

# SCIENGINEER



## Tutorial

### Steel Building – Analysis, Meshing & Results

---

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2016 SCIA Group nv. All rights reserved.

Release: SCIA Engineer 16  
Revision: 06/2016

## Nemetschek SCIA Offices

Latest version is available on: <https://www.scia.net/en/contact/offices>

### Belgium Headquarters

Nemetschek SCIA nv  
Industrieweg 1007  
B-3540 Herk-de-Stad  
Tel.: +32 13 55 17 75  
Fax: +32 13 55 41 75  
E-mail: [info@scia.net](mailto:info@scia.net)

Support Phone  
CAE (SCIA Engineer)  
Tel: +32 13 55 09 90

CAD (Allplan)  
Tel: +32 13 55 09 80

Support e-mail:  
[support@scia.net](mailto:support@scia.net)

### Austria

SCIA Datenservice Ges.m.b.H  
Dresdnerstrasse 68/2/6/9  
A-1200 Wien  
Tel.: +43 1 7433232-11  
Fax: +43 1 7433232-20  
[info@SCIA.at](mailto:info@SCIA.at)

Support  
Tel: +43 1 7433232-12  
E-mail: [support@SCIA-online.com](mailto:support@SCIA-online.com)

### Brazil

Nemetschek SCIA do Brasil  
Rua Dr. Luiz Migliano, 1896 – sala  
702, CEP  
SP -05711-001 São Paulo  
Brasil  
Tel: +55 11 4314-5880  
E-mail: [brasil@SCIA-online.com](mailto:brasil@SCIA-online.com)

### Czech Republic

Nemetschek SCIA s.r.o.  
Evropská 2591/33d  
160 00 Praha 6  
Tel.: +420 226 205 600  
Fax: +420 226 201 673  
E-mail: [info.praha@SCIA.cz](mailto:info.praha@SCIA.cz)

Nemetschek SCIA s.r.o.  
Slavickova 827/1a  
638 00 Brno  
Tel.: +420 530 501 570  
Fax: +420 226 201 673  
E-mail: [info.brno@SCIA.cz](mailto:info.brno@SCIA.cz)

### France

Nemetschek SCIA sarl  
Centre d'Affaires  
29, Grand' Rue  
FR-59100 Roubaix  
France  
Tel.: +33 3.28.33.28.67  
Fax: +33 3.28.33.28.69  
[france@SCIA-online.com](mailto:france@SCIA-online.com)

### Germany

SCIA Software GmbH  
Emil-Figge-Strasse 76-80  
D-44227 Dortmund  
Tel.: +49 231/9742586  
Fax: +49 231/9742587  
[info@SCIA.de](mailto:info@SCIA.de)

### Netherlands

Nemetschek SCIA bv  
Wassenaarweg 40  
NL- 6843 NW Arnhem  
Tel.: +31 26 320 12 30  
Fax: +31 26 320 12 39  
[info@SCIA.nl](mailto:info@SCIA.nl)

### Slovakia

Nemetschek SCIA s.r.o.  
Murgašova 1298/16  
SK - 010 01 Žilina  
Tel.: +421 415 003 070-1  
Fax: +421 415 003 072  
[info@SCIA.sk](mailto:info@SCIA.sk)

### Switzerland

Nemetschek SCIA Switzerland  
Dürenbergstr. 24  
CH-3212 Gurmels  
Tel.: +41 26 341 74 11  
Fax: +41 26 341 74 13  
[info@SCIA.ch](mailto:info@SCIA.ch)

### USA

SCIA Inc. North America  
7150 Riverwood Drive  
Columbia, MD (USA)  
Tel.: +1 443-671-1431  
Fax: +1 410-290-8050  
[usa@scia.net](mailto:usa@scia.net)

# Table of Contents

Table of Contents.....	4
Introduction.....	5
Finite Element Mesh & Analysis .....	6
Mesh Setup .....	6
Generate Mesh.....	7
Solver Setup.....	8
Run Analysis.....	10
Results.....	11
Reactions .....	13
3D Displacement.....	14

# Introduction

This tutorial aims at introducing the user to the SCIA Engineer functionalities which are used in the design of a commercial steel building structure. The functionality in this tutorial are consistent with those found in the Fundamentals Edition of SCIA Engineer. Additionally, in order to properly understand the tutorial, the user must be familiar with the basic principles of the Finite Element Method. For more detailed information on the Finite Element Method and how it is utilized in SCIA Engineer, refer to the [FEM Training Manual](#).

In all, this tutorial consists of 7 parts including:

Part 1 – Modeling

Part 2 – Loading

**Part 3 – Analysis, Meshing & Results**

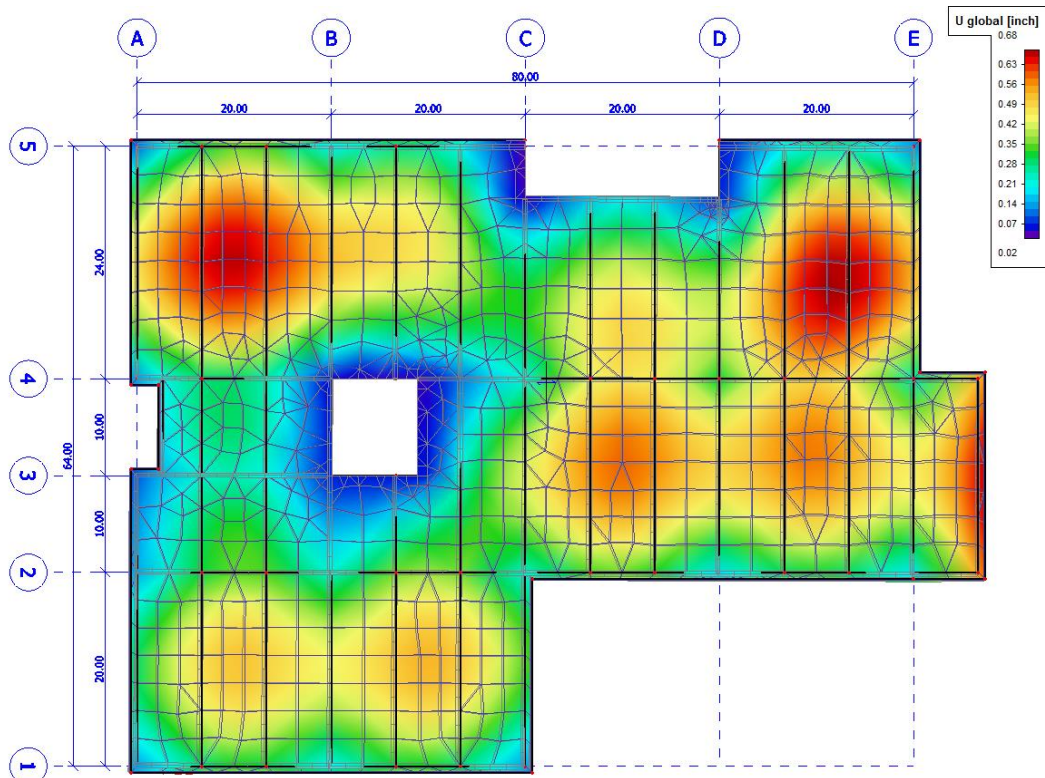
Part 4 – Steel Design (Columns, Braces, Joists)

Part 5 – Composite Steel Design

Part 6 – Concrete Shear Wall Design


Part 7 – Engineering Report

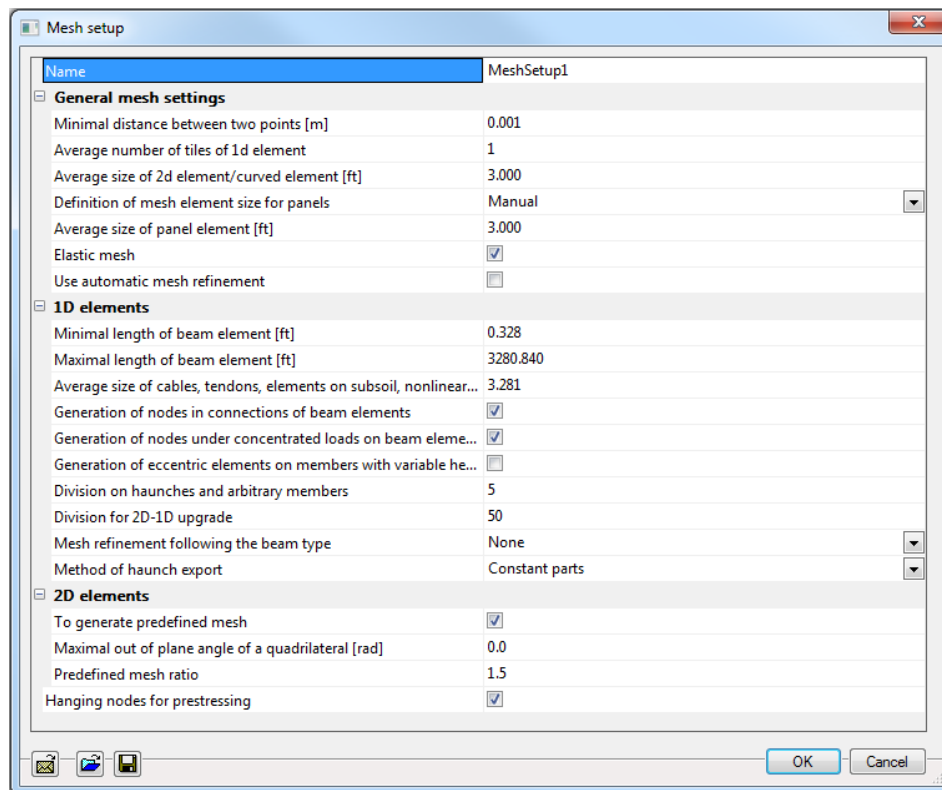
In Part 3 of the Commercial Steel Building tutorial, the functionality and features that are available to create a finite element mesh, run an analysis and evaluate structure results will be explored.



# Finite Element Mesh & Analysis

## Mesh Setup

Now that all load cases, load combinations and loading are added to the structure, the finite element mesh must be generated in order to properly distribute load applied to 2D elements (plates and walls). Before the mesh can be generated, the criteria of the mesh and solver need to be established. To do this, expand the  **Calculation, mesh** group and double click on **mesh setup**. The mesh setup includes settings for the general mesh, 1D elements and 2D elements.



For general models, most of the parameters within the mesh setup can be left as default. Below are mesh settings which should be reviewed before any mesh is generated or analysis is run:

- Average size of 2d element/curved element
  - o Modifies the size of mesh for plate/wall elements
  - o Mesh can be refined using local mesh refinements
  - o For this project a mesh size of **3ft** will be used.
- Definition of mesh element size for panels
  - o Options for Automatic or Manual
  - o Sets the mesh size for the load panels (which distribute wind load)
  - o Set as **manual** for this project



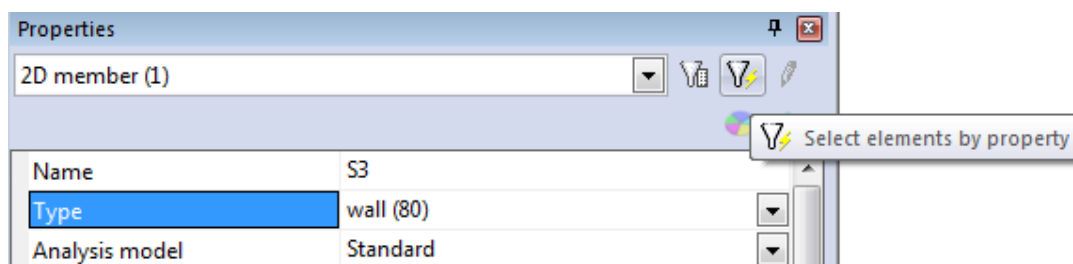
- Average size of panel element
  - o Only applicable when definition of mesh element size is set to **manual**
  - o For this project a panel mesh size of **3ft** will be used.
- Elastic mesh
  - o Checkbox to enable or disable elastic mesh
  - o Elastic mesh allows for the use of **automatic mesh refinement**
  - o Checkbox is enabled for this project
- Use automatic mesh refinement
  - o Checkbox to enable or disable automatic mesh refinement
  - o When enabled **target error percentage** and **load cases for refinement** can be set
  - o Checkbox is disabled for this project

For more information about automatic mesh refinement, visit the SCIA Resource Centre (<http://resources.scia.net/en/home.htm>) and search **automatic mesh refinement**.

## Generate Mesh

With the mesh settings established, the finite element mesh can be generated. Before generating the mesh, it is possible to apply **local mesh refinements**, to various 2D elements within the structure. In this example, the mesh for the concrete walls in the structure will be revised.

First select all concrete walls. The best way to do this is to select any wall element, select the property **type** and then click the **select elements by property tool**. This should select all the walls within the project.

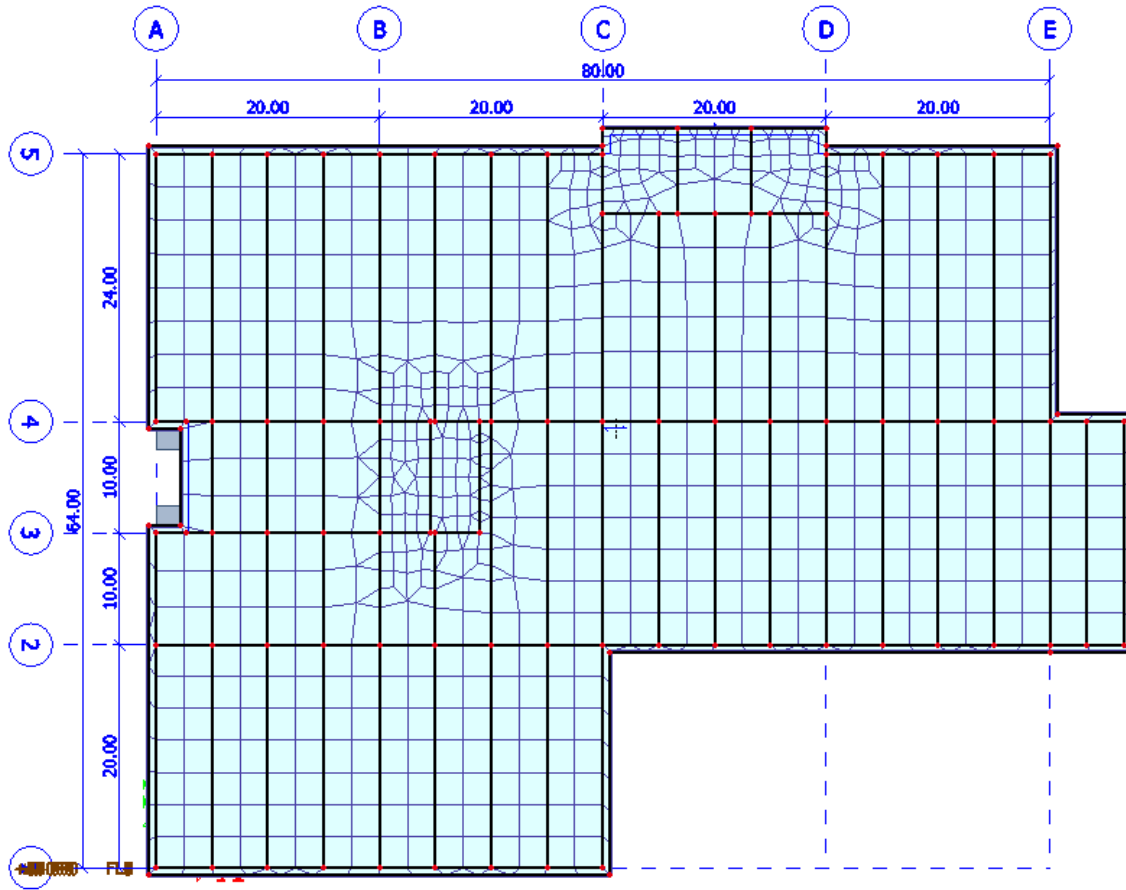


With all the walls selected, double click on **local mesh refinement**, found in the **Calculation, mesh** group. With local mesh refinement open, select **surface mesh refinement** and change the size to **1ft**. The mesh refinement attribute will be added to all the wall elements. To hide the attribute, right click in the workspace and select **Set View Parameters for All**. Under the **Structure** tab uncheck the box for **draw refinements** and then click OK.




With the refinements finished, the mesh can be generated. To do so, double click on **mesh generation** from within the **Calculation, mesh** group. The mesh will be generated and then the visibility of the

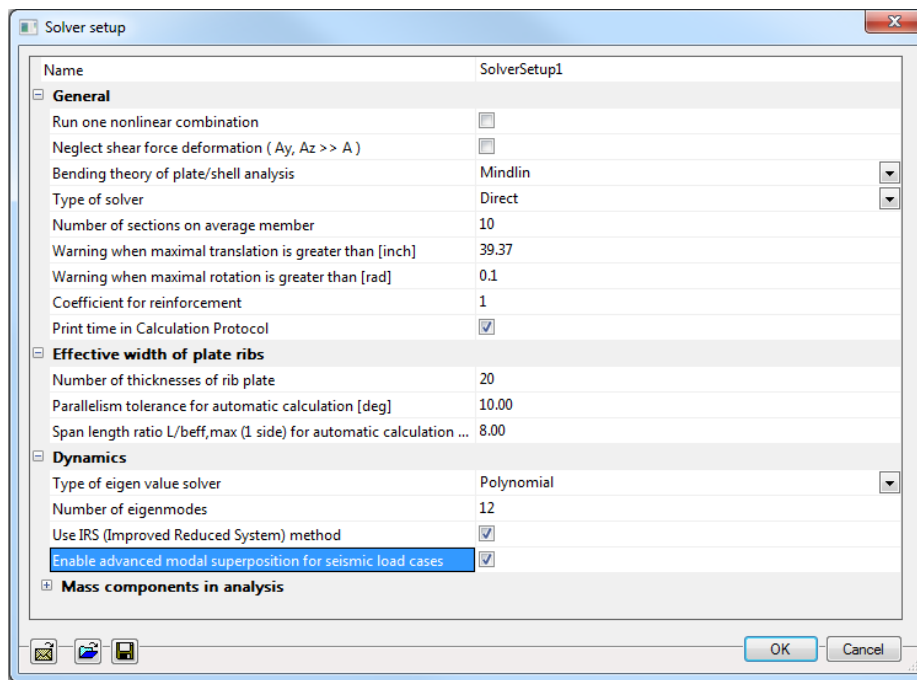
mesh can be enabled by selecting the **Fast Adjustment of View Parameters** button from the toolbar above the command line and then selecting **Mesh** from the **Structure** section.



## Solver Setup

With the mesh settings created, it is also necessary to review the solver settings. To do this, expand the  **Calculation, mesh** group and double click on **solver setup**. The solver setup includes information about the average number of section cuts on a 1D member as well as advanced settings for nonlinear and dynamic analysis.






For most models, the majority of the parameters do not need to be modified and can be left as default. For this project, the only settings that need to be evaluated are those associated with **dynamics** which will be used to generate proper seismic forces. The below dynamic settings should be reviewed before running the analysis:

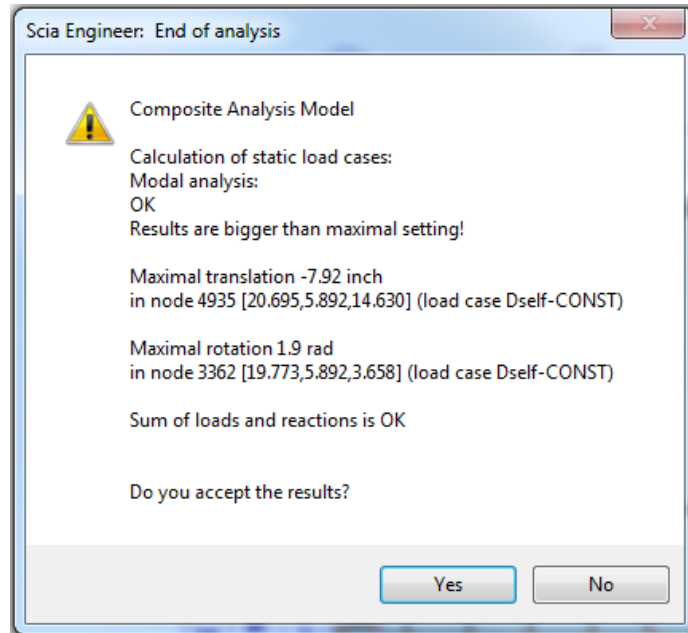
- Type of eigenvalue solver
  - o Available options: Lanczos, subspace iteration, iterative ICGC & polynomial
  - o **Polynomial** is the preferred method, based on finding roots of characteristic polynomials
- Number of eigenmodes
  - o User defined number of modes (eigen frequencies) that should be calculated
  - o Number is dependent on model size and number of degrees of freedom in the mode
  - o For this project, 12 eigenmodes will be sufficient
- Use IRS (Improved Reduced System) method
  - o Checkbox to enable or disable IRS Method
  - o IRS Method is required for use of the **Equivalent Lateral Force Procedure (ELF)** as well as **Etools** (concrete shear wall design)
- Enable advanced modal superposition for seismic load cases
  - o Checkbox to enable or disable advanced modal superposition
  - o Required for seismic analysis (using seismic spectrums)

For more information about the IRS Method, visit the corresponding SCIA Engineer Help section: [http://help.scia.net/16.0/en/#sr/seismic\\_analysis\\_of\\_buildings/seismic\\_analysis\\_of\\_buildings\\_02\\_reducedmodel.htm](http://help.scia.net/16.0/en/#sr/seismic_analysis_of_buildings/seismic_analysis_of_buildings_02_reducedmodel.htm)


With the mesh and solver settings established, it is now possible to run the analysis and evaluate the results of the structure.

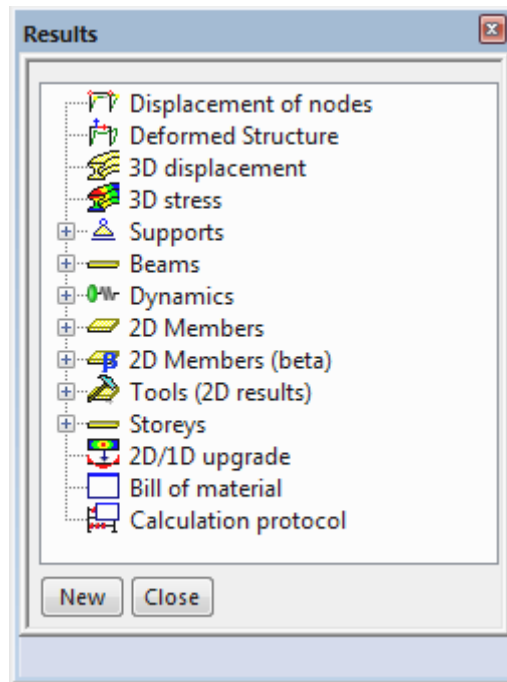
## Run Analysis

To run the linear analysis, click on the **Calculation** button  on the **Project** toolbar, or from the **Calculation, Mesh** group in the **Main** service tree. Make sure **linear calculation** is selected and then click **OK**. While the analysis is being run, various solver windows should appear showing the progress of the calculation. When the analysis is complete, the dialogue window showing the basic results will appear and the results can be accepted. Before moving on to reviewing the results, **save** the model.

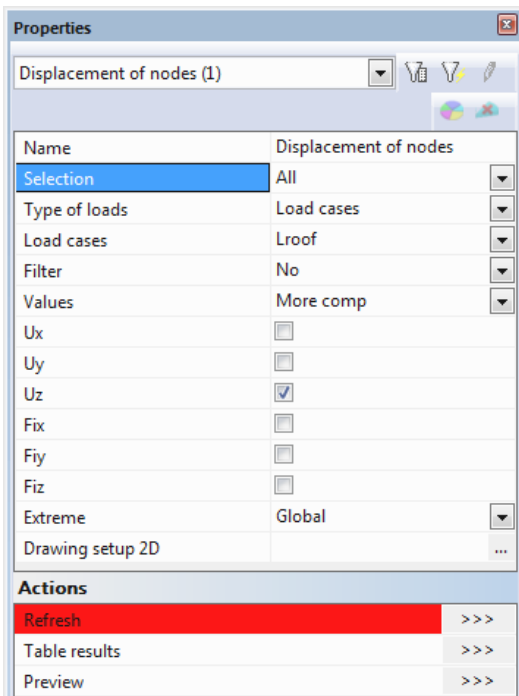


# Results

With the analysis complete, the  Results are now available in the **Main** service tree. When the results service is open, the main tree window now lists the various result types that can be accessed.



Additionally, when a result is selected, the **properties** window is used to modify the settings of the result to be viewed. Each property of the result can be modified to customize the result output.



**Selection:** All, current (what is selected) or Named Selection

**Type of loads:** load cases, combinations, mass combinations (for modal results) and classes

**Filter:** similar to selection; only shows result on elements that meet the filter criteria

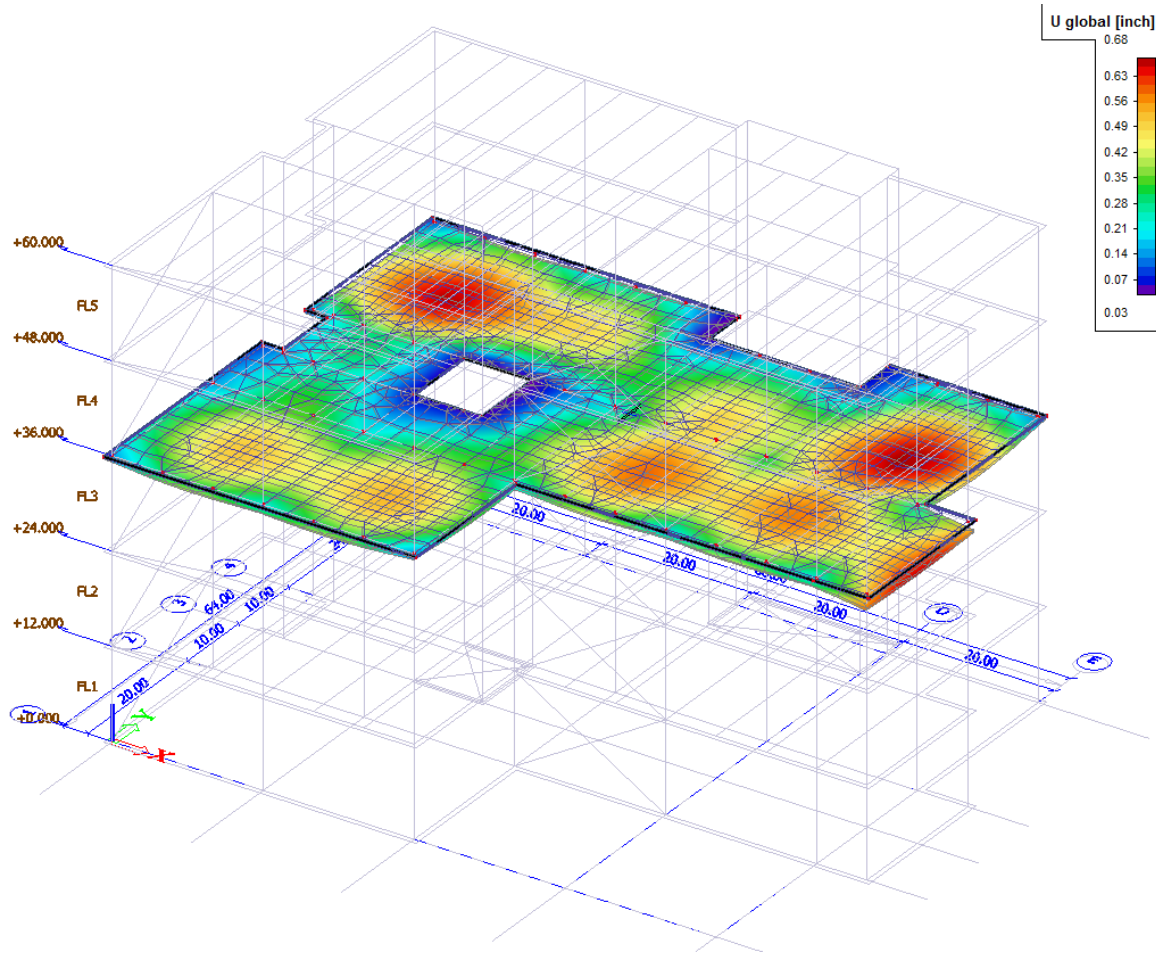
**Values:** result output values; select one specific value or choose "more comp" to select multiple values

**Extreme:** sets the result extreme; allows for extreme values to be shown graphically and in table form for the entire project (Global), based on a member or cross section (Member or Cross-Section) or to just show every value (No)

**Drawing Setup:** allows for manipulations for output on members with ability to enable current units, various graphical representations, min-max result limits, etc.

**Actions:** refresh, table results and preview

When the properties of a specific output have been established, the **refresh** action button must be clicked in order to update the graphical display.



In addition to viewing the results graphically, two separate table based results can be viewed as well. The **table results** are a dynamic table that allows filtering, column selection, sorting and easy copy/paste with Excel.

Name	Mesh	x [ft]	y [ft]	z [ft]	Case	ux+ [inch]	ux- [inch]	uy+ [inch]	uy- [inch]	uz+ [inch]	uz- [inch]	U global+ [inch]	U global- [inch]
1	S15	Node: 174	79.999	19.332	36.000 1.2D+1.6L+0.5Lr/1	<b>-0.14</b>	-0.04	0.07	0.07	-0.26	-0.26	0.30	0.27
2	S15	Node: 3435	60.000	59.733	36.000 1.2D+1.6L+0.5Lr/1	-0.06	-0.06	<b>-0.02</b>	-0.02	-0.02	-0.02	0.07	0.07
3	S15	Node: 3333	37.714	-0.668	36.000 1.2D+1.6L+0.5Lr/1	-0.04	-0.02	<b>0.11</b>	0.10	-0.17	-0.17	0.21	0.20
4	S15	Node: 3993	71.141	49.435	36.000 1.2D+1.6L+0.5Lr/1	-0.08	-0.08	0.03	0.03	<b>-0.68</b>	-0.68	0.68	<b>0.68</b>
5	S15	Node: 3971	36.230	58.655	36.000 1.2D+1.6L+0.5Lr/1	-0.03	-0.02	0.02	0.00	-0.19	-0.19	0.19	0.19
6	S15	Node: 3986	63.835	58.136	36.000 1.2D+1.6L+0.5Lr/1	-0.07	-0.06	0.00	0.02	-0.24	-0.24	0.25	0.25
7	S15	Node: 3322	20.000	-0.668	36.000 1.2D+1.6L+0.5Lr/1	-0.05	<b>0.05</b>	0.10	0.10	-0.29	-0.29	0.31	0.31
8	S15	Node: 3250	19.667	64.667	36.000 1.2D+1.6L+0.5Lr/1	0.04	-0.04	0.01	0.00	-0.30	-0.30	0.30	0.30
9	S15	Node: 181	20.000	40.000	36.000 1.2D+1.6L+0.5Lr/1	-0.01	-0.01	0.04	0.04	-0.02	-0.02	0.05	0.04
10	S15	Node: 165	40.000	64.667	36.000 1.2D+1.6L+0.5Lr/1	-0.03	-0.03	-0.01	-0.01	<b>-0.01</b>	-0.01	0.03	<b>0.03</b>

The **preview** result is a static table that matches the output that is being shown graphically. This table is also the table that would be included if the result was input into the Engineering Report.

### 3D displacement

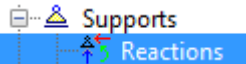
Linear calculation  
 Combination: 1.2D+1.6L+0.5Lr  
 Selection: S15  
 Location: In nodes avg., System: LCS mesh element  
**Results on 2D member:**  
 Extreme 2D: Global

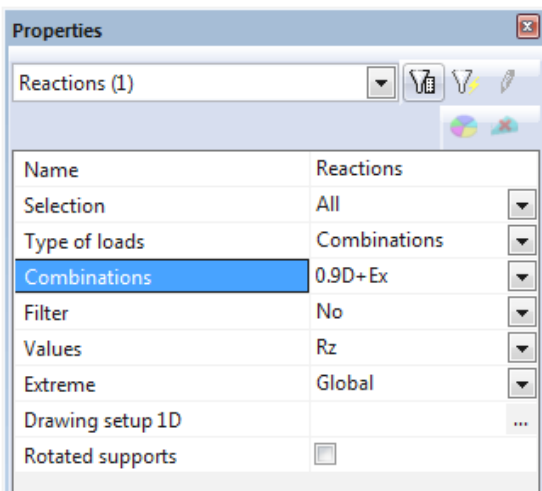
Name	Mesh	Position [ft]	Case	ux+ [inch] ux- [inch]	uy+ [inch] uy- [inch]	uz+ [inch] uz- [inch]	φ <sub>x</sub> [rad]	φ <sub>y</sub> [rad]	φ <sub>z</sub> [rad]	U global+ [inch] U global- [inch]
S15	Node: 174	79.999 19.332 36.000	1.2D+1.6L+0.5Lr/1	-0.14 -0.04	0.07 0.07	-0.26 -0.26	0.0	0.0	0.0	0.30 0.27
S15	Node: 3435	60.000 59.733 36.000	1.2D+1.6L+0.5Lr/1	-0.06 -0.06	-0.02 -0.02	-0.02 -0.02	0.0	0.0	0.0	0.07 0.07
S15	Node: 3333	37.714 -0.668 36.000	1.2D+1.6L+0.5Lr/1	-0.04 -0.02	0.11 0.10	-0.17 -0.17	0.0	0.0	0.0	0.21 0.20
S15	Node: 3993	71.141 49.435 36.000	1.2D+1.6L+0.5Lr/1	-0.08 -0.08	0.03 0.03	-0.68 -0.68	0.0	0.0	0.0	0.68 0.68
S15	Node: 3971	36.230 58.655 36.000	1.2D+1.6L+0.5Lr/1	-0.03 -0.02	0.02 0.00	-0.19 -0.19	0.0	0.0	0.0	0.19 0.19
S15	Node: 3986	63.835 58.136 36.000	1.2D+1.6L+0.5Lr/1	-0.07 -0.06	0.00 0.02	-0.24 -0.24	0.0	0.0	0.0	0.25 0.25
S15	Node: 3322	20.000 -0.668 36.000	1.2D+1.6L+0.5Lr/1	-0.05 0.05	0.10 0.10	-0.29 -0.29	0.0	0.0	0.0	0.31 0.31
S15	Node: 3250	19.667 64.667 36.000	1.2D+1.6L+0.5Lr/1	0.04 -0.04	0.01 0.00	-0.30 -0.30	0.0	0.0	0.0	0.30 0.30
S15	Node: 181	20.000 40.000 36.000	1.2D+1.6L+0.5Lr/1	-0.01 -0.01	0.04 0.04	-0.02 -0.02	0.0	0.0	0.0	0.05 0.04
S15	Node: 165	40.000 64.667 36.000	1.2D+1.6L+0.5Lr/1	-0.03 -0.03	-0.01 -0.01	-0.01 -0.01	0.0	0.0	0.0	0.03 0.03


Name	Combination key
1.2D+1.6L+0.5Lr/1	1.20*Dself-FINAL + 1.20*Dsup + 1.60*L + 0.50*Lroof

With the basics of results established, it is possible to review various result types (displacements, reactions, stresses and internal forces). For this example, two different types of results will be reviewed and added to the Engineering Report.

## Reactions

Select **Reactions** from the  group with the following result properties:



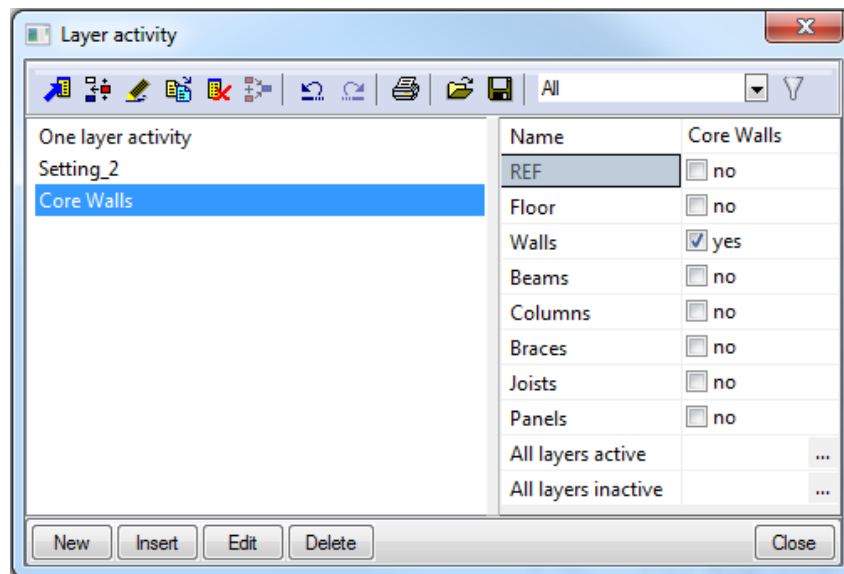
With the result displayed graphically, it is possible to add the result to the inbox of the Engineering Report as a **table**. To do this, right click in the workspace and select  **Table to Engineering report**. Once the dialogue for inserting objects into the Engineering report opens, click **Insert & Close** to add the item into the Inbox of the report for later use.


## 3D Displacement

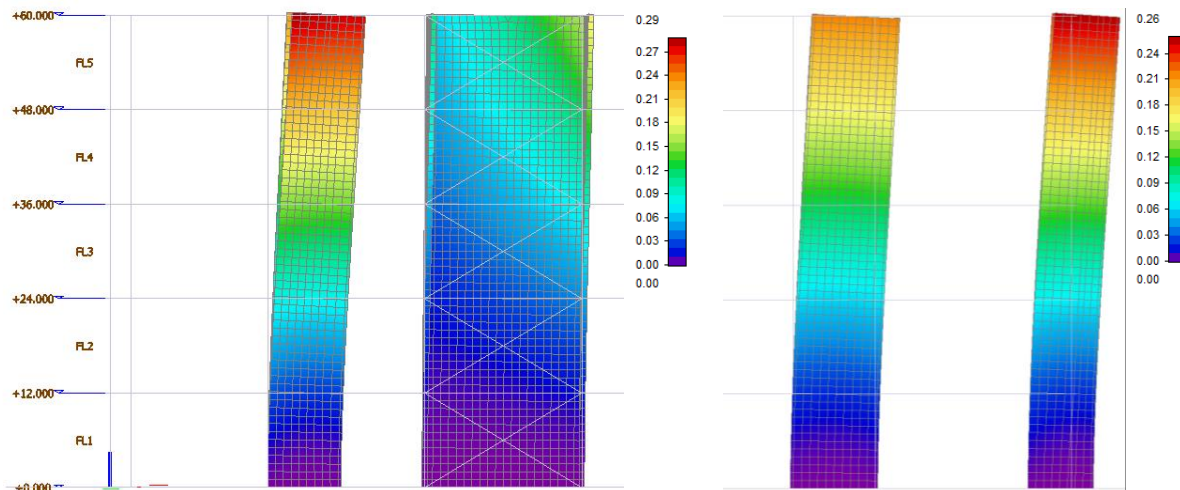
In the previous section, the mesh settings for the concrete core walls were refined. With the mesh refined, the result for 3D displacement due to seismic load can be reviewed. In order to create a view where only the walls are active, select the **layer activity** button from the **Activity** toolbar.



Layer activities (just like all other activities) are used to filter to viewable elements in the model, in this case the model can be filtered based on layers (which were previously created in Part 1 of the tutorial). To create a new layer activity, click **new** and then enable the checkbox for the layer **walls**, as shown in the picture below. When the activity is properly created, click OK.




With the layer activity set, select the  3D displacement result for load case **Ex** and value **U global**. When the parameters of the result are set, click **refresh**.




Displacement for Seismic Load in X

Displacement for Seismic Load in Y

---

With the result displayed, right click in the workspace and select  **Live picture into Engineering report** . A live picture is inserted to the Engineering Report as a “dynamic” image meaning that if the model changes in anyway the picture will update when the report is updated. Additionally, since the picture is “dynamic” it is also possible to change the parameters of the pictures results within the report without needing to create a new picture. Finally, any of the picture properties can be modified either from the insert dialogue or once the picture is inserted into the Engineering Report. To add the item, click **Insert & Close** so the picture will be available in the Inbox.

Once this picture is added, repeat the process for the load case **Ey**, with value **U global**.

Note: It is also possible to add a  **Screenshot into Engineering report** which is a static image that will not update due to changes in the model. Additionally, this type of picture does not allow for changes to be made to the parameters of the picture’s result.

Now that the two different result types have been added to the Engineering Report for use later in the tutorial, additional results can be reviewed as desired. When that is finished, it would be wise to save the model (**File > Save**) and then proceed to **Part 4 – Steel Design (Columns, Braces, Joists)** of the tutorial by visiting the [Manuals and Tutorials](#) section of the website to download the PDF.