

SCIENGINEER



Tutorial

Steel Building – Loading

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

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

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

Support
Tel: +43 1 7433232-12
E-mail: 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

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

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

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

Germany

SCIA Software GmbH
Emil-Figge-Strasse 76-80
D-44227 Dortmund
Tel.: +49 231/9742586
Fax: +49 231/9742587
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

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

Switzerland

Nemetschek SCIA Switzerland
Dürenbergstr. 24
CH-3212 Gurmels
Tel.: +41 26 341 74 11
Fax: +41 26 341 74 13
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

Table of Contents

Table of Contents.....	4
Introduction.....	5
Create Load Groups & Load Cases	6
Gravity Loads.....	9
Lateral Loads	11
Load Combinations & Classes.....	17

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

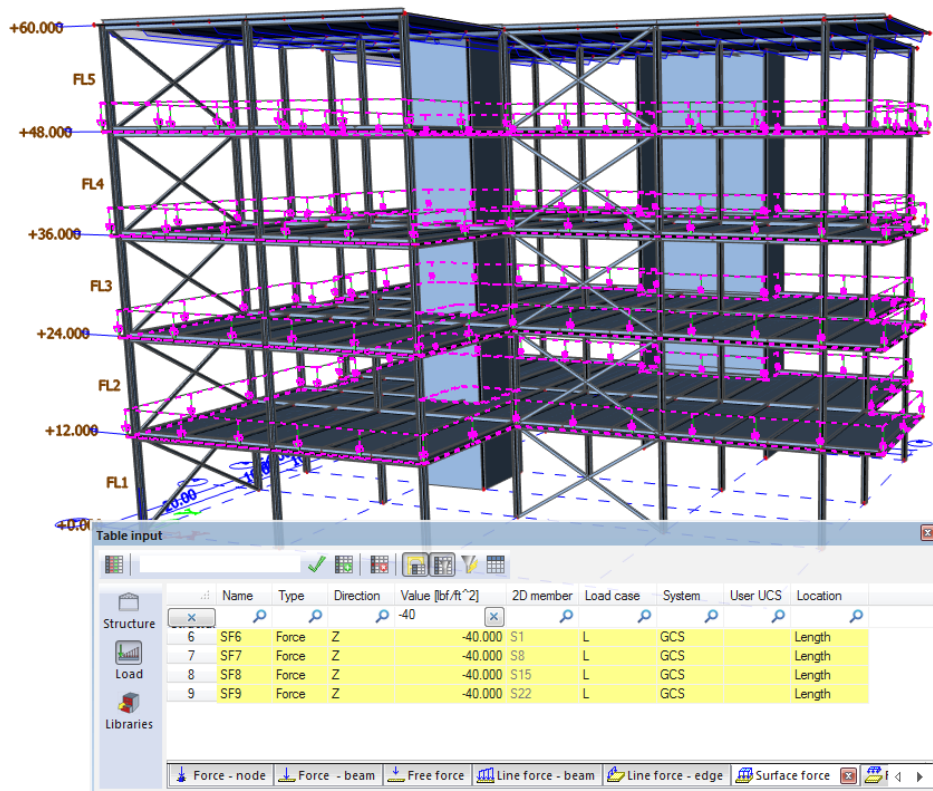
Part 4 – Steel Design (Columns, Braces, Joists)

Part 5 – Composite Steel Design

Part 6 – Concrete Shear Wall Design

Part 7 – Engineering Report

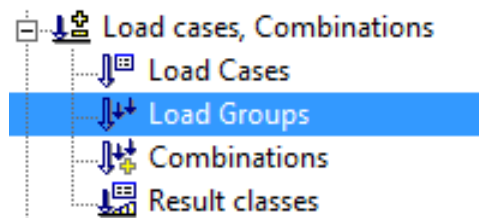
In Part 2 of the Commercial Steel Building tutorial, the functionality and features that are available to create load cases and combinations as well as add load to the structure will be discussed.



Create Load Groups & Load Cases

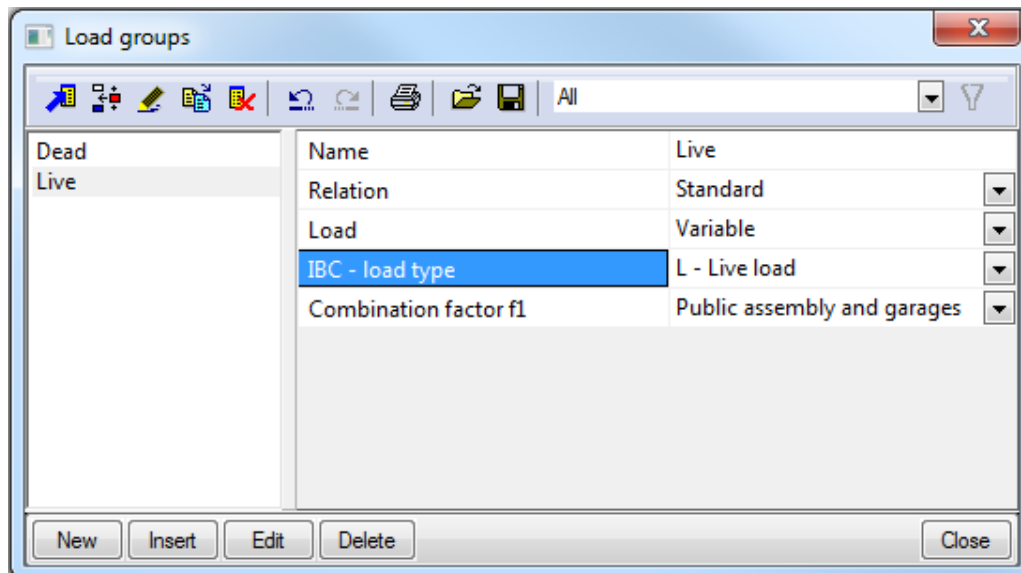
Load Groups

Before adding load cases to the project it is necessary to create **load groups**. Load Groups are used to combine individual load cases when they are inserted into a load combination. These are especially important when using the automatic load combinations based on the ASCE. To access the functionality, double click on **Load Groups** from **the Load Cases, Combinations** service found in the **Main** service tree.



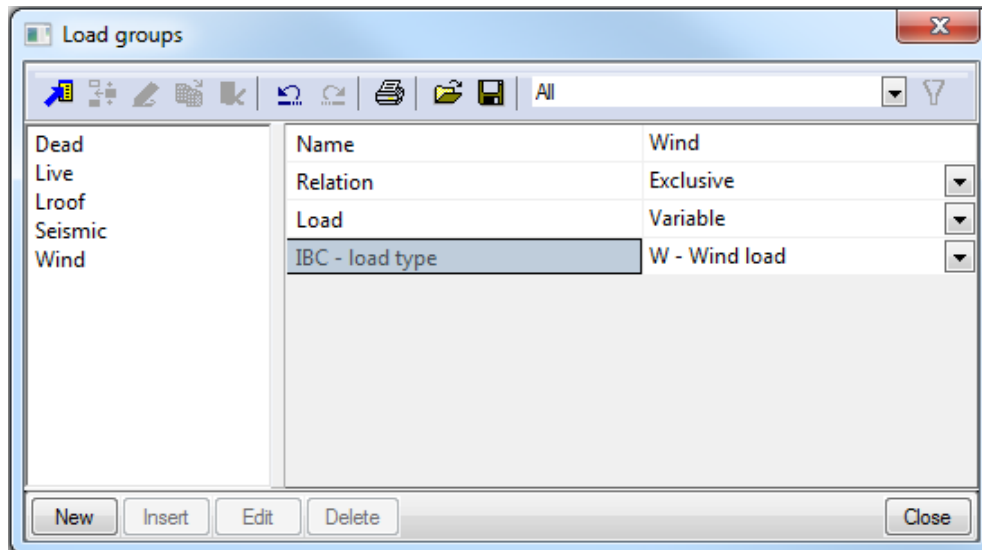
With the load group manager open, the program creates a load group for **permanent load** (dead load) by default. Change the name of **LG1** to **Dead** and then click the **New** button to create additional load groups. The following parameters for each load group can be modified:

- Name: name of the specific load group
- Relation: how the cases within a group will be combined.
 - Standard: Load case sorting and combination will be used when creating load combinations.
 - Exclusive: Load cases of the same load group are always inserted into a new combination.
 - Together: Two load cases from the same load group will never appear in the same combination.
- Load: Permanent (dead loads) and variable (all other loads). Accidental is not used for IBC code.
- IBC – load types: As defined in ASCE 7 Section 2.2.



It is now possible to create the following load groups:

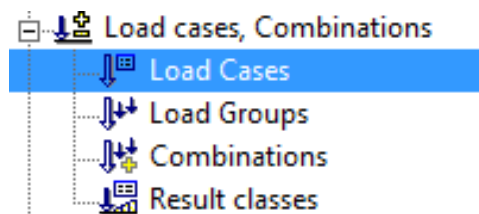
- **Live:** Relation = Standard, Load = Variable, IBC – load type = L – Live Load
- **Roof:** Relation = Standard, Load = Variable, IBC – load type = Lr – Roof Live Load
- **Seismic:** Relation = Exclusive, Load = Variable, IBC – load type = E – Earthquake Load
- **Wind:** Relation = Exclusive, Load = Variable, IBC – load type = W – Wind Load



With all the necessary load groups created, it is now possible to create individual load cases.

Load Cases

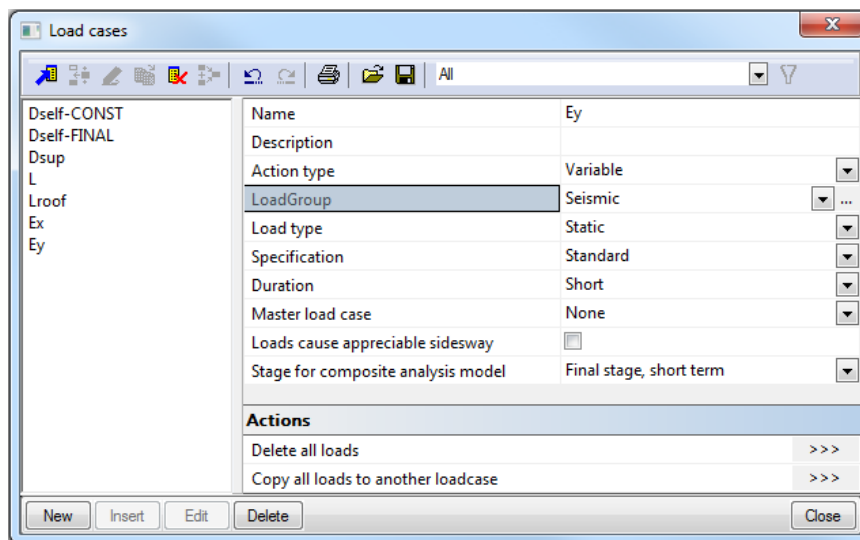
Now that the structure has been modelled properly and before an analysis can be executed, individual load cases need to be created in order to add various load types to the model. To begin adding load cases double click on **Load Cases** from the **Load Cases, Combinations** service found in the **Main Service** tree.



With the load case dialogue open, new load cases can be added to the existing self-weight load case (LC1) that is created by default in every project. To add load cases, click **New**. For each individual load case, the **Name**, **Action type**, **Load Group** and **Load type** need to be set. For this example, all load cases (with the exception of the self-weight case) will be created with Action type **Variable**. Additionally, when modifying the parameters of each load case, make sure that the name that is being set matches the correct **Load Group**. It is now possible to create the following **Load Cases**:


- **Dself-CONST:**
 - Action Type = Permanent
 - Load Group = Dead
 - Load Type = Self-Weight
 - Stage for Composite Analysis Model = Construction Stage

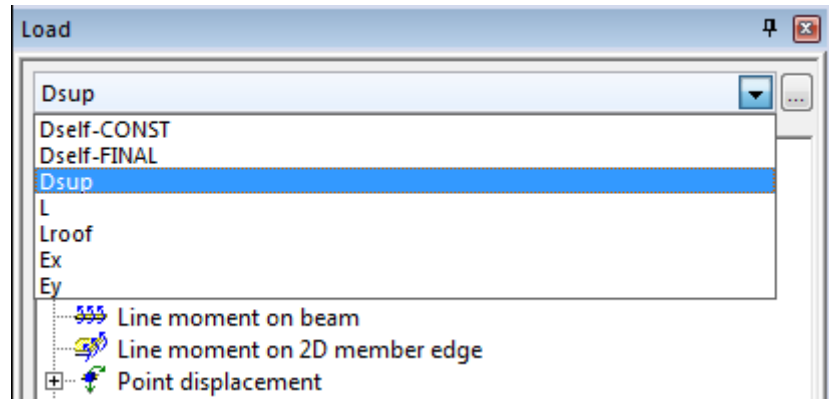
- **Dself-FINAL:**
 - Action Type = Permanent
 - Load Group = Dead
 - Load Type = Self-Weight
 - Stage for Composite Analysis Model = Final state, long term
- **Dsup**
 - Action Type = Permanent
 - Load Group = Dead
 - Load Type = Standard
 - Stage for Composite Analysis Model = Final state, long term
- **L**
 - Action Type = Variable
 - Load Group = Live
 - Load Type = Static
 - Stage for Composite Analysis Model = Final state, short term
- **Lroof**
 - Action Type = Variable
 - Load Group = Lroof
 - Load Type = Static
 - Stage for Composite Analysis Model = Final state, short term
- **Ex**
 - Action Type = Variable
 - Load Group = Seismic
 - Load Type = Static
 - Stage for Composite Analysis Model = Final state, short term
- **Ey**
 - Action Type = Variable
 - Load Group = Seismic
 - Load Type = Static
 - Stage for Composite Analysis Model = Final state, short term




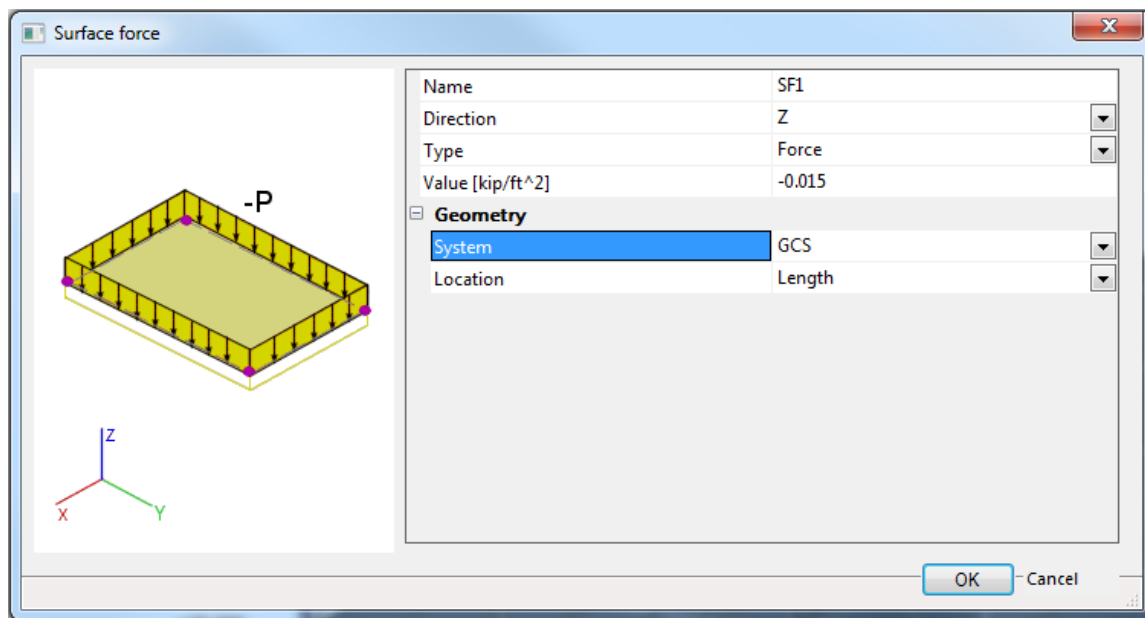
Note: Wind load cases will be created automatically by way of the 3D wind load generator.

Gravity Loads

With load cases and groups created, it is now possible to add the required gravity loads to the structure. To do this, double click on  Load to open the **Load** service. When the **Load** service opens different types of loading can be applied to the created **load cases** by selecting them from the drop down box.



To begin this process, select **Dsup** from the drop down menu and then click the [+] to expand the **surface load** functionality. Next, double click on  on 2D member to open the **surface force** dialogue. When the dialogue opens, configure the properties of the surface force as shown in the picture below and click **OK**.



The **command line** now instructs the user to select the specific 2D members in which the load should be applied. Select all **5** plate elements (4 composite floors and 1 metal deck roof) to apply the load. After the selection is complete, **right click** and select **OK** or press **ESC** to end the command.

Additional gravity loads will be added in a similar fashion. Apply the following surface loads on 2D members as follows:

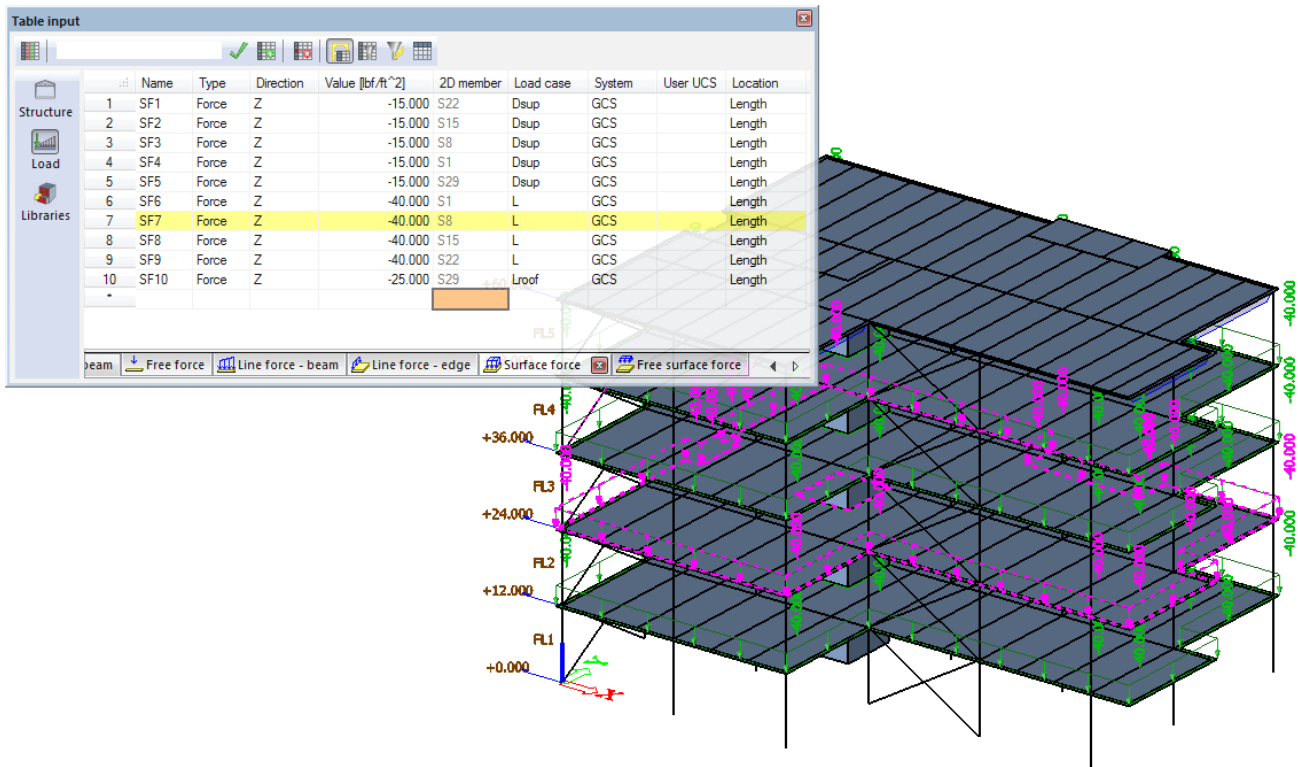
Live Load (L) @ FL1 to FL4:

- Value = -40 PSF, Direction = Z

Roof Live Load (Lroof) @ FL5:

- Value = -25 PSF, Direction = Z

Note: Other load types (ie. point loads, line loads, etc.) can be added through the **load** service as well. Additionally, the easiest way to review all applied loads and make quick modifications is using the **Table Input**. The table input is available by navigating to View > Toolbars and selecting table input. Within the table input, various **structural elements**, **library items** and **applied loads** can be viewed and manipulated. Additionally, items selected within the table will also be selected in the graphical interface.

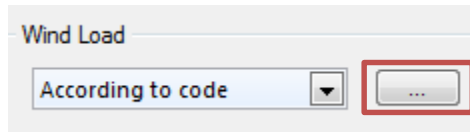


Lateral Loads

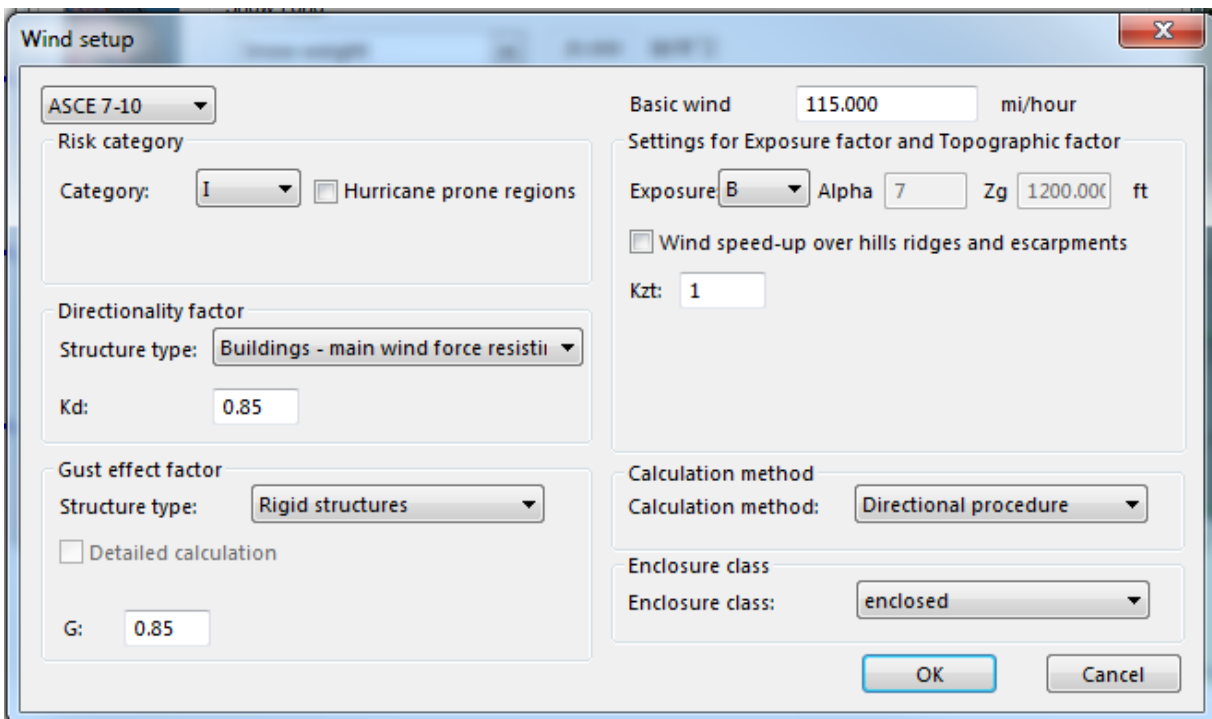
3D Wind Loading

With the gravity loads applied to the structure, it is possible to add wind loading to the structure using the 3D wind load generator. Before loads can be added the functionality needs to be enabled from the **Project** menu. To do this, double click on **Project** in the **Main Service** tree and navigate to the **Functionality** tab. Click to enable **Climatic loads** and then switch to the **Loads** tab.

Within the **Loads** tab, change the combo box under Wind Load to **according to code** and click on the ellipsis button to access the code setup.

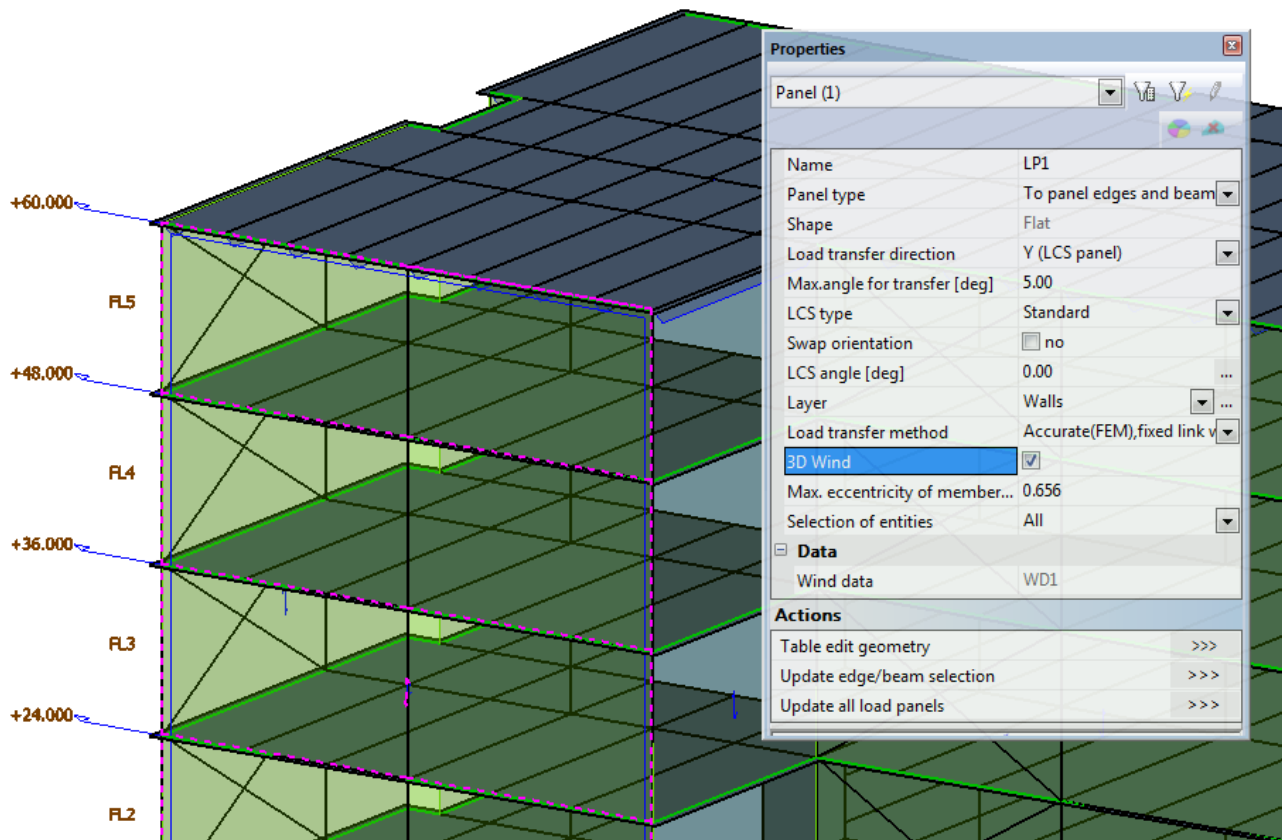


The various properties within the wind setup can then be modified to match what is shown in the below picture. Within SCIA Engineer, it is possible to utilize the wind provisions from ASCE 7-05 and 7-10 while also choosing between the low-rise method (envelope procedure) or all heights method (directional procedure).

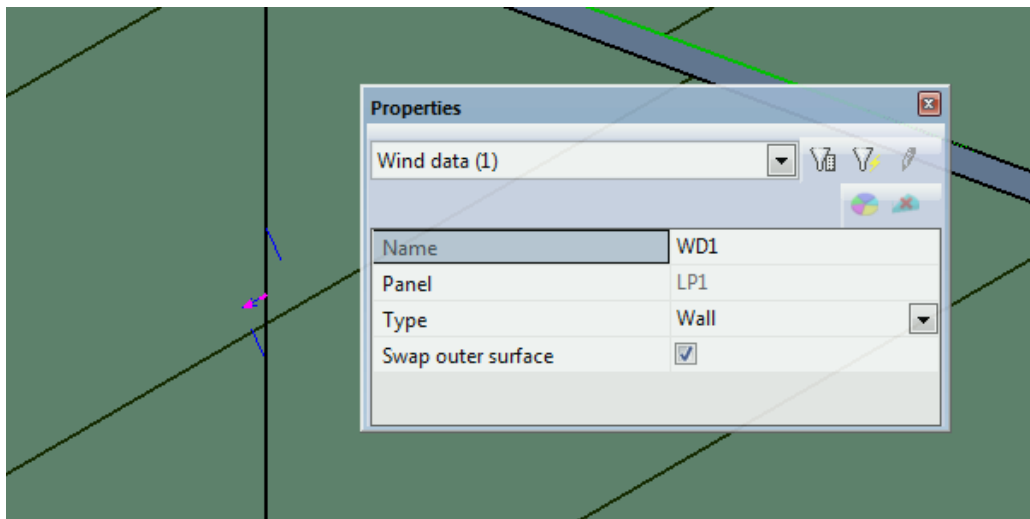


Once the properties are set, click OK and then set the Snow Load to **Snow Weight**. This is required in order to properly utilize the 3D wind loading within the project.

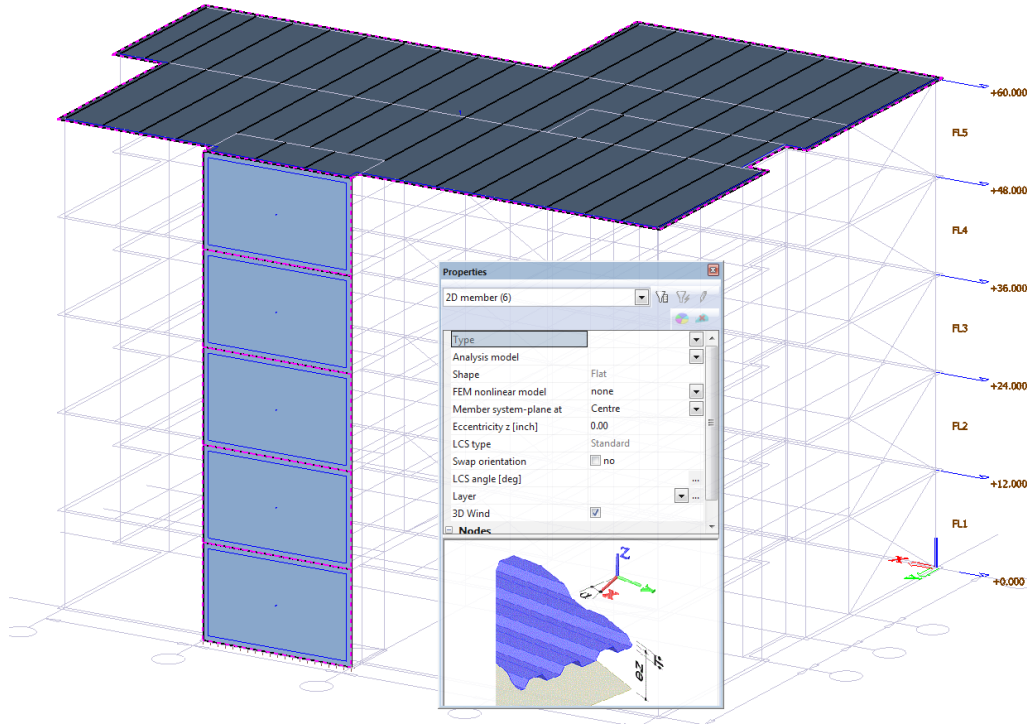
With the properties of the load set, the surfaces (plates & load panels) that will have the 3D wind load applied to it need to be specified. This is done by selecting a surface and clicking to enable the 3D wind in the properties of the panel or surface. For this example, select each of the load panels (13 in total) and enable the 3D wind checkbox in the member properties.



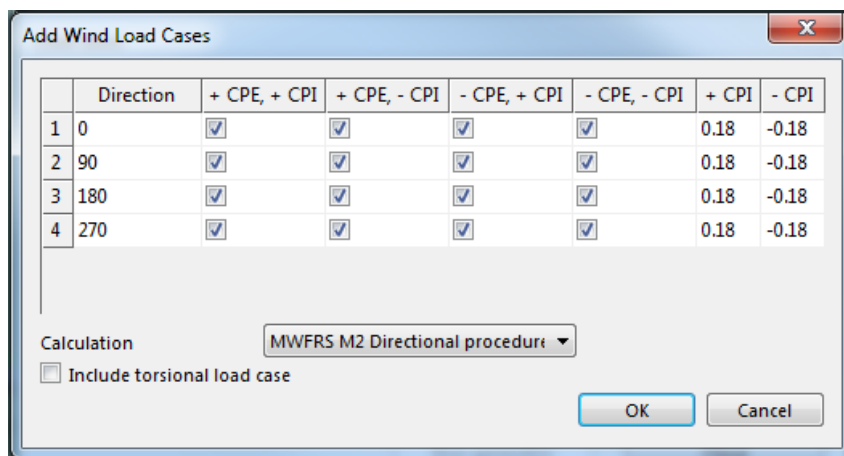
Additionally, it is a requirement of the 3D wind load engine that the outer surface of the element be properly defined based on the direction of the **wind data arrow**. This is done so that the wind generator can properly assign the internal and external pressure coefficients to the structure.



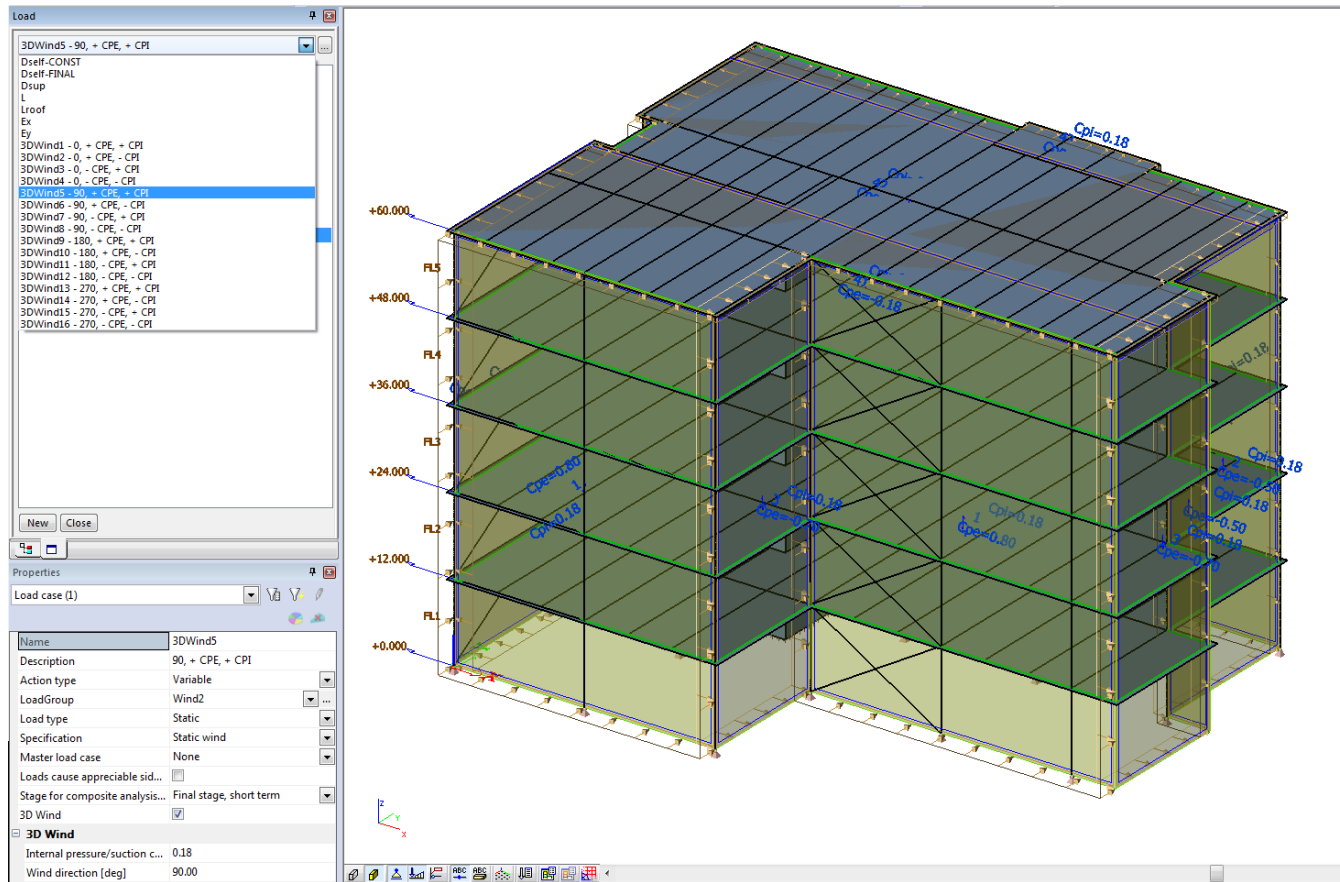
In this example, the **swap outer surface** checkbox needs to be enabled for 6 of the load panels (where the wind direction arrow on the panel is pointing inward). In addition to enabling 3D wind on the load panels, it should also be enabled on the exterior shear wall and the roof plate in order to ensure that the entire exterior of the structure is enabled for 3D wind load application.



With the 3D wind load enabled on all exterior surfaces, the wind load cases can be generated from the **Load** service by double clicking on **3D Wind Generator**. In the first dialogue, the mean roof height should be set to **60'-0"**. Next, click the **Add load cases** button. The subsequent dialogue will list the active wind directions and combinations of internal (C_{pi}) and external (C_{pe}) pressure coefficients. Additional wind directions can be added by right clicking to **insert a new line**. Click OK to create the load cases and then click **Run generator** to create the load cases.



Once the wind generator has been successfully run, it is possible to evaluate individual wind load cases in order to assess that the proper coefficients were applied in accordance with the ASCE 7 code.

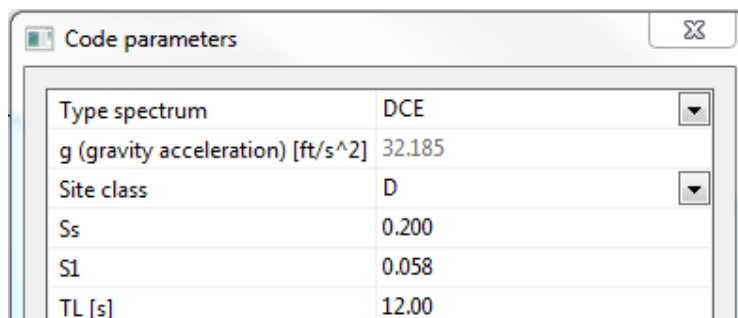


Note: To enable the labels of wind data (and pressure coefficient), right click in the workspace and select **Set View Parameters for All**. Then make sure that **wind data** and **labels of wind data** are enabled within the **Loads/Masses** tab.



Seismic Loading (Equivalent Lateral Force Procedure)

In addition to adding wind loading to the structure, seismic loading can be added by way of creating **response spectrums** and using them as seismic **load cases**. Before seismic loads can be added, it is necessary to make sure that the **Dynamic** and **Seismic** functionality is enabled. To do this, double click on **Project** in the **Main Service** tree and navigate to the **Functionality** tab. Click to enable **Dynamics** and **Seismic**.

With all necessary functionality enabled, the seismic response spectrum can be defined by navigating to **Libraries > Loads > Seismic Spectrums**. When the dialogue opens, close the **predefined database** in order to access the **seismic spectrum** input dialogue. Then change the **input type** to **IBC** and click **code parameters** to set the ASCE code provisions of the spectrum. In this example, the values for the spectrum can be defined as shown below:



Once the spectrum is defined, masses need to be added to the structure in order to properly execute the seismic analysis. In SCIA Engineer, masses can be added in two different ways:

1. By adding them manually using  **Masses** found in the **Dynamics** tree. Masses can be added at nodes, beams or surfaces.
2. Automatically from previously created linear loading via  **Mass groups**.

For this example, the masses will be created automatically, first double click on **mass groups** and enter the following mass groups.

Dead:

- Bound to load case: Yes
- Load case: Dsup

Live:

- Bound to load case: Yes
- Load case: L

Roof:

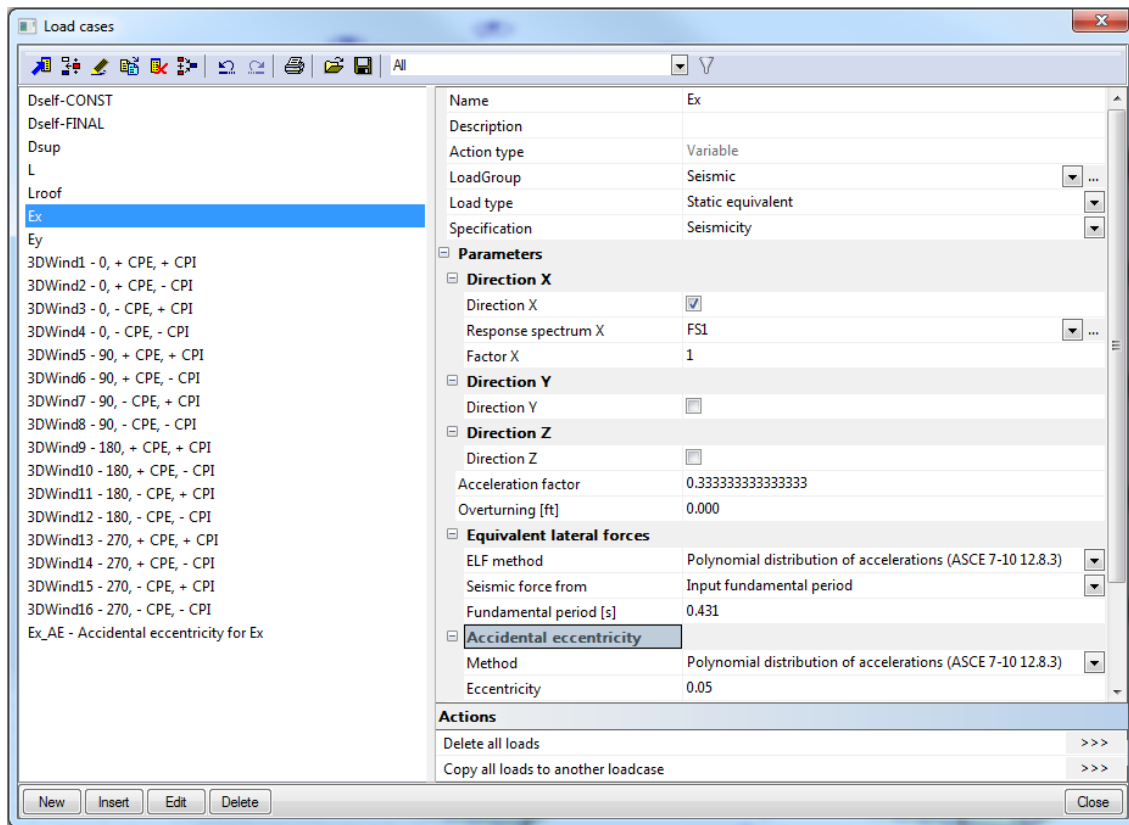
- Bound to load case: Yes
- Load Case: Roof

After adding each mass group, press the action button **create masses from load case** to generate the masses that are associated with the already applied linear loading.

After **mass groups** are created, it is required that they be combined into a **Combination of mass groups**. Double click on the functionality and add the three available mass groups (Dead, Live & Lroof) to the contents of the combination and rename the combination **D+0.3L**. Click OK to finish the input and then change the load factor of **mass groups** Live and Lroof to be **0.3**.

Note: Self-weight of the structure is always taken into account as mass and does not need to be added as a specific mass group or into the combination of mass groups.

With the seismic spectrum and mass combination created, specific **seismic load cases** can be defined. To do this, double click on **Load Cases** and select the previously created **Ex** load case. Input the parameters of the seismic load cases as shown below. Similar parameters can be used for the **Ey** load case as well, with the only change required is enabling the seismic spectrum in the **Direction Y**.




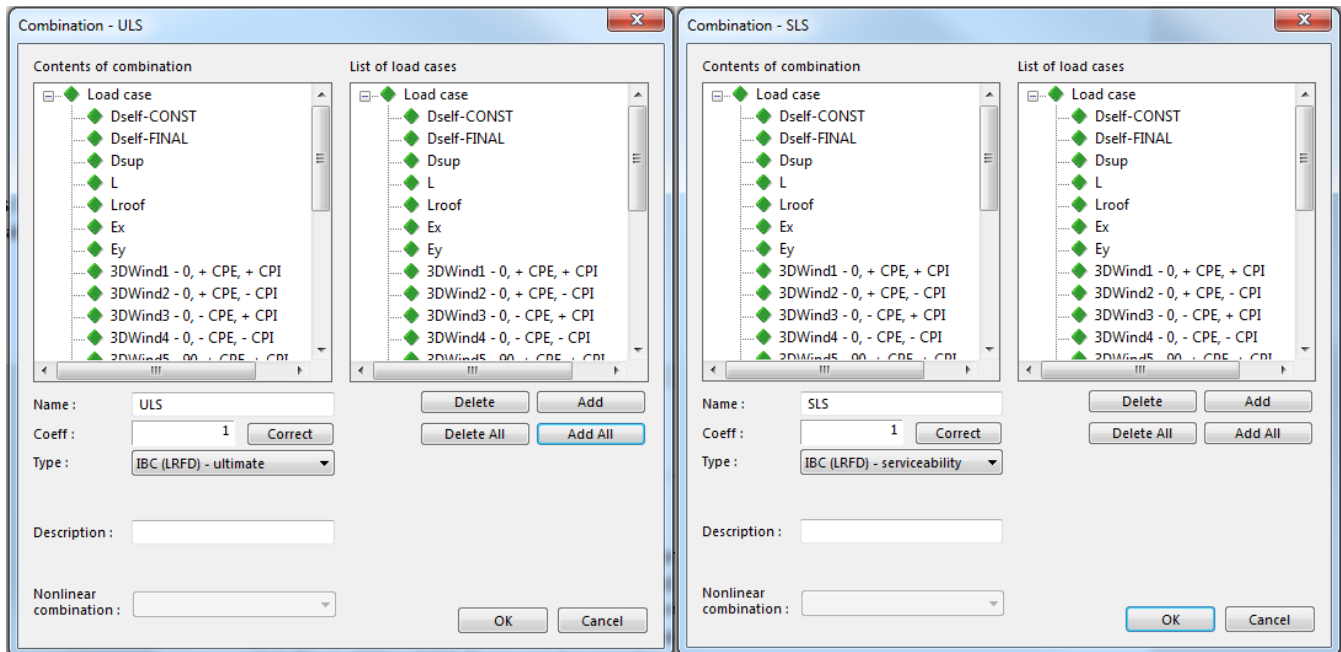
Notes:

- When setting the appropriate **load type**, first select **Dynamic** with specification **Seismicity**. Then enable the **Equivalent Lateral Forces**. This will automatically change the load type to **static equivalent**.
- The acceleration factor is the used to reduce the effects of the seismic spectrum. In this way, the value is equal to 1/R.
- The inputted fundamental period is based on the equation $T_a = C_t h_n^x$
- Enabling the accidental eccentricity requires the user to set the percentage of load associated with eccentricity. When eccentricity is enabled an addition **accidental eccentricity** load case is created. This load case will be automatically combined with the seismic load case so that the user can review all seismic load effects in a chosen direction.

Load Combinations & Classes

Load Combinations

With all gravity and lateral load cases created and applied to the structure **load combinations** should be created in order to properly combine the loading for analysis and design purposes. To access the load combinations double click on  **Combinations**. When the dialogue opens, you will initially see two previously created combinations (for seismic load + seismic eccentricity). To create new combinations, click **New**. Within SCIA Engineer, it is possible to create various types of load combinations including envelope, linear and code based envelopes. For this example, we will create two IBC code based envelopes for strength and service level design conditions. With the load combination dialogue open, the **ULS** and **SLS** combinations can be created as shown below (with all load cases added to each combination).



In addition to the envelope combinations, create two linear combinations for the evaluation of specific load combination. For this example, create the following:

1.2D+1.6L+0.5Lr

- Type: linear ultimate
- Load cases: Dself-Final, Dsup, L & Lroof
- Combination Factors: 1.2 on all dead load, 1.6 on live load & 0.5 on roof live load


0.9D+Ex

- Type: envelope ultimate
- Load cases: Dself-Final, Dsup & Ex
- Combination Factors: 0.9 on all dead loads

Notes:

- The IBC envelope is created using ASCE 7-10 combinations. If you want to use the combinations from ASCE 7-05, navigate to Project > Code Setup and change the **code for combinations**.
- Code based combination envelopes and other envelopes can be exploded in order to review the individual combinations that make up the envelope. To do this, select one of the **envelope** combinations and select the **Action** button **Explode to linear**.

Results Classes

With the load combinations created, it is also possible to create a **result class**. Result classes are used to group cases and combinations together for a more in-depth comparison. A good example of this are the sixteen different wind load cases that were created by the 3D wind load generator earlier in the example. In order to properly evaluate the wind on the structure and to easily determine which wind load cases are controlling, all wind load cases should be added into a new class. To do this, double click on  **Result classes** from within the **Load cases, combinations** service and click **New**. Add all 3D wind load cases into the contents of the class and name the class **All Wind**.

With all loads, load cases, combinations and classes created the model is ready for mesh/solver setup and analysis. At this point in the tutorial it is wise to save the model (**File > Save**) and then proceed to **Part 3 – Analysis, Meshing & Results** of the tutorial by visiting the [Manuals and Tutorials](#) section of the website to download the PDF.