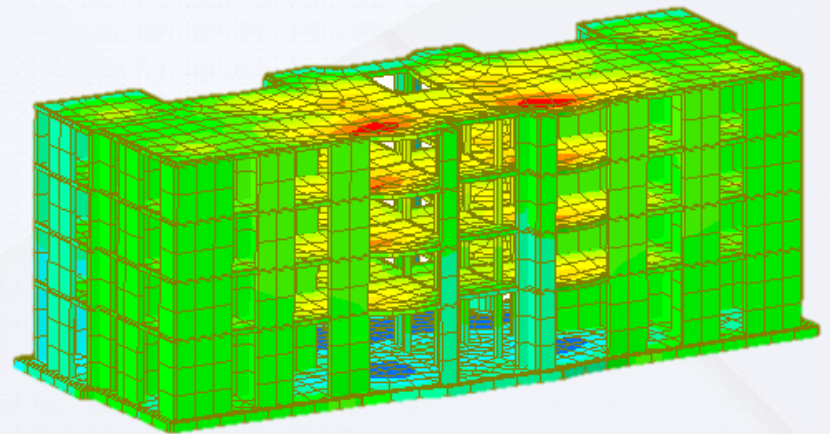
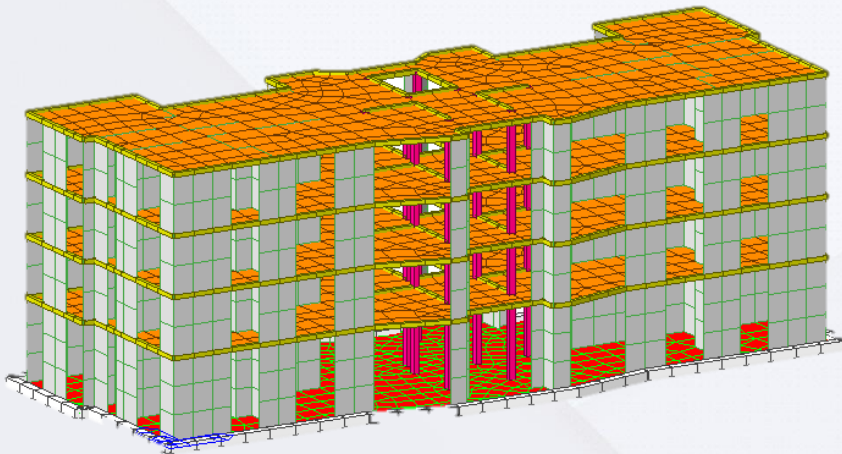
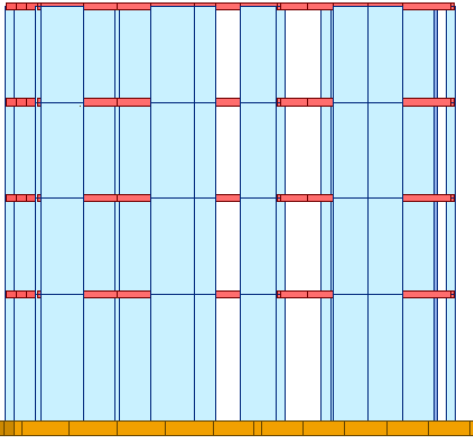


# SLAB AND WALL ANALYSIS AND DESIGN

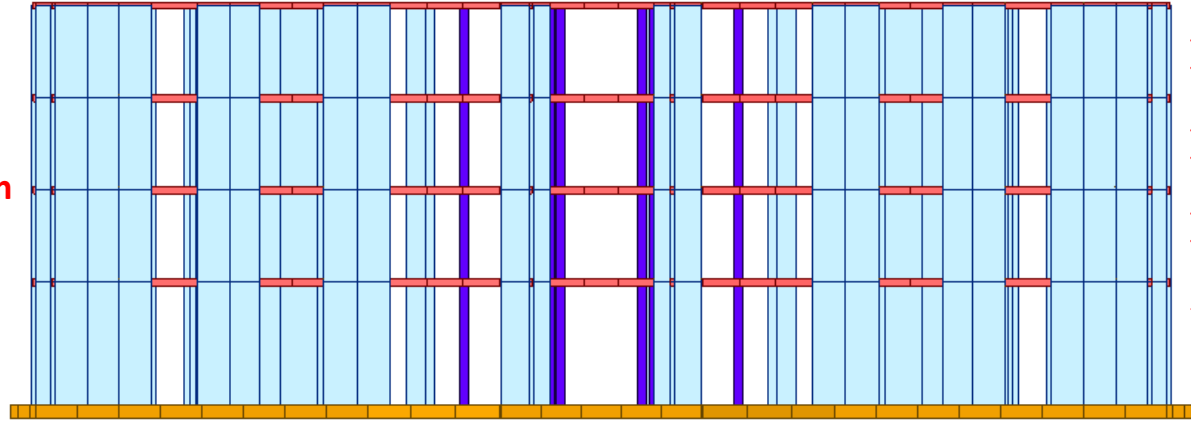


# Dimensions

8m



8.8m



2m

2m

2m

2.8m



24 m

## Unit System

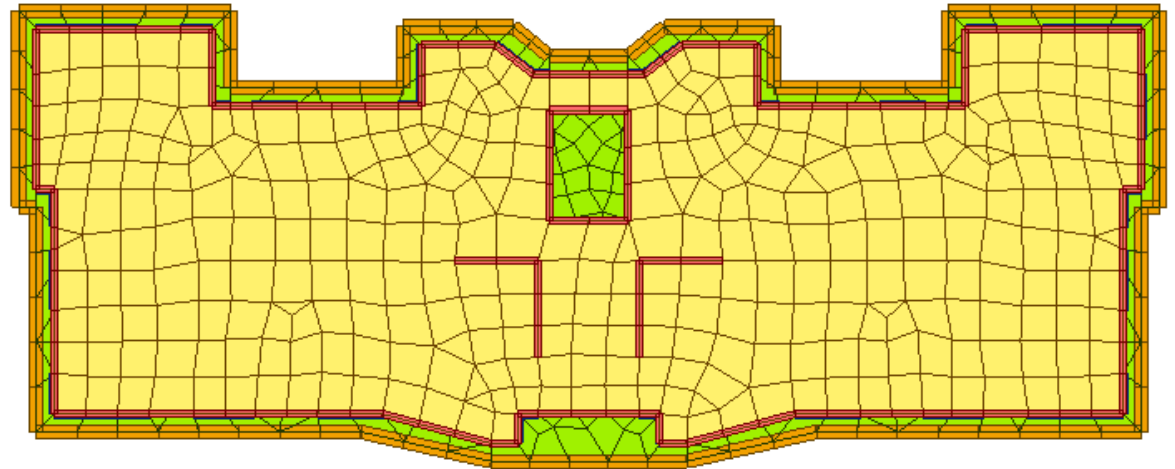
### Length

- m
- cm
- mm
- ft
- in

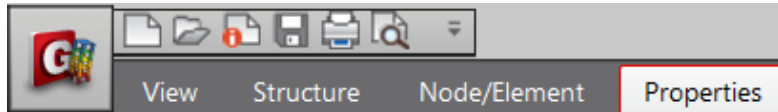
### Force (Mass)

- N (kg)
- kN (ton)
- kgf (kg)
- tonf (ton)
- lbf (lb)
- kips (kips/g)

9m



# Define Properties



Properties

ID	Name	Type	Standard	DB
1	Grad...	Conc...	ASTM(RC)	Grade C4500



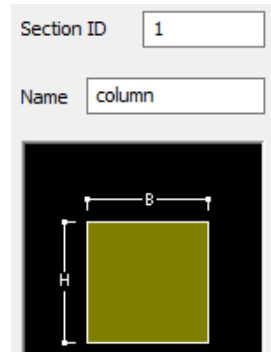
Material Properties  
Material

- Define Material  
-Concrete ASTM C4500

- Define 4 rectangle Sections as shown

Properties

ID	Name	Type	Shape
1	column	User	SB
2	slab girder	User	SB
3	raft girder	User	SB
4	wall marker	User	SB



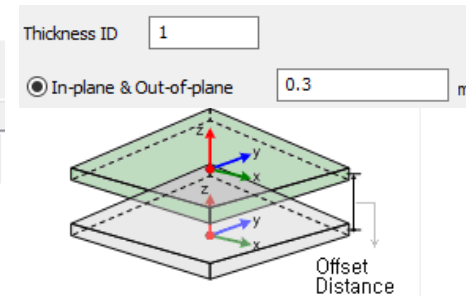
Section Properties

	H	B
Column	0.2 m	0.2 m
Slab girder	0.15 m	0.15 m
Raft girder	0.3 m	0.3 m
Wall marker	.001 m	.001 m

- 3 thicknesses as shown

Properties

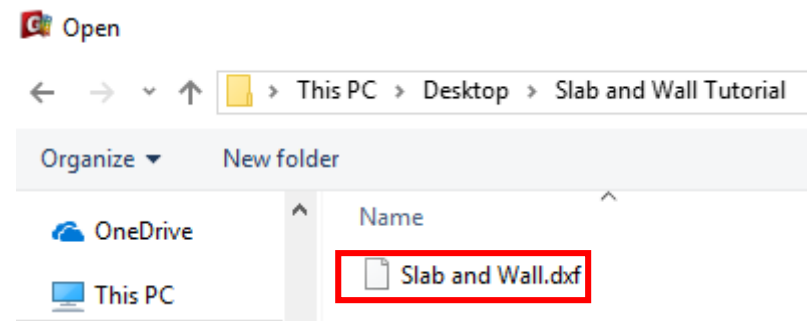
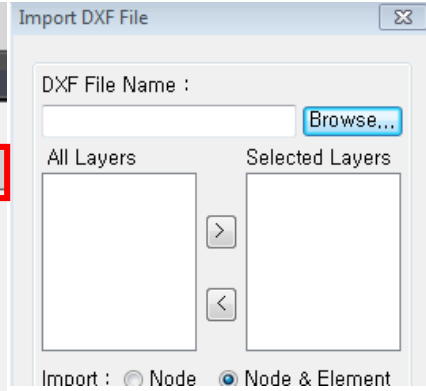
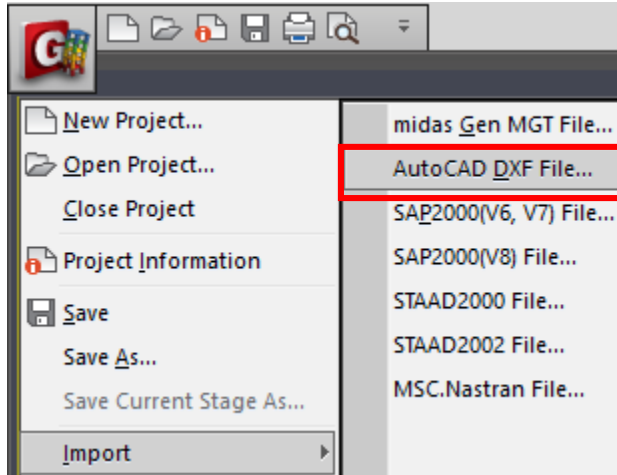
ID	Type	Thickness(m)
1	Value	0.300000
2	Value	0.200000
3	Value	0.150000



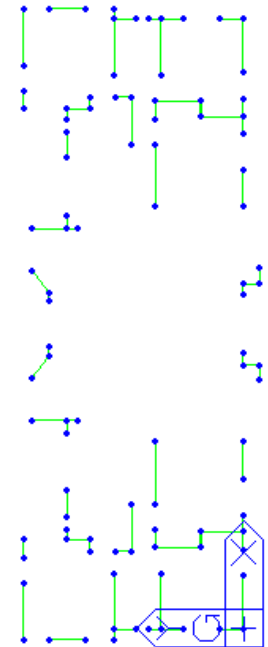
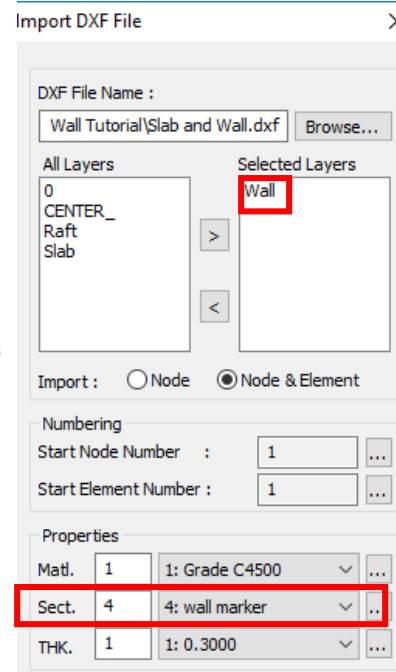
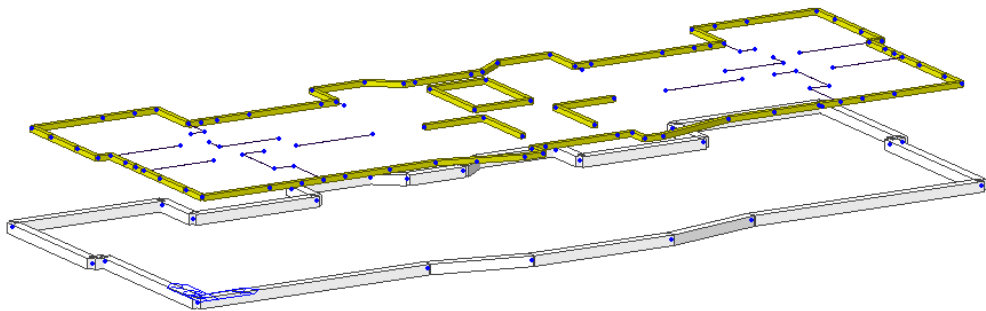
Thickness

	Thickness
Raft	0.3 m
Wall	0.2 m
Slab	0.15 m

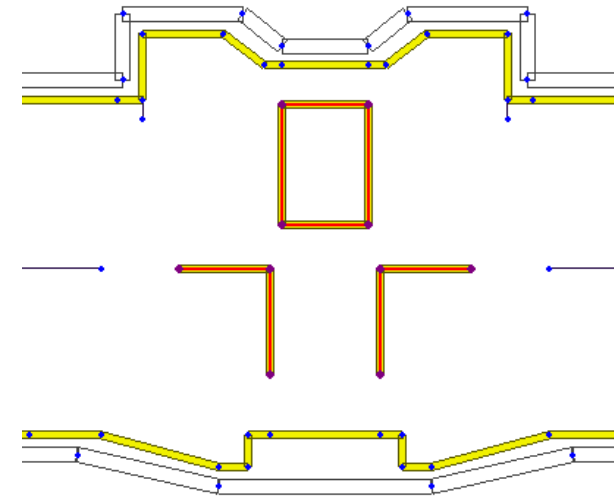
# Import DXF CAD Layers



- Import Slab and Wall DXF
- Select Wall Layer and assign wall marker section
- Select Raft Layer and assign raft girder section
- Select Slab Layer and assign slab girder section



# Extrude Columns



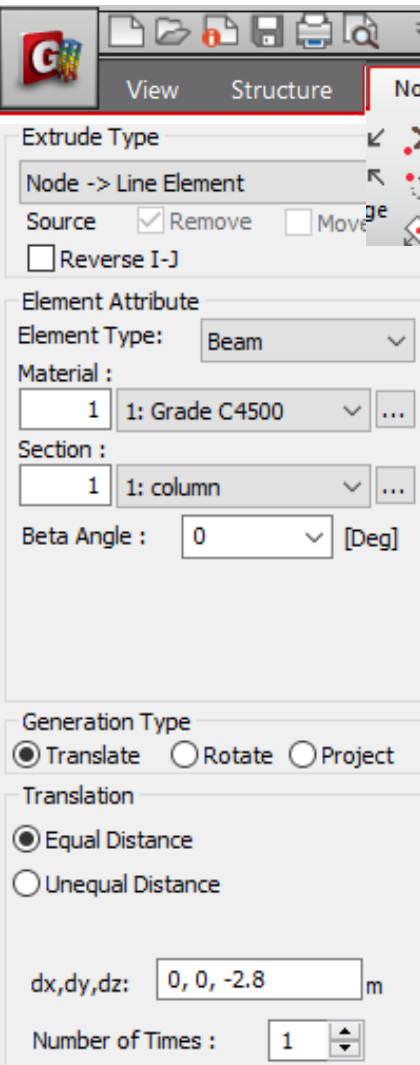
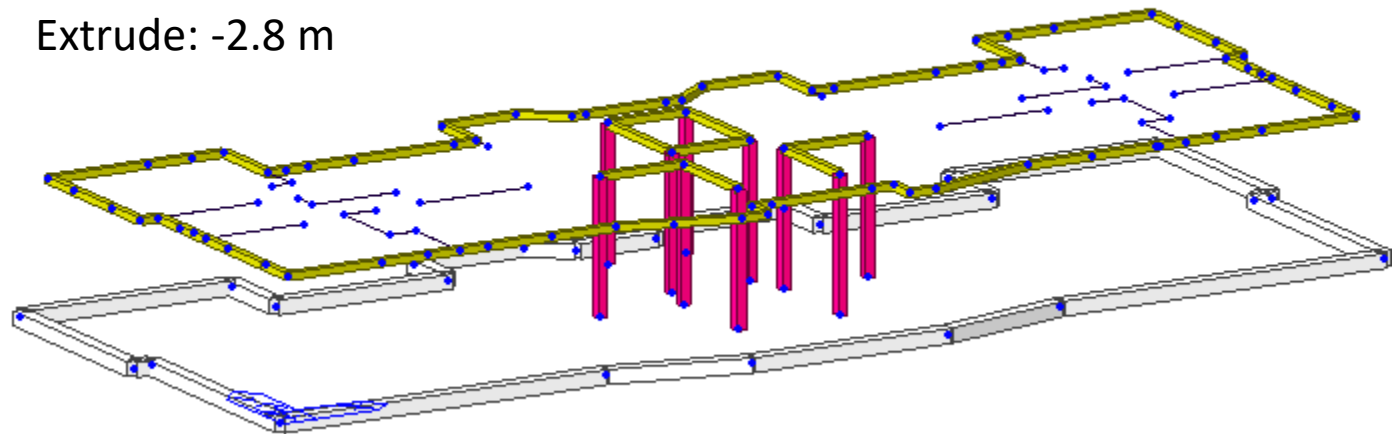
## Extrude columns

Assign Element Type: General beam

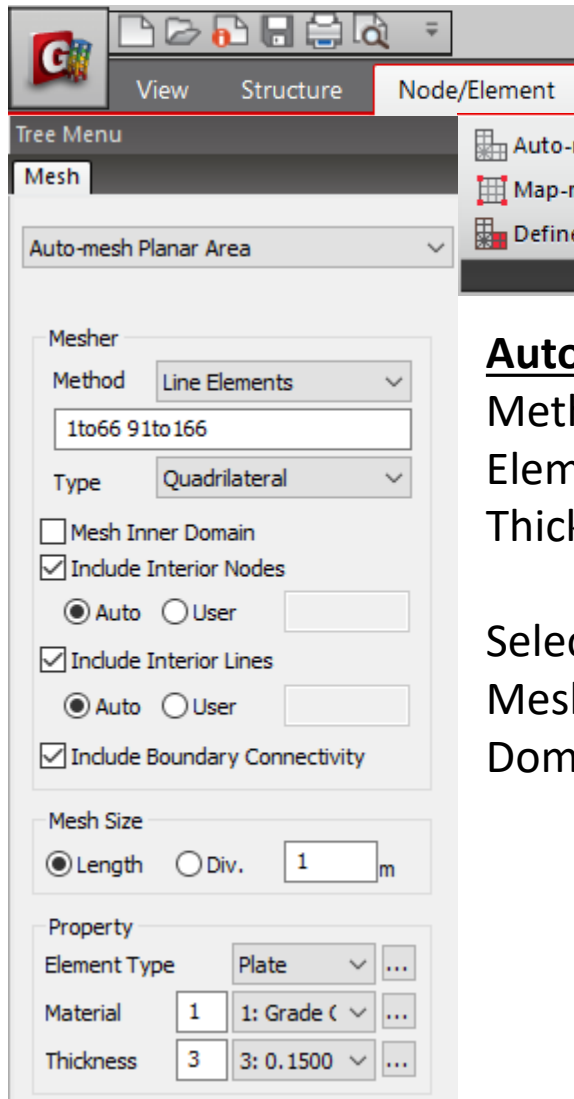
Section: Column

**Select center slab layer nodes as shown**

Extrude: -2.8 m



# Mesh Slab



## Auto Mesh Planar Area

Method: Line Elements

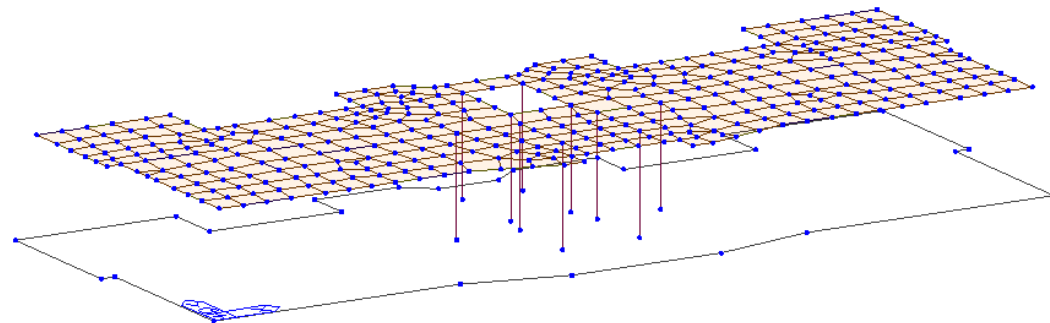
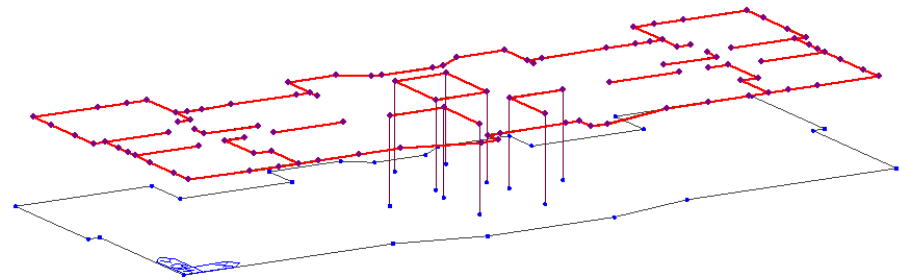
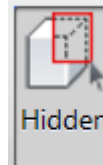
Element Type: Plate

Thickness: 0.15

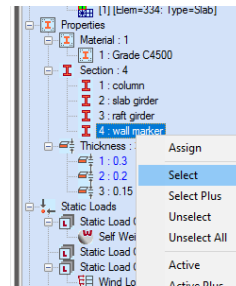
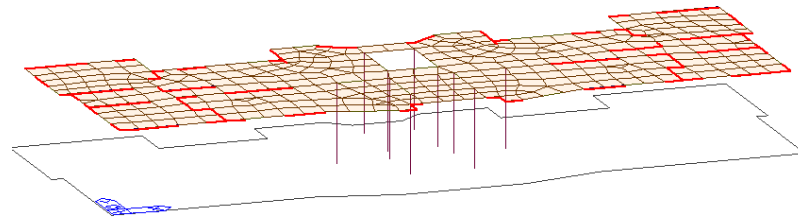
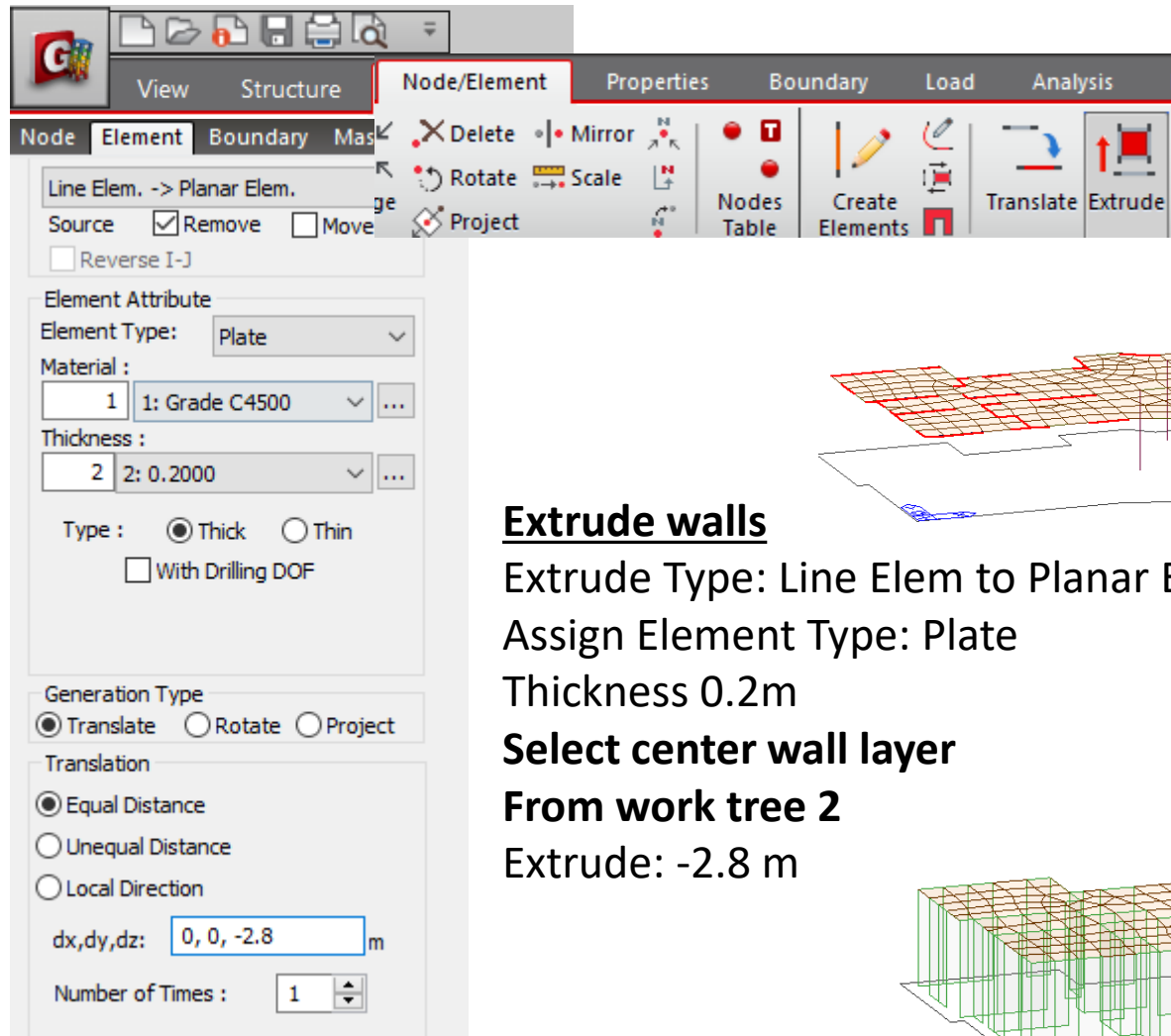
Select Slab and Wall layers

Mesh size 1m

Domain: Slab



# Extrude Walls



## Extrude walls

Extrude Type: Line Elem to Planar Element

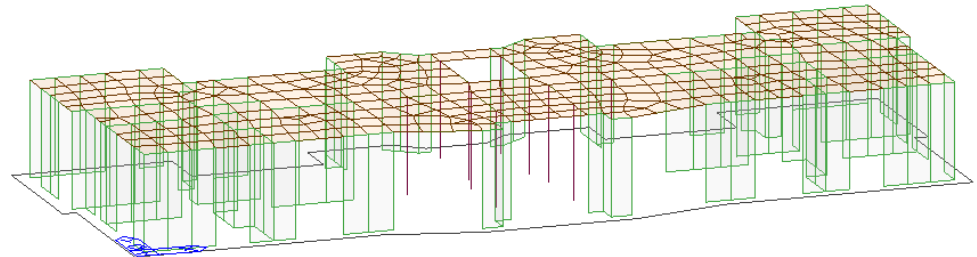
Assign Element Type: Plate

Thickness 0.2m

Select center wall layer

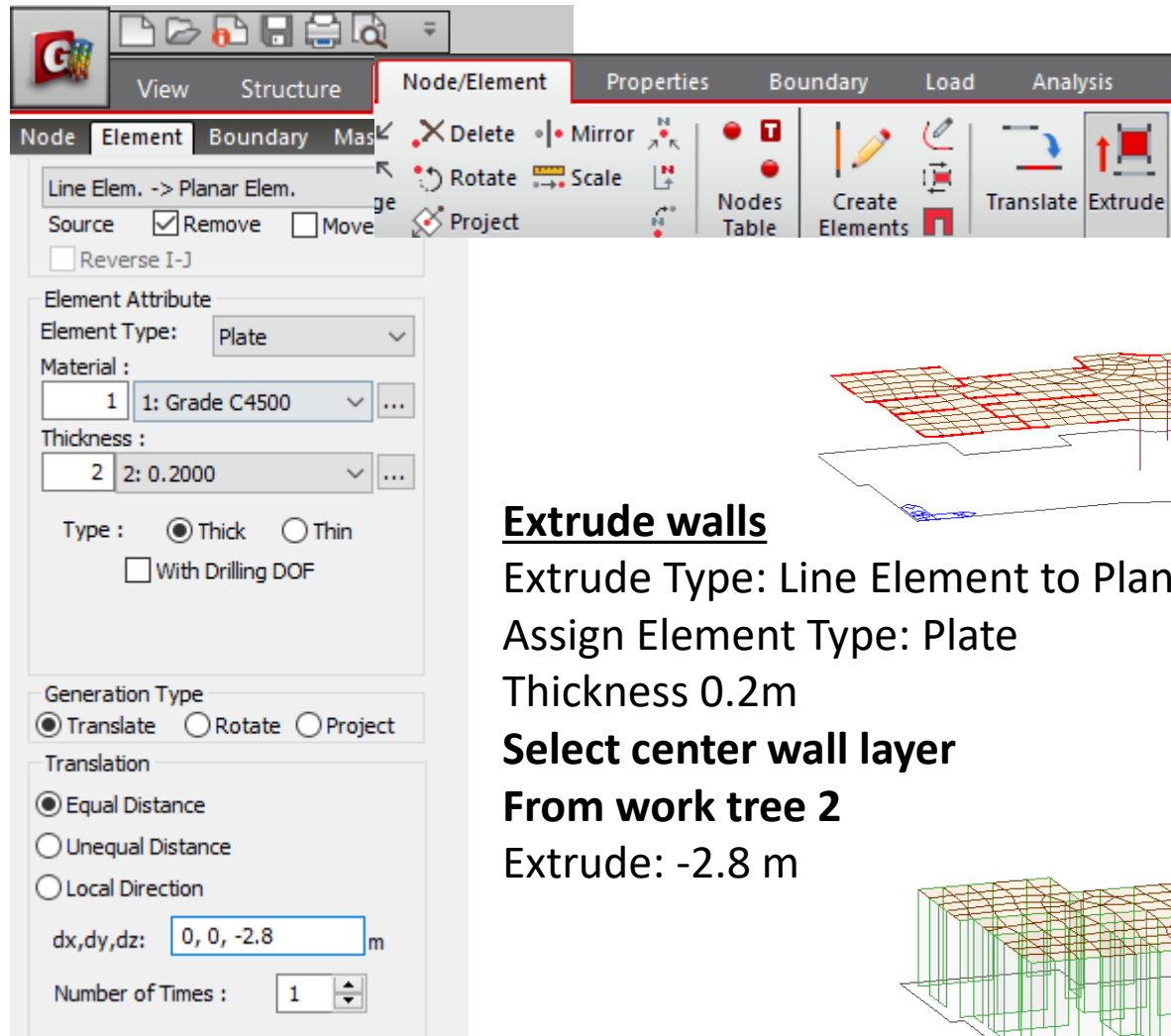
From work tree 2

Extrude: -2.8 m





# Extrude Walls



View Structure Node/Element Properties Boundary Load Analysis

Node Element Boundary Mas

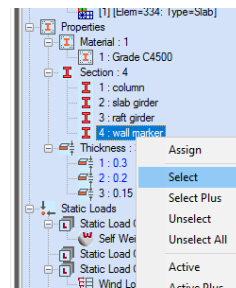
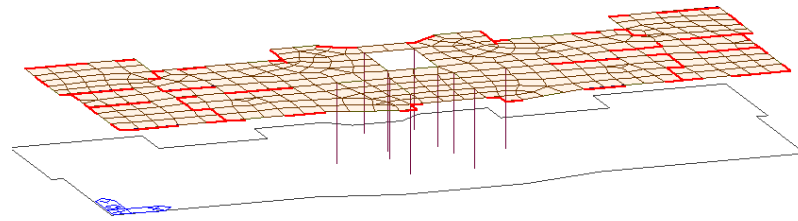
Delete Mirror Rotate Scale Project Nodes Table Create Elements Translate Extrude

Line Elem. -> Planar Elem.  
Source  Remove  Move  Reverse I-J

Element Attribute  
Element Type: Plate  
Material: 1 1: Grade C4500  
Thickness: 2 2: 0.2000  
Type:  Thick  Thin  
 With Drilling DOF

Generation Type  
 Translate  Rotate  Project

Translation  
 Equal Distance  
 Unequal Distance  
 Local Direction  
dx,dy,dz: 0, 0, -2.8 m  
Number of Times: 1



## Extrude walls

Extrude Type: Line Element to Planar Element

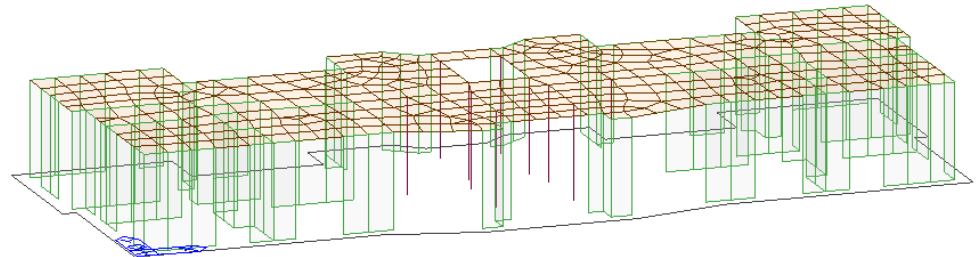
Assign Element Type: Plate

Thickness 0.2m

Select center wall layer

From work tree 2

Extrude: -2.8 m





# Mesh Raft

The screenshot shows the midas Gen software interface. The 'Node/Element' menu is open, displaying options: 'Auto-mesh', 'Map-mesh', and 'Define Domain'. The 'Auto-mesh Planar Area' option is selected. Below the menu, the 'Mesh' settings panel is visible, containing the following configuration:

- Mesh**
  - Method: Line Elements
  - 67to90
  - Type: Quadrilateral
  - Mesh Inner Domain
  - Include Interior Nodes
    - Auto  User
  - Include Interior Lines
    - Auto  User
  - Include Boundary Connectivity
- Mesh Size**
  - Length  Div.  m
- Property**
  - Element Type: Plate
  - Material: 1: Grade C
  - Thickness: 1: 0.3000

## Auto Mesh Planar Area

Method: Line Elements

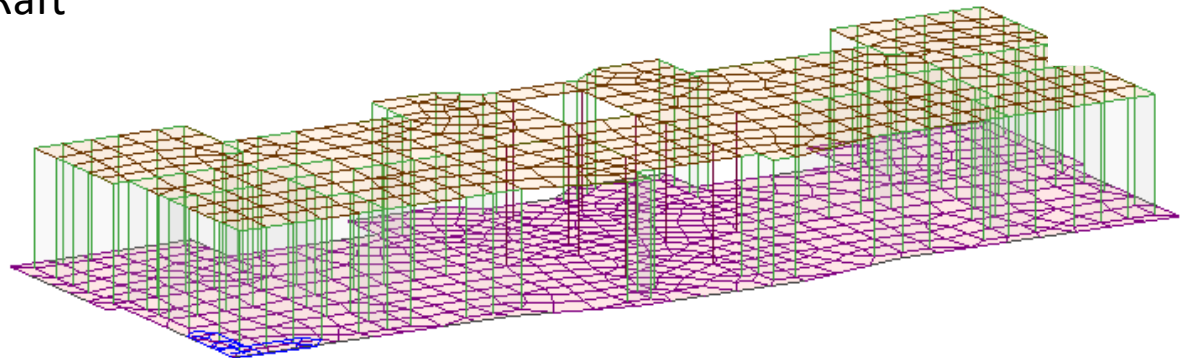
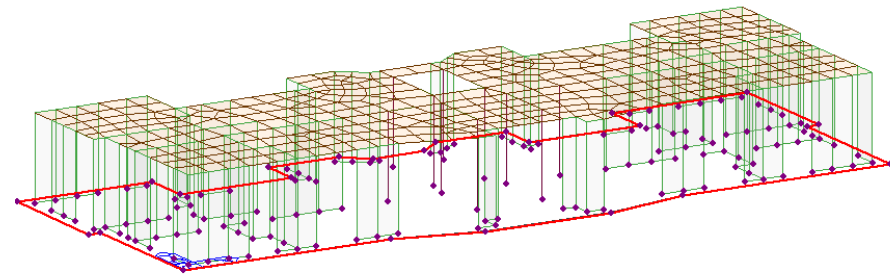
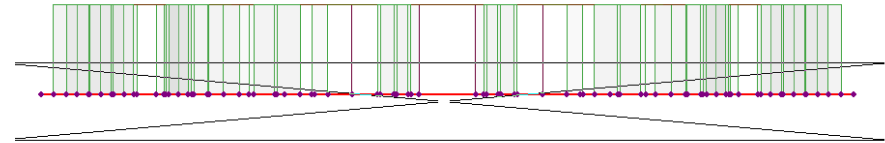
Element Type: Plate

Thickness: 0.3m

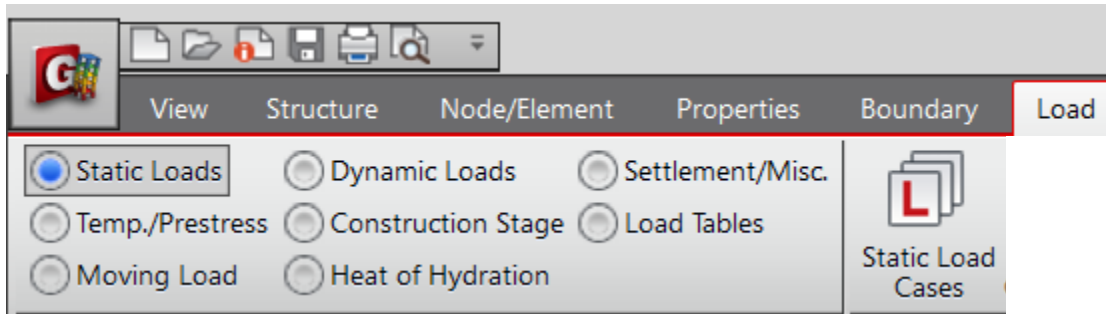
Select All Bottom nodes

Mesh size 1m

Domain: Raft



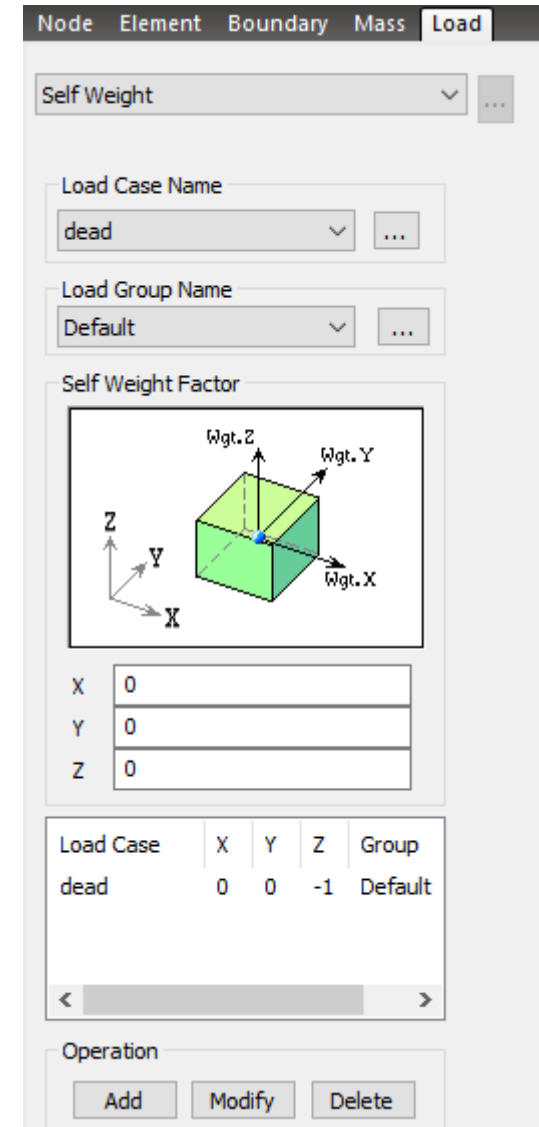
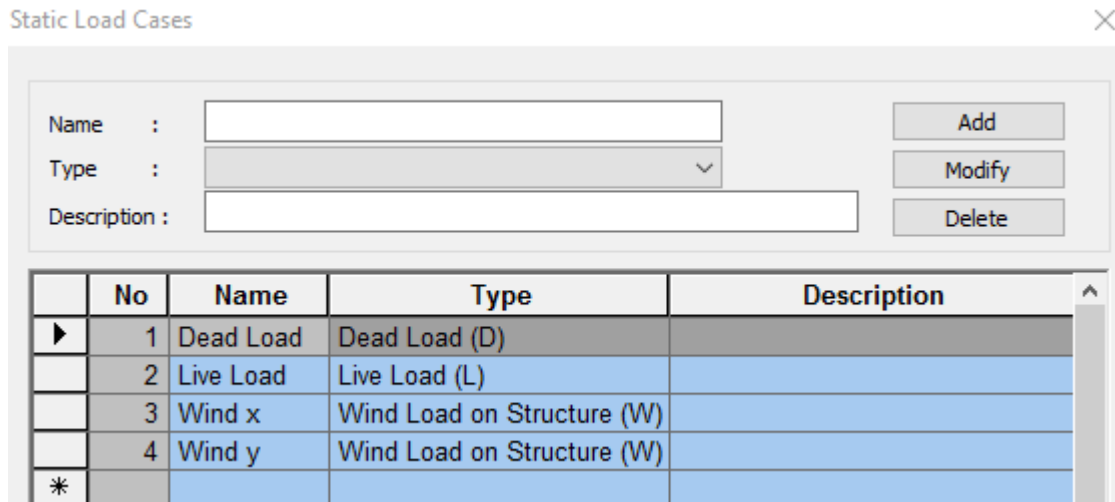
# Load Cases



## Static Load Cases

Create 4 load cases

Assign Self Weight to dead load case



# Assign Pressure Loads

The screenshot shows the midas Gen software interface. The 'Load' menu is open, showing options: 'Pressure Loads', 'Hydrostatic Pressure', and 'Assign Plane Loads'. The 'Pressure Loads' option is selected, and a sub-menu is visible with 'Pressure Load' highlighted. The 'Pressure Loads' dialog box is open, showing the following settings:

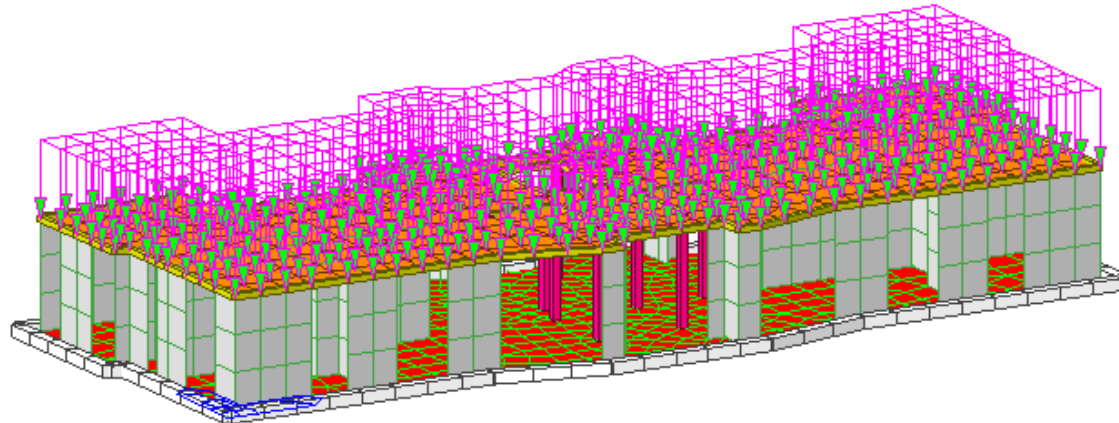
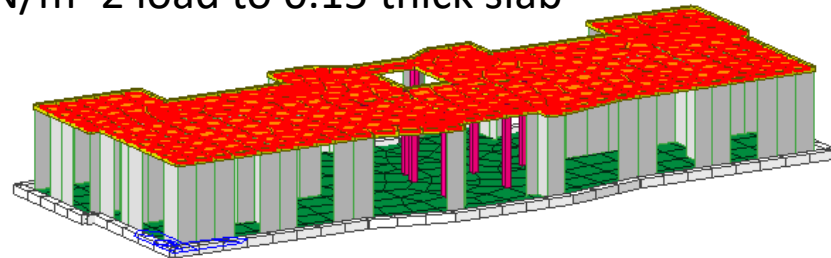
- Type:  Load Case  Load Type
- Load Case Name: live
- Element Types: Plate/Plane Stress(Face)
- Pressure on Plate:
- Selection:  Node  Element
- Pressure Face: Face #1
- Direction: Local z
- Vector: 0, 0, 0
- Projection:  Yes  No
- Loads:  Uniform  Linear
- P1: -7 kN/m<sup>2</sup>

## Assign pressure load to slab

Load Case: Live

Element type: Plate

Assign a -7 kN/m<sup>2</sup> load to 0.15 thick slab



The Properties panel shows the assigned load case and element type for the slab. The 'Load Case' is set to 'live' and the 'Element Type' is 'Plate/Plane Stress(Face)'. The 'Thickness' is set to 0.15.

Property	Value
Material	1
Section	4
Thickness	3 : 0.15

# Building Generation

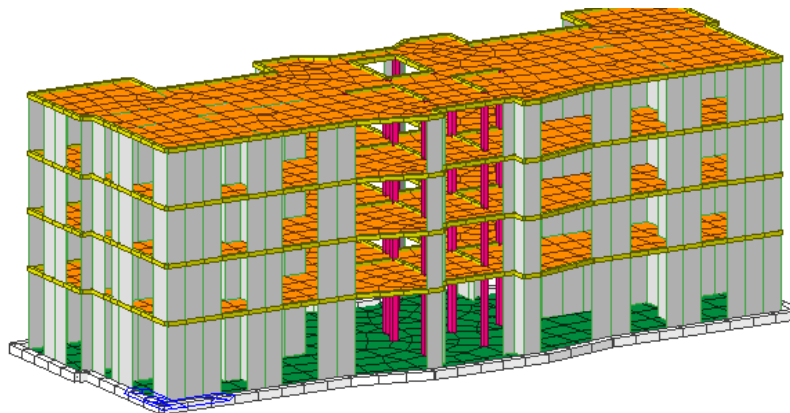
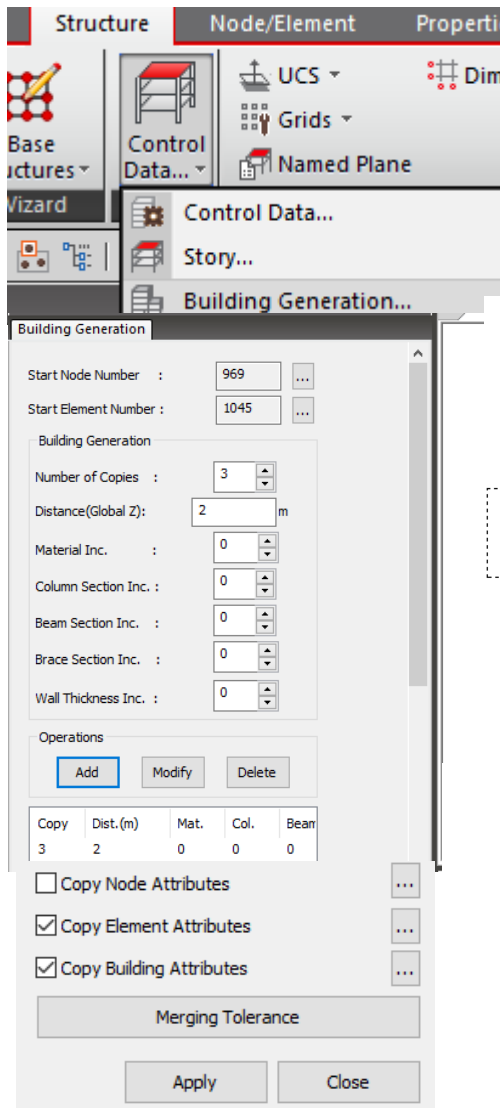
## Make copies of the first floor

Select All except raft elements

Copy 3 times at 2m

Click Add

Click Apply



# Mesh Wall

View Structure Node/Element

Auto-mesh Planar Area

Meshers

Method **Planar Elements**

594to693 1803to1902 2372to2471

Type **Quadrilateral**

Mesh Inner Domain

Include Interior Nodes

Auto  User

Include Interior Lines

Auto  User

Include Boundary Connectivity

Mesh Size

Length  Div.  m

Property

Element Type **Plate**

Material **1** 1: Grade C

Thickness **2** 2: 0.2000

Domain

Name **wall**

Auto-mesh Define Sub-Domain

Map-mesh

Define Domain

**Mesh**

## Auto Mesh Planar Area

Method: Planar Elements

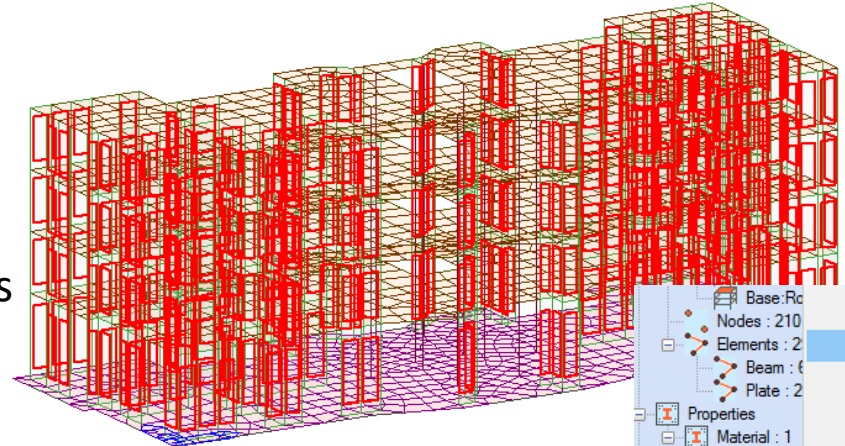
Element Type: Plate

Thickness: 0.2 m

Select All 0.2m thick elements

Mesh size 1m

Domain: Wall



Base:Ro

Nodes : 210

Elements : 2

Beam : 6

Plate : 2

Properties

Material : 1

1 : Grad

Section : 4

1 : colum

2 : slab g

3 : raft g

4 : wall r

Thickness :

1 : 0.3

**2 : 0.2**

Assign

Select

Select Plus

Unselect

Unselect All

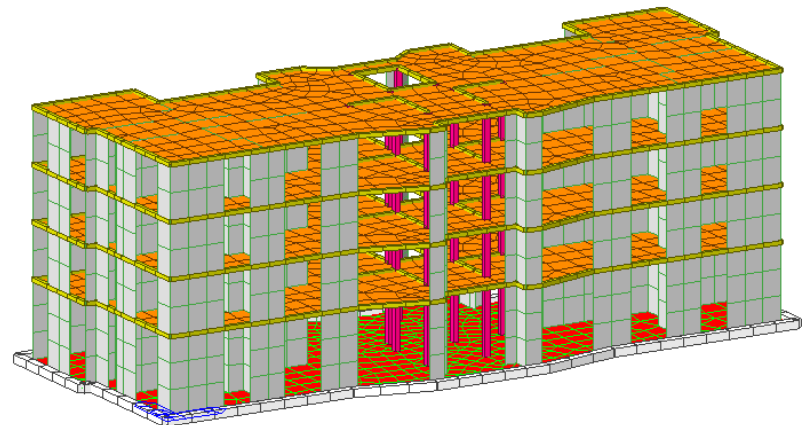
Active

Active Plus

Inactive

Active All

Delete



# Generate story data

## Auto Generate Story Data

Automatic Generation of Story Data

Unselected List

No	Level
2	0.933333
3	1.86667
5	3.8
7	5.8
9	7.8

Selected List

No	Name	Level	Height
1	1F	-1.77636...	2.8
4	2F	2.8	2
6	3F	4.8	2
8	4F	6.8	2
10	Roof	8.8	0

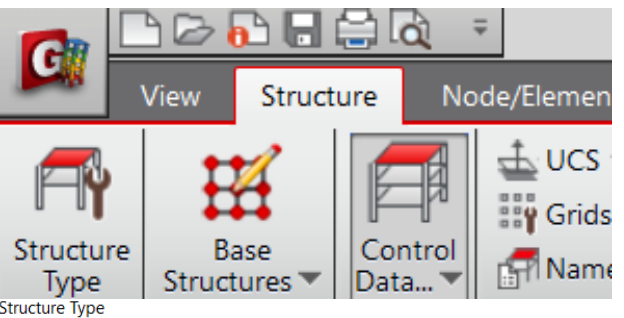
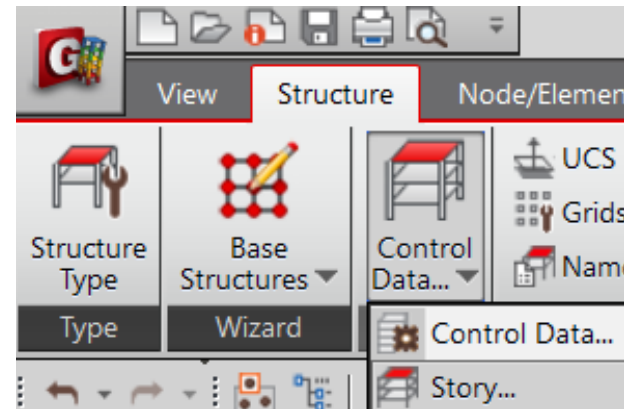
Include Seismic Accidental Eccentricity :  % of Plan Dimension  
 Include Wind Eccentricity :  % of Plan Dimension

Story Data

Ground Level  m

	Module Name	Story Name	Level(m)	Height(m)	Floor Diaphragm
▶	Base	Roof	8.80	0.00	Do not consider
	Base	4F	6.80	2.00	Do not consider
	Base	3F	4.80	2.00	Do not consider
	Base	2F	2.80	2.00	Do not consider
	Base	1F	-0.00	2.80	Do not consider
*					

## Convert Self weight into Masses



Structure Type

3-D  
  X-Z Plane  
  Y-Z Plane  
  X-Y Plane  
  Constraint RZ

Mass Control Parameter

Lumped Mass
   
 Consider Off-diagonal Masses
   
 Considering Rotational Rigid Body Mode for Modal Participation Factor

Consistent Mass

Convert Self-weight into Masses
   
 Convert to X, Y, Z  
  Convert to X, Y  
  Convert to Z

# Wind Load X

## Add Wind Load

Load > Static Loads > Lateral > Wind Loads > Click [Add]

Load Case: Wind X

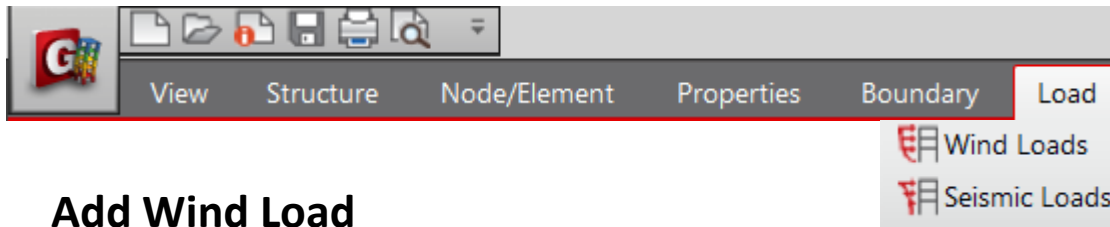
Wind Load Code: IBC 2012

Exposure: C

X Direction Scale Factor 1

Accidental Eccentricity X Direction Positive

Inspect Wind Load Profile



Add/Modify Wind Load Specification

Load Case Name : wind x

Wind Load Code : IBC2012(ASCE7-10)

Description :

Alternate Method  Directional Procedure

Wind Load Parameters

Basic Wind Speed : 85 mile/h

Exposure Category : C

Mean Roof Height : 8.8 m

Topographic Effects : ...

Directional Factor : Kdx 0.85 Kdy 0.85

Rigid Structure  Flexible Structure

Gust Effect Factor : Gx 0.85 Gy 0.85

Load Evaluation Using Force Coefficient

Force Coefficient : 1

Wind Eccentricity

X-Dir. (Wx) :  Positive  Negative  None

Y-Dir. (Wy) :  Positive  Negative  None

Wind Load Direction Factor (Scale Factor)

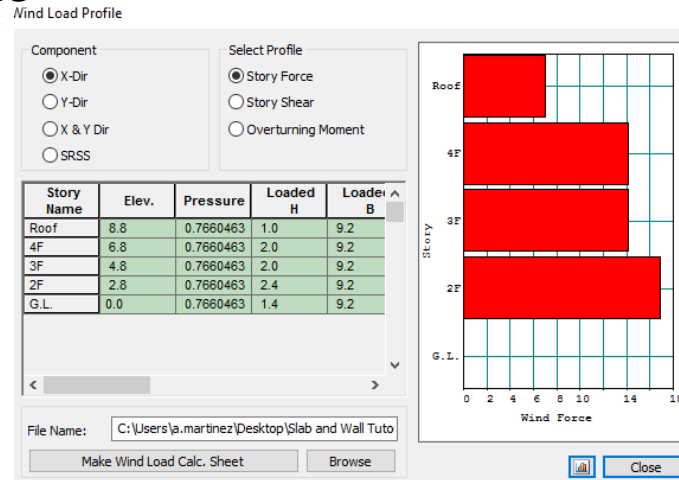
X-Dir. 1 Y-Dir. 0 Z-Rot. 0

Additional Wind Loads (Unit:kN,m)

Story	Add.-X	Add.-Y	Add.-RZ

Wind Load Profile...

OK Cancel Apply





# Wind Load Y

## Add Wind Load

Load > Static Loads > Lateral > Wind Loads > Click [Add]

Load Case: Wind Y

Wind Load Code: IBC 2012

Exposure: C

Y Direction Scale Factor 1

Accidental Eccentricity Y Direction Positive

Inspect Wind Load Profile

Add/Modify Wind Load Specification

Load Case Name : wind y

Wind Load Code : IBC2012(ASCE7-10)

Description :

Alternate Method  Directional Procedure

Wind Load Parameters

Basic Wind Speed : 85 mile/h

Exposure Category : C

Mean Roof Height : 8.8 m

Topographic Effects : ...

Directional Factor : Kdx 0.85 Kdy 0.85

Rigid Structure  Flexible Structure

Gust Effect Factor : Gx 0.85 Gy 0.85

Load Evaluation Using Force Coefficient

Force Coefficient : 1

Wind Eccentricity

X-Dir. (Wx) :  Positive  Negative  None

Y-Dir. (Wy) :  Positive  Negative  None

Wind Load Direction Factor (Scale Factor)

X-Dir. 0 Y-Dir. 1 Z-Rot. 0

Additional Wind Loads (Unit:kN,m)

Story	Add.-X	Add.-Y	Add.-RZ

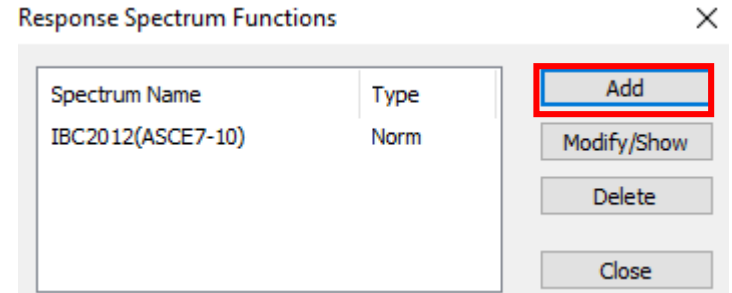
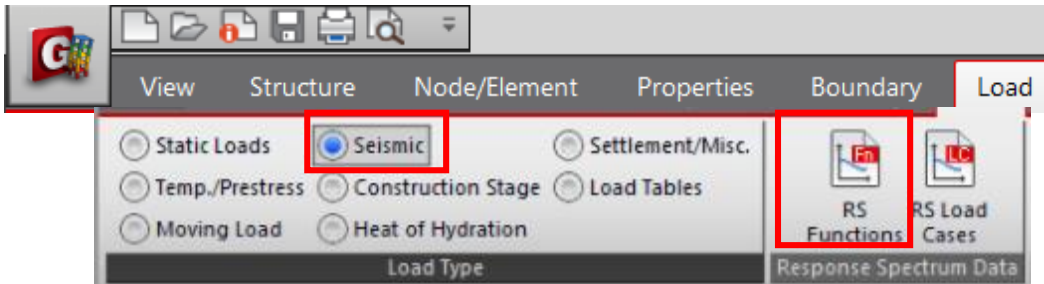
Wind Load Profile...

OK Cancel Apply

Wind Load Profile



# Response Spectrum Functions



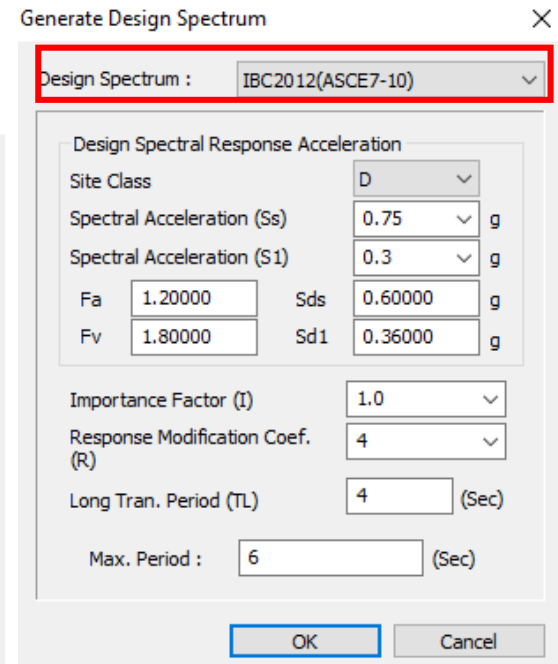
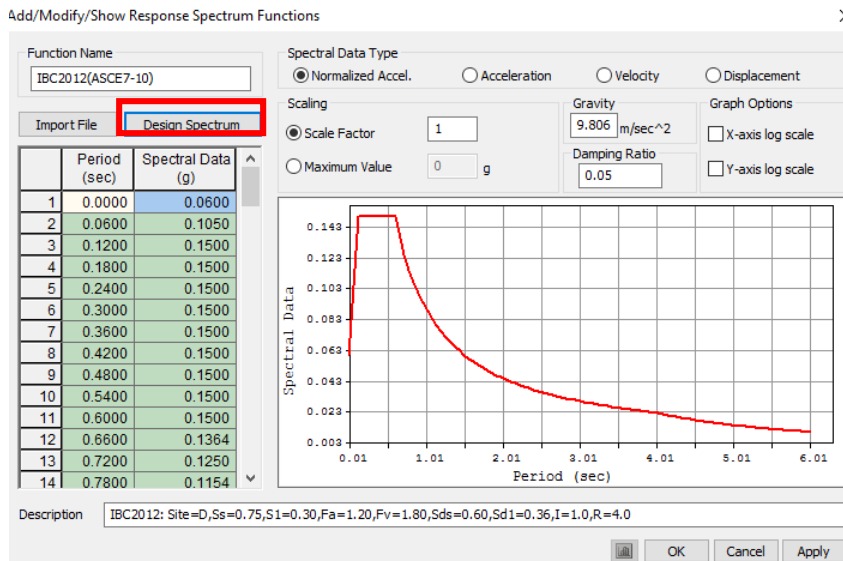
## Seismic Load

Load > Seismic > Response Spectrum Data > Response Spectrum Functions

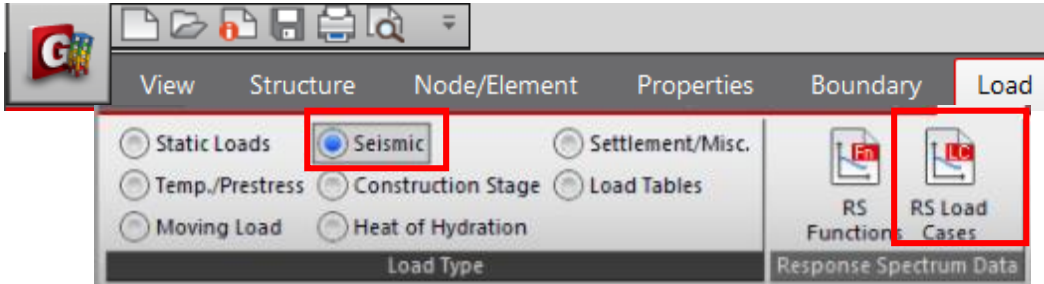
Click Add

Click Design Spectrum

Generate Design Spectrum: IBC2012(ASCE7-10)



# Response Spectrum Load Cases



## Seismic Load

Load > Response Spectrum Data >  
Response Spectrum Load Cases

**Load Case Name: RX**

**Excitation Angle : 0**

Check : IBC2012(ASCE7-10)

Check: Accidental Eccentricity

Click [Add]

**Load Cases Name : RY**

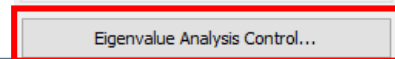
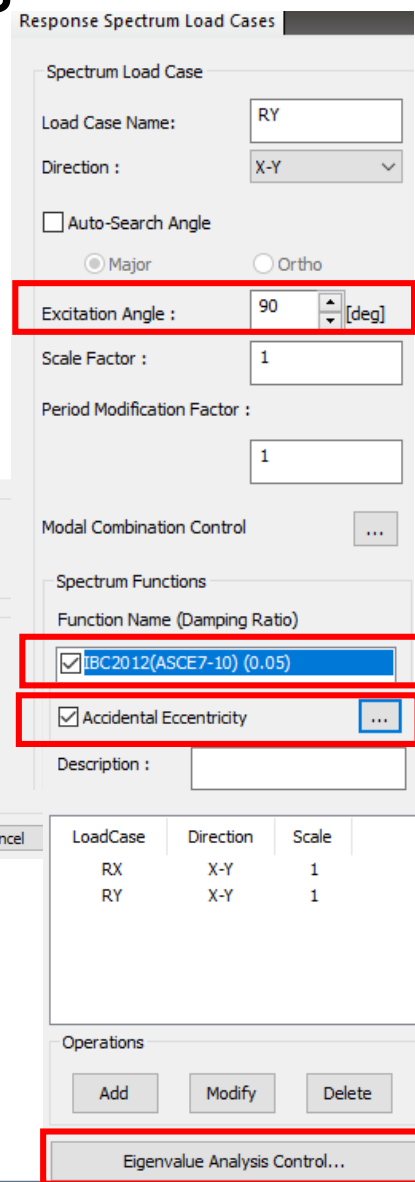
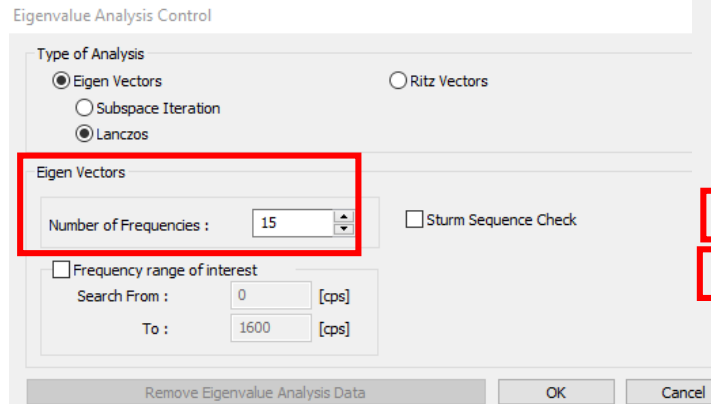
**Excitation Angle : 90** > Click [Add]

Check: Accidental Eccentricity

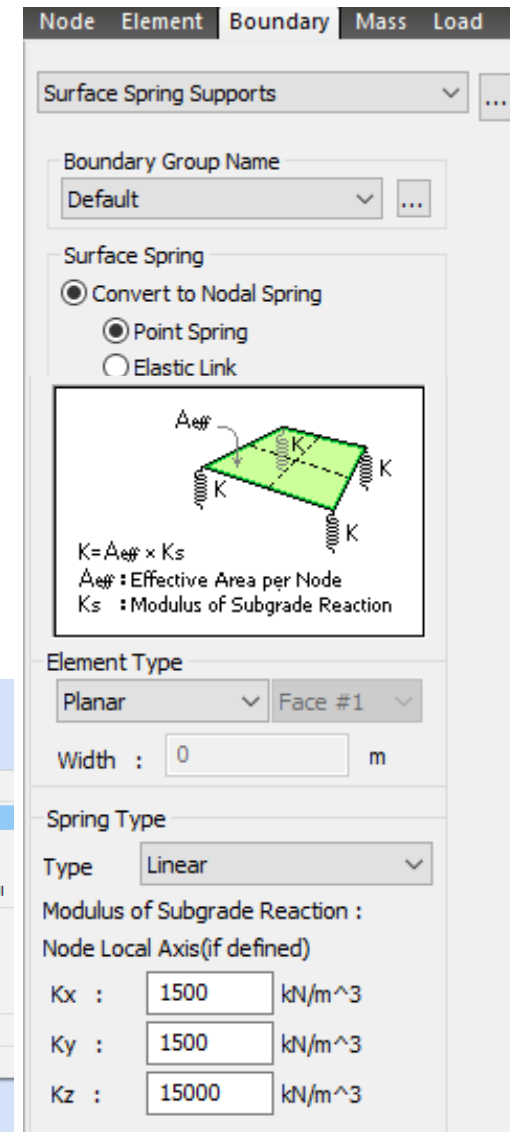
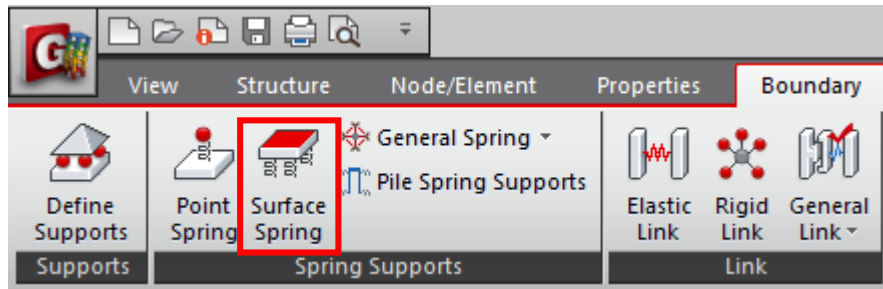
Click [Eigenvalue Analysis control]

Number of Frequencies: 15 > Click [OK]

Click [Close]



# Boundary Condition



## Add Spring Supports

Add Surface Springs

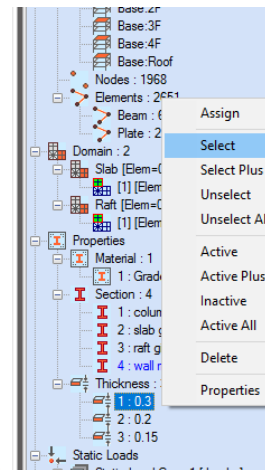
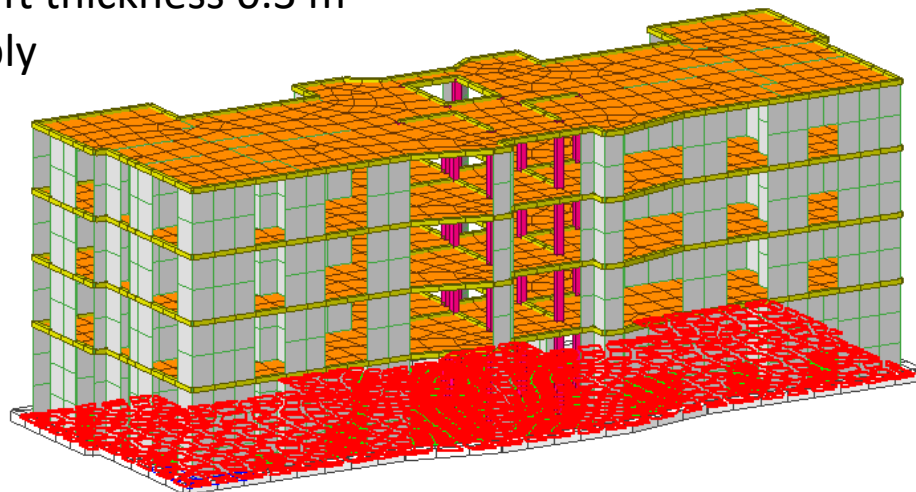
Element Type: Planar

Spring Type: Linear

$K_x = K_y = 1,500$   $K_z = 15,000$  kN/m<sup>3</sup>

Select Raft thickness 0.3 m

Click Apply



# Load combination

Automatic Generation of Load Combinations

Option  
 Add  Replace

Code Selection  
 Steel  Concrete  SRC  
 Cold Formed Steel  Footing

Design Code :

Scale Up of Response Spectrum Load Cases  
 Scale Up Factor :   

Factor	Load Case

Wind Load Factor  
 Strength-level  Service-level

Consider Lateral Soil Pressure Factor  
 Load Factor :

Manipulation of Construction Stage Load Case  
 ST : Static Load Case  
 CS : Construction Stage Load Case  
 ST Only  CS Only  ST+CS

Consider Orthogonal Effect

100 : 30 Rule  
 SRSS(Square-Root-of-Sum-of-Squares)

Generate Additional Load Combinations  
 for Special Seismic Load  
 for Vertical Seismic Forces

Will Execute Construction Stage Analysis  
 Consider Losses for Prestress Load Cases

Transfer Stage :   
 Service Load Stage :

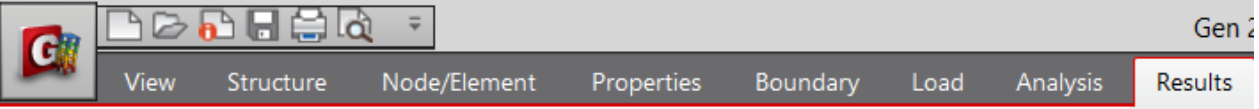
## Generate Load Combo

Results > Combinations > Concrete Design > Auto Generation

Select Concrete and Design

Auto Generation

Design Code: ACI318-14



Load Combinations

General | Steel Design | **Concrete Design** | SRC Design | Cold Formed Steel Design | Footing Design

Load Combination List

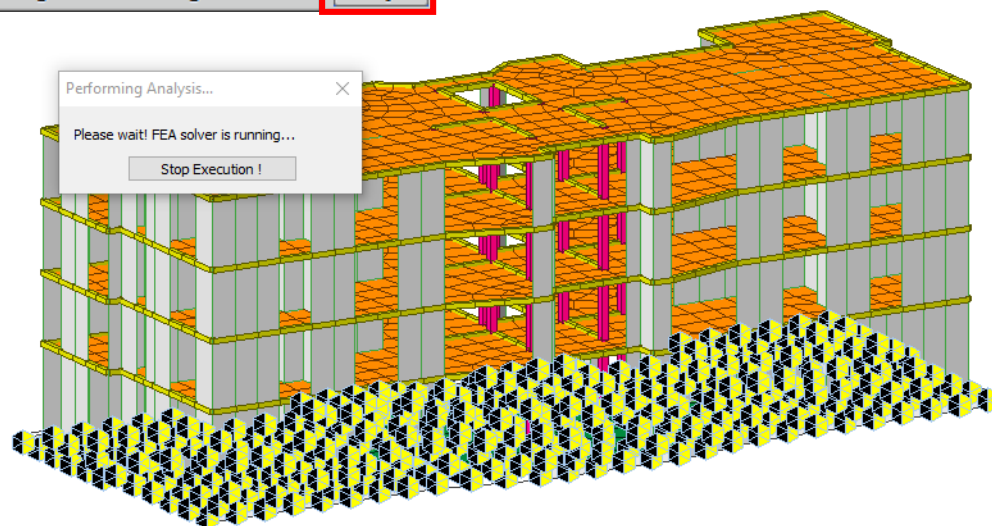
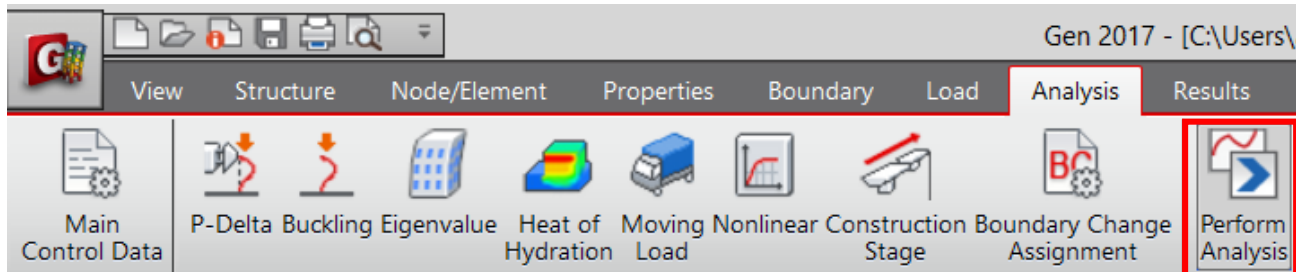
No	Name	Active	Type	Description
1	cLCB1	Stren	Add	1.4(D)
2	cLCB2	Stren	Add	1.2(D) + 1.6(L)
3	cLCB3	Stren	Add	1.2(D) + 1.0wind x + 1.0(L)
4	cLCB4	Stren	Add	1.2(D) + 1.0wind y + 1.0(L)
5	cLCB5	Stren	Add	1.2(D) - 1.0wind x + 1.0(L)
6	cLCB6	Stren	Add	1.2(D) - 1.0wind y + 1.0(L)
7	cLCB7	Stren	Add	1.2(D) + 1.0(1.0)(RX(RS)+RX(E
8	cLCB8	Stren	Add	1.2(D) + 1.0(1.0)(RX(RS)-RX(ES
9	cLCB9	Stren	Add	1.2(D) + 1.0(1.0)(RY(RS)+RY(E
10	cLCB10	Stren	Add	1.2(D) + 1.0(1.0)(RY(RS)-RY(E
11	cLCB11	Stren	Add	1.2(D) - 1.0(1.0)(RX(RS)+RX(ES
12	cLCB12	Stren	Add	1.2(D) - 1.0(1.0)(RX(RS)-RX(ES
13	cLCB13	Stren	Add	1.2(D) - 1.0(1.0)(RY(RS)+RY(E
14	cLCB14	Stren	Add	1.2(D) - 1.0(1.0)(RY(RS)-RY(ES
15	cLCB15	Stren	Add	0.9D + 1.0wind x
16	cLCB16	Stren	Add	0.9D + 1.0wind y
17	cLCB17	Stren	Add	0.9D - 1.0wind x
18	cLCB18	Stren	Add	0.9D - 1.0wind y
19	cLCB19	Stren	Add	0.9(D) + 1.0(1.0)(RX(RS)+RX(E
20	cLCB20	Stren	Add	0.9(D) + 1.0(1.0)(RX(RS)-RX(ES
21	cLCB21	Stren	Add	0.9(D) + 1.0(1.0)(RY(RS)+RY(E

Load Cases and Factors

LoadCase	Factor
dead(ST)	1.4000
*	

File Name: C:\Users\j.a.martinez\Desktop\Slab and Wall Tutorial\Slab and Wall Tutorial\Load Combination Sheet

# Perform Analysis



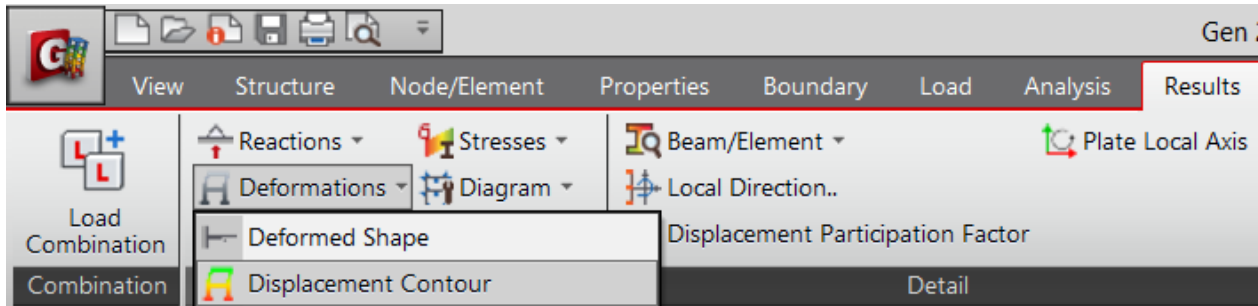
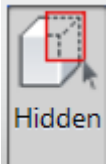
Message Window

```

MULTI-FRONTAL SOLUTION HAS BEEN COMPLETED.
DISPLACEMENT/FORCE/STRESS OUTPUT FOR ECCENTRIC MOMENT LOADING.
ACCIDENTAL ECCENTRIC MOMENT   ELEM. :   3440 OF   3440

-----SOLUTION TERMINATED
YOUR MIDAS JOB IS SUCCESSFULLY COMPLETED.....C:\Users\a.martinez\Desktop\Slab
TOTAL SOLUTION TIME..:   49.16 [SEC]
    
```

# Results: Deformations



Reactions | **Deformations** | Forces | Stresses

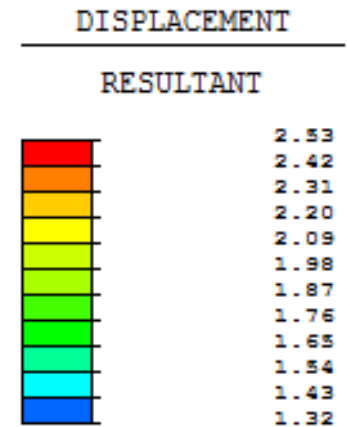
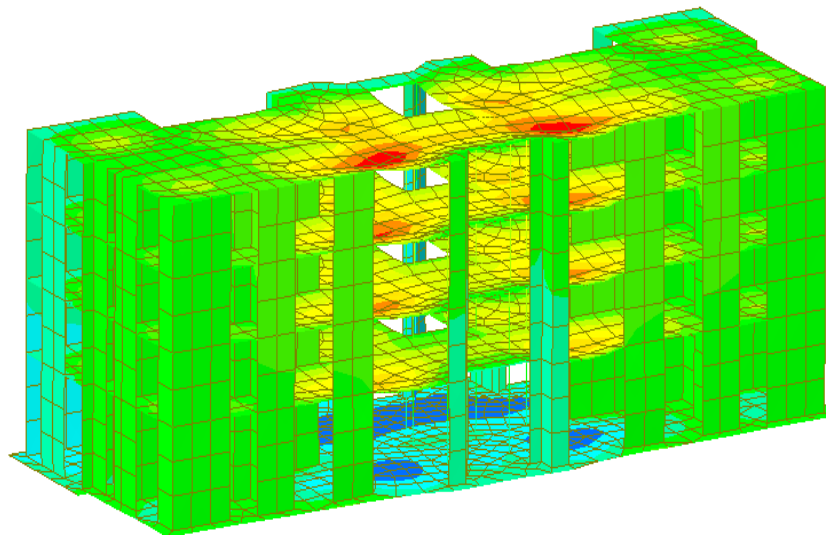
Displacement Contour

Load Cases/Combinations  
 ST: live  
 Step

Displacement    Velocity  
 Acceleration  
 Absolute Acceleration

Components  
 DX    DY    DZ  
 RX    RY    RZ  
 RW  
 DXY    DYZ    DXZ  
 DXYZ  
 Local (if defined)

Type of Display  
 Contour    Deform  
 Values    Legend



ST: LIVE

---

MAX : 2337  
 MIN : 1179

---

FILE: SLAB AND W~  
 UNIT : mm



# Results: Axial Plate Forces

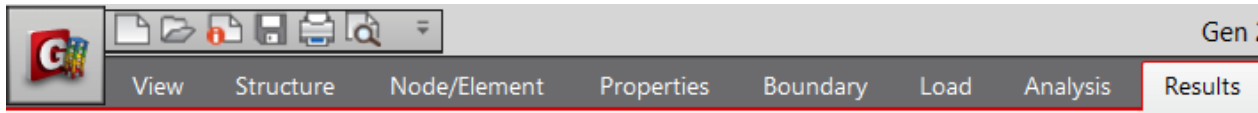


Plate Forces/Moments

Load Cases/Combinations  
 ST: live  
 Step

Plate Force Options  
 Local  
 UCS Current UCS  
 Print UCS Axis

Element  Avg. Nodal  
 Avg. Nodal Active Only

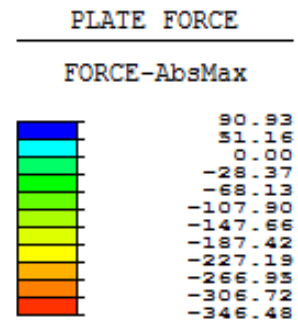
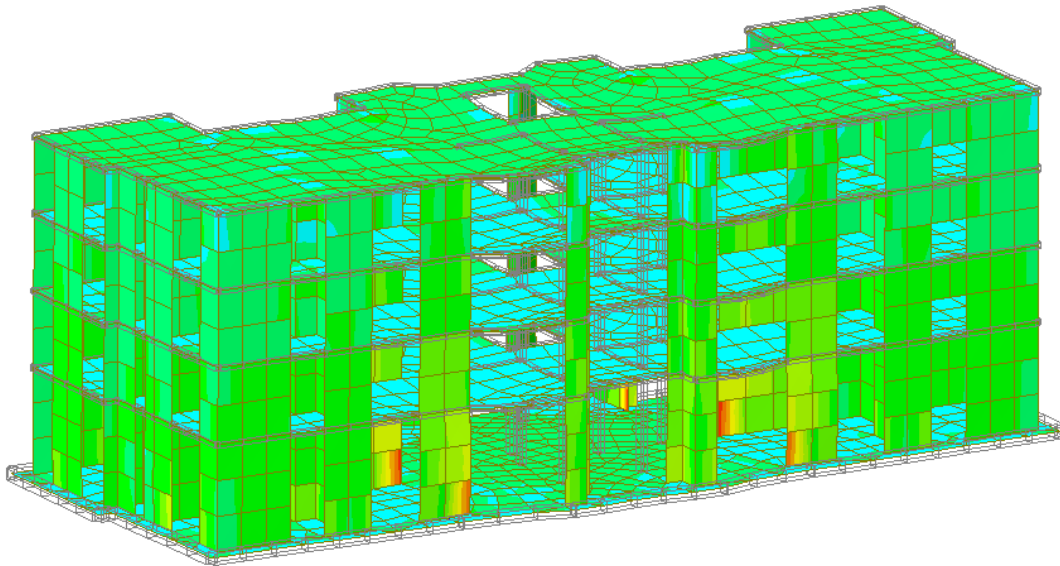
Components  
 Fxx  Fyy  Fxy  
 Fmax  Fmin  FMax  
 Mxx  Myy  Mxy  
 Mmax  Mmin  MMax  
 Vxx  Vyy  VMax  
 Wood Armer Moment

Fvector  Mvector  
 Positive  Negative

Vector Scale Factor  
 Length: 1.000000  
 Thickness: 2

Type of Display  
 Contour  Values  Deform  Legend

- Reactions
- Stresses
- Deformations
- Diagram
- Forces
  - Truss Forces
  - Beam Forces/Moments
  - Beam Diagrams
  - Plate Forces/Moments
- HY Results



SCALEFACTOR=  
 5.0141E+002

ST: LIVE  
 ELEMENT

---

MAX : 3069  
 MIN : 3529

---

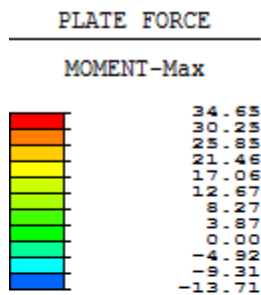
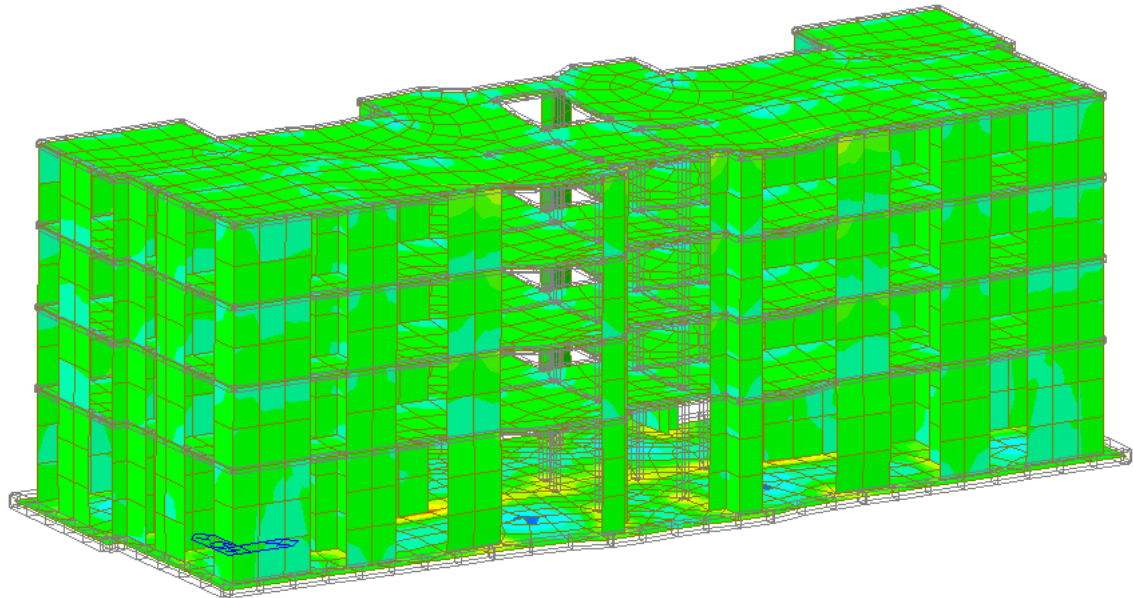
FILE: SLAB AND W~  
 UNIT: kN/m

# Results: Moments Y

The screenshot shows the midas Gen software interface with the 'Results' menu open. The menu options are: Reactions, Deformations, Forces, Stresses, Truss Forces, Beam Forces/Moments, Beam Diagrams, and Plate Forces/Moments. The 'Plate Forces/Moments' option is selected.

On the left side, there are several configuration panels:

- Plate Forces/Moments:** A dropdown menu showing 'Plate Forces/Moments'.
- Load Cases/Combinations:** 'ST: live' is selected.
- Plate Force Options:** 'Local' is selected under UCS, and 'Element' is selected under display options.
- Components:** 'Mmax' is selected.
- Vector Scale Factor:** Length is 1.000000 and Thickness is 2.
- Type of Display:** 'Contour' and 'Deform' are checked.



SCALEFACTOR=  
5.0141E+002

ST: LIVE  
ELEMENT

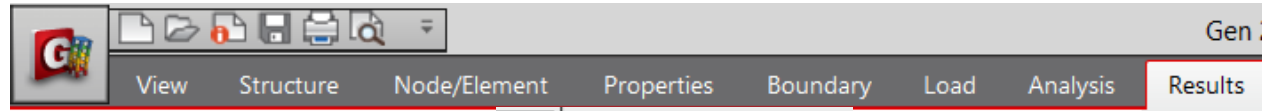
---

MAX : 823  
MIN : 2820

---

FILE: SLAB AND W-  
UNIT: kN\*m/m

# Results: Shear Forces



Plane-Stress/Plate Stresses

Load Cases/Combinations

ST: live

Step

Stress Options

Local  
 UCS Current UCS  
 Print UCS Axis

Element  Avg. Nodal  
 Avg. Nodal Active Only

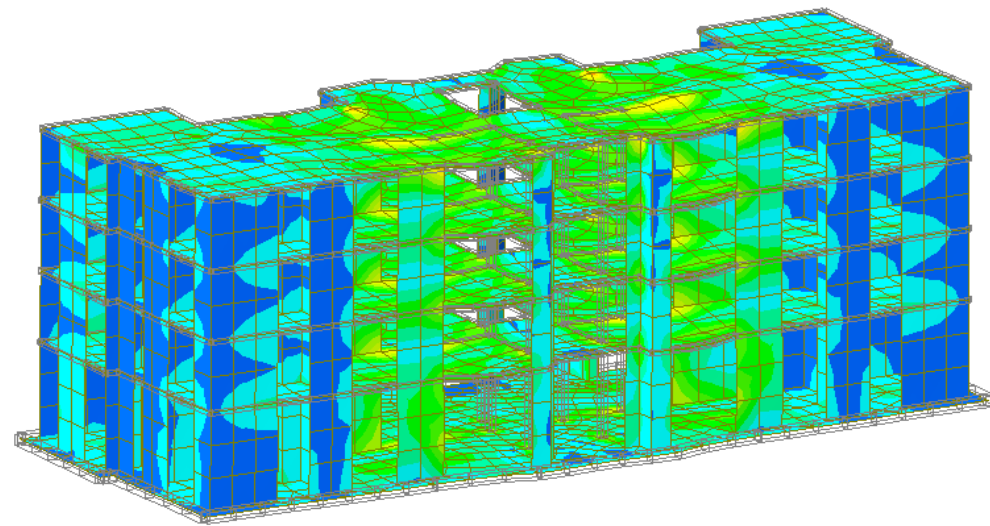
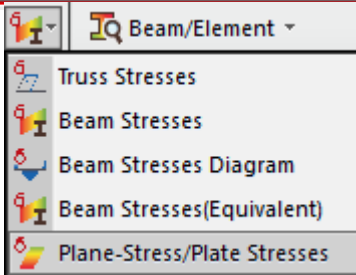
Top  Bottom  
 Both Sides  Abs Max

Components

Siq-xx  Siq-yy  Siq-zz  
 Siq-xy  Siq-yz  Siq-xz  
 Siq-max  Siq-min  Siq-eff  
 Max-Shear

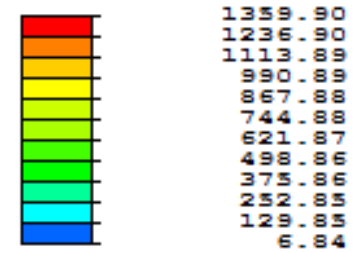
Type of Display

Contour  Deform  
 Values  Legend  
 Animate  Undeformed



PLN STS/PLT STRS

MAX-SHEAR TOP



SCALEFACTOR=

5.0680E+002

ST: LIVE

AVG NODAL

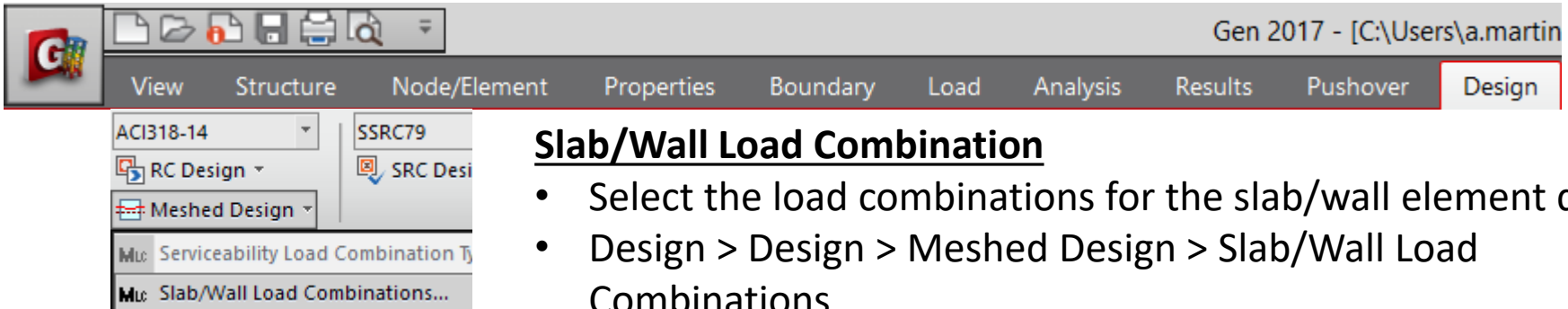
MAX : 2574

MIN : 3505

FILE: SLAB AND W~

UNIT: kN/m<sup>2</sup>

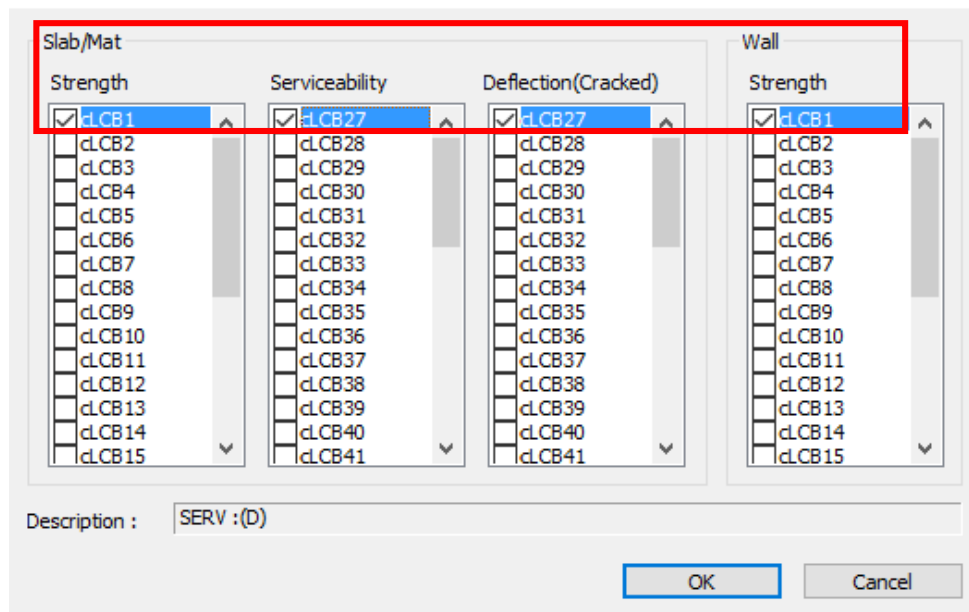
# Slab and wall load combinations



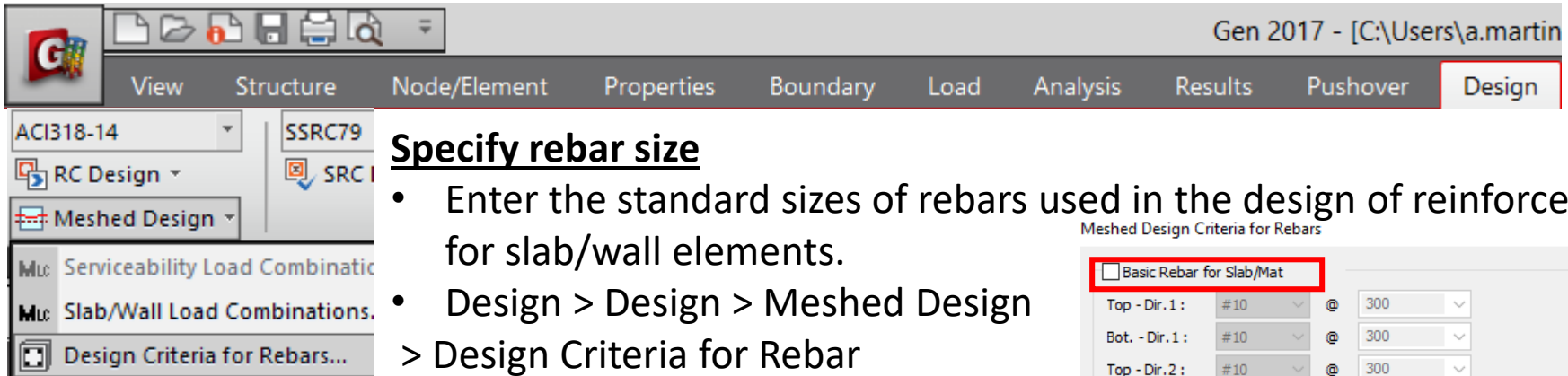
## Slab/Wall Load Combination

- Select the load combinations for the slab/wall element design.
- Design > Design > Meshed Design > Slab/Wall Load Combinations
- Select the 1<sup>st</sup> load combination in each column to consider during the slab/wall design.

Meshed Slab/Wall Load Combinations

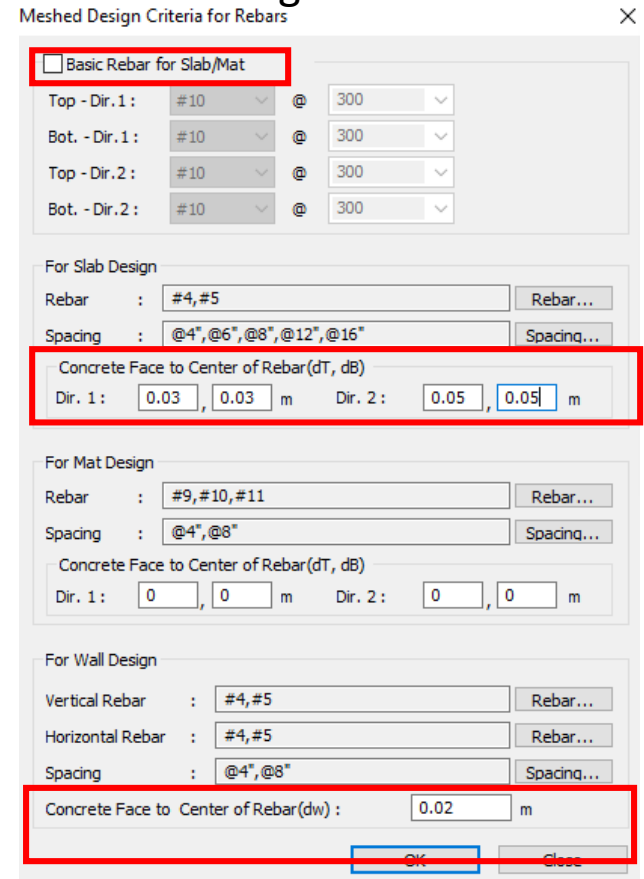


# Define Design Criteria for Rebar

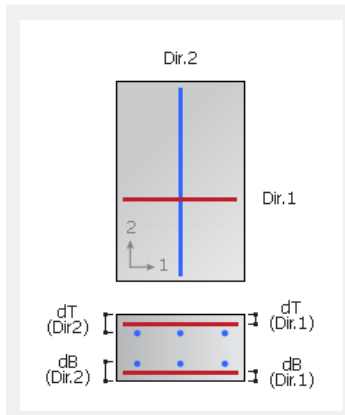


## Specify rebar size

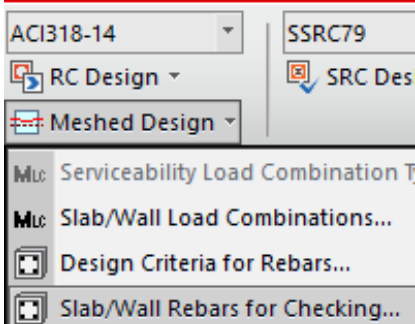
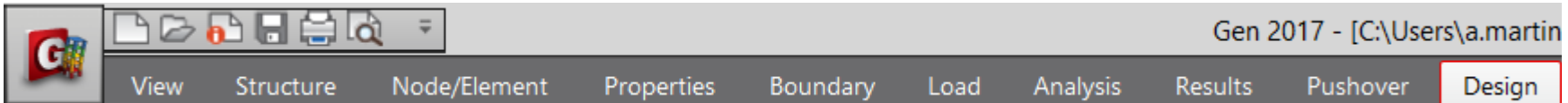
- Enter the standard sizes of rebars used in the design of reinforcement for slab/wall elements.
- Design > Design > Meshed Design > Design Criteria for Rebar
- Check off [Basic Rebar for Slab]
- For Slab Design:
  - Dir. 1 : 0.03 m, 0.03 m
  - Dir. 2 : 0.05 m, 0.05 m
- For Wall Design
  - Face to Center Rebar 0.02m



Slab/Mat

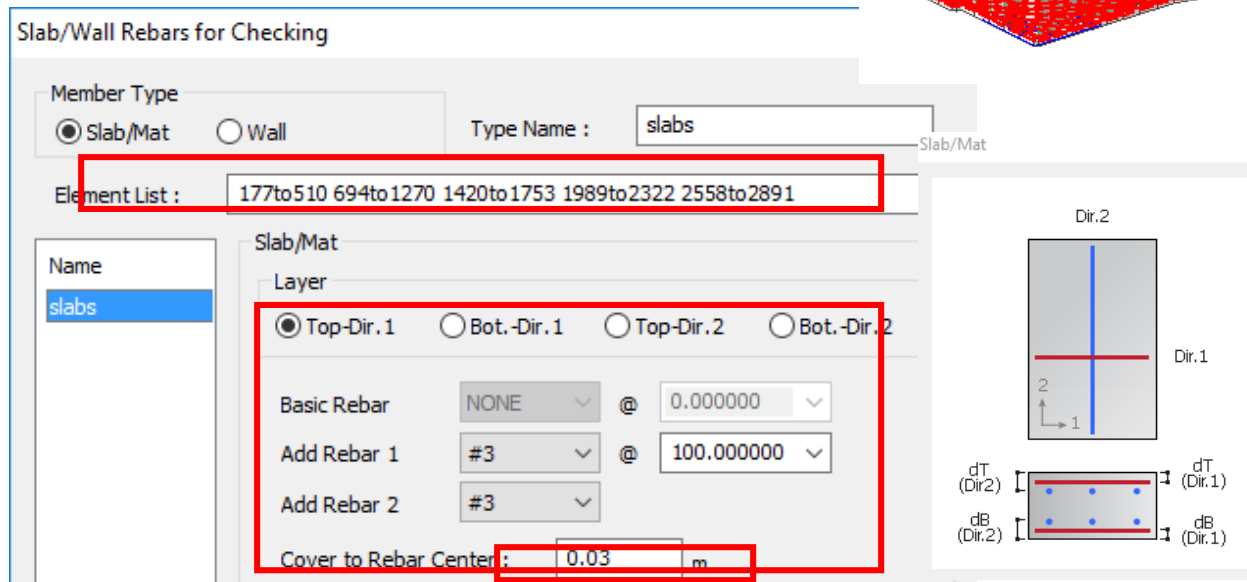
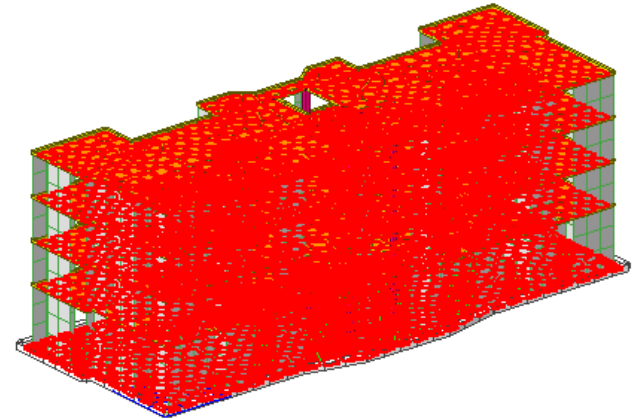


# Slab/Wall Rebar Checking Data

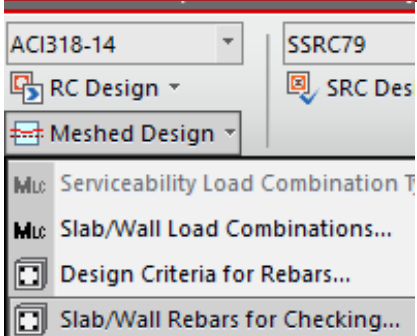
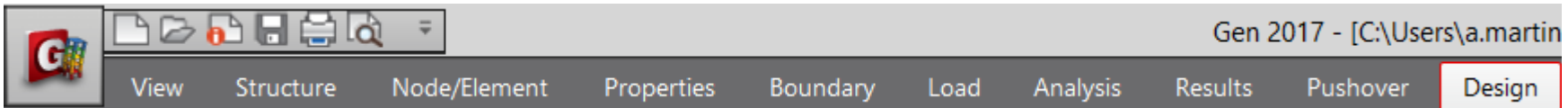


## Specify rebar size

- Select all slabs from tree menu
- Layer Top Dir 1
- Add rebar 1: #3 @ 100mm
- Add rebar 2: #3
- Add/Replace

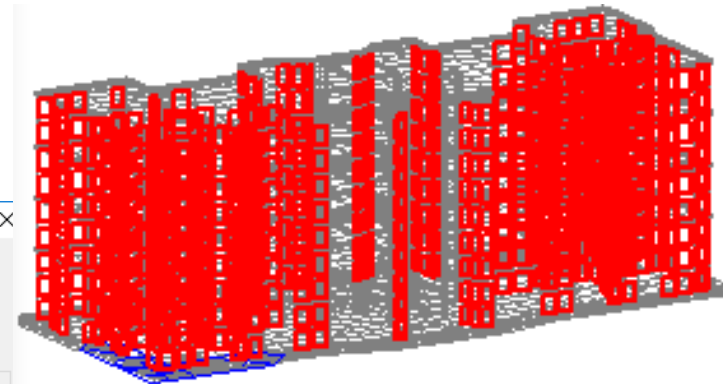
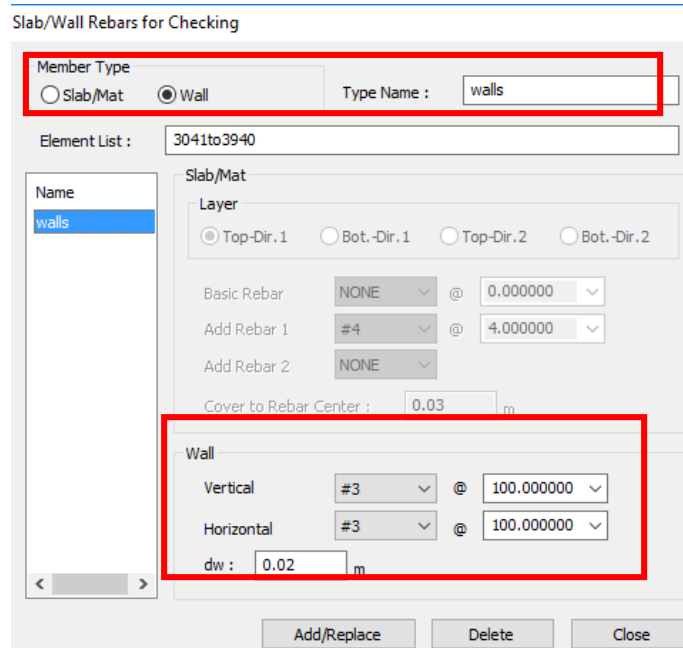


# Slab/Wall Rebar Checking Data



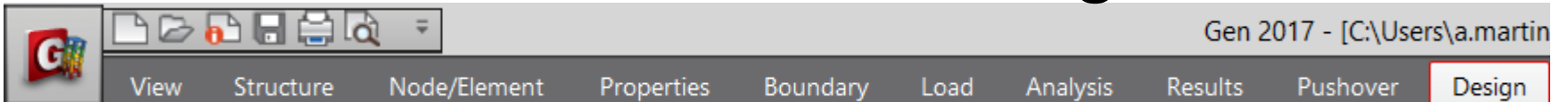
## Specify rebar size

- Select all 0.2m walls from tree menu
- Layer Top Dir 1
- Vertical 1: #3 @ 100mm
- Horizontal 2: #3 @ 100mm
- Add/Replace





# Slab Flexural Design



**Design**

Slab Flexural Design

Load Cases/Combinations  
ALL COMBINATION

Flexural Design

Element  Avg. Nodal

Element  Width 1 m

Top  Bottom  Both

Dir. 1  Dir. 2

Type of Display  
 Contour  Legend  
 Values

One-Way Flexural Design

Element Edge  
 Both  Left  Right

Rebar  
 As\_req (m<sup>2</sup>/m)  
 Rho\_req  
 x/d  
 Resistance Ratio  
 Wood Armer Moment

Meshed Design

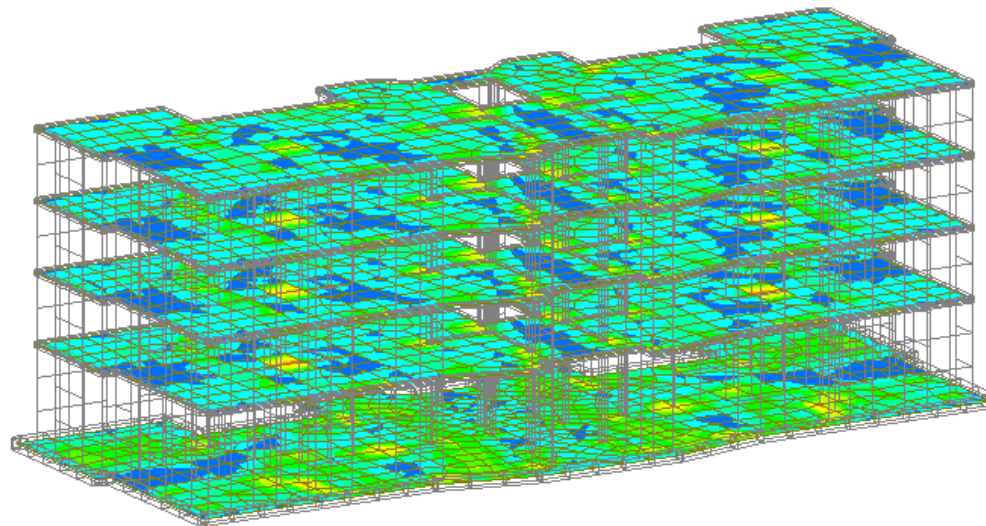
- Serviceability Load Combination Type...
- Slab/Wall Load Combinations...
- Design Criteria for Rebars...
- Slab/Wall Rebars for Checking...
- Serviceability Parameters...
- Slab Flexural Design...**

## Run Design

Select Avg. Nodal

Dir. 1

Resistance Ratio : The ratio of the design moment to the moment resistance when the designed rebar spacing is applied.



**SLAB DESIGN**

Position:  
Top & Bot

Smoothing:  
Element (Element)

Component:  
Direction 1  
Resistance Ratio

# Slab Flexural Design

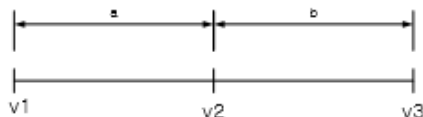
## Procedure

### [Smoothing]

Design > Meshed Slab/Wall Design >  
Slab Flexural Design



**Width smoothing :**  
weighted average method



weighted average for 'v2' =

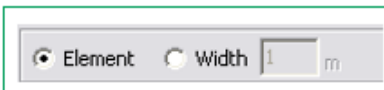
$$\frac{(v1+v2) \times a/2 + (v3+v2) \times b/2}{a+b}$$

For practical design, smooth moment distributions are preferred. By selecting the smoothing option, the program can consider the smooth moment in slab design.



**Element:** Design results are displayed using the internal forces calculated at each node of elements. (no smoothing)

**Avg. Nodal:** Design results are displayed using the average internal nodal forces of the contiguous elements sharing the common nodes.



**Element:** Design results are produced for moments at each node of slab elements. (no smoothing)

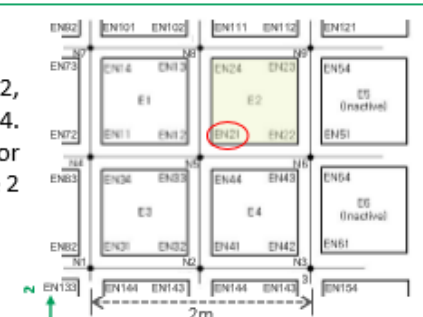
**Width:** Design result of slab elements at each node is produced using the average of the bending moments of the contiguous slab elements with the specified width.

### (Example) Design force for Node. EN21

In one plate element, 4 internal forces exist. For the element E2, member forces exist at the node EN21, EN22, EN23 and EN24. Following equations show how the smoothing option works for the node EN21. (Assume that rebar direction is selected as Angle 2 for Width smoothing direction.)

- (1) **Element + Element:** EN21
- (2) **Avg. Nodal +Element:** (EN12+EN21+EN33+EN44)/4
- (3) **Element + Width 2m (dir. 1):**

$$\begin{aligned} & \{ (EN21+EN92) \cdot 1m/2 + (EN21+EN101) \cdot 1m/2 + (EN21+EN73) \cdot 1m/2 + (EN21+EN14) \cdot 1m/2 \\ & + (EN21+EN72) \cdot 1m/2 + (EN21+EN11) \cdot 1m/2 + (EN21+EN83) \cdot 1m/2 + (EN21+EN34) \cdot 1m/2 \\ & + (EN21+EN82) \cdot 1m/2 + (EN21+EN31) \cdot 1m/2 + (EN21+EN133) \cdot 1m/2 + (EN21+EN144) \cdot 1m/2 \\ & + (EN21+EN112) \cdot 1m/2 + (EN21+EN121) \cdot 1m/2 + (EN21+EN23) \cdot 1m/2 + (EN21+EN154) \cdot 1m/2 \\ & + (EN21+EN22) \cdot 1m/2 + (EN21+EN151) \cdot 1m/2 + (EN21+EN43) \cdot 1m/2 + (EN21+EN64) \cdot 1m/2 \\ & + (EN21+EN42) \cdot 1m/2 + (EN21+EN61) \cdot 1m/2 + (EN21+EN143) \cdot 1m/2 + (EN21+EN154) \cdot 1m/2 \} / (1m \cdot 24) \end{aligned}$$



# Slab Flexural Design

## Procedure

[Design strength of  
flexural member]

*Design Strength*  $\geq$  *Required Strength*

$\Phi$ (Nominal Strength)  $\geq U$

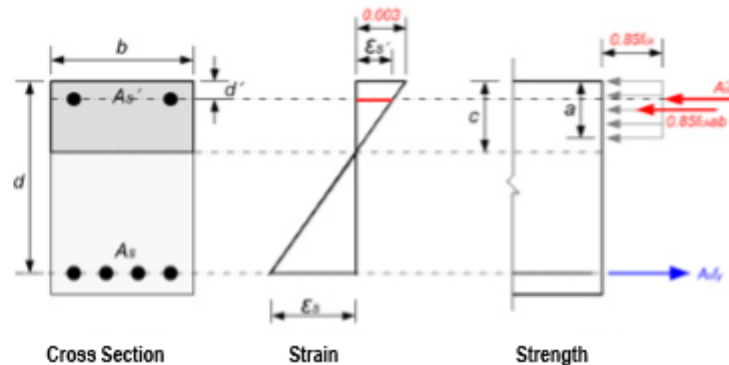
### 1. Design Strength

□ Flexural strength of meshed slab is calculated based on the doubly reinforced beam design method.

Doubly Reinforced:  $M_{n1} = A_s' f_y (d - d')$

$$M_{n2} = (A_s - A_s') f_y \left(d - \frac{a}{2}\right) \quad \text{where, } a = \frac{(A_s - A_s') f_y}{0.85 f_{ck} b}$$

$$\Phi M_n = \Phi (M_{n1} - M_{n2}) = \Phi [A_s' f_y (d - d') + (A_s - A_s') f_y \left(d - \frac{a}{2}\right)]$$



# Slab Flexural Design

## Procedure

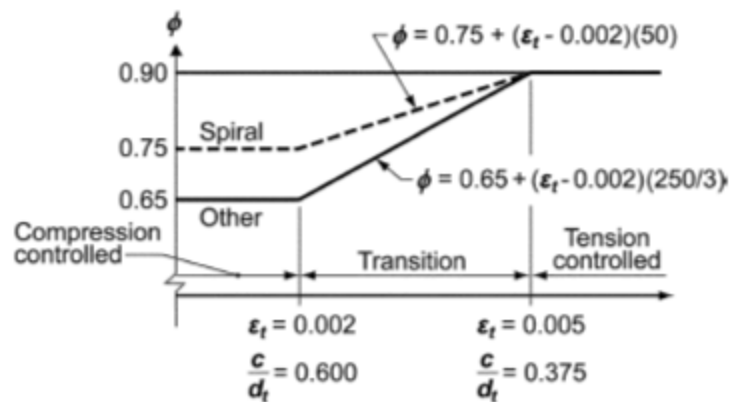
[Design strength of  
flexural member]

$$\text{Design Strength} \geq \text{Required Strength}$$

$$\Phi(\text{Nominal Strength}) \geq U$$

## 2. Strength reduction factor

Strength reduction factor needs to be calculated based on the tensile strain in extreme tension steel.



Interpolation on  $c/d_t$ :  
 Spiral  $\phi = 0.75 + 0.15[(1/c/d_t) - (5/3)]$   
 Other  $\phi = 0.65 + 0.25[(1/c/d_t) - (5/3)]$

Fig. R9.3.2—Variation of  $\phi$  with net tensile strain in extreme tension steel,  $\epsilon_t$ , and  $c/d_t$  for Grade 60 reinforcement and for prestressing steel.

➡ Strength reduction factor is uniformly applied as 0.9 in midas Gen.

# Slab Flexural Design

## Procedure

[Design strength of  
flexural member]

### 3. Minimum reinforcement of flexural members

$$A_{s,min} = 0.002bh \quad \text{for } f_y = 40\text{ksi or } 50\text{ksi}$$

$$A_{s,min} = 0.0018bh \quad \text{for } f_y = 60\text{ksi}$$

$$A_{s,min} = \frac{0.0018 \times 60000}{f_y} bh \quad \text{for } f_y > 60\text{ksi}$$

➡ Above limitation is applied in midas Gen. If  $f_y > 60\text{ksi}$ ,  $A_{s,min}$  is the smaller of 0.0014 and  $\frac{0.0018 \times 60000}{f_y} bh$ .

### 4. Maximum reinforcement of flexural members

#### B.10.3 — General principles and requirements

**B.10.3.3** — For flexural members and members subject to combined flexure and compressive axial load where  $\phi P_n$  is less than the smaller of  $0.10f'_c A_g$  and  $\phi P_b$ , the ratio of reinforcement,  $\rho$ , provided shall not exceed 0.75 of the ratio  $\rho_b$  that would produce balanced strain conditions for the section under flexure without axial load. For members with compression reinforcement, the portion of  $\rho_b$  equalized by compression reinforcement need not be reduced by the 0.75 factor.

➡ In midas Gen, maximum rebar ratio is limited as 75% of balanced rebar ratio as per Appendix B10.3.3.

### 5. Minimum Spacing Limit

Rebar spacing shall not be less than the smaller of "3\*slab thickness" and 18in.

# Slab Flexural Design

## Procedure

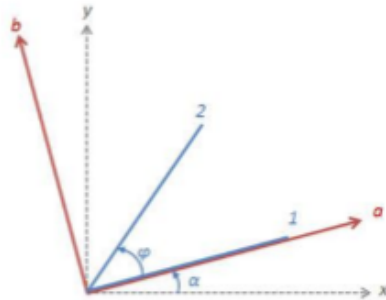
### [Wood Armer Moment]

#### 6. Required Moment Strength calculated from Wood Armer moment

From the analysis results, following plate forces about the local axis are calculated

- $m_{xx}$
- $m_{yy}$
- $m_{xy}$

In order to calculate design forces in the reinforcement direction, angle  $\alpha$  and  $\varphi$  will be taken as following figure:



x, y: local axis of plate element

1, 2: reinforcement direction

$\alpha$ : angle between local x-direction and reinforcement direction 1

$\varphi$ : angle between reinforcement direction 1 and reinforcement direction 2

Firstly, internal forces ( $m_{xx}$ ,  $m_{yy}$  and  $m_{xy}$ ) are transformed into the a-b coordinate system.

$$m_a = \frac{m_{xx} + m_{yy}}{2} + \frac{m_{xx} - m_{yy}}{2} \cos 2\alpha + m_{xy} \sin 2\alpha$$

$$m_b = \frac{m_{xx} + m_{yy}}{2} - \frac{m_{xx} - m_{yy}}{2} \cos 2\alpha - m_{xy} \sin 2\alpha$$

$$m_{ab} = -\frac{m_{xx} - m_{yy}}{2} \sin 2\alpha + m_{xy} \cos 2\alpha$$

# Slab Flexural Design

## Procedure

### [Wood Armer Moment]

Then, Wood-Armer moments are calculated as follows:

#### [Bottom Rebar]

$$m_{ud1} = m_a - 2m_{ab} \cot \varphi + m_b \cot^2 \varphi + \left| \frac{m_{ab} - m_b \cot \varphi}{\sin \varphi} \right|$$

$$m_{ud2} = \frac{m_b}{\sin^2 \varphi} + \left| \frac{m_{ab} - m_b \cot \varphi}{\sin \varphi} \right|$$

When  $m_{ud1} < 0$  and  $m_{ud2} > 0$ ,

$$m_{ud1} = 0$$

$$m_{ud2} = \max \left\{ 0, \frac{m_b + |(m_{ab} - m_b \cot \varphi)^2 / (m_a - 2m_{ab} \cot \varphi + m_b \cot^2 \varphi)|}{\sin^2 \varphi} \right\}$$

When  $m_{ud1} > 0$  and  $m_{ud2} < 0$ ,

$$m_{ud1} = \max \left\{ 0, m_a - 2m_{ab} \cot \varphi + m_b \cot^2 \varphi + \left| \frac{(m_{ab} - m_b \cot \varphi)^2}{m_b} \right| \right\}$$

$$m_{ud2} = 0$$

When  $m_{ud1} < 0$  and  $m_{ud2} < 0$ ,

$$m_{ud1} = 0$$

$$m_{ud2} = 0$$

#### [Top Rebar]

$$m'_{ud1} = m_a - 2m_{ab} \cot \varphi + m_b \cot^2 \varphi - \left| \frac{m_{ab} - m_b \cot \varphi}{\sin \varphi} \right|$$

$$m'_{ud2} = \frac{m_b}{\sin^2 \varphi} - \left| \frac{m_{ab} - m_b \cot \varphi}{\sin \varphi} \right|$$

When  $m'_{ud1} > 0$  and  $m'_{ud2} < 0$ ,

$$m'_{ud1} = 0$$

$$m'_{ud2} = \min \left\{ 0, \frac{m_b - |(m_{ab} - m_b \cot \varphi)^2 / (m_a - 2m_{ab} \cot \varphi + m_b \cot^2 \varphi)|}{\sin^2 \varphi} \right\}$$

When  $m'_{ud1} < 0$  and  $m'_{ud2} > 0$ ,

$$m'_{ud1} = \min \left\{ 0, m_a - 2m_{ab} \cot \varphi + m_b \cot^2 \varphi - \left| \frac{(m_{ab} - m_b \cot \varphi)^2}{m_b} \right| \right\}$$

$$m'_{ud2} = 0$$

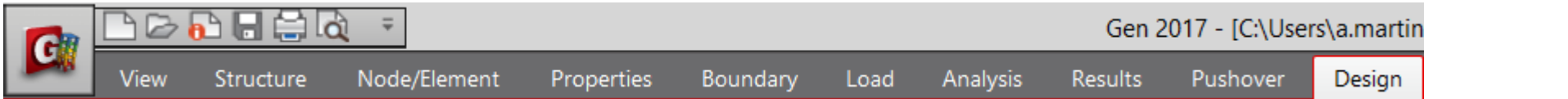
When  $m'_{ud1} > 0$  and  $m'_{ud2} > 0$ ,

$$m'_{ud1} = 0$$

$$m'_{ud2} = 0$$



# Slab Serviceability Check



**Design**

Slab Serviceability Checking

Load Combinations: ALL COMBINATION

Element Avg. Nodal

Element Width 1 m

Top Bottom Both

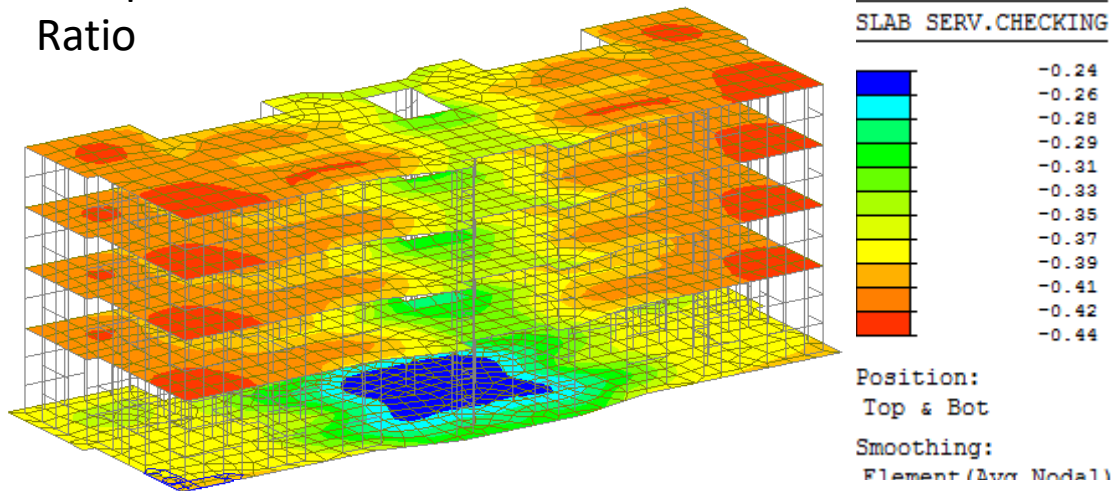
Dir. 1 Dir. 2

Type of Display:  Contour  Legend

Deflection:  Uncracked  Creep (Phi: 3)

Value Ratio

**Run Check**  
 Check UnCracked  
 Creep Phi : 3  
 Ratio



**TABLE 9.5(b) — MAXIMUM PERMISSIBLE COMPUTED DEFLECTIONS**

Type of member	Deflection to be considered	Deflection limitation
Flat roofs not supporting or attached to nonstructural elements likely to be damaged by large deflections	Immediate deflection due to live load <b>L</b>	$l/180^*$
Floors not supporting or attached to nonstructural elements likely to be damaged by large deflections	Immediate deflection due to live load <b>L</b>	$l/360$
Roof or floor construction supporting or attached to nonstructural elements likely to be damaged by large deflections	That part of the total deflection occurring after attachment of nonstructural elements (sum of the long-term deflection due to all sustained loads and the immediate deflection due to any additional live load) <sup>†</sup>	$l/480^{\ddagger}$
Roof or floor construction supporting or attached to nonstructural elements not likely to be damaged by large deflections	That part of the total deflection occurring after attachment of nonstructural elements (sum of the long-term deflection due to all sustained loads and the immediate deflection due to any additional live load) <sup>†</sup>	$l/240^{\S}$

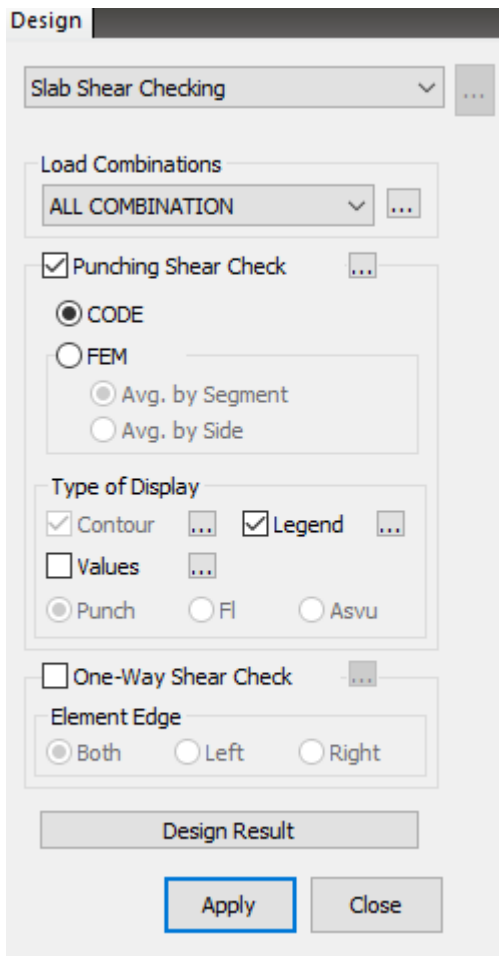
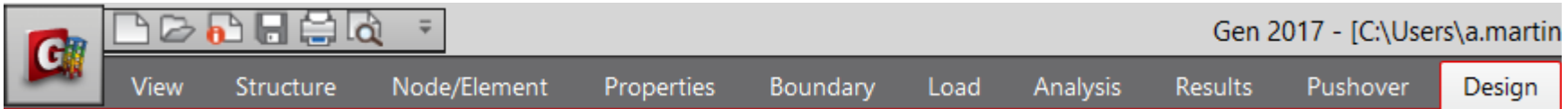
\*Limit not intended to safeguard against ponding. Ponding should be checked by suitable calculations of deflection, including added deflections due to ponded water, and considering long-term effects of all sustained loads, camber, construction tolerances, and reliability of provisions for drainage.

<sup>†</sup>Long-term deflection shall be determined in accordance with 9.5.2.5 or 9.5.4.3, but may be reduced by amount of deflection calculated to occur before attachment of nonstructural elements. This amount shall be determined on basis of accepted engineering data relating to time-deflection characteristics of members similar to those being considered.

<sup>‡</sup>Limit may be exceeded if adequate measures are taken to prevent damage to supported or attached elements.

<sup>§</sup>Limit shall not be greater than tolerance provided for nonstructural elements. Limit may be exceeded if camber is provided so that total deflection minus camber does not exceed limit.

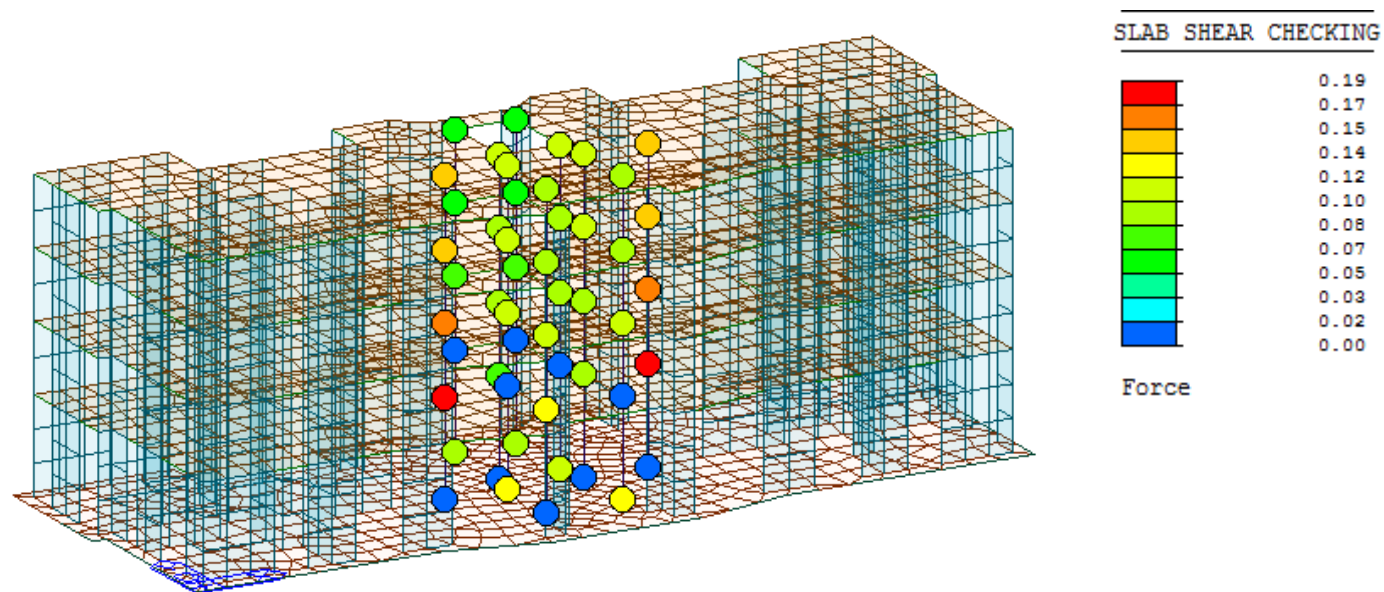
# Slab Shear Checking



## Run Check

Check on Punching Shear Check

Slab Shear Checking Produce the two-way shear (punching shear) check results at the supports of slab elements or at concentrated loads and the one way shear check results along the user-defined Shear Check Lines



# Slab Shear Checking

## Procedure

[Shear strength]

[Punching Shear Check(By CODE)]

$$\Phi V_n \geq V_u$$

$$V_n = V_c + V_s$$

Where,  $V_c$  : nominal shear strength provided by concrete

$V_s$  : nominal shear strength provided by shear reinforcement

➡ Shear strength reduction factor is applied as 0.75.

## 1. Shear strength of Concrete, $V_c$

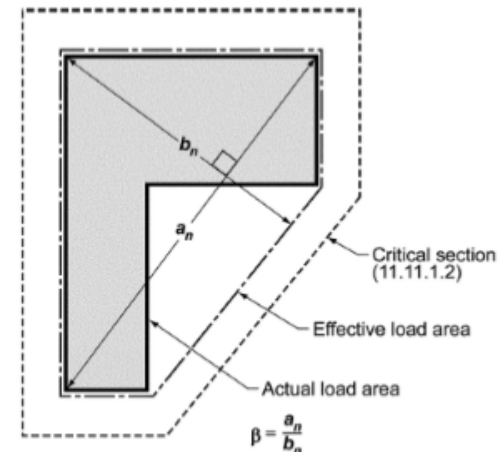
$$V_c = \min \left\{ \begin{array}{l} \Phi \left( 2 + \frac{4}{\beta} \right) \lambda \sqrt{f_{ck}} \\ \Phi \left( 2 + \frac{\alpha_s d}{b_o} \right) \lambda \sqrt{f_{ck}} \\ \Phi 4 \lambda \sqrt{f_{ck}} \end{array} \right.$$

where,  $\beta$ : Ratio of the maximum to the minimum dimension of a column or wall

$b_o$ : Critical perimeter

$\alpha_s$ : 40(Interior column), 30(Edge column), 20(Corner column)

$\lambda$ : 1.0 (normal weight concrete)



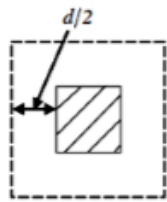
# Slab Shear Checking

## Procedure

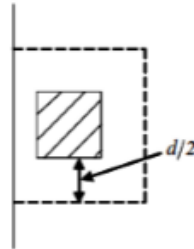
### [Punching Shear Check(By CODE)]

#### Punching shear perimeter for calculating concrete shear strength

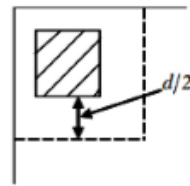
In this method, the program takes the axial force in the column supporting the slab as the shear force ( $V_u$ ). The basic control perimeter is taken at a distance  $d/2$  from the column face (as shown in the diagram below).



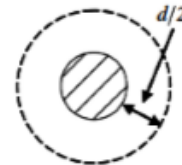
Interior Column



Edge Column



Edge Column



Circular Column

#### Maximum Shear Strength by Concrete (ACI318-11 11.1.3.1)

$$V_n \leq 6\sqrt{f_{ck}} b_o d$$

$$V_c \leq 2\lambda\sqrt{f_{ck}} b_o d$$

➡ In midas Gen, the above limitation is applied when slab thickness is larger than 200mm.

# Slab Shear Checking

## 2. Shear strength of reinforcement, $V_s$

$$V_s = \frac{A_v f_y d}{s}$$

$$V_{s,\min} = 4\sqrt{f_{ck}} b_w d$$

### Shear rebar spacing limit

$$s \leq 0.5d$$

$$s \leq \begin{cases} 0.75d & \text{for } v_u \leq 6\phi\lambda\sqrt{f_{ck}} \\ 0.50d & \text{for } v_u > 6\phi\lambda\sqrt{f_{ck}} \end{cases}$$

$$g \leq 2d$$

### Minimum Shear Rebar Area

$$\frac{1}{2}\phi V_c < V_u \leq \phi V_c$$

$$A_{v,\min} = 0.75\sqrt{f_{ck}} \frac{b_w s}{f_y} \quad \text{but shall not be less than } (50b_w s) / f_y.$$



In midas Gen, required rebar area is calculated by “ $V_s = V_n - V_c$ ”.  
Shear rebar spacing limit and minimum shear rebar area are not applied.

# Slab Shear Checking

## Procedure

[Punching Shear Check(By CODE)]

### 3. Required Shear Strength, $V_u$

Unbalanced moment between a slab and column by flexure

$$\gamma_v = (1 - \gamma_f)$$

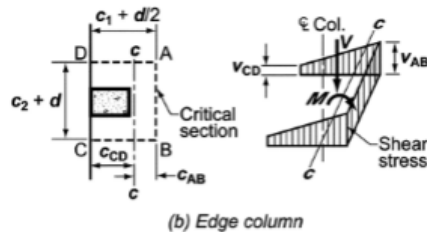
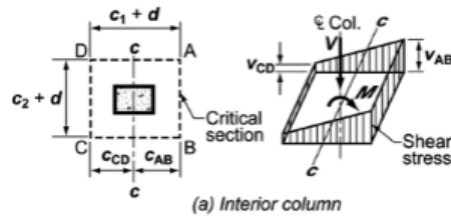
Unbalanced moment between a slab and column by eccentricity of shear

$$\gamma_f = \frac{1}{1 + (2/3)\sqrt{b_1/b_2}}$$

Factored shear stress

$$v_{f(AB)} = \frac{V_u}{A_c} + \frac{\gamma_v M_u c_{AB}}{J_c}$$

$$v_{f(CD)} = \frac{V_u}{A_c} - \frac{\gamma_v M_u c_{CD}}{J_c}$$



Assumed distribution of shear stress.

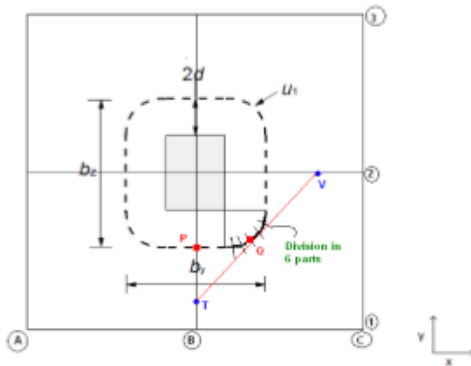
Case	Area of critical section, $A_c$	Modulus of critical section		$c$	$c'$
		$J/c$	$J/c'$		
A	$(b_1 + 2b_2)d$	$\frac{b_1 d(b_1 + 6b_2) + d^3}{6}$	$\frac{b_1 d(b_1 + 6b_2) + d^3}{6}$	$\frac{b_1}{2}$	$\frac{b_1}{2}$
B	$2(b_1 + b_2)d$	$\frac{b_1 d(b_1 + 3b_2) + d^3}{3}$	$\frac{b_1 d(b_1 + 3b_2) + d^3}{3}$	$\frac{b_1}{2}$	$\frac{b_1}{2}$
C	$(2b_1 + b_2)d$	$\frac{2b_1^2 d(b_1 + 2b_2) + d^3(2b_1 + b_2)}{6b_1}$	$\frac{2b_1^2 d(b_1 + 2b_2) + d^3(2b_1 + b_2)}{6(b_1 + b_2)}$	$\frac{b_1^2}{2b_1 + b_2}$	$\frac{b_1(b_1 + b_2)}{2b_1 + b_2}$
D	$(b_1 + b_2)d$	$\frac{b_1^2 d(b_1 + 4b_2) + d^3(b_1 + b_2)}{6b_1}$	$\frac{b_1^2 d(b_1 + 4b_2) + d^3(b_1 + b_2)}{6(b_1 + 2b_2)}$	$\frac{b_1^2}{2(b_1 + b_2)}$	$\frac{b_1(b_1 + 2b_2)}{2(b_1 + b_2)}$

# Slab Shear Checking

## Procedure

### [Punching Shear Check(By FEM)]

In these methods (The FEM Method), the Shear force along the critical section is taken and divided by the effective depth to calculate shear stress. Therefore there is no need to calculate  $\beta$  (Beta), to consider moment transferred to the column.



(There are 4 plate elements intersecting at nodes. The nodes are marked by nomenclature of Grid Lines. As the center node is denoted by B2 , B on x-Axis and 2 on Y-Axis)

When slab is defined as the plate element, the program calculated stresses only at the nodes, in the analysis. So we have the stresses at B1, B2, C2 etc. (see the figure above) are calculated by the program.

Case 1 - To calculate stresses at the critical section that is  $u_1$  in the given figure, for example we take the point P in the figure which lies in a straight line. The stress at B1 and B2 are known. The values at these nodes are interpolated linearly to find the stress at point P .

Case 2- Now if the point lies in the curve such as the point Q, then the software will divide the curve into 6 parts. At each point such as Q a tangent which intersects B1-B2 and C2-B2. The value of stresses at T and V are determined by linear interpolation of stresses which are known at for T (at B1 and B2) and for V (at C2 and B2). After knowing stresses at T and V the stress at Q is determined by linear interpolation of stresses at T and V.



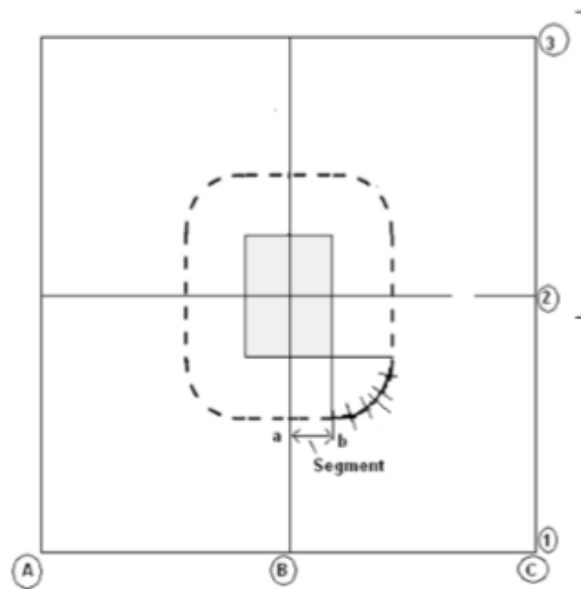
# Slab Shear Checking

## Procedure

### [Punching Shear Check(By FEM)]

#### (Method 1: Average by elements.)

In this method the stresses at all the critical points is determined. The critical points divide the critical section into segments. The average value for all these segments is determined by dividing the stresses at the two ends of the segment by 2. After determining the average value for each segment, **the maximum** average value from all of the segments is reported as the Stress value for the critical Section.



a,b are stresses at the segment ends.

Average value for the segment will be  $(a+b)/2$ , and such average value for each segment is determined.

# Slab Shear Checking

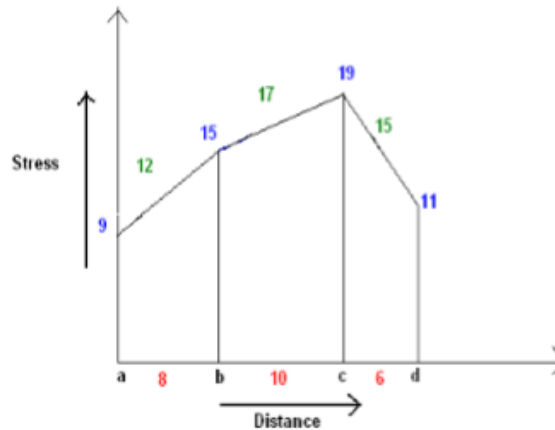
## Procedure

### [Punching Shear Check(By FEM)]

#### (Method 2: Average by Side)

In this method stresses at all critical points is determined and then average stress value is calculated by weighted mean.

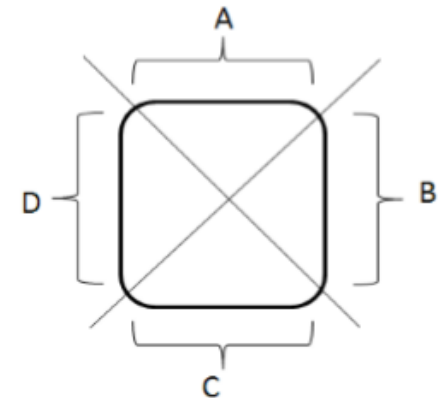
To calculate weighted mean, For example we have 4 critical points a, b, c, d.



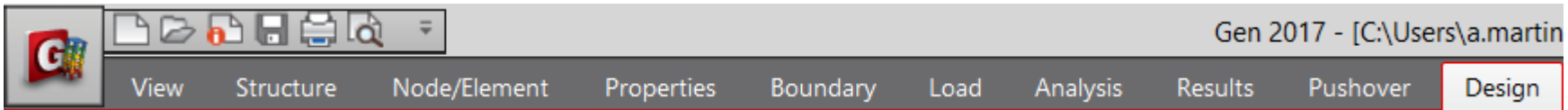
- **Stress at critical points:** For example at 'a' its 9
- **Average of the segment:** For example in 'a' and 'b' its  $(15+9)/2 = 12$
- **Distance Between the critical points:** For example between 'a' and 'b' its 8
- **Final Stress** =  $(12 * 8 + 17 * 10 + 15 * 6) / (8+10+6)$ , which is the weighted average.

We divide the Critical section into 4 sides as shown in figure.

The weighted mean value for each side is determined and then the maximum value out of the 4 sides A, B, C, D is reported as the stress value.



# Wall Design



**Design**

Wall Design

Load Combinations  
ALL COMBINATION

Element  Avg. Nodal

Element  Width 1 m

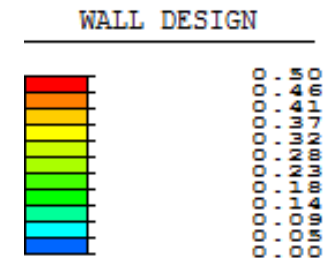
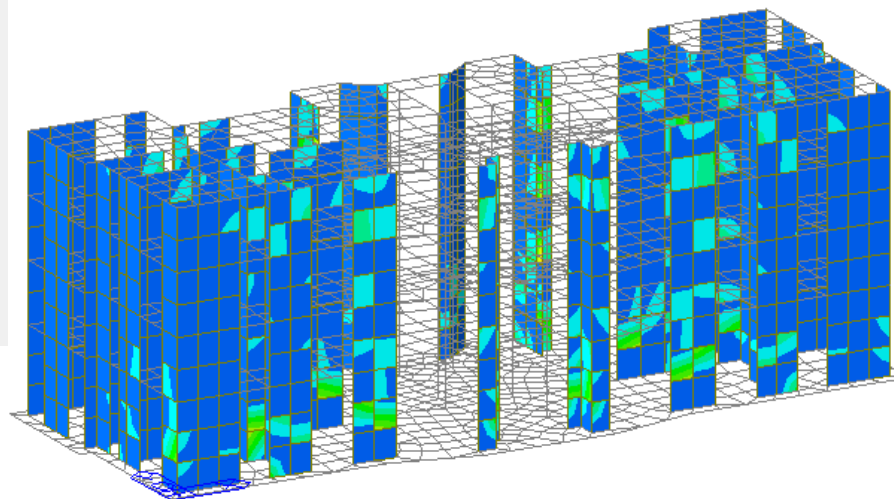
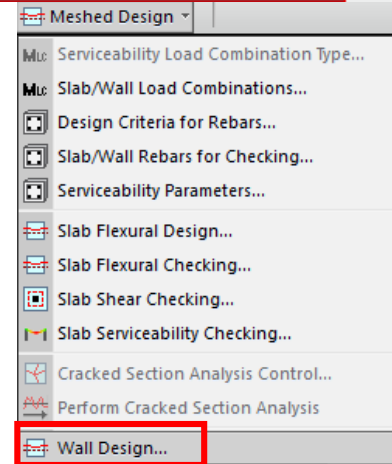
Horizontal  Vertical

Sig\_cd (concrete)

Type of Display  
 Contour  Legend  
 Values

Rebar  
 As\_req (m<sup>2</sup>/m)  
 Rho\_req  
 Resistance Ratio

**Run Design**  
 Specify Design Criteria as  
 Resistance Ratio



Smoothing:  
 Element (Element)

Component:  
 Horizontal  
 Resistance Ratio

# Wall Design

## Procedure

### Wall Design

Wall design forces and tension reinforcements are obtained in an element subject to in-plane orthogonal stress.

The tension reinforcement in an element subject to in-plane orthogonal stresses  $\sigma_{Edx}$ ,  $\sigma_{Edy}$  and  $\tau_{Edxy}$  can be calculated as shown below. Compressive stresses should be taken as positive, with  $\sigma_{Edx} > \sigma_{Edy}$ , and the direction of reinforcement should coincide with the x and y axes.

$$f_{tdx} = \rho_x f_{yd} \text{ and } f_{tdy} = \rho_y f_{yd}$$

where,  $\rho_x$  and  $\rho_y$  are the geometric reinforcement ratios, along the x and y axes respectively.

In locations where  $\sigma_{Edy}$  is tensile or  $\sigma_{Edx} \cdot \sigma_{Edy} \leq \tau_{Edxy}^2$ , reinforcement is required. The optimum reinforcement, indicated by superscript ', and related concrete stress are determined by:

FOR  $\sigma_{Edx} \leq |\tau_{Edxy}|$

$$f'_{tdx} = |\tau_{Edxy}| - \sigma_{Edx}$$

$$f'_{tdy} = |\tau_{Edxy}| - \sigma_{Edy}$$

$$\sigma_{cd} = 2|\tau_{Edxy}|$$

FOR  $\sigma_{Edx} > |\tau_{Edxy}|$

$$f'_{tdx} = 0$$

$$f'_{tdy} = \frac{\tau_{Edxy}^2}{\sigma_{Edx}} - \sigma_{Edy}$$

$$\sigma_{cd} = \sigma_{Edx} \left( 1 + \left( \frac{\tau_{Edxy}}{\sigma_{Edx}} \right)^2 \right)$$

Wall design using wall element is also supported in midas Gen.

# Wall Design

## Procedure

### Wall Design

Minimum reinforcement for vertical and horizontal rebar is considered in accordance to ACI318-11, 14.3.2 and 14.3.3. Maximum ratio of vertical reinforcement are applied as “0.04” and it can be modified in Design > Concrete Design Parameter > Limiting Maximum rebar Ratio.

#### [Minimum ratio of vertical reinforcement area]

**14.3.2** — Minimum ratio of vertical reinforcement area to gross concrete area,  $\rho_t$ , shall be:

- (a) 0.0012 for deformed bars not larger than No. 5 with  $f_y$  not less than 60,000 psi; or
- (b) 0.0015 for other deformed bars; or
- (c) 0.0012 for welded wire reinforcement not larger than W31 or D31.

#### [Minimum ratio of horizontal reinforcement area]

**14.3.3** — Minimum ratio of horizontal reinforcement area to gross concrete area,  $\rho_t$ , shall be:

- (a) 0.0020 for deformed bars not larger than No. 5 with  $f_y$  not less than 60,000 psi; or
- (b) 0.0025 for other deformed bars; or
- (c) 0.0020 for welded wire reinforcement not larger than W31 or D31.

#### [Maximum ratio of vertical reinforcement area]

0.04

Limiting Maximum Rebar Ratio

Design Code : ACI318-11

Rebar Ratio

Shear Wall Design (Rhow)	: 0,04
Column Design (Rhoc)	: 0,03
Brace Design (Rhor)	: 0,03

OK Cancel



End

