here]

☒ Required reinforcement results package

Model name: Model1

Analysis name: Required reinforcement

Package description: [Enter additional description for required reinforcement results package here]

[Help](#) OK Cancel

Selecting and configuring naming for required reinforcement results package storage will give you access to these results to inform your design.

The Revit extensions, part of the Subscription Advantage Pack, are a set of pretty amazing little tools to help automatically generate real 3D rebar in your model. For the most part they will help you model standard reinforcement but now that there is support for required reinforcing results packages, we can now take advantage of these results to inform our reinforcement layout and modeling. In actuality, this functionality has been available with the extensions for some time though the internal mechanisms of storage have been refined and streamlined. Selecting a beam, for instance, and starting the reinforcing of beams extensions, we see a familiar interface where we can configure top bars, bottom bars, stirrups, and additional bars.

Reinforcement of beams

File Help

Geometry

Stirrups

Stirrup distribution

Bars - main

Additional top bars

Additional bottom bars

Bar division

Reinforcement areas

Identification

Family name: Concrete-Rectangular Beam

Type: 16 x 32

Geometric parameters

Number of spans: 1

b = 1' - 4"

h = 2' - 8"

l_b = 42' - 0"

h_s = 0' - 6"

Element parameters

Number of beams: 1

Level: Level 2

Offset: e_0 = 0' - 0"

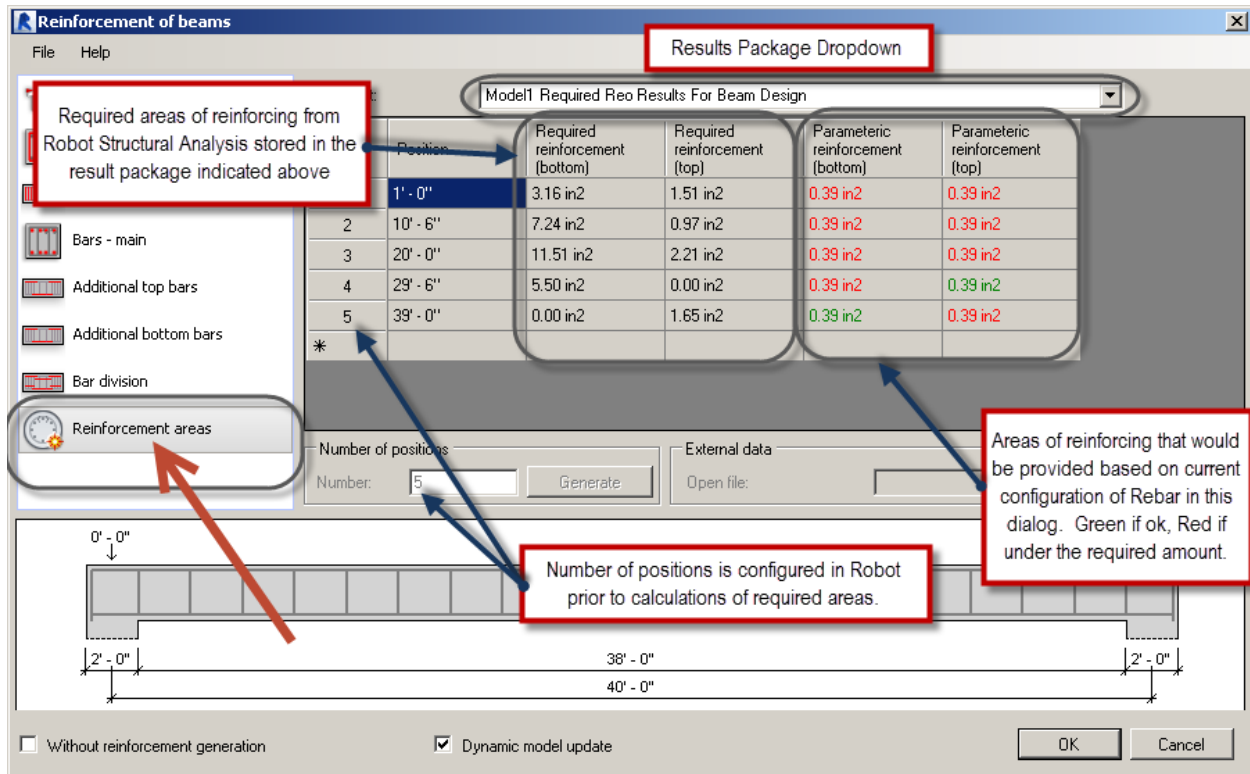
Offset: e_1 = 0' - 0"

☐ Save data without reinforcement generation

☐ Without reinforcement generation ☒ Dynamic model update

OK Cancel

One thing you may or may not have noticed at the bottom of this dialog is a tab for “Reinforcement areas”. This often overlooked tab is where the magic of integration happens in this workflow. If we click on that tab, we’ll see a table with the required areas of reinforcing, the currently provided reinforcing at those same sections along the member (based on current rebar configuration in this very dialog) with control over which results package is currently displayed:



This is actually a pretty impressive integration of required areas of steel from design code based analysis with fully automated 3D rebar detailing.

The obvious advantages of this workflow are that your designs are being informed by real code-based calculations of required reinforcing, you are modeling all rebar for much more accurate rebar quantities, you can do reinforcement interference checking to manage complexity of rebar taking into account real bend radii, diameters of elements, and cover spacings. Even this out-of-the-box+Revit Extensions workflow is fairly compelling, especially if you would like to model 3D rebar in your model.

SOFiSTiK Reinforcement Generation (Labs)

The second solution we will investigate utilizes not only the Revit API but also harnesses the Analysis Visualization Framework to provide very powerful and compelling reinforcement design and analysis inside Revit. This first example is currently a "labs" style offering meaning that it is in the early stages of development similar to a beta release. It currently cannot leverage result packages stored in Revit though the hope is that it will be able to access those result packages in the release version of the tool. This tool is made by SOFiSTiK a well-respected analysis software provider based in Germany. SOFiSTiK has been providing advanced analysis and design since 1987 and has been providing analysis for Revit via a link they have developed. In 2010 SOFiSTiK launched a subsidiary company BIMOTiON to focus on driving analysis and

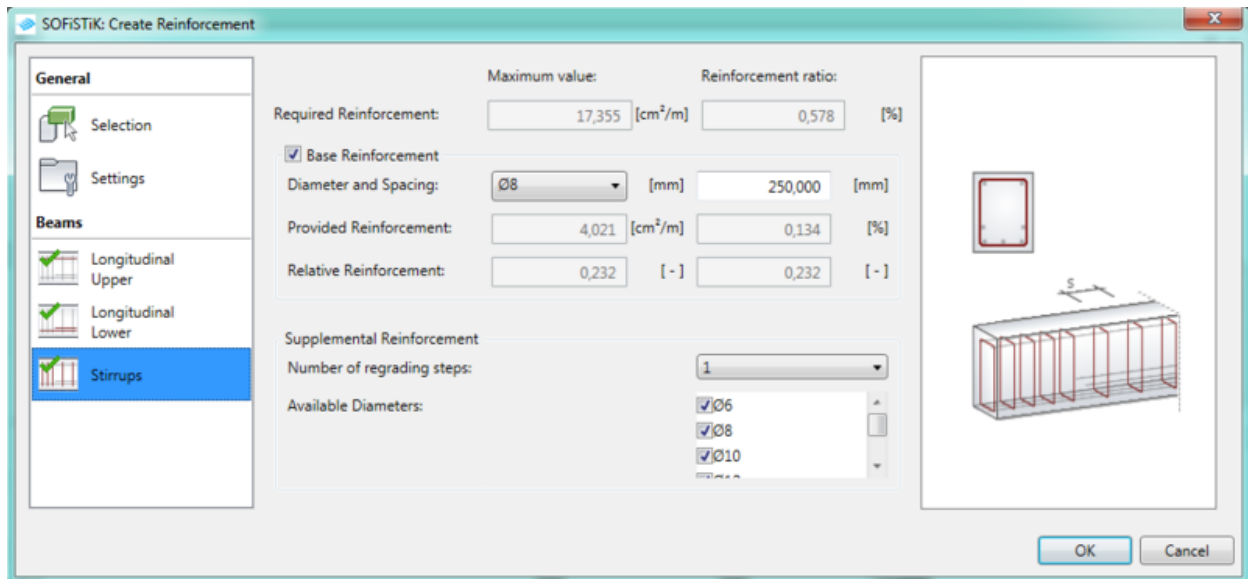
design technology into fully integrated solutions for structural engineering and design. I would encourage you to check them out on the web at www.sofistik.com/en

The SOFiSTiK Reinforcement Generation tool is working to provide the following functionalities on top of Revit Structure powered by SOFiSTiK's own powerful and full featured FEA analysis software.

Results Storage: Reinforcement Generation can import analysis and design results from SOFiSTiK's CDB database and store those results as Revit Result Packages.

Again, all of the fully integrated analysis and design functionality relies on using the latest technology for storing results packages in the Revit model. SOFiSTiK's Reinforcement Generation tool interoperates with their FEA tool to create the results packages necessary in Revit to drive this tool's amazing capabilities.

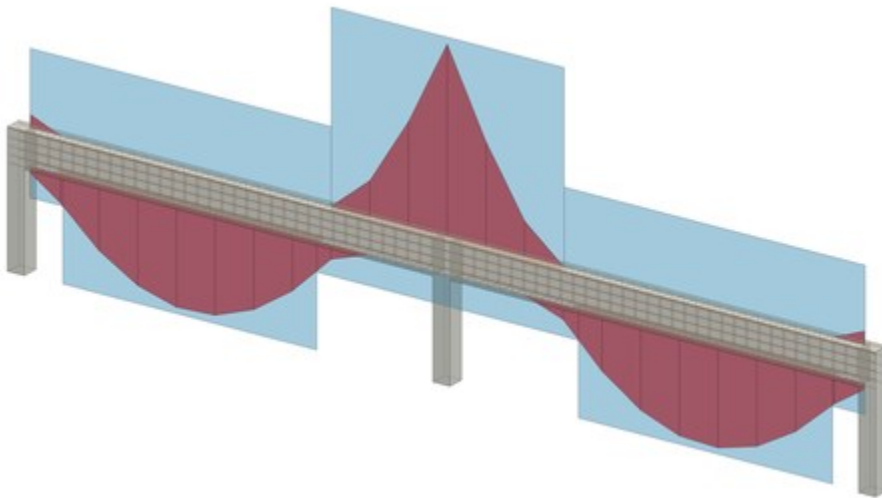
Automated Reinforcement Generation: Automatic generation of reinforcement for beams, columns, walls and slabs is possible with this tool and it fully utilizes results from analysis and design. It currently only works with SOFiSTiK FEA results but in the future may be able to consume results packages generated by other tools.



You'll notice that this dialog somewhat resembles the Revit Extensions in that it offers the user a dialog to configure reinforcement generation. This reinforcement generation is performed based on required areas of reinforcing calculated by SOFiSTiK's FEA software. You will notice the required reinforcement values at the top of this tab of the dialog.

Visualization of Capacity and Demand: The last main feature of Reinforcement Generation is the ability to visualize required (theoretical) and capacity of modeled (physical) reinforcing.

This particularly powerful feature included with SOFiSTiK Reinforcement Generation is visualization of not only required but also provided capacity using the Autodesk Analysis Visualization Framework to display multiple results on a member at the same time. Here is a picture of the required reinforcing (in Red) with a diagram of provided capacity overlaid in light blue:



This capacity diagram is based on the actual rebar placed in 3D in the model! If that wasn't exciting enough as it is, the coolest thing about this tool is that it fully utilizes the Revit API's Dynamic Update technology and responds almost instantly to actual changes in the reinforcing... You can, literally, modify the reinforcing of this beam member, lengthen bars, change bar diameters, etc. and the diagram nearly instantly updates in response. You can fully investigate bar length cut-offs and configurations live and real-time. This is concrete design functionality and BIM integration like you've never seen before!

SOFiSTiK Reinforcement Generation is a free download for Revit 2015 from the Revit Exchange AppStore: Check it out!

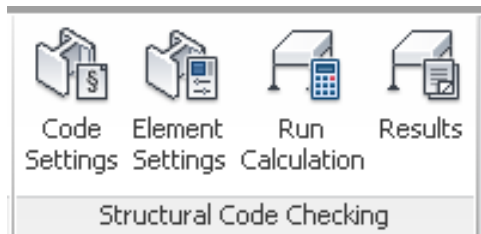
Marsh API Concrete Code Checking per ACI 318-11 and CSA-A23.3-04

The last solution is a tool that is still in the development phases and is pioneering use of the Autodesk Code Checking Framework which was published by Autodesk along with Revit 2014. As we have previously mentioned, the Code Checking Framework provides a full featured framework to facilitate per member code checking and design in Revit. We'll first look at the

user interface portion of this powerful toolkit in the way Marsh API has implemented it and then we'll look at some of the sample output and talk about future directions.

The Autodesk Code Checking Framework

Once you have installed the Structural Analysis Toolkit for Revit that there is a new panel added to the Analyze tab called "Structural Code Checking" which contains the following tools:



These tools work collectively to provide a consistent user experience for users of any future code checking application. Any number of code checking applications can be developed for Revit and this is the main idea of the code checking framework: Autodesk would like to encourage development of code checking utilities by experts around the world. This is a bold commitment to democratization of the code-checking and structural design workflow in Revit.

Each code checking application is installed as an external application built on the Revit API in conjunction with the Structural Analysis SDK. Each of these tools (Code Settings, Element Settings, Run Calculations, and Results) are designed to provide a common location and consistent user experience when interacting with one or more code checking applications regardless of who develops the code-checking/design functionality.

Marsh API Concrete Required Reinforcing

In order to give you an idea of how each of these tools works and provide an example of an implementation, we will look at how each of these Code Checking Framework components work together to facilitate a code checking application.

Code Settings: provides access to configure overall code parameters for any code currently installed in Revit. Clicking on Code Settings brings up the Code Settings dialog where you have access to select which, of the installed code checks, you want to enable (check boxes on the left) and to access the settings for any particular code checking application by selecting the application's entry on the left hand side of the dialog:

Code Settings

☒ ACI 318-11
☐ CSA-A23.3-04

Description
In-house build# 2015.0.5408.37347

Calculation points
Division type: ☒ Constant number of points
☐ Constant segment length
Number of points: 11
☐ Add points at maximum calculated forces

Tolerances
Axial Force Tolerance: 0.10 kip
Bending Moment Tolerance: 0.10 kip-ft

Analysis
Input package: ⚠ Results are out of date
Model name: T-Beams integral with floor2
Analysis name: With Load Combinations
Description: [empty]
Output package:
Model name: T-Beams integral with floor2
Analysis name: With Load Combinations-ACI 318-11 test
Description: [empty]

Load cases and combinations
Load cases: [empty]
Load combinations: DL; DL + LL; DL + LL + WL; DL + LL + WL2; DL + WL;

OK Cancel

While most of the settings in this dialog are particular to the application two of the settings are provided as part of the code checking framework to facilitate selection of input package (results package) and which load cases/combinations are active for design/code-check.

As additional code checking applications are installed you would see them here in this dialog and be able to control their overall settings. Only code check/design applications checked will

be executed when calculation are run. This allows the user to select multiple code checking applications to compare results or simply to design check different materials.

Element Settings: This is where design/code checking parameters specific to individual elements are configured and managed. For each installed code checking application there will be a tab at the top of this dialog to allow you to configure settings particular to an individual design code. As each code checking application may be unique and each code supported may have unique settings this layout is completely custom for each code checking application.

Element settings are organized by material and category. Dropdowns at the top of the dialog filter and control display of settings based on both of these items. In this example below, you see the settings for “Default3” which pertain to concrete elements of category “Analytical Beams”. These settings are all part of the code checking application for ACI 318-11 and CSA-A23.3-04 that we have created. There could be as many code checking applications that implement ACI318-11 as the user is interested in obtaining. This also means that you could create your own code checking applications internally as we have previously mentioned.

Here we see the settings for Analytical Beams available. The user has control over which member forces will be considered in design, whether the beam will consider slabs above as T-Beam interaction for calculation of required reinforcing, and an initial bar size and material for the longitudinal reinforcing. There are also settings for stirrup bar/leg configuration and how the user would like to treat torsion as well as how shear should be treated near supports, all pretty standard per element controls for performing calculations of required reinforcing, and all governed by the installed code-checking application in question:

Element Settings

Material: Concrete Category: Analytical Beams

ACI 318-11 CSA-A23.3-04

Default3

Calculation Parameters

Calculation for:

- ☐ Axial load
- ☐ Lateral shear force
- ☒ Shear force
- ☐ Torsional moment
- ☒ Bending moment
- ☐ Lateral bending moment

Creep coefficient: 2

Slab-Beam Interaction:

Flange Width/Slab Depth: 4

Longitudinal reinforcement

Material: Rebar, ASTM A615, Grade 60

Minimum yield stress: 60.00 ksi

Rebar bar: #4

Deformation: Deformed

Bar diameter: 0.50"

☐ Use symmetric reinforcement

Eccentricity <: 0.25

Transverse reinforcement

Stirrup Legs Vertical: 3

Material: Rebar, ASTM A615, Grade 60

Minimum yield stress: 60.00 ksi

Rebar bar: #4

Deformation: Deformed

Bar diameter: 0.50"

Other

Width of end support for beam: 0' - 0"

☐ Use Shear at d from Face of Support

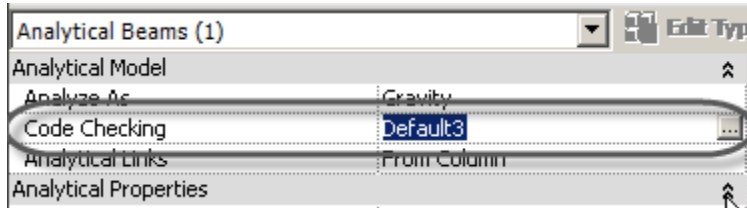
☐ Equilibrium Torsion per 11.5.2.1

OK Cancel

**Element
settings
configurations
listed here**

**Tabbed sheets
allow
configuration
of element
settings**

After element settings have been configured, they must be assigned to individual elements. Analytical elements in Revit have a new property as of Revit 2014 called “Code Checking” under “Analytical Model” section of the analytical element properties as shown here:

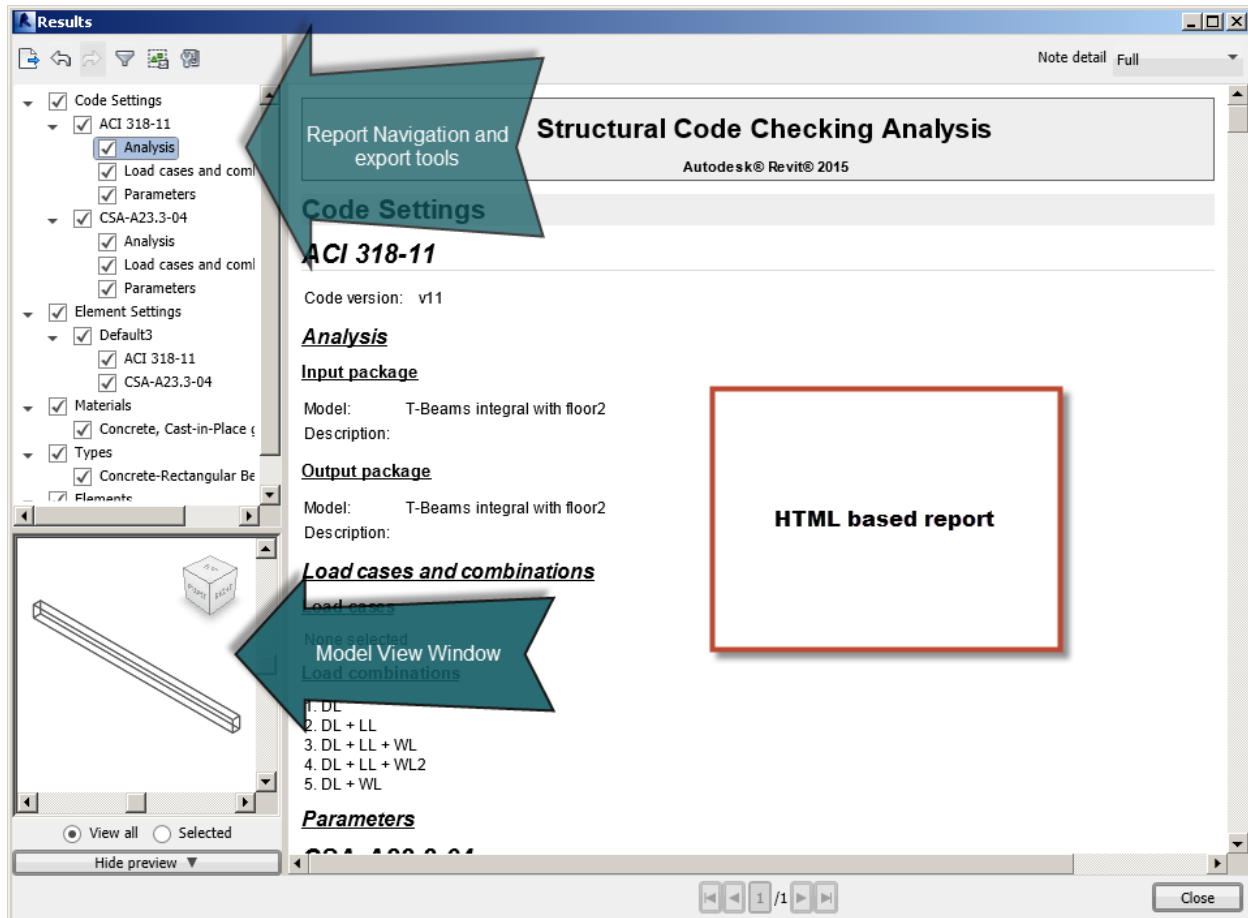


Pressing the ellipsis button will bring up the element settings dialog where you can select an element settings configuration from the list you have previously configured, or even modify or add a new element settings configuration for a particular element. All Revit analytical elements now have a “Code Checking” parameter. Note that it is the Analytical element not the physical element that hosts this parameter.

Run Calculations: Once code settings have been set, the code-checking app selected to run in Code Settings dialog, and Element settings have been configured and assigned to elements of interest, pressing “Run Calculations” will proceed to execute all code-checking applications currently configured for all elements selected by the user.

Each code checking application will perform its own calculations and prepare its own output for you to review and utilize in your documentation. The individual code checking applications may additionally store new results packages in the Revit model to facilitate further downstream analysis, design, review, or additional code-checking.

Here is a sample of the results viewer which is provided by the Code Checking Framework with results from a calculation of a beam for ACI 318-11:



The main components of the report viewer provided by the Code Checking Framework are

1. Report Navigation and Export Tools
 - a. Provide quick navigation through the report which can be somewhat lengthy
 - b. Allow the user to export and save a copy of the HTML report
 - c. Adjust the sorting order of the report
 - d. Filter the report
2. Model View Window
 - a. Elements currently displayed in the HTML report are will be highlighted in the model view area. The elements are all listed by element ID so having some visual feedback in terms of the model view is very helpful in understanding which element you are viewing results for.
3. HTML Report area
 - a. In addition to the wealth of set-up information available in the standard report, individual code-checking applications will generate their own result information for the elements.

Conclusion

We have thoroughly reviewed key technologies which facilitate integrated structural design and code-checking inside Revit. We have looked at the capabilities of A360 analysis and seen how both A360 Analysis as well as Robot Structural Analysis store analysis results directly in the Revit model. We have also seen how this capability is open to any third party application which would like to harness this functionality. We have also reviewed the capabilities and little known functionality included with the Analysis Visualization Framework and looked at the results management and exploration in Revit which is powered by the Analysis Visualization Framework. Finally we have reviewed three different tools what aim to provide concrete reinforcing design directly inside Revit. Two of these applications are dedicated to full realization of 3D reinforcing in the model based on real required reinforcing. One is taking advantage of the new Autodesk Code Checking Framework to deliver advanced solutions for reinforced concrete required areas of steel for North America and Canada in particular.

I expect that we will continue to see development in this area which is ripe for driving fully integrated design and code-checking workflows directly in the Revit environment. No longer can it be said that Revit is a drafting tool, it's a true BIM tool and these key technologies and the tools that leverage them are paving the way to fully realizing the vision of BIM for structural engineering.