ColumnBase User's Manual, Version 4.0

Integrated Software for Analysis and Design of Column-Base Connections using Three Dimensional Finite Element Modeling

Last updated: Aug 23, 2018

PCEESoft Inc.
Professional Civil & Earthquake Engineering Software Incorporation

Imam Ali; who his greatness not yet known, said:
Knowledge is a great treasure which does not come to an end.

Copyright

PCEESoft Inc.
Address: 1005, 3489 Ascot Place, Vancouver, BC, Canada.
Email: info@pceesoft.com
Website: www.pceesoft.com

Table of Content

1. Introduction

1.1 What is ColumnBase Software?

ColumnBase is integrated software for analysis and design of column-base connections using three dimensional finite element modeling. Since 1950, many laboratory studies have been done and many theories have been proposed in order to study the behavior of these connections. Researchers have tried to find an applicable solution for engineer designers by applying simplified hypotheses. The results of these studies have been reflected in designing handbooks and instructions. These principles, inevitably, impose some hypotheses and constraints (rigidity of column-base and anchors, strain compatibility, and other assumptions) to analyses that are far from reality. Most of these designing principles have not been tested in laboratory and are based on analytical methods, few experimental data, and engineering judgments [Gomez et al 2010]. Indeed, simplified hypotheses that are employed in column-base designation are far different from real behavior of these connections, and cannot respond to diverse problems that designers confront with.
ColumnBase software has been invented to solve these problems in structural engineering. This software provides a three dimensional model of connection parts (including foundation, grout, column, base plate, stiffeners, etc.) for finite element analysis, and by applying contact modeling analysis to connection parts in simultaneous loading cases, analyzes connection model and finally applies its results according to designing handbooks and standards. Outputs of this software have been proved by the most reliable laboratory experiments, and its accuracy is incomparable.

1.2 Why Use ColumnBase Software?

-High Accuracy in Prediction of Actual Behavior of Connection: Comparing output results of software with reliable laboratorial tests shows that ColumnBase has unique accuracy in modeling and estimating actual behavior of column base plate connection.

-No Limit in Modeling and Simultaneous Loadings: ColumnBase doesn't have any limitation in geometrical modeling of connection and is able to solve any case study with any arrangement by applying three dimensional finite element modeling. Simultaneous loadings (two directional bending, compressive axial forces and uplift, shear forces in two directions) and different combinations of loading are the other options of this software.

-High Speed and Ability in Process: Obviously, finite element modeling of foundation, column, and contact analysis for each case of loading combination increases the complexity of problem solution. ColumnBase benefits from smart powerful algorithms that enable ColumnBase to analyze behavior of connection effectively. Accuracy of output results is comparative to laboratorial results and finite element software such as ABAQUS, ANSYS, etc.

-Design of Connection Components According to Designing Handbooks and Standards: Accurate results of connection analysis are used for designation of components. Designing and checking of base plate, anchors, foundation and etc. are applied by software according to standards, and detailed calculations for every combination of loading can be reported.

-Predesigned Database and Supporting Various Systems of Units: ColumnBase has prepared all required data such as materials, column sections, and anchor bolts for software users around the globe. It also supports common system of units, properly.

-Visual and User-friendly Interface: One of the attractive specifications of ColumnBase is user-friendly and visual environment. 2D and 3D views of connection model, display of stress contours and etc. are provided in this software. Visual environment decreases input errors and makes the browsing of output data easier.

2. General

2.1 System Requirements

Before installing this software, it is important that you verify that your computer meets, or exceeds, the minimum system requirements for operation. If your system does not meet the minimum requirements, problems can occur while running the product, and at the operating system level.

Minimum Requirements	
Operating System	Microsoft® Windows® 10 Microsoft Windows 8.1 Microsoft Windows 7 SP1
CPU Type	32-bit: 1.6 Gigahertz (GHz) or faster 32-bit (x86) processor 64-bit: 1.6 Gigahertz (GHz) or faster 64-bit (x64) processor
Memory	32-bit: 2 GB 64-bit: 4 GB
Display Resolution	1360 x 768
Disk Space	Minimum 4.0 GB
.NET Framework	.NET Framework Version 3.5

Recommended Requirements	
Operating System	Microsoft® Windows® 10 Microsoft Windows 8.1 Microsoft Windows 7 SP1
CPU Type	2.6 Gigahertz (GHz) or faster processor
Memory	32-bit: 4 GB 64-bit: 8 GB
Display Resolution	1920 x 1080
Disk Space	Minimum 8.0 GB
.NET Framework	.NET Framework Version 3.5

2.2 How to install/Uninstall program

Before installing the software, check the minimum system requirements. Download the latest version from this link. After downloading, by clicking setup file, the software will be installed automatically. For removing the program from your computer, go to Control Panel/Programs and features. Select the program and click to uninstall. The software will be uninstalled automatically.

2.3 How to use the full version

For using the full version of software, you must purchase the product. For more information, view this link. After purchasing the product, you will receive step by step help on how to activate the full version of the software.

2.4 Program Workspace

ColumnBase's Workspace includes three main boxes: Inputs, 2D View and 3D View are shown in Figure 2-1. There are seven menu items at menu bar containing File, Define, Analysis, Display, Design, and Reporting and help menu. Tool strip menu is below the menu bar and contains: New Project, Open Project, Save Project, Set Analysis Options, Select Load Combos, Run Analysis, Lock/Unlock Model and Set design preferences. Unit system button is shown in Figure 2-1.

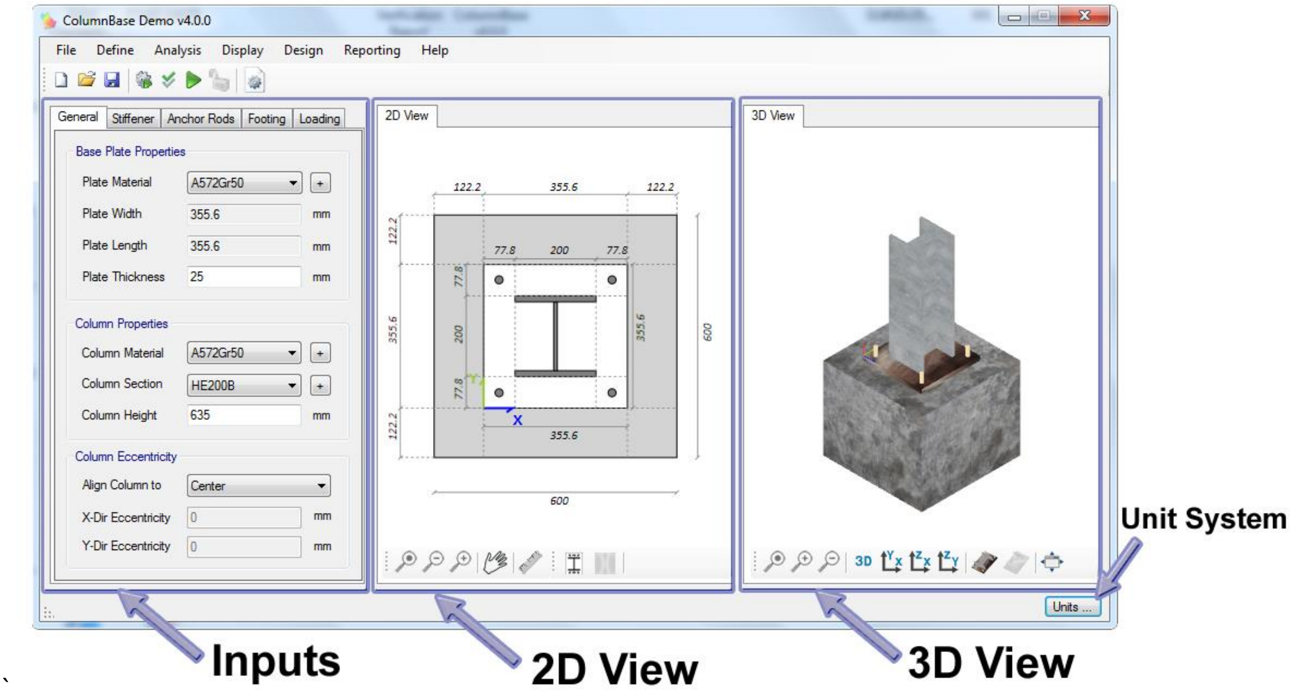


Figure 2-1, Program Workspace

2.5 Working with Units

Several unit systems are supported in the software such as Metric SI, Metric MKS, I.R. & U.S. Imperial units. Furthermore, user can customize units for any of the components. Available units for all components are shown below. User can switch between units at any time and all inputs/results will update, automatically. To change the units, click units' button in the right side of status bar. Figure 2-2 is shown the **Units** form.

Item	Available Units
Length	Inch , mm , cm , m
Force	Kips , lbs , KN, N , Tonf , Kgf
Moment	Kip.ft , Kip.in , lb.ft , lb.in , KN.m , N.m , Tonf.m , Kgf.m , Kgf.cm
Stress	ksi , psi , MPa , Tonf/m² , kgf/m² , kgf/mm²

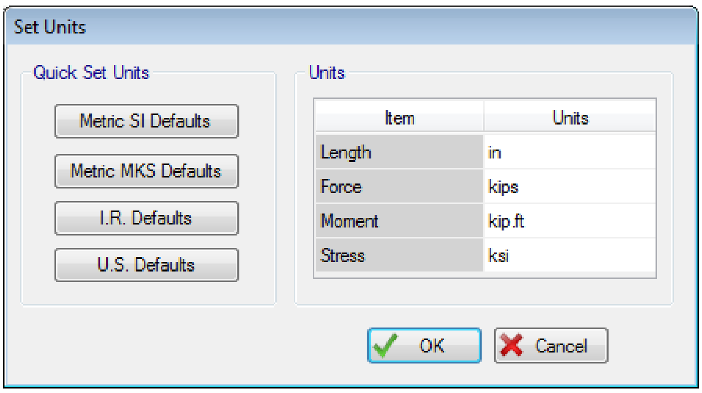


Figure 2-2, Set Units Form

3. Modeling

3.1 Required Definition

3.1.1 Define Materials

For defining the materials in the main window of the software, use **Define menu**> **Material Properties** command to access **Define Materials** Form, (Figure 3-1). In this Form, three types of materials including Steel, Concrete and Anchor Rod can be defined. All of these materials are linearly elastic with isotropic behavior. The parameters defining these materials are explained below.

Here, there are the possibilities for defining new materials, editing defined materials, deleting and also importing them from a specified software database.

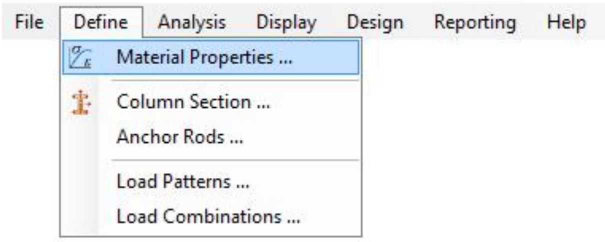


Figure 3-1, Open Define Materials Form

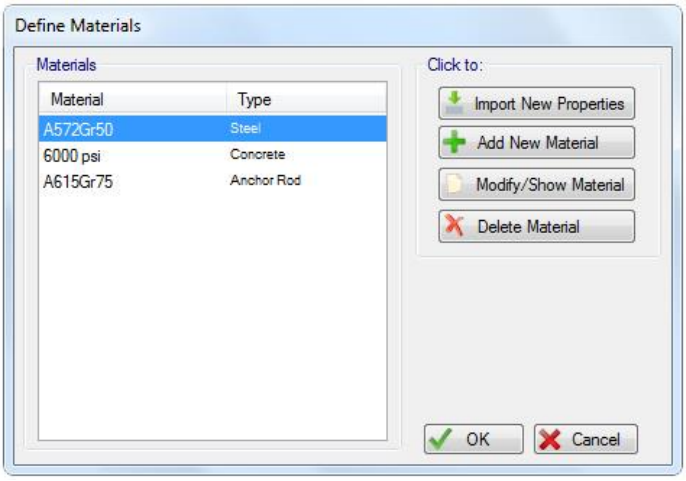


Figure 3-2, Define Materials Form

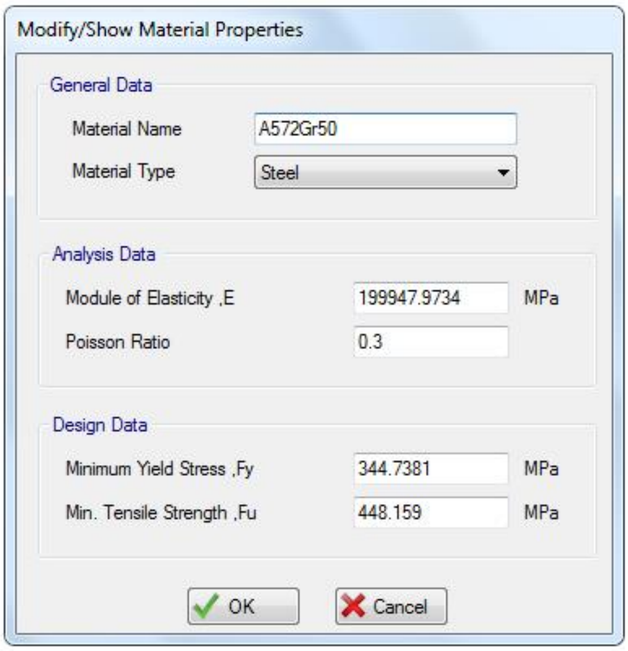


Figure 3-3, Modify/Show Steel Material Form

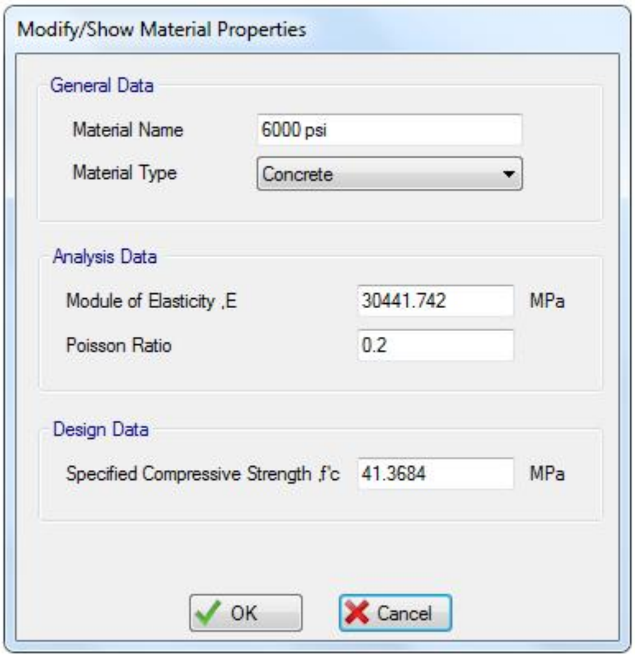


Figure 3-4, Modify/Show Concrete Material Form

- Steel: is used to model the column, plate and stiffeners. The parameters used to model steel material consist of modulus of elasticity, Poisson ratio, yield stress and ultimate stress (Figure 3-3). The first two parameters are used in finite element modeling. Yield and ultimate stresses are the required parameters for designing.

- Concrete: concrete material is used for modeling the footing and grout. Modulus of elasticity and Poisson's ratio are the parameters for finite element modeling and Specified concrete compressive strength is used to control the bearing capacity of footing/grout (Figure 3-4).

- Anchor rod: for modeling the behavior of anchor rod, only the modulus of elasticity is used. Poisson's ratio of anchor rods is supposed to be 0.3. Yield stress and tensile strength of anchor rods are the required parameters for design. Ultimate stress is used to calculate the tensile and shear capacity of anchor rods.

Modulus of elasticity is a positive value and Poisson's ratio is between 0 and 0.5. Controlling of these restrictions is done automatically.

- Import Materials

The information of current global materials is put in the software's database for the sake of easy modeling. The available databases in this version of software are listed in the tables below. For importing new materials in the **Define Material** Form (Figure 3-2) click **Import New Properties** button.

Steel Type Materials	
Iranian Database	ST37 , ST52
Indian Database	Fe250 ,Fe345
Italian Database	S235 , S275 , S355 , S450
Chinese Database	Q235 , Q345 , Q390 , Q420
US Database	A36 , A53GrB , A500GrB42 , A500GrB46 , A572Gr50 , A913Gr50 , A992Fy50-1
Europe Database	S235 , S275 , S355 , S450 , S275 N/NL , S355 N/NL, S420 N/NL , S460 N/NL , S275 M/ML , S355 M/ML , S420 M/ML , S460 M/ML , S235 W , S355 W , S460Q/QL/QL1

Concrete Type Materials	
Iranian Database	C20 , C25 , C28 , C30 , C35 , C40 , C50
Indian Database	M15 , M20 , M25 , M30 , M35 , M40 , M45 , M50 , M55 , M60
Chinese Database	C15 , C20 , C25 , C30 ,C35 , C40 , C45 , C50 , C55 , C60 , C65 , C70 , C75 , C80
US Database	3000 psi , 4000 psi , 5000 psi , 6000 psi

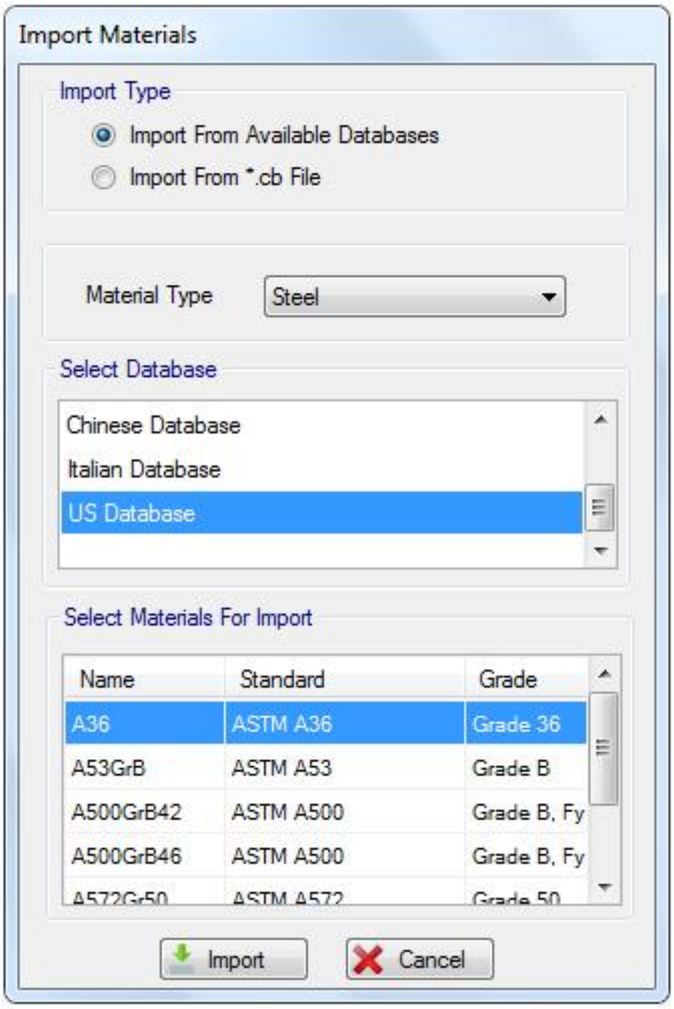


Figure 3-5, Import Material Form

3.1.2 Define Column Sections

For defining the column sections, use **Define menu**>**Column Section** command (Figure 3-6) to access **Define Column Section** Form (Figure 3-7). The definable column sections are I-Shaped, I/Wide Flanges, Tube Box and Double I Section.

In the **Define Column Section** Form, there are four buttons to **Import**, **Add**, **Modify or Delete** column sections. The required geometric parameters to define these sections are explained below.

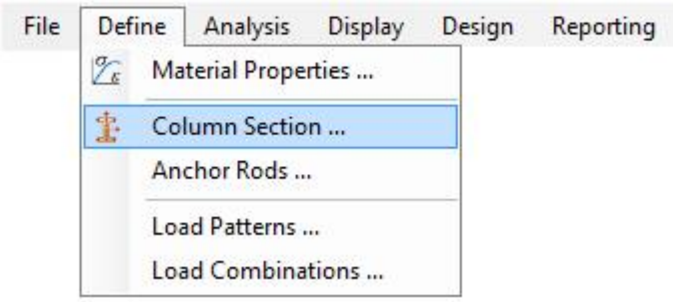


Figure 3-6, Open Define Column Section Form

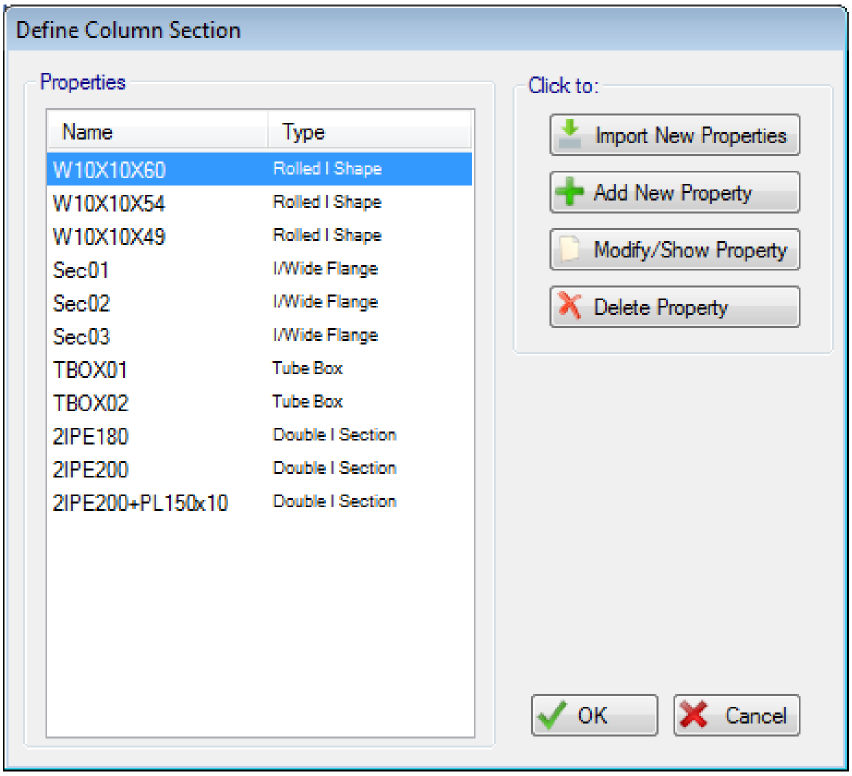


Figure 3-7 , Define Column Section Form

I-Shaped Sec. Parameter	Description
- Outside Height	It is the outer distance between flanges of cross-section which must be greater than the sum of thicknesses of two flanges.
- Flange Width	It is the flange width which must be greater than web thickness.
- Flange Thickness	It is the flange thickness and the sum of both flanges thickness must be less than the outside height of section.
- Web Thickness	It is the web thickness which must be less than the flange width.

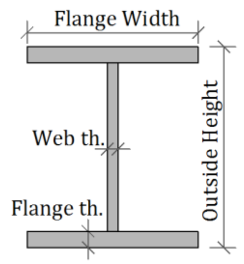


Figure 3-8, I-Shaped Sec.

I/Wide Flange Sec. Parameter	Description
- Outside Height - Top Flange Width - Top Flange Thickness - Web Thickness - Bottom Flange Width - Bottom Flange Thickness	All parameters of I/Wide Flange section are the same as I-shaped section except that dimensions of each flange can be defined, separately (Figure 3-9).

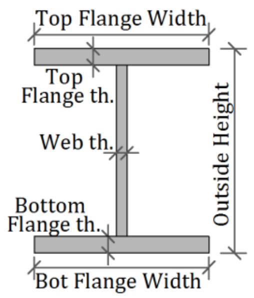


Figure 3-9 , I/Wide Flange Sec.

Tube Box Sec. Parameter	Description
Web Height	see Figure 3-10
Flange Width	see Figure 3-10
Flange Thickness	see Figure 3-10
Web Thickness	see Figure 3-10
Web-Flange Overlap	It is the overlap length of web and flange plates. The maximum value is equal to the flange thickness and the minimum value is zero.

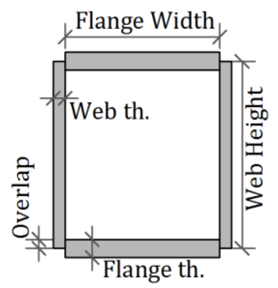


Figure 3-10 , Tube Box Sec.

Double I Sec. Parameter	Description
Initial Dimensions	The initial dimensions of this section are those of I Rolled Shape section.
Center to Center/Closest Distance	To determine the distance of each I shape section to the other, one can use the center to center distance or closest distance.
Include Flange Cover Plate	It is possible to define the cover plates which are laid above and below the flanges.
Cover Plate Width	The cover plate width must be greater than the closest distance between two single sections.
Cover Plate Thickness	-

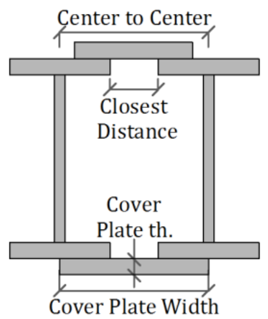


Figure 3-11 , Double I Sec.

- Import Section

The property data of current global steel sections are gathered in software's database for the sake of simplicity in modeling. The current database consists of:

Database	Available Sections
Arcelormittal Products for Europe	IPE , IPEA , IPEAA , IPEO , HEAA , HEA , HEB , HEBC , HEM , HE , HL , HD , HP
Arcelormittal Products for Russia	B
Arcelormittal Products for Japan	H
Arcelormittal Products for USA (ASTM A913)	W , HP
Arcelormittal Products for USA (ASTM A913M)	W , HP
Arcelormittal Products for USA (ASTM A992)	W , HP
Arcelormittal Products for USA (ASTM A992M)	W , HP
Chinese GB08 database	HW , HM , HN , HT , YB-H , YB-LWH , YB-WH , JG-LH , GB-I , YB-I
Euro Database	ILS , IPE , IPEO , IPER , IPEV , HEA , HEB , HLS , H
Indian Database	ISJB , ISLB , ISMB, ISWB , ISHB
Iranian Database	IPE , HEB , INP

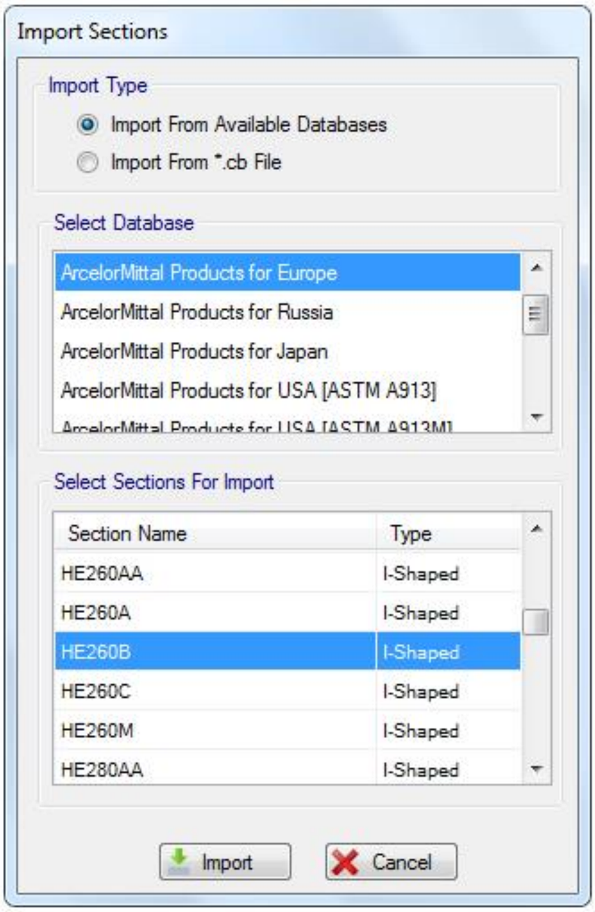


Figure 3-12, Import Sections Form

3.1.3 Define Anchors

For defining the anchor rods, use Define menu>Anchor Rods command (Figure 3-13) to access Define Anchor Rods Form (Figure 3-14). In the Define Anchor Rods Form, there are three buttons to Add, Delete or Import items from available databases.

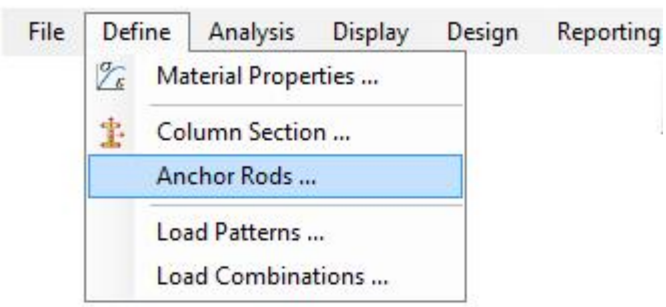


Figure 3-13, Open Define Anchor Rods Form

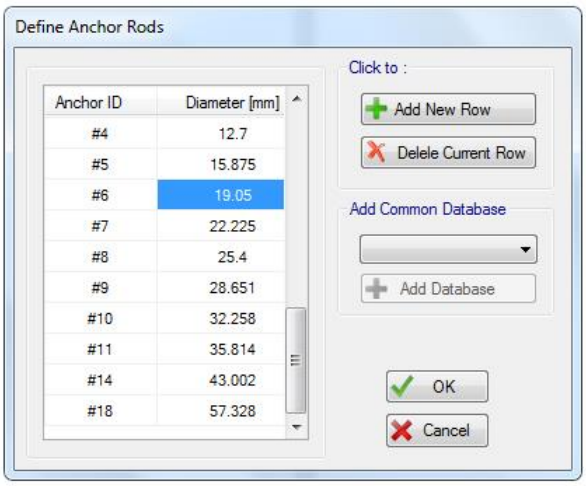


Figure 3-14, Define Anchor Rods Form

3.1.4 Define Load Patterns & Load Combinations

The definition of Load Patterns and Load Combinations is necessary to assign loads on a connection. Load Patterns are used to define Load Combinations. After defining the load combinations, user can select load combinations for analysis.

For defining the Load Patterns in the software's main window, use **Define menu**> **Load Patterns** command (Figure 3-15) to access **Define Load Patterns** Form, (Figure 3-16).

In the **Define Load Patterns** Form, there are the possibilities for **Adding to**, **Deleting or Modifying** (name or type) of a load pattern. For modifying a defined load pattern, user can double-click on the intended cell and change values to the desired ones. The type of load can be Dead, Live, Earthquake, Wind, Snow, etc.

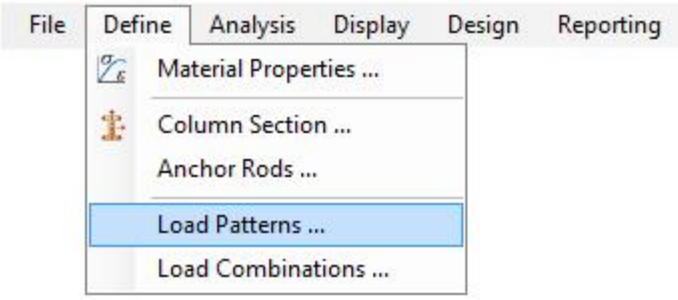


Figure 3-15, Open Define Load Patterns Form

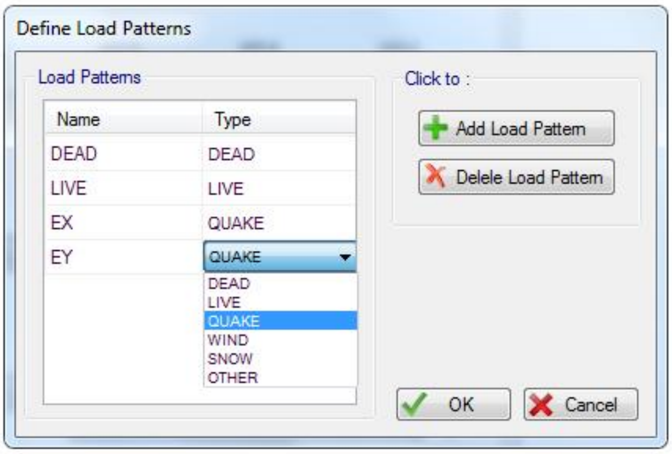


Figure 3-16, Define Load Patterns Form

If a repetitious name for a Load pattern is set by user, it will be modified automatically by the software.

For defining the Load Combinations in the software's main window, use **Define menu**> **Load Combinations** command (Figure 3-17) to access **Define Load Combinations** Form, (Figure 3-18).

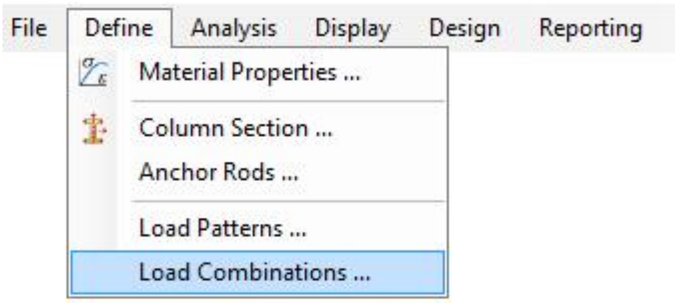


Figure 3-17, Open Define Load Combinations Form

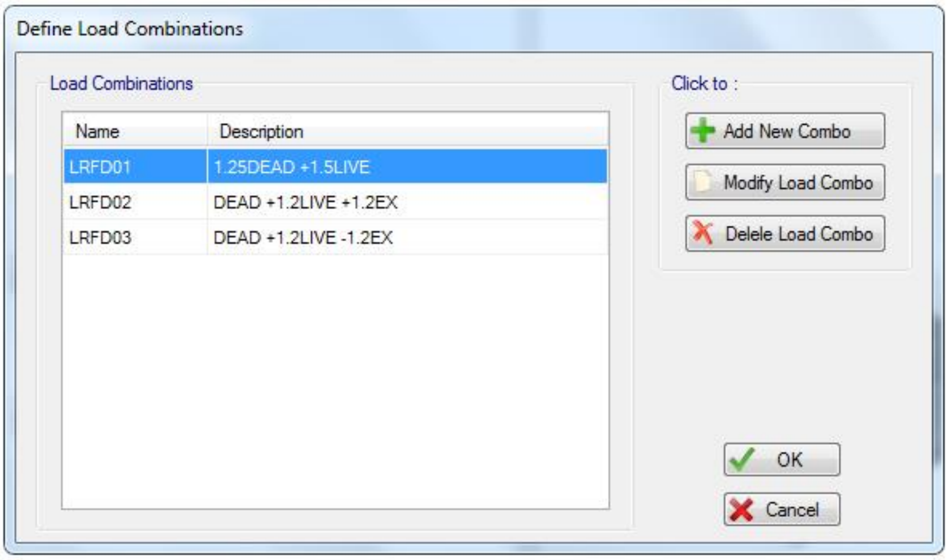


Figure 3-18, Define Load Combinations Form

Name and description of each load combination are shown In the Load Combination Table (Figure 3-18).

There are the possibilities for **Adding, Modifying or Deleting** a Load combination. Clicking the Add New Combo button, the Load Combinations Form is appeared (Figure 3-19) in which the name of load combination, type of load combination and load patterns coefficient can be defined. One can modify the name or load patterns coefficients by double-clicking the intended load combination or click on Modify Load Combo button.

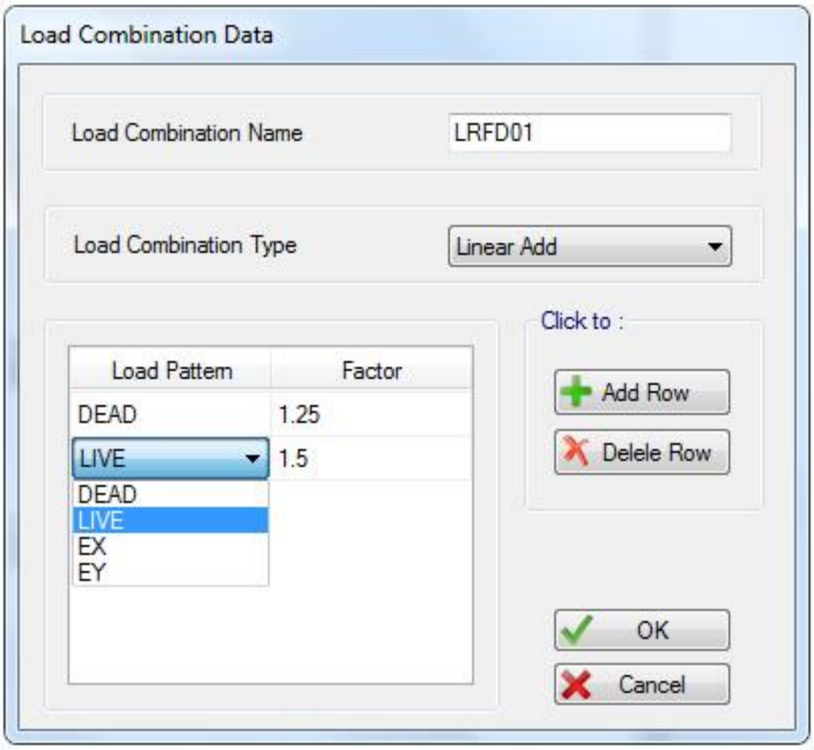


Figure 3-19, Load Combination Data Form

If the name of a new load combination is repetitious, an alarm massage is appeared.

3.2 Start Model Definition

The connection modeling is done by **Inputs** part shown in Figure 2-1. The connection modeling consists of assigning the properties of plate, column, stiffeners, anchor rods, footing, and grout and finally, load assigning for the connection. There are five main tabs in **Inputs** part (Figure 2-1) consists of General, Stiffeners, Anchor Rods, Footing and Loading tab, respectively.

3.2.1 Base Plate Properties

In General tab, the material, dimensions and thickness of plate are determined (Figure 3-20)

-Plate Material: Only steel type materials are shown in this section. It should be mentioned that the modulus of elasticity of base plate must be greater than zero; otherwise the analysis will not run. For defining the materials, refer to section 3.1.1.

-Plate Width, Length & Thickness: In the current version, only the rectangular base plates can be modeled. Figure 3-21 shows the base plate dimensions (width and length). The width and length of base plate cannot be less than the total width and height of column. The maximum width and length of base plate is limited by 100 inches (2540 mm) and the maximum thickness of base plate is 4 inches (101.6 mm). The limitations are applied to limit the required analysis time.

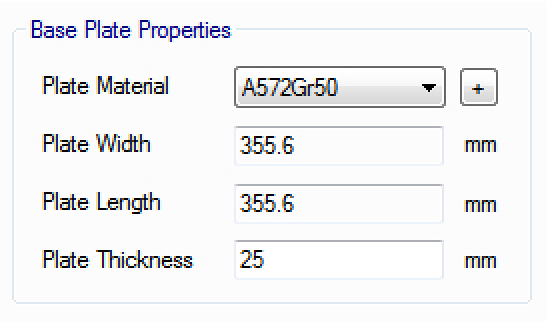


Figure 3-20, Assign Base Plate Properties

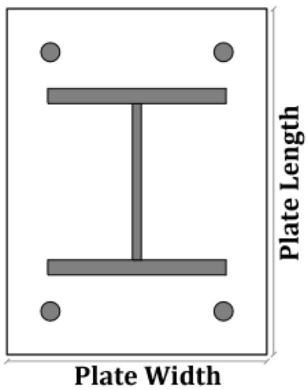


Figure 3-21, Plate Dimensions definition

3.2.2 Column Properties

Column properties can be assigned in **General** tab and Material, section and height of the column are determined. Furthermore, eccentricity for column with several typical positions and custom eccentricity can be defined (Figure 3-22).

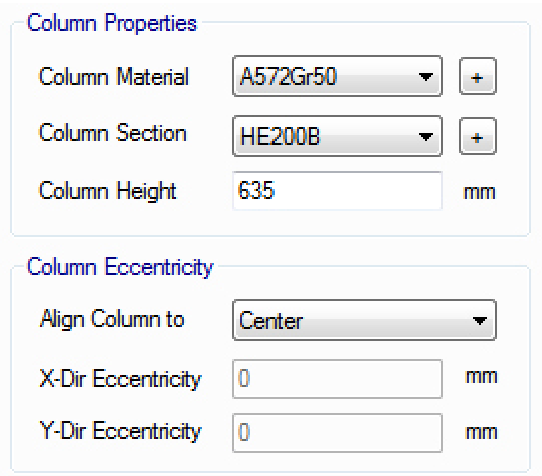


Figure 3-22, Assign Column Properties

- Column Material: This section is used to determine the column material. Only steel type materials are shown in this section. If the modulus of elasticity of column is set to zero, the column will be not modeled in finite element model, then loads of connection is applied above the base plate (in the intersection of column section and base plate). Although this trick leads to a faster analysis, it is not recommended because of low precision respect to the real behavior of connection.

- Column Section: Column sections defined in section 3.1.2 are shown in Column Section list (Figure 3-22). By changing the cross section of column, geometrical check will be done automatically and column dimensions, anchor bolts distance, and assigned stiffeners arrangement will be checked. If the dimensions of column are greater than those of base plate, the base plate dimensions are increased such that the column can be laid on it.

- Column Eccentricity: User can determine the position of column on the base plate without any restriction. Setting the position of column at the Center, Top Edge, Right Edge, Bottom Edge and Left Edge of the base plate is performed. In the Custom Eccentricity mode, one can enter desired value of eccentricity in both X and Y directions. These values of eccentricities correspond to the distance between the center of base plate and the center of a rectangle which surrounding the column (Figure 3-23)

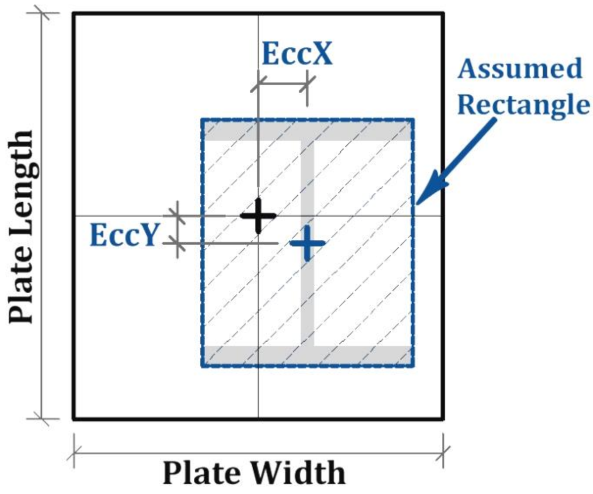


Figure 3-23, Column eccentricity definition

In 2D View tool strip menu, by selecting **Set Display Dimensions> Show Column Location on Plate**, column position related to plate edges will be shown. One can set the position of column respect to the base plate edges.

3.2.3 Stiffeners Properties

Modeling and analysis of rectangular stiffeners in the connection model is possible. This feature is optional and the user decides whether to model stiffeners or not. In the Stiffener tab, user can determine stiffener material, height and arrange them in connection model (Figure 3-24).

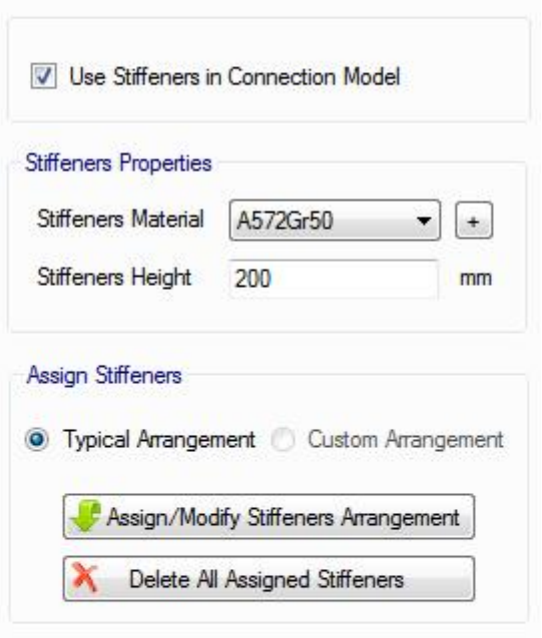


Figure 3-24, Assign Stiffeners Properties

3.2.3.1 General

Modeling of stiffeners in the connection model is optional. By checking 'Use Stiffeners in Connection Model' checkbox, modeling of stiffeners can be done. If one or more stiffeners were assigned previously and then, user unchecks this checkbox, all of the previous assigned stiffeners will be deleted.

- Stiffeners Material: This section is used to determine the stiffeners material. Only steel type materials are shown in this section. If the modulus of elasticity of stiffeners is set to zero, the stiffeners will be not modeled in finite element model.

- Stiffeners Height: This section is used to determine the stiffeners height. As mentioned above, only rectangular stiffeners can be modeled and heights of all assigned stiffeners are equal and cannot be different.

3.2.3.2 Typical Arrangement

Now, only typical arrangement of stiffeners is provided. By clicking 'Assign/Modify Stiffeners Arrangement', 'Assign Typical Stiffeners' Form is appeared. For each type of column section (Defined in section 3.1.2), special typical arrangement for stiffeners is available. The available stiffeners arrangements will be updated automatically when the column section type changes.

- Stiffeners Thickness: In this section, thickness of stiffeners is determined. In typical arrangement, the possibility to assign of different thicknesses for stiffeners is not provided.

- Stiffeners Arrangement: Due to chosen thickness for stiffeners, column section type, column dimensions and its eccentricity, program determines available stiffeners arrangements for connection model.

> If one or more stiffeners are assigned previously, by clicking OK, a message box will appear to inform you that by assigning new stiffeners arrangement, previous stiffeners will be deleted. Also, if one or more stiffeners violated with anchors, user must decide to delete violated anchors or choose another stiffeners arrangement.

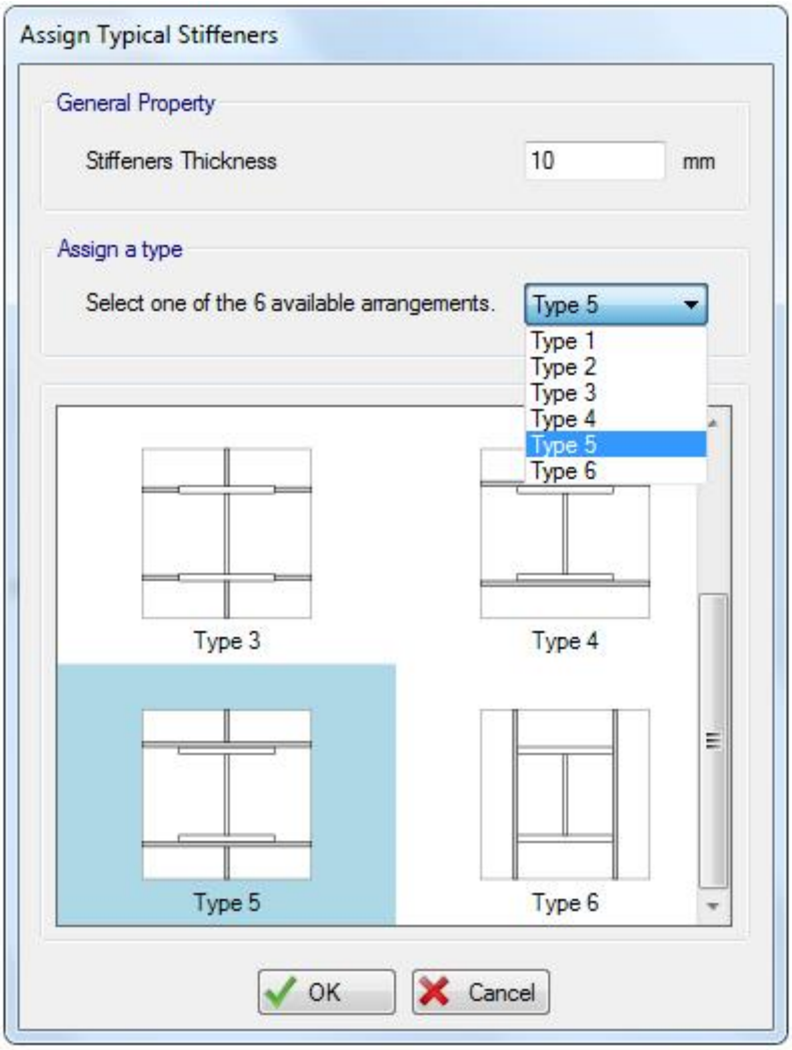


Figure 3-25, Assign Typical Stiffeners Form

3.2.4 Anchor Bolts Properties

The possibility to model, analysis and design of cast-in place anchors is provided. In the **Anchor Rods** tab, Anchors material, geometric properties and arrangements of anchors on connection are determined (Figure 3-26)

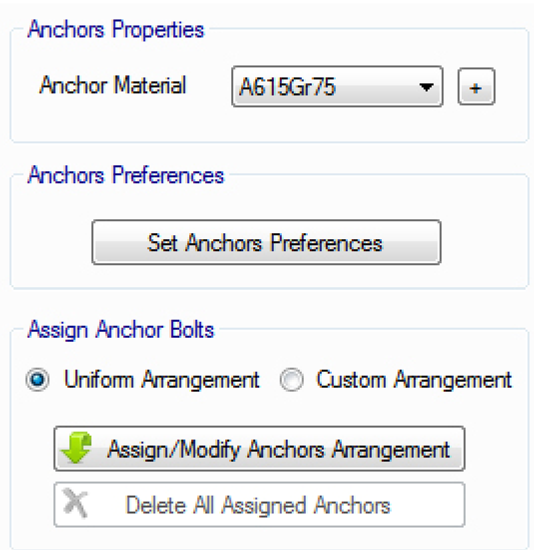


Figure 3-26, Assign Anchor Bolt Properties

3.2.4.1 General

- Anchor Material: Only those materials defined as Anchor Bolt are shown in this section (Figure 3-26). The modulus of elasticity of anchors must be greater than zero otherwise the analysis will not run.

- Anchor Preferences: By clicking on Set Anchor Preferences button in Anchor Rods Tab (Figure 3-26), Anchor Preferences Form is appeared (Figure 3-27). In this Form, Anchors type, length, and spacing control provisions are determined.

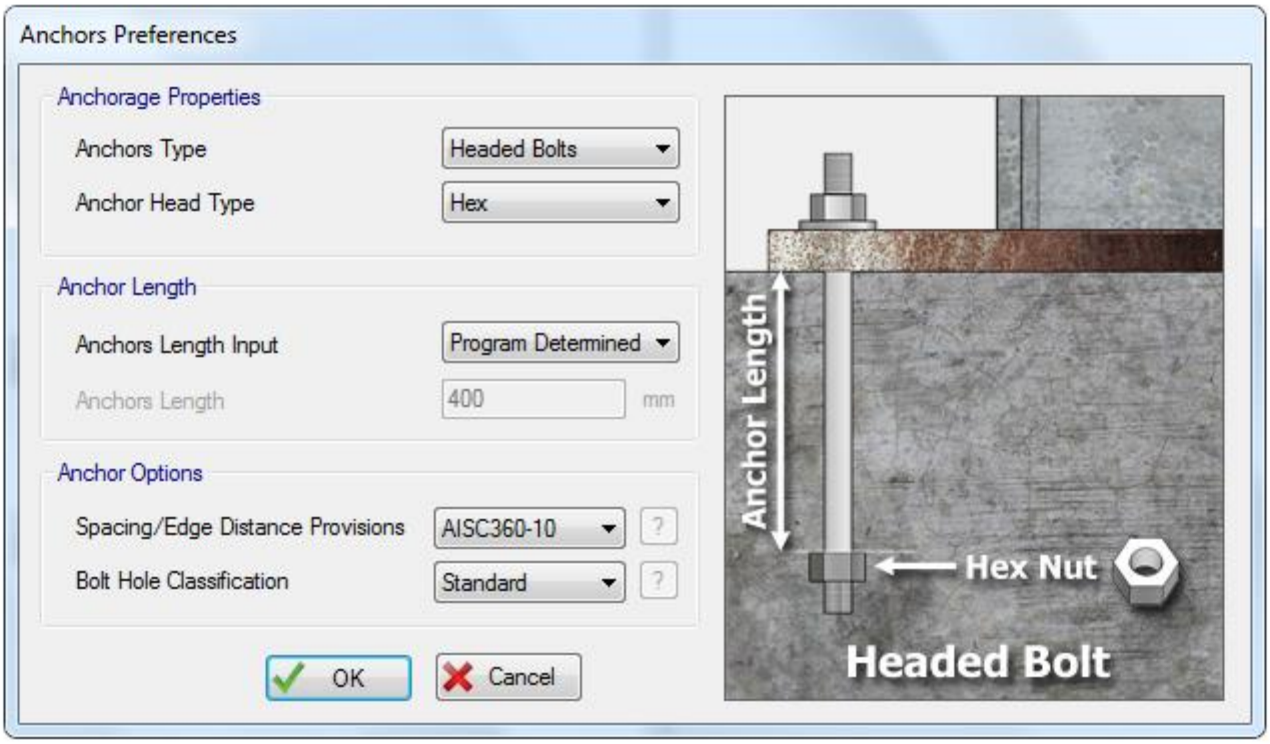
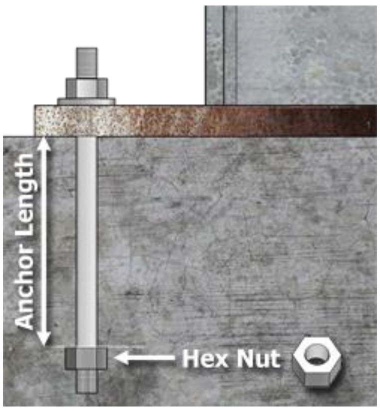
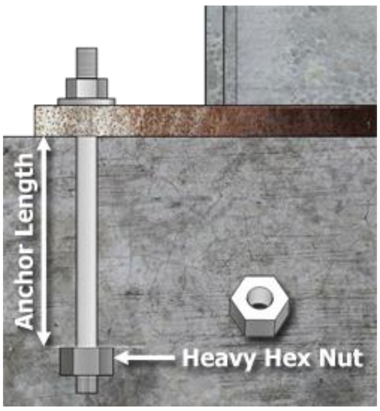


Figure 3-27, Anchor Preferences Form

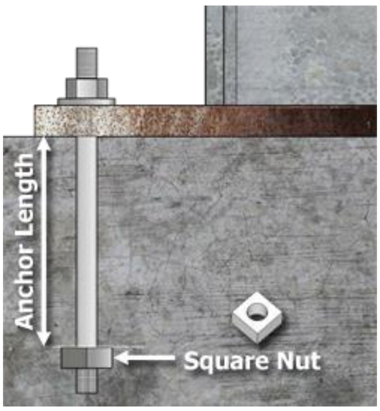
- Anchor Type: Both headed and hooked anchors can be modeled. Although, the performance of headed anchors because of their high capacity especially in seismic regions is more than that of the hooked ones, using the hooked anchors with larger diameter can be more economic than the Headed ones. So for covering all cases, modeling of both anchors can be done in this software.



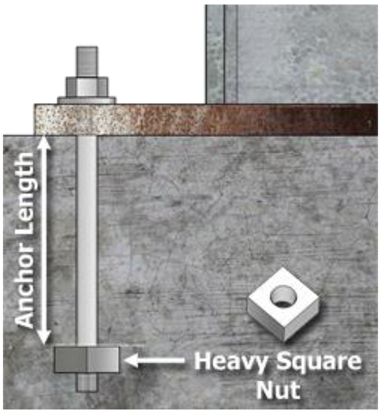
a) Headed Bolt with Hex nut



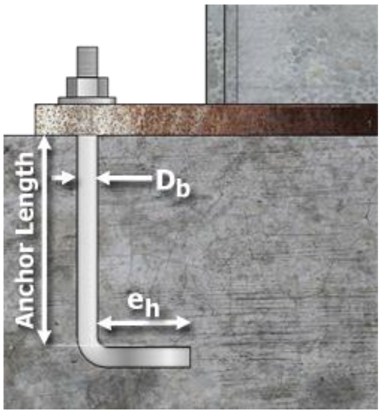
b) Headed Bolt with Heavy Hex nut



c) Headed Bolt with Square nut



d) Headed Bolt with Heavy Square nut



e) Hooked L Bolt

Figure 3-28, Anchor Bolts Types

-Anchor Head Type: If the Headed Anchors option is chosen, the anchor head type buried in concrete should be selected, too. User can select any kind of head type such as Hex, Heavy Hex, Square and Heavy Square. For more information about Anchor Heads refer to ACI318-14 & PCA 2008.

eh/Db Ratio: If the Hooked Anchors option is chosen, the eh/Db ratio should be determined. The 'eh' parameter is shown in Figure 3-28(e) for L-shaped anchors. This parameter is used to calculate pullout strength of hooked anchor and in ACI318 code one restriction is mentioned for it: 3Db<eh<4.5Db. The reason for determining such minimum and maximum value for eh is that the boundaries of experimental specimens tested by Shaikh and Kuhn 1996 have such values and consequently, the code relations are derived from the result of these tests. For more information refer to R17.4.3.5 ACI318-14 Commentary. It should be mentioned that one can choose a value greater than the maximum, but for calculating the pullout strength of anchors, the maximum allowable value for eh (i.e. 4.5Db) will be used.

- Anchor Length: In the connection model, the buried lengths of all anchor rods are considered as equal. These lengths for different anchor bolt types are shown in Figure 3-28. If user chooses the Program Determined option for anchor length, the software will determine the lengths of rod anchors. In this mode, software considers the lengths of anchor rods 20 times as equal as the maximum diameter of rods and this length is shown in the Default Length field. If user chooses the Specified By User option then the anchors length must be specified by user.

In the analysis procedure, it is assumed that the failure mode of anchors is occurred because of steel failure in tension and shear modes. So after the analysis and design of the connection, user must check the concrete capacity for avoiding the brittle rupture.

- Space/Edge Distance Provisions: For anchor rod positioning on the base plate, the minimum distance of anchor rod from the plate, column and stiffeners edge and the other anchor rods must be observed. Now, these restrictions are checked based on AISC360-10, J3.3 and J3.4. If user chooses None option, the aforementioned restrictions are not checked. In this mode the minimum space of anchor rods from each other and edges is limited to Db (i.e. the anchor rod diameter).

- Hole Classification: It is necessary to determine the hole classification type, if AISC360-10 is chosen for Spacing Provisions section. Now, two types of holes (Standard and Oversized) can be chosen. If in the Spacing Provisions section, the 'None' option is selected, only standard hole is available.

- Assign Anchor Bolts: To model the anchor rods of a connection, one can use Uniform Assignment or Custom Assignment (Figure 3-26). Uniform Assignment is used for positing of anchor rods with ease and speed, but also with some restrictions. On the other hand, the Custom Assignment is a general method without any restriction in which all coordinates of each anchor rod can be determined. In both modes, the graphical environment leads to easy positioning of anchor rods.

3.2.4.2 Uniform Arrangement

By choosing Uniform Arrangement in Anchor Rods tab and clicking Assign/Modify Anchors Arrangements button, Uniform Arrangements Form is appeared (Figure 3-29). Respect to column position, base plate surface divided into four areas (Zones 1, 2, 3 & 4). The whole column is embedded in a virtual rectangle and edges of virtual rectangle are assumed as column edges. Based on this assumption, the distances of anchor rods are determined.

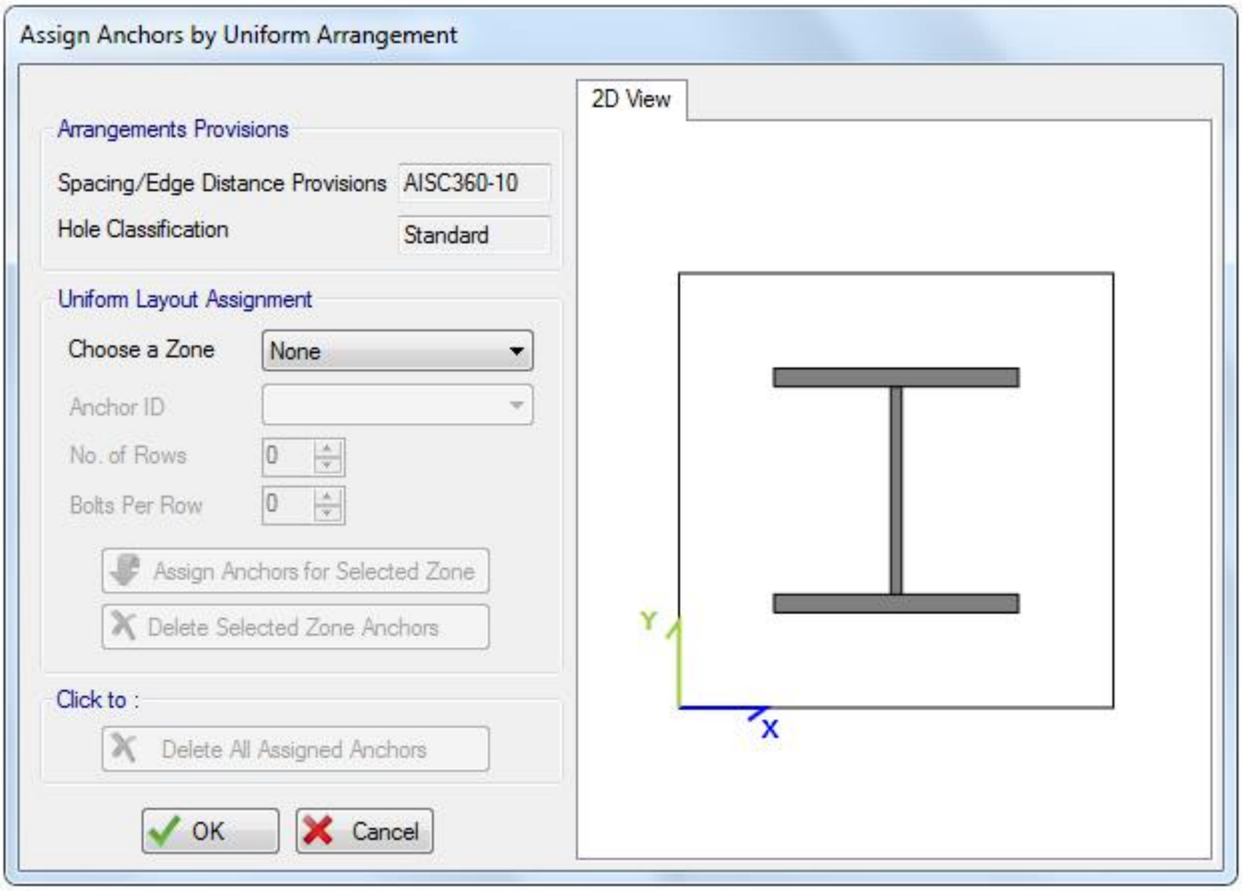


Figure 3-29, Assign Anchors by Uniform Arrangement From

The restriction of Uniform Assignment method is that it can not be used when one or more stiffeners are assigned. Also user has to assign anchors with the same diameter in each zone and assign of anchors outward of defined zone is not performed.

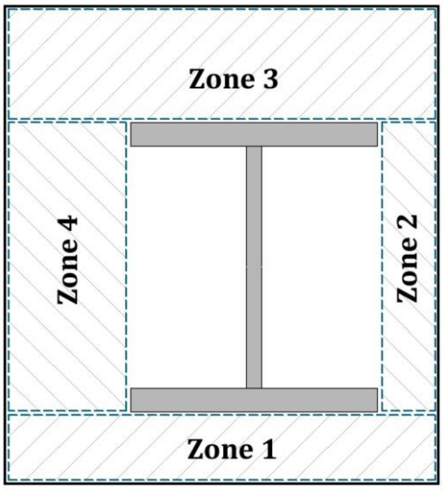


Figure 3-30, Defined zones for uniform assignment of anchors

- Choose a Zone: After selecting a zone, it becomes highlighted and then one can assign the anchors in that. It should be mentioned that in the Uniform method in each zone, the diameter of all anchor rods are equal.

- Anchor ID: Those anchor rods defined in the section 3.1.3 are shown in this list. From this list, any Anchor ID can be chosen for assigning to selected zone.

- No. of Rows/Bolts per Row: The number of anchor rod rows and the number of anchor rods in each row can be determined in each zone. The max and min values are calculated and applied automatically by program. These values depend on the selected anchor rod diameter, the criteria of controlling the anchor rod distances, type of holes and the zone's dimensions.

-Assign Anchors for Selected Zone: In each zone for assigning the anchor rods to connection model, after selecting the intended zone and determining the diameter and number of anchor rods, click on this button.

-Delete Selected Zone Anchors: If User wants to delete or modify the assigned anchor rods in each zone, first he must select the intended zone and then click on the Delete Selected Zone Anchors.

- Delete All Assigned Anchors: For deleting all of the assigned anchor rods in the model connection, user can click on Delete All Assigned Anchors.

3.2.4.3 Custom Arrangement

By choosing Custom Arrangement in Anchor Rods tab and clicking Assign/Modify Anchors Arrangements button, Custom Arrangements Form is appeared (Figure 3-31). It is a general method for assigning the anchor rods that their positions are determined based on the coordinates entered by user.

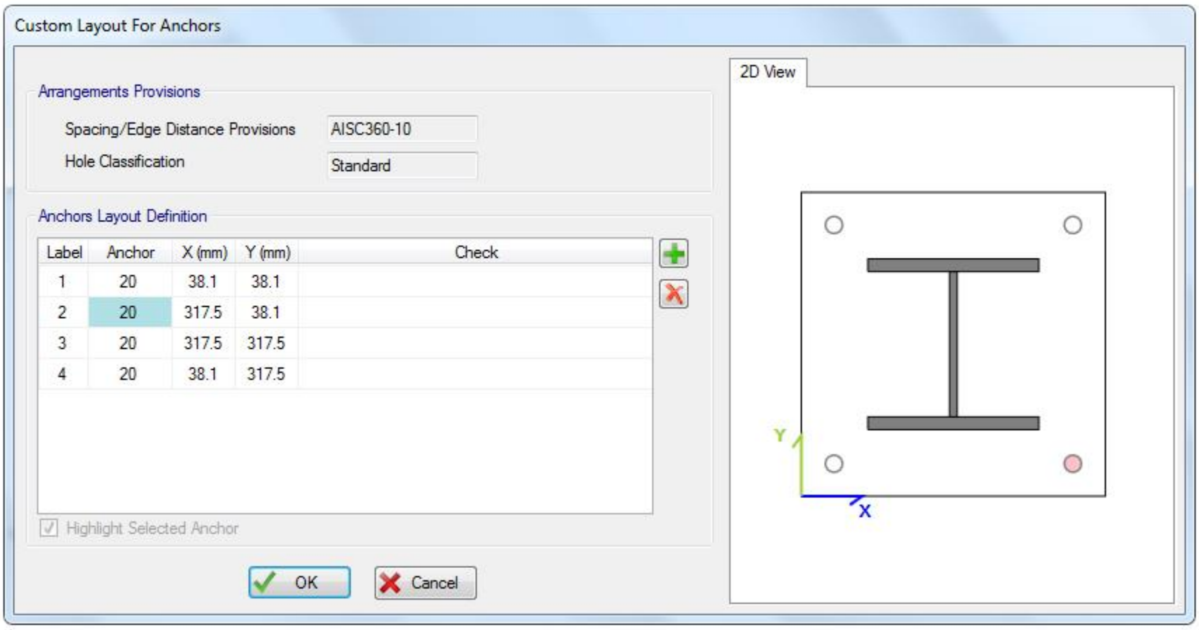


Figure 3-31, Assign Anchors by Custom Arrangement From

By clicking on (+) button, a new anchor rod is added to the connection. For deleting each anchor rod, first select the intended anchor rod and then click on the (x) button. After selecting each row of Anchors Layout Definition table, the related anchor rod is highlighted. The columns of Anchor Layout Definition are explained below.

- Label: The first column of this table shows the index of anchor rods. The indexing is done automatically and is based on the sequence of definition.

- Anchor: In this column, the anchor's ID defined in section 3.1.3 is specified. For each anchor, user can modify the Anchor's ID by double clicking on this cell.

- X, Y: In these columns, the coordinates of anchor rod respective to the origin can be determined.

- Check: Based on the distance check provision of anchor rods, the position of anchor rod respective to column, stiffeners, plate edges and the other anchor rods are investigated and if an error is found, the error message appears in this column. The error messages are described in table below.

Message	Description
Minimum edge distance has not been met!	If the minimum distance from edges of column, stiffener and plate is violated, this error message would appear.
Minimum spacing with anchor # has not been met!	If the minimum distance between two anchors is violated, this error message would appear.
Overlapped with anchor #	If two anchor rods have overlap with each other, this error message would appear.
Overlapped with column section	If the anchor rod overlaps the column section, this error message would appear.

If the positions of one or more anchor rods have an error, move/delete it or them respectively.
By clicking OK, the program will delete illegal anchors if any error found.

3.2.5 Footing Properties

Concrete footing is modeled as a separate part in connection model and no simplification using independent springs or etc. is not done in modeling. Common simplifications in using independent springs are not global and decrease in accuracy of modeling compared to real behavior of connection. In the Footing tab, footing properties can be determined (Figure 3-32).

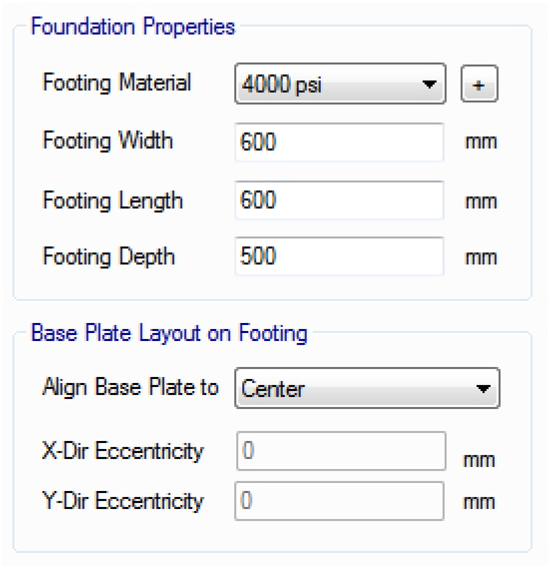
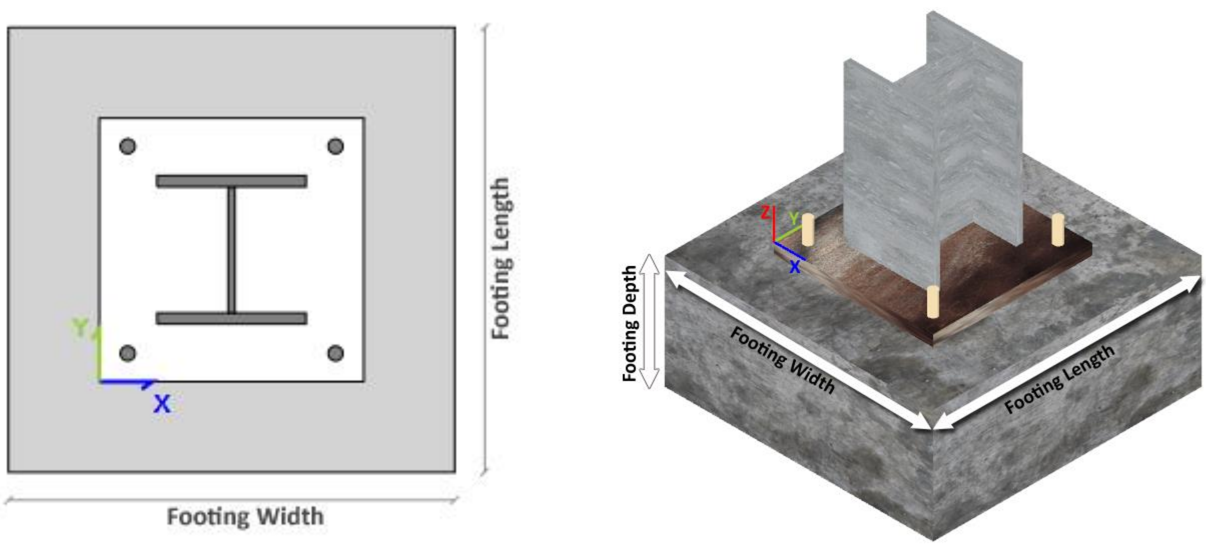


Figure 3-32, Assign Footing Properties

- Footing Material: Only concrete type materials are shown in this section. The analysis would not run if the footing modulus of elasticity is set to zero.

- Footing Width, Length & Depth: Now, only rectangular footing can be modeled. The length and width of such footing are shown in figure 3-33. The maximum and minimum width and length ratio of footing dimensions correspond to base plate ones is limited to 10 and 1 respectively.



a) 2D View b) 3D View

Figure 3-33, Footing Dimensions

-Base Plate Layout on Footing: The position of column-base connection on footing is determined without any restriction. User can put the base plate at middle, top edge, left edge, right edge and bottom edge of the footing with ease. User can also put the base plate on the footing based on the desired eccentricity in X and Y direction using Custom Eccentricity mode. The distance between the base plate center and footing center is interpreted as eccentricity.

In 2D View tool strip menu, by selecting **Set Display Dimensions> Show Plate Location on Footing**, plate position related to footing edges will be shown. One can set the position of plate respective to footing edges.

3.2.6 Grout Properties

This feature provides the possibility of modeling and analysis of concrete grout below base plate. It is optional and the user decides whether to model this part or not. Similar to footing, concrete grout is modeled as a separate part and no simplification is made in its modeling. In the Footing tab, by choosing 'Use Built-up Grout … ' the user can determine the grout material and grout average thickness in the connection model (Figure 3-34). For removing concrete grout from connection model, uncheck the 'Use Built-up Grout …' checkbox.

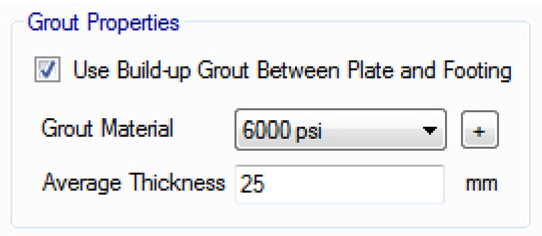


Figure 3-34, Assign Grout Properties

- Grout Material: Only those materials were defined as concrete ones are shown in this list. The grout will not be modeled if modulus of elasticity of the grout is zero. Due to standards recommendations, grout concrete strength shall be at least two times of footing concrete strength.

- Average Thickness: This section is used to determine the average thickness of grout. Grout will not be modeled if the grout thickness is zero.

3.3 Loading

In the loading Phase, one can apply the tension forces (Uplift), compression forces, biaxial moment and shear in both directions on the connection. Also the Loads can be applied in both top of the column and plate surface. Assigning loads and their preferences are done in the Loading Tab (Figure 3-35).

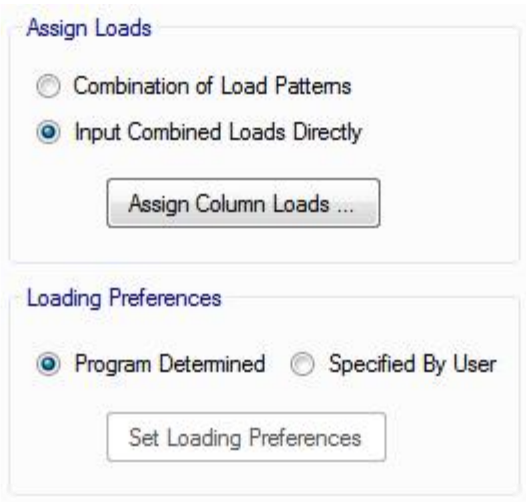


Figure 3-35, Assign Loads

3.3.1 Assign Loads

Two loading methods are provided in the program: Combination of load patterns and entering combined loads directly. In the first method, load patterns applied on the column (such as dead loads, live loads, seismic loads, etc.) are assigned for each load component (axial force, flexural moments and shear loads) and based on the to definition of load combinations, the ultimate loads are calculated by the program. In the second method, the combined loads in each load combination are calculated by user and then, the combined loads for each load combination are assigned to the connection, directly. After choosing the loading method, click the 'Assign Column Loads …' button. Figure 3-36 and figure 3-37 show the 'Assign Column Loading' Form.

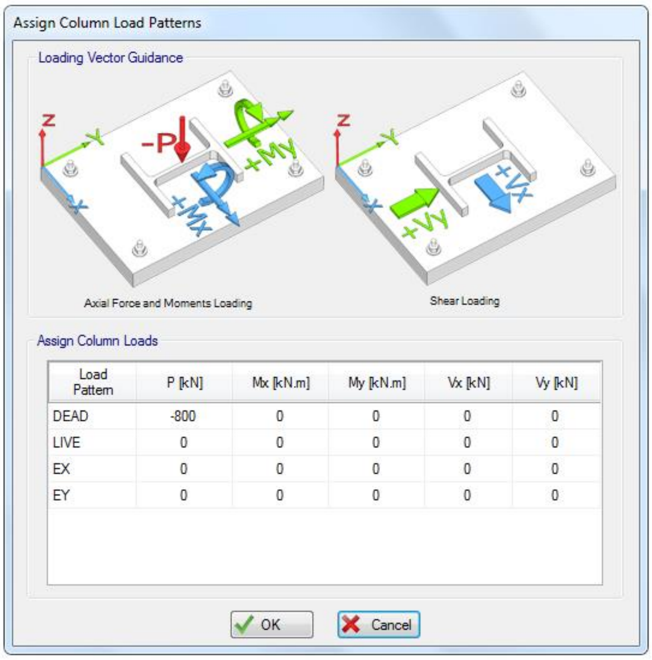


Figure 3-36, Loading by Column Load Patterns

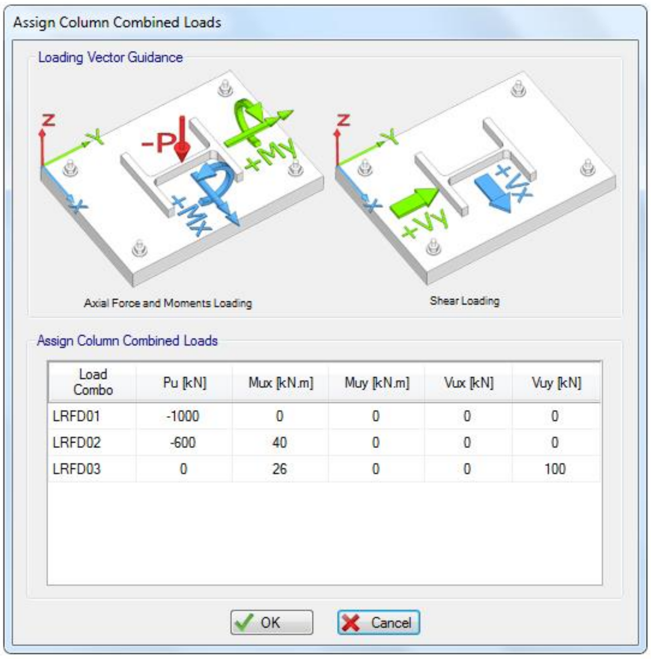
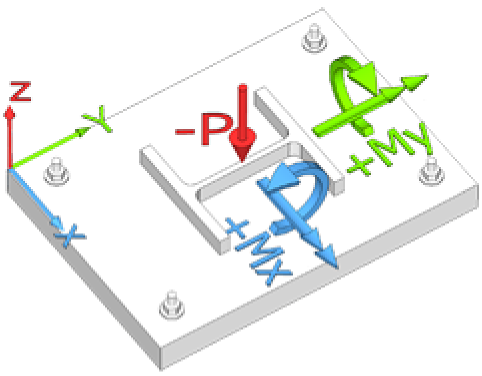
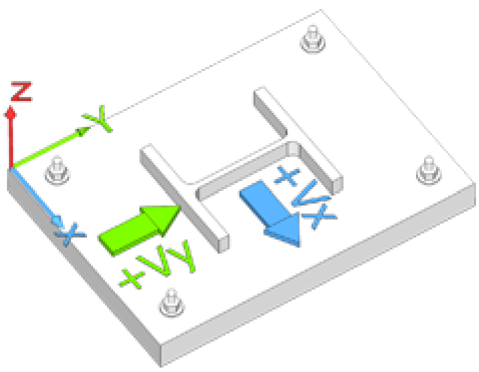


Figure 3-37, Loading by Column Combined Loads

Signs of loading vectors are shown in Figure 3-38. Compression axial force is represented by negative sign and tension axial force is represented by positive sign. The sign of flexural moments in both directions is determined according to right hand rule. When thumb is in positive direction of an axis, rotation of fingers shows the positive sign of flexural moment (Figure 3-38a). Positive sign of shear forces is determined by positive direction of axis, according to Figure 3-38b.



a) Normal forces and flexural moments



b) shear loads

Figure 3-38, Loading vector guidance

3.3.2 Loading Preferences

This section is used to set the elevation for applying loads. As mentioned above, loads can be applied at top of the column or at the plate-column interface. By choosing 'Program Determined' option in 'Loading Preferences' (Figure 3-35), axial force and flexural moments are applied at the top of column and shear forces are applied at the plate surface. For custom preferences, choose 'Specified by User' in 'Loading Preferences' (Figure 3-35) and click 'Set Loading Preferences' button to show the 'Set Loading Preferences' Form (Figure 3-39).

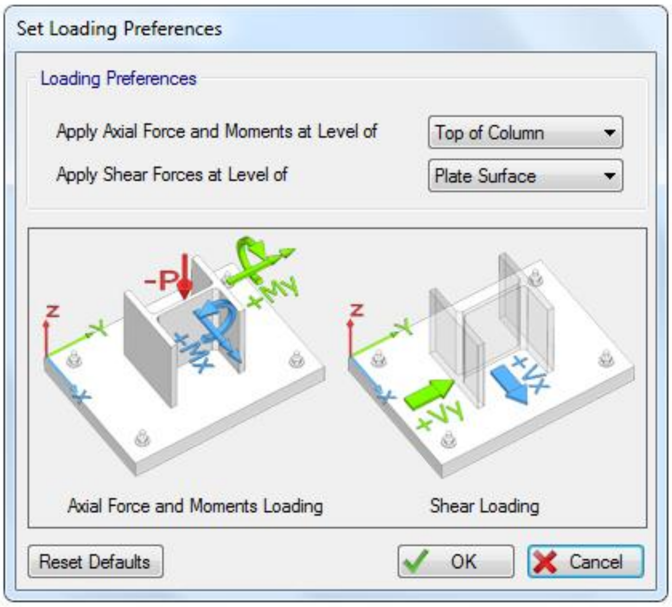
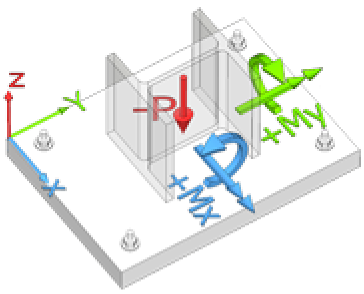
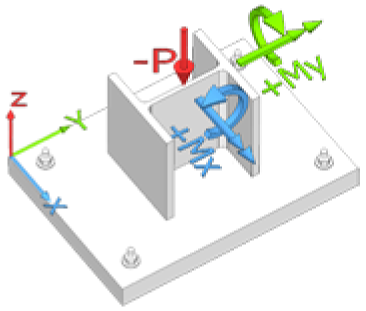


Figure 3-39, Loading Preferences Form

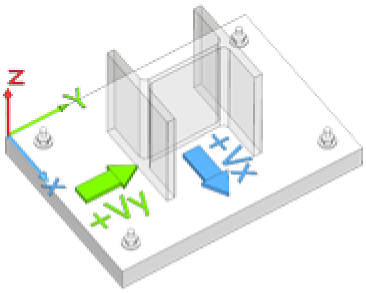
It is recommended to set the elevation level for applying axial force and flexural moments to Top of Column. By choosing this option, analysis results for column and stiffeners (if assigned) are real deformations and stresses and results can be used for the design of column and stiffeners (if assigned) according to standards/codes. For shear forces, considering this point is important that if shear forces are applied at top of the column, additional flexural moments are generated at the level of plate-column interface. So for shear forces, it is recommended always to use Plate Surface for elevation level of applying forces.



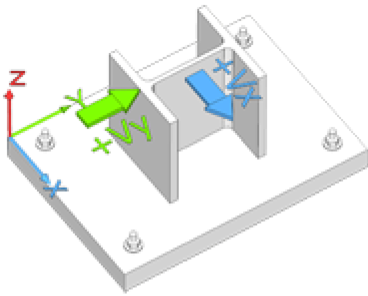
a) Apply Axial Force and Moments at Plate-Column Interface



a) Apply Axial Force and Moments at Top of Column



c) Apply Shear Forces at Plate-Column Interface



d) Apply Shear Forces at Top of Column

Figure 3-40, Loading Preference Options

4. Analysis

Analysis of column-base model is based on elastic behavior of material and infinitesimal deformations. For considering contact between plate and footing/gout interfaces, contact analysis using penalty method is performed.

4.1 Analysis Options

To view Analysis Options Form, in the main window of the program, click Analysis menu and select 'Set Analysis Options' or press F2 Key (Figure 4-1).

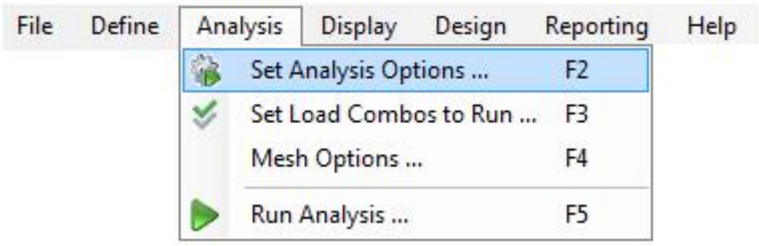


Figure 4-1, Open Set Analysis Options Form

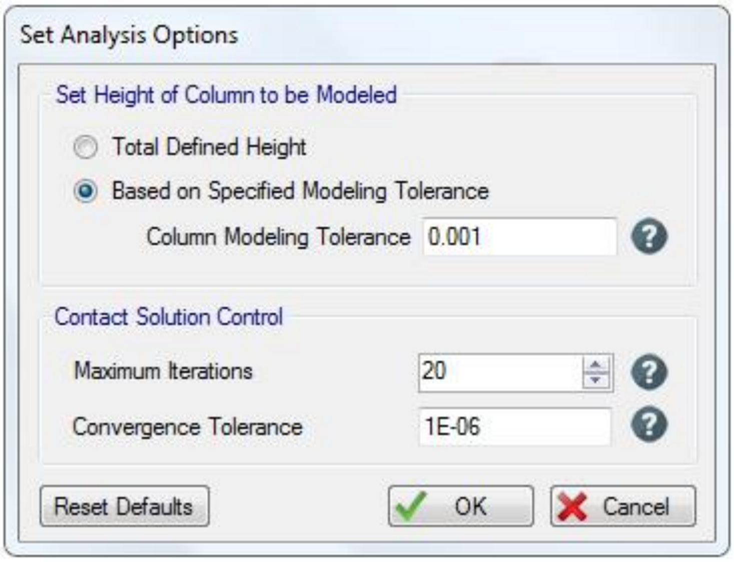


Figure 4-2, Set Analysis Options Form

- Set Height of Column to be modeled: Two options are provided for modeling of column part in the connection model. One can force program to model total height of column that is expensive in terms of time and speed. Another option is to apply an intelligent algorithm to model specified height of column that reaches acceptable tolerance. Surveys show that it is not require to model total height of column in many cases and by setting column modeling tolerance less than or equal to 0.001, analysis results achieve very good accuracy compared to when total height of column is modeled. By this method, less time is required for analysis to achieve the same accuracy and it is recommended by the program.

- Contact Solution Control: As mentioned above, the contact between plate and footing/grout interfaces is modeled using penalty method in finite element modeling. Therefore, it requires several iterations to converge to acceptable accuracy. Maximum Iterations value sets the maximum iteration number used for trying to achieve acceptable accuracy. It is recommended to set this value at least equal to 20 iterations.

After any iteration in contact analysis, residual tolerance is calculated and it is checked by an acceptable convergence tolerance. Convergence Tolerance in Analysis Options (Figure 4-2), is the same acceptable tolerance and is set by user. It is recommended to set this value less than or equal to 1E-6.

- Reset Defaults: To return all items to the program default values, click Reset Defaults button.

4.2 Setting Load Combos to Run

To set load combinations to be involved in connection analysis, in the main window of program, select Analysis menu and then click Set Load Combos to Run or press F3 Key (Figure 4-3).

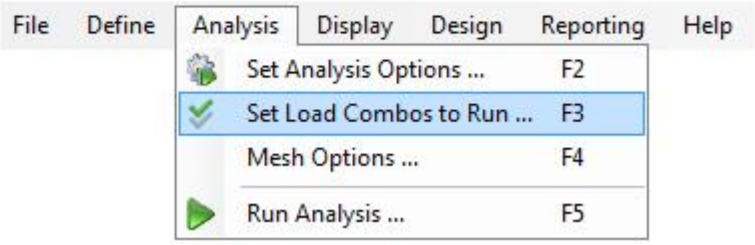


Figure 4-3, Open Set Load Combos to Run Form

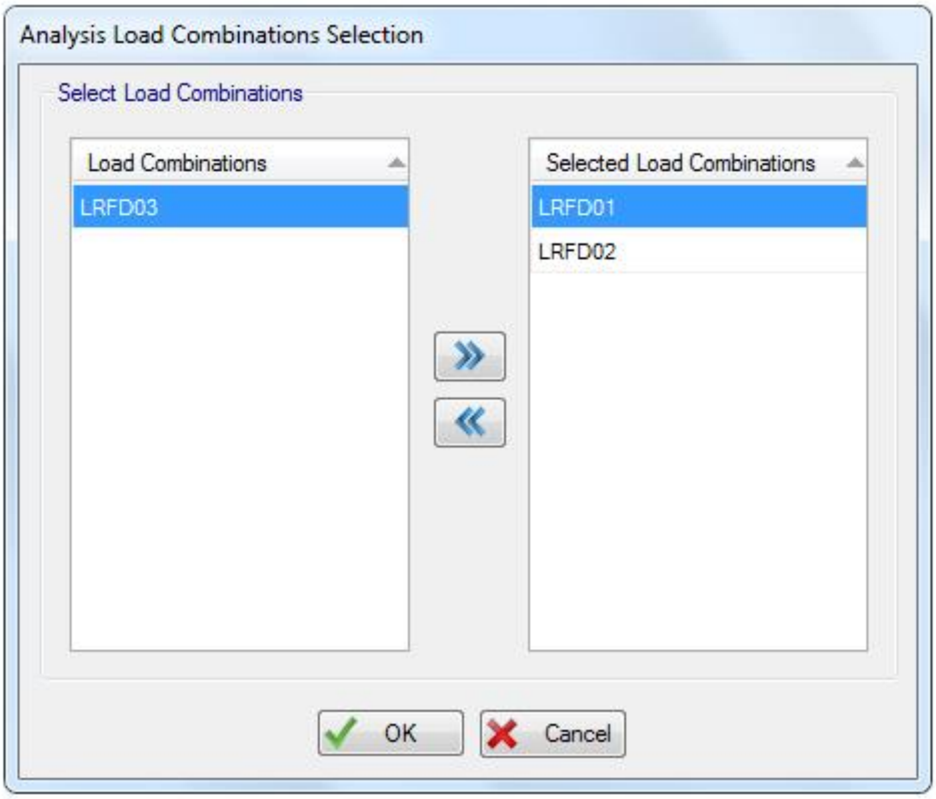


Figure 4-4, Analysis Load Combinations Selection Form

All defined load combinations are shown in the left panel and the selected load combinations are shown in the right panel. One can select one or more load combinations for analysis. There is no restriction in selecting the number of load combinations.

At least one load combination should be selected in the above box, otherwise analysis could not start.

4.3 Mesh Options

Mesh size of all parts is determined with the internal intelligent algorithm before starting the finite element analysis. One can change the recommended mesh sizes determined by the program. For setting the mesh size, from Analysis menu click Mesh Options or press F4 key (Figure 4-5).

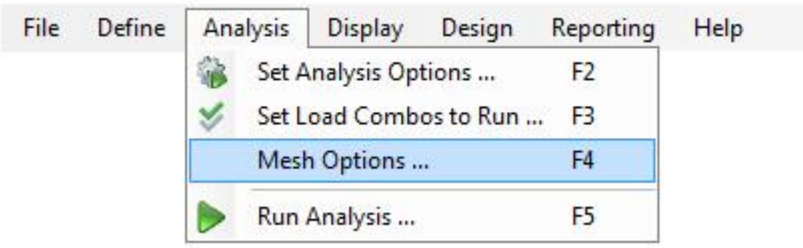


Figure 4-5, Open Mesh Options Form

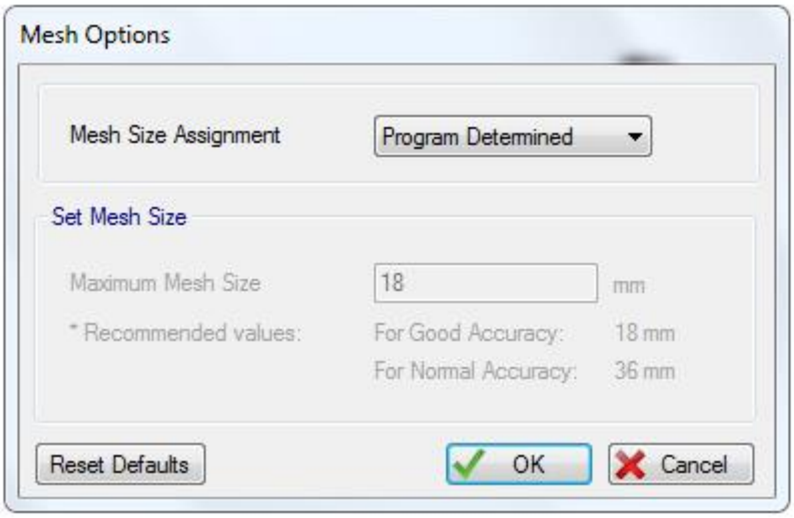


Figure 4-6, Mesh Options Form

In the Mesh Options Form, recommended mesh sizes for good accuracy and normal accuracy are shown. These values depend on dimensions of connection parts and the distance between connected parts (Such as anchors to plate, column and stiffeners edges distance and etc.). By choosing Program Determined option, recommended mesh size for good accuracy is used. Otherwise, one can enter custom mesh size in this field. Entering values more than recommended value for normal accuracy is not allowed.

By clicking Reset Defaults button, mesh size assignment is set to Program Determined and maximum mesh size is modified according to it.

The important point is that the required memory and analysis time depend on the chosen mesh size. Choosing small values for it may causes problems in successful completion of analysis.

4.4 Running Analysis

For running analysis from Analysis menu, click Run Analysis or press F5 key (Figure 4-7)

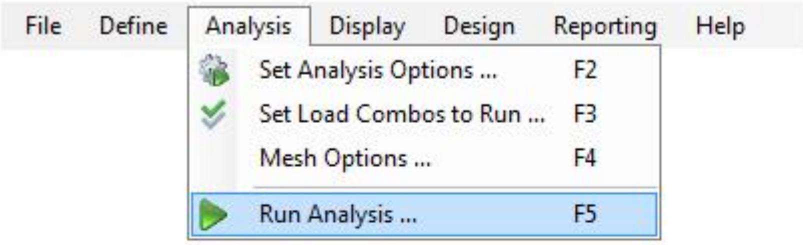


Figure 4-7 Open Run Analysis Form

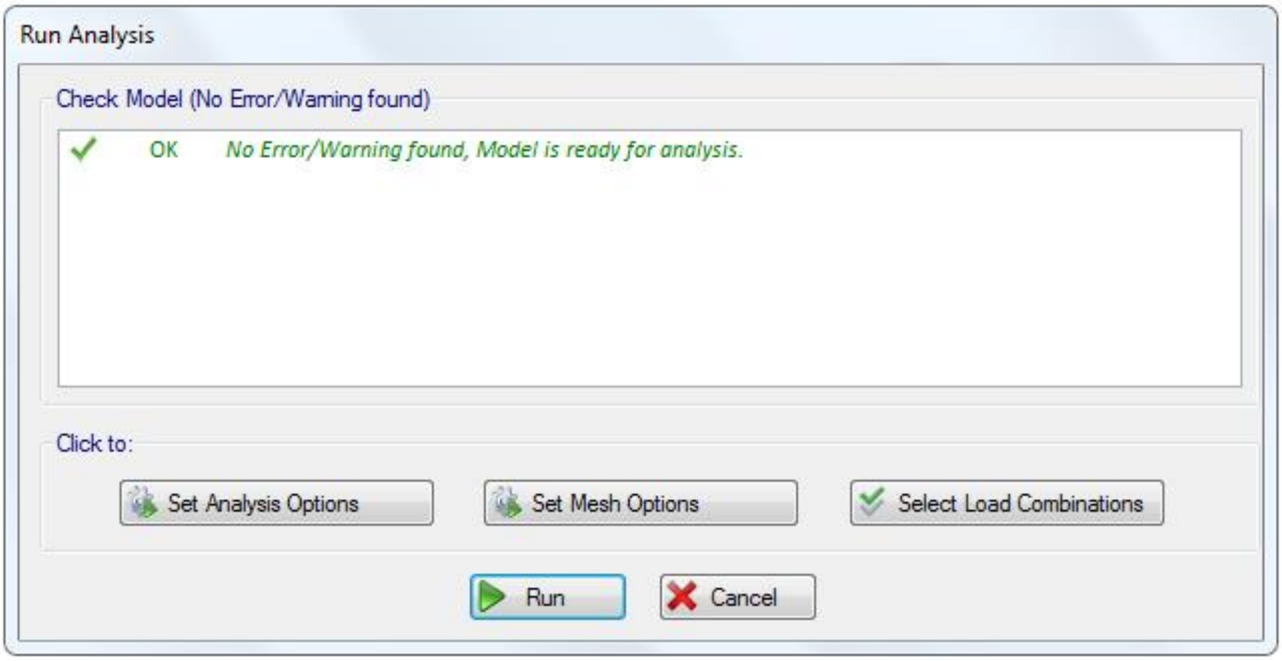


Figure 4-8, Run Analysis Form

Run Analysis Form is shown in figure 4-8. After loading this Form, overall checking is performed, automatically. Model is checked for any geometric problem, problem in user-defined values, estimating pre-analysis problems, etc. Three types of messages may be shown after model checking.

1) Notes: These messages show more information about FEA. These types of messages don't cause any problem in continuing running analysis. For example, if one sets grout thickness to zero, a message note is appeared that informs that grout will not be modeled in FEA, because its thickness is set to zero.

2) Warnings: These messages inform a problem distinguished by the program and don't cause any problem in continuing running analysis. For example, if all components of combined loads for one or more load combinations are zero, a warning message informs this issue.

3) Errors: These messages inform that serious problems are found in the model. As long as the error message appears on the screen, no analysis is permitted to run. For example, if modulus of elasticity for plate material is zero, an error message informs this problem.

Some of messages that may appear after model checking are listed in the table below.

Error/Warning Note	Solution
Error, Assigned material for Plate has zero Modulus of Elasticity.	Modulus of elasticity of plate material should be modified or another material should be assigned for the plate. Modulus of elasticity for steel materials is about 210 GPa or 30000 Ksi.
Error, Assigned material for Plate has incorrect Poisson Ratio.	If poisson ratio of plate material is of unacceptable value, this message is appeared. To fix the error, modify poisson ratio of plate material or assign another material to the plate. Poisson ratio for steel material is about 0.3
Error, Assigned material for Footing has zero Modulus of Elasticity.	Modulus of elasticity of footing material should be modified or another material should be assigned for it. Modulus of elasticity for concrete materials corresponds to its strength. Importing of concrete materials from available databases in program is recommended.
Error, Assigned material for Footing has incorrect Poisson Ratio.	Poisson ratio of footing material is of unacceptable value. To resolve the error, modify the poisson ratio of concrete material or assign another material to footing. Poisson ratio for concrete material is about 0.15 to 0.2
Error, No Anchor Rod assigned, Connection is unstable!	No anchor bolts assigned. So connection is unstable for lateral forces and analysis of model is impossible. For solving the problem, assign anchor bolts to connection model.
Error, Assigned material for Anchor Rods has zero Modulus of Elasticity.	Modify the modulus of elasticity of anchor bolts material. Modulus of elasticity for anchor bolt material is nearly the same as that of steel ones, about 210 GPa or 30000 Ksi.
Error, No load combo selected!	If no load combination is selected for analysis, this message wolud appear. For resolving the error, at least one load combination should be selected.
Warning, For selected load combo ,All factored load components are zero.	If all components of each selected load combo is zero, this message would appear. Check the connection loading for the mentioned load combos.

After model checking, if no error is found, analysis can start. To run analysis, click Run button. By starting the analysis, inputs interface will be locked and one cannot change the inputs data. After analysis completion, one can click Unlock Model in the strip menu to change the inputs. It is important that after unlocking model, all analysis results will be deleted.

4.5 Analysis Run Log

During the analyzing of model, dynamic reporting on what procedure is being done is available. To open this Form during the analysis, click 'Show Details …' in the status bar. After completing the analysis, this Form is available from Display menu> Last Analysis Run Log (Figure 4-9)

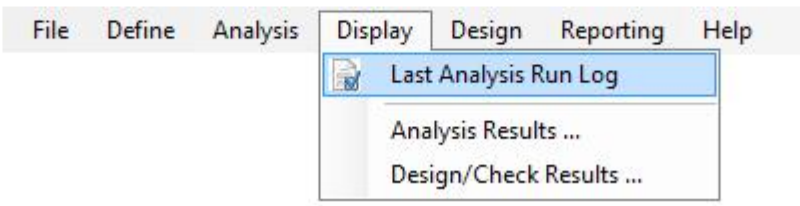


Figure 4-9, Open Analysis Run Log

Details of modeling, analysis, contact analysis iterations and post-processing are reported in this Form (Figure 4-10).

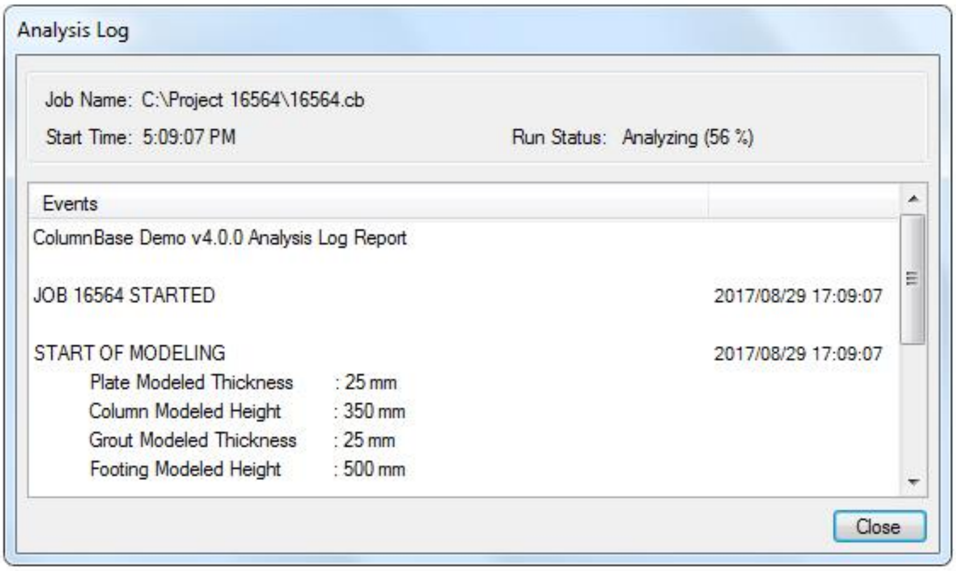


Figure 4-10, Analysis Log Form

5. Post-Processing

In this section, analysis outputs and design results have been explained.

5.1 Analysis Outputs

5.1.1 General

Analysis outputs include deformations and stresses of connection components; anchor bolts forces; final check and reporting of finite element analysis. To see the Analysis Results Form, from Display menu click Analysis Results (Figure 5-1) or click 'Show Analysis Results' button in the status bar.

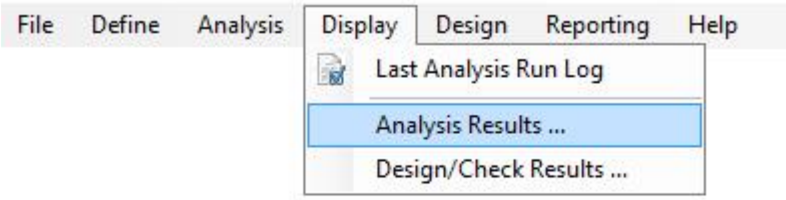


Figure 5-1, Open Analysis Results From

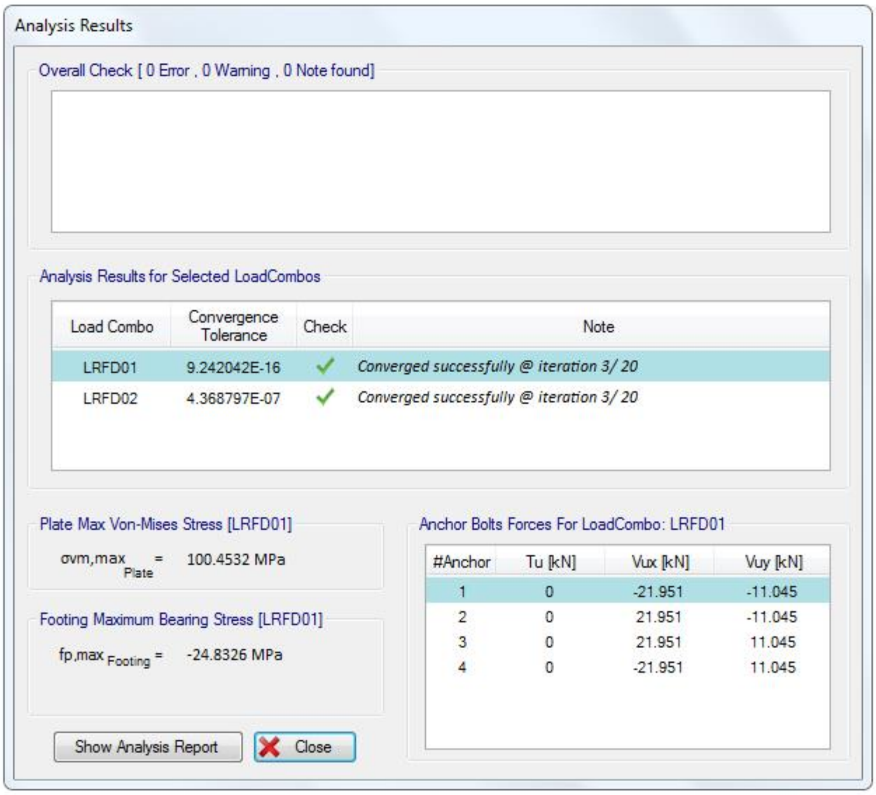


Figure 5-2, Analysis Results Form

Overall checking for analysis process has been done after analysis completion and founded errors/warnings are shown in the Analysis Results Form (Figure 5-2). Some of messages that may appear are listed in the table below.

Error/Warning Note	Description/Solution
Error, For some of load combinations, achieved convergence tolerance is greater than the allowable value.	For solving the error, firstly view the residual force for mentioned load combination that is reported in the analysis report. If residual force is small compared to connection assigned loads (For example residual force is 0.001kN or 0.0002Kips, and loads applied to connection are in order of 100KN or 22Kips and higher) , ignore the error message. Else, set mesh size to smaller values and then run analysis again.
Error, More Iterations required for achieving analysis convergence for 2 load combos.	For solving the error, set maximum iterations in analysis options to higher values. It is recommended to always set this value more than or equal to 20 iterations.
Error, Analysis results have insufficient accuracy! because maximum mesh size is set more than recommended values.	If mesh size is set to more than the program recommended value for normal accuracy, this message is appeared. In this case, surely analysis results have insufficient accuracy. For resolving the error, use recommended values for mesh size.
Note, Accuracy of analysis results can improve by setting smaller mesh size.	If mesh size is set to more than the program recommended value for good accuracy, this message is appeared. User can improve analysis results accuracy by modifying the mesh size to at least the recommended value for good accuracy.
Note, Column is not modeled, because height of column has been set to zero.	This message informs that column is not modeled in the finite element modeling. In this case, analysis results differ from realistic behaviour of column-base connection.
Note, Column is not modeled, because modulus of elasticity for this part has been set to zero.	The same as above.
Note, Stiffeners are not modeled, because modulus of elasticity for them has been set to zero.	Message note is clear.

In the Analysis Results Form (Figure 5-2), by choosing any load combination, its analysis outputs will be shown (such as plate maximum von-mises stress, footing/grout maximum bearing stress and anchor bolts forces).

By clicking 'Show Analysis Report' button in this Form, a comprehensive report on analysis will be shown. One can save this report to an XPS or PDF format file.

5.1.2 Plot Contours

2D and 3D contour viewing is performed in the program.

For viewing 2D contours, click 'Plot Contours on 2D View' in the 2D View panel. The Form below will be shown (Figure 5-3).

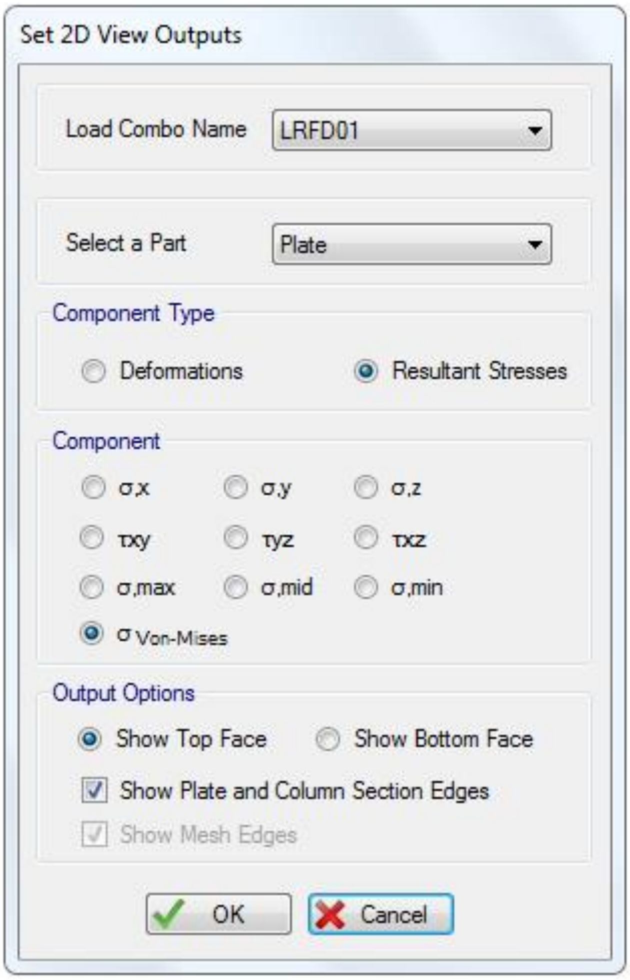


Figure 5-3, Set 2D View Outputs

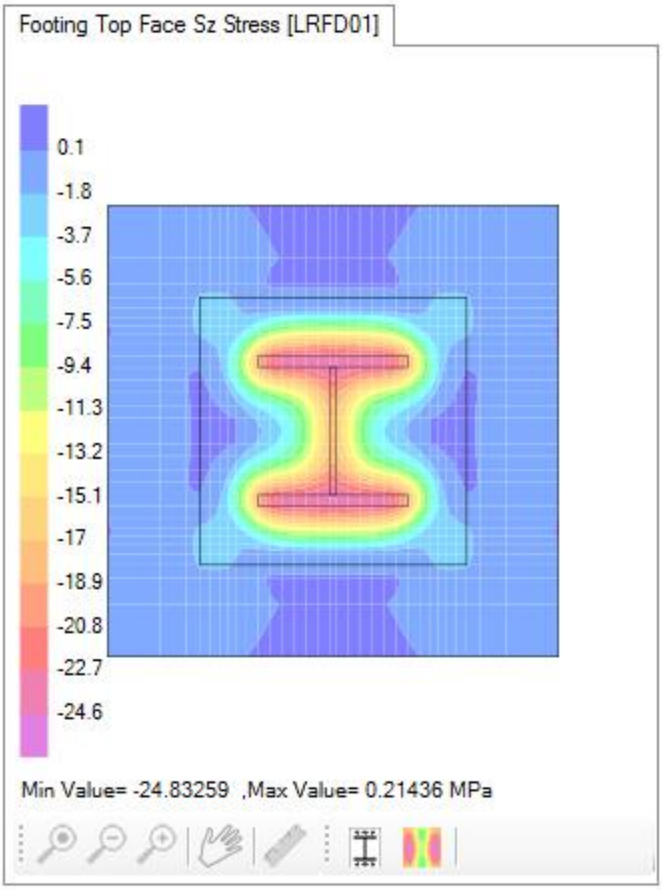


Figure 5-4, Sample 2D Contour (Bearing Stress for Footing)

By choosing any load combination, one can plot 2D contour of deformations/stresses of any part of the connection. In 2D contour options, the view of top face or bottom face of chosen part is provided. After clicking OK, 2D contour will be plotted in 2D view window. By moving mouse position on the plotted contour, coordinates and output values will be shown.

One can change the program units at any time. By changing the units, plotted contours will be updated automatically.

To view 3D contours, click 'Plot Contours on Undeformed Shape' in the 3D View panel. The below Form will be shown (Figure 5-4).

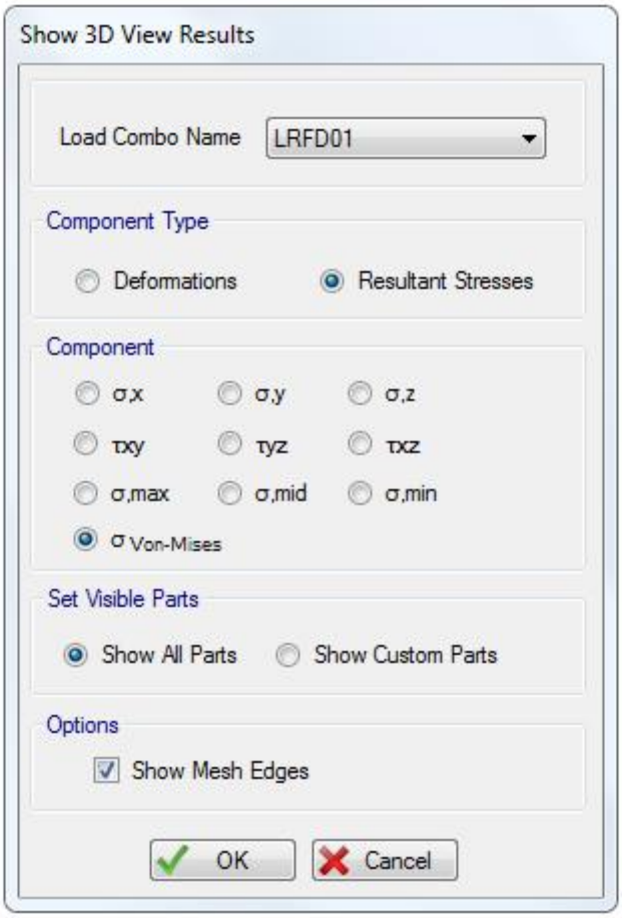


Figure 5-5, Show 3D View Results

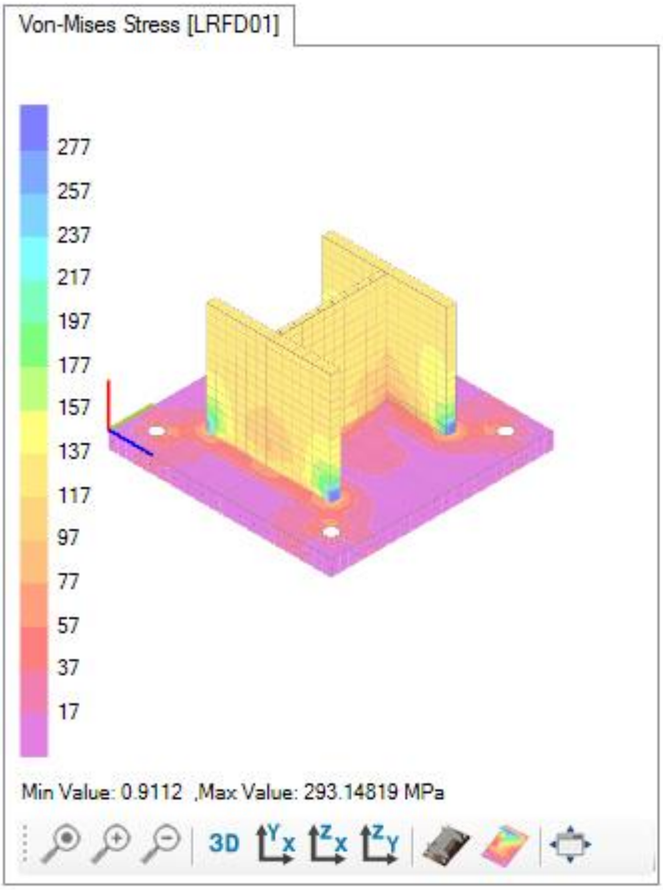


Figure 5-6, Sample 3D Contour (Von-Mises Stress)

By choosing any load combination, contours for any selected component can be plotted. One can choose 'Show All Parts' for showing all connection parts in the contour plot or select 'Show Custom Parts' to show custom parts of connection in the contour plot. In the Option part, by choosing 'Show Mesh Edges', mesh edges for all parts will be shown, else mesh edges will be hidden.

Upper and lower limit of contour guide in 3D view are set by maximum and minimum output value of shown parts. By changing shown parts, the upper and lower limit of contour guide will be updated.

For parts with concrete material such as footing and grout, Von-Mises stress cannot be defined. For these parts, use bearing stress or principal stresses. In 3D contour, when concrete parts as well as parts with steel material are shown, viewing Von-Mises stress is not true.

5.2 Design

Design procedure is independent of the analysis process. Therefore, after analysis completion, one can modify design preferences and design results for all connection parts will be updated automatically.

5.2.1 Setting Design Preferences

For setting design preferences, click Design menu> Design Preferences (Figure 5-6).

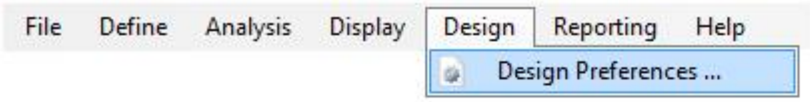
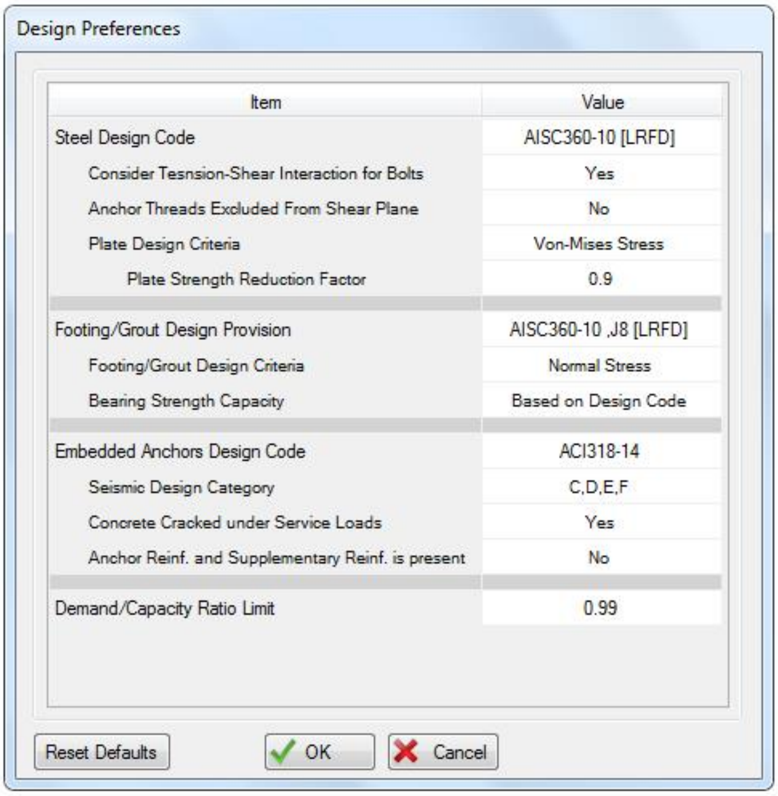


Figure 5-7, Open Design Preferences Form

In Design Preferences Form (Figure 5-7), steel design code, footing/grout design provision and embedded anchors design code can be chosen. Parameters of design provisions are shown in Figure 5-7. Now for steel design code only AISC360-10 [LRFD] is available. In this code, design of anchor bolts is performed for both tension and shear. Also one can set design procedure to consider tension-shear interaction or not.

Plate is designed by Von-Mises stress criteria. The user enters strength reduction factor for designing of the plate. Values of 0.9 to 1.0 are recommended for it. To design of footing and grout, AISC360-10, Section J8 provisions is available. One can set footing/grout bearing strength capacity to Design Code or User Specified. If 'Based on Design Code' option is chosen, bearing capacity is calculated according to AISC360-10 Section J8, else bearing capacity stress is entered by user.


Design Preferences

Item	Value
Steel Design Code	AISC360-10 [LRFD]
Consider Tesnsion-Shear Interaction for Bolts	Yes
Anchor Threads Excluded From Shear Plane	No
Plate Design Criteria	Von-Mises Stress
Plate Strength Reduction Factor	0.9
Footing/Grout Design Provision	AISC360-10 ,J8 [LRFD]
Footing/Grout Design Criteria	Normal Stress
Bearing Strength Capacity	Based on Design Code
Embedded Anchors Design Code	ACI318-14
Seismic Design Category	C,D,E,F
Concrete Cracked under Service Loads	Yes
Anchor Reinf. and Supplementary Reinf. is present	No
Demand/Capacity Ratio Limit	0.99

Reset Defaults | OK | Cancel

Figure 5-8, Design Preferences Form

Design of anchor bolts is done by ACI318-14 standard. Design of anchors for concrete breakout failure, concrete side-face blowout failure and checking pullout strength is provided. Demand/Capacity Ratio Limit sets the acceptable ratio for design of all connection parts. It is recommended to set this value to about 0.99 or equal to one.

5.2.2 Design Results

To view design results, click Display menu> Design/Check Results (Figure 5-8) or click 'Show Design Results' button in the status bar.

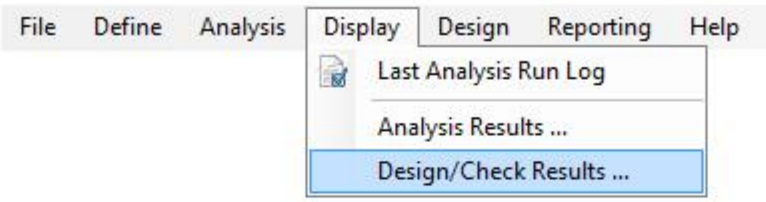


Figure 5-9, Open Design/Check Results Form

'Design/Check Results' Form is shown in figure 5-8. By changing design check points, design results for all load combinations and design summery for selected ones will be updated automatically. Also user can modify design preferences in this Form and view updated results, immediately.

A comprehensive report is provided for design results. For view design report for selected load combination, click 'Show Report For Selected Load Combo' button and to view envelope design report, click 'Show Envelope Design Report' button.

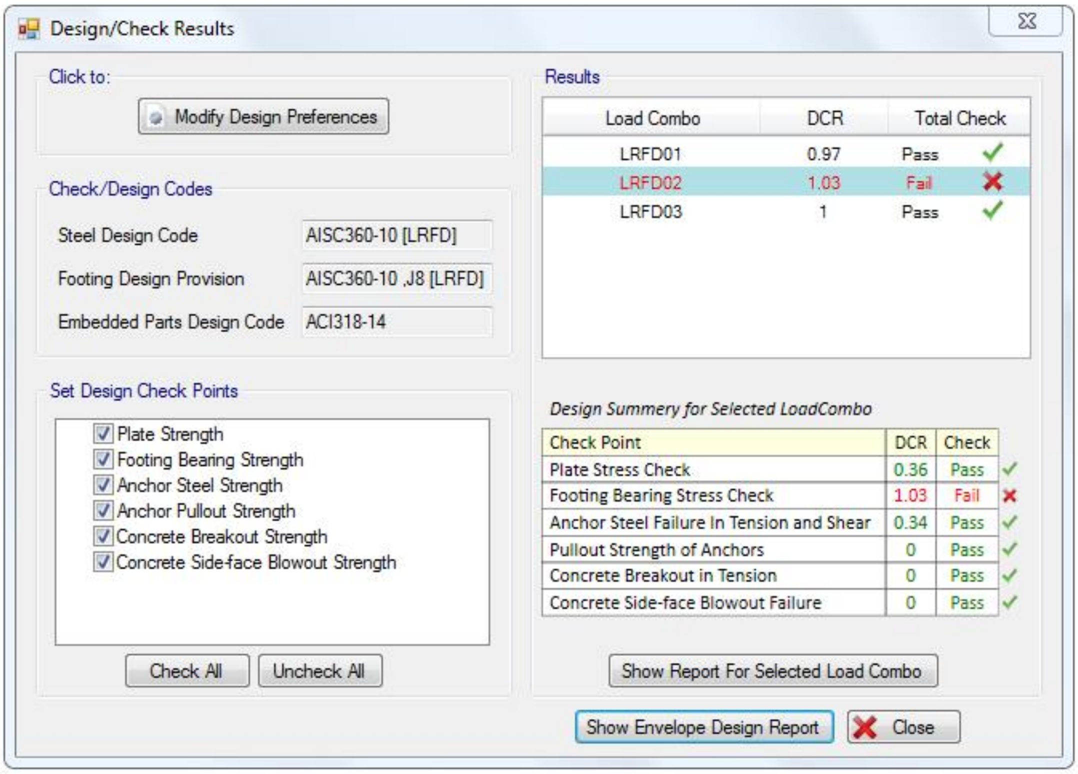


Figure 5-10, Design/Check Results Form