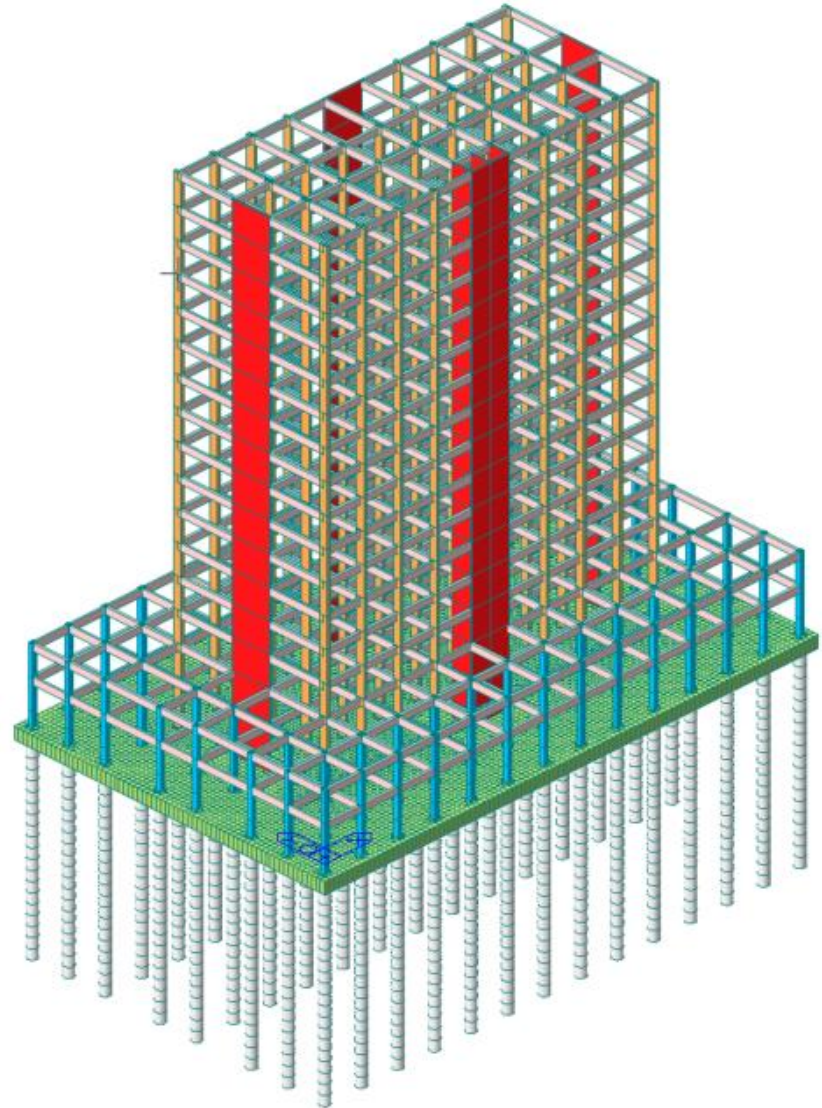


Overview

1. Graphical User Interface
2. Project Information
3. Foundation Overview
4. Midas Gen Model
 - a. MGT Command Shell
5. Analysis & Results
6. Isolated Footing Design using Design+
 1. Pile Cap Design
7. Mat and Pile Modelling
8. Analysis & Results
9. Pile Analysis
 - a. P-y Curve
10. Pile Design

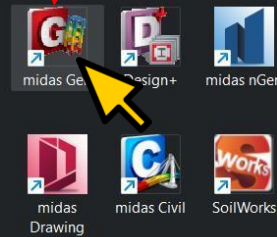


- Folder Dbase
- Folder DgnLanguage
- Folder DgnPlugIn
- Folder Gen Manual
- Folder Gen Verification
- Folder Html
- Folder Language
- Folder Manual
- Folder midas GSD
- Folder midas link for Revit Structure
- Folder midas link for Tekla Structure
- Folder midasCAD
- Folder Protection drivers
- Folder Resources
- Folder SeisDevCtrl
- Folder Tutorial

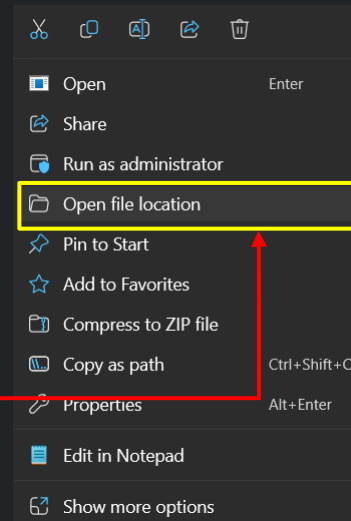
3. *midas Gen Program File folder* will open.

You can check folders like **Manual** and **Tutorial** for practice and others.

1. Right-Click *midas Gen* icon.



2. Click *Open file location*.

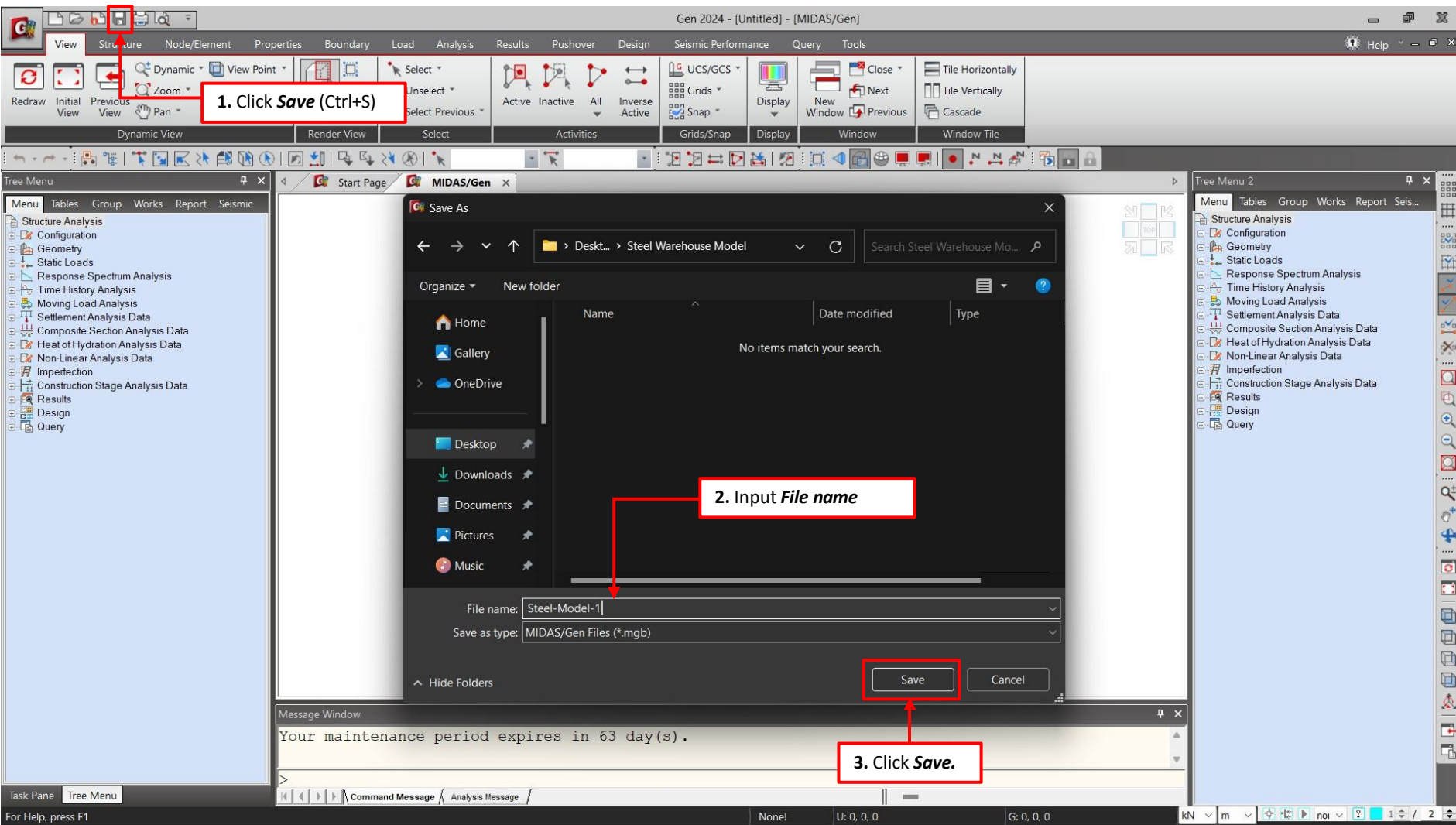


The screenshot displays the MIDAS Gen 2024 software interface. The top menu bar includes options like File, Edit, Structure, Node/Element, Properties, Boundary, Load, Analysis, Results, Pushover, Design, Query, and Tools. The 'File' menu is open, showing options like New, Open, Save, and Print. A red box highlights the 'New' button in the top-left corner of the software window. Below the menu bar, there are several toolbars for various functions. The main workspace shows a 'Welcome to MIDAS' message with links to 'Seismic Performance Evaluation', 'Wind Load as per ASCE 7-16 for the Warehouse with Gable Hip Roof', and 'Go to MIDAS Customer Online Support'. Below this, there is a 'Recent' section with a grid of project thumbnails. The first thumbnail is labeled 'New Project' and has a red box around it with an arrow pointing to it. A red box also highlights the 'New Project' button in the 'Recent' section. The bottom of the interface shows a 'Message Window' and a 'Task Pane'.

1. Click **New** (Ctrl+N) OR

Click **New Project**





Gen 2024 - [Untitled] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

1. Click **Save** (Ctrl+S)

2. Input **File name**

3. Click **Save**.

Save As

< > > Desk... > Steel Warehouse Model

Organize New folder

Home Gallery OneDrive Desktop Downloads Documents Pictures Music

No items match your search.

File name: Steel-Model-1

Save as type: MIDAS/Gen Files (*.mgb)

Hide Folders

Save Cancel

Message Window

Your maintenance period expires in 63 day(s).

Task Pane Tree Menu

For Help, press F1

None! U: 0, 0, 0 G: 0, 0, 0 kN m 1 2

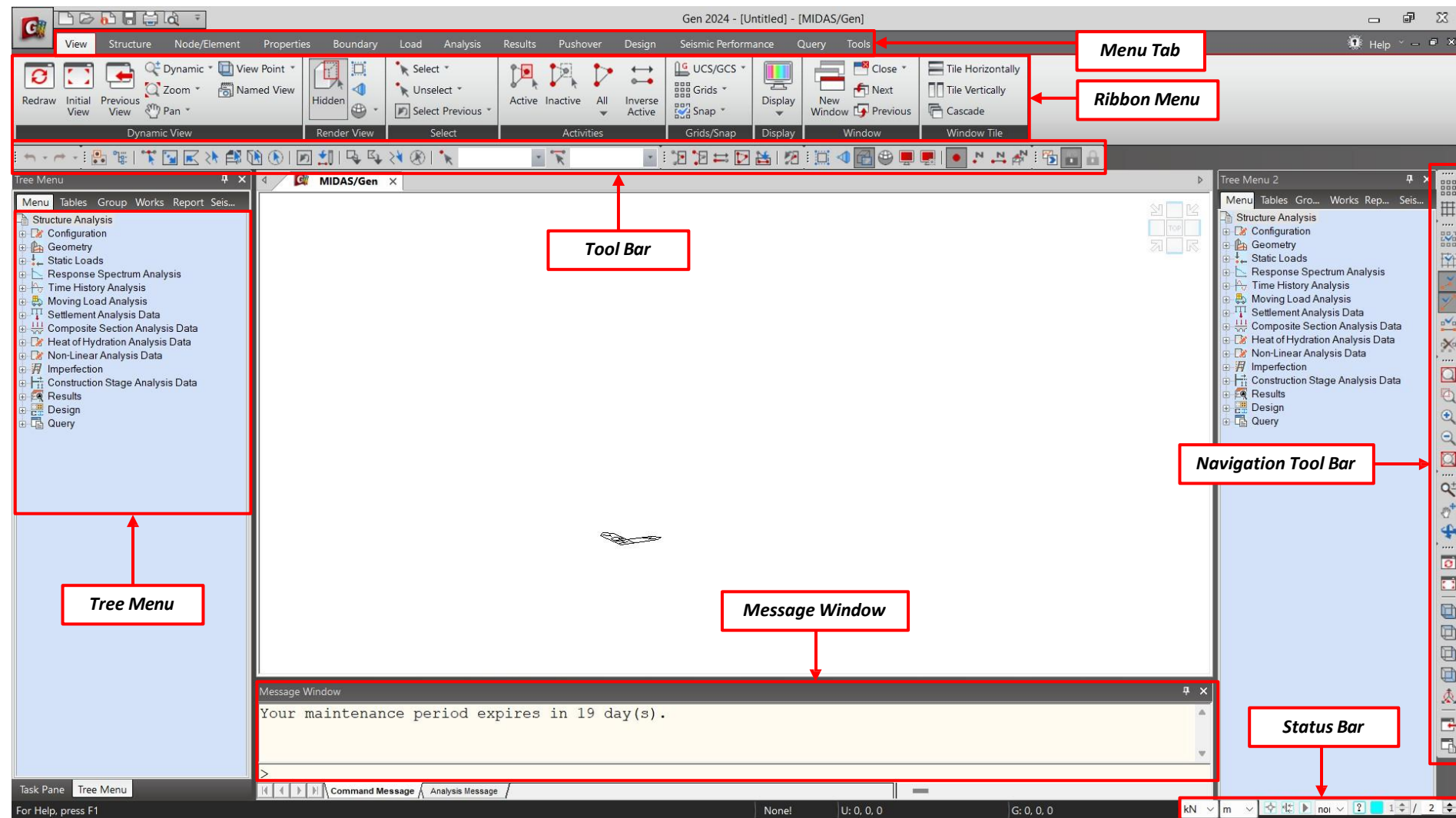
NOTES

Always save in a folder so temporary files will be contained.



GRAPHICAL USER INTERFACE (GUI)

5



GRAPHICAL USER INTERFACE (GUI)

6

The screenshot displays the MIDAS/Gen software interface. The top menu bar includes View, Structure, Node/Element, Properties, Boundary, Load, Analysis, Results, Pushover, Design, Seismic Performance, Query, and Tools. Below the menu bar is a toolbar with icons for various functions. Red boxes and arrows highlight specific areas:

- Undo/Redo:** Located in the top-left toolbar.
- Selection Commands:** Located in the top toolbar, below the Undo/Redo icons.
- Node/Element Selection by List:** Located in the top toolbar, below the Selection Commands icons.
- Activation Commands:** Located in the top toolbar, below the Node/Element Selection by List icons.
- Display:** Located in the top toolbar, below the Activation Commands icons.
- Nodes/Elements Activation View:** Located in the top toolbar, below the Display icons.

On the right side, the Tree Menu 2 is visible, showing a hierarchical list of analysis and design options. A red box and arrow point to the bottom right corner of the interface, highlighting the following options:

- Units
- Selection Filter
- Snap Division

The bottom status bar shows the current unit as kN, and the bottom right corner displays the command 'nor' and the snap division '1/2'.



The screenshot shows the MIDAS/Gen software interface. The 'Tools' menu is highlighted in the top bar. The 'Preferences' dialog box is open, and the 'Design Code' tab is selected. The 'Design Code' section shows the following settings:

	Steel	Concrete
Design Code	IS:800-2007	IS456:2000
Rebar Material	IS(RC)	IS(RC)

Annotations in the image include:

- 1. Go to **Tools**
- 2. Click **Preferences**
- 3. Click **Design/Load Code**
- 4. Set the following parameters under **Design Code**
- 5. Go to **Design Code**

NOTES

Under the **Preferences**, you can set the desired codes as the default ones.



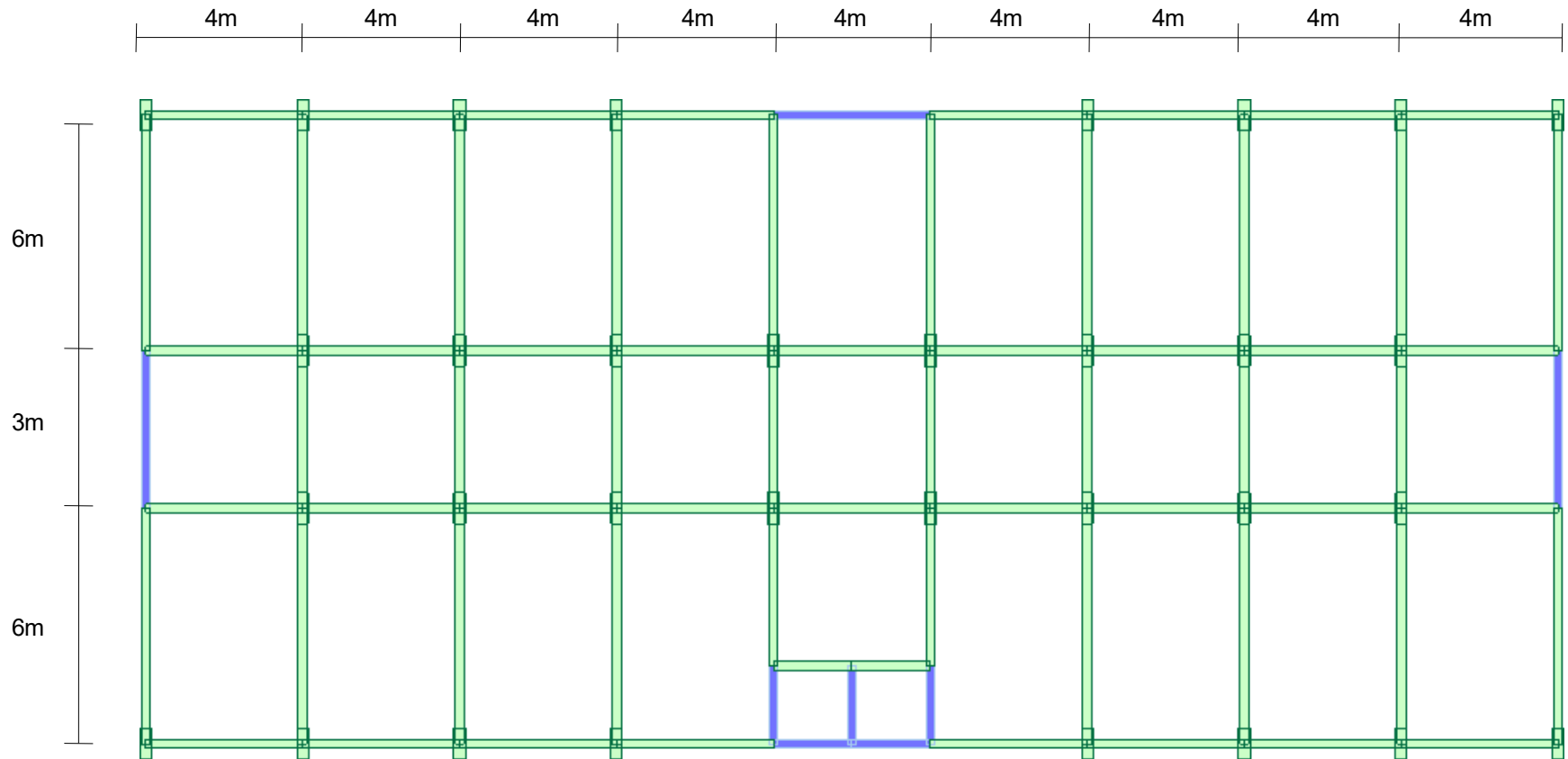
1. Set the following parameters under **Load Code**

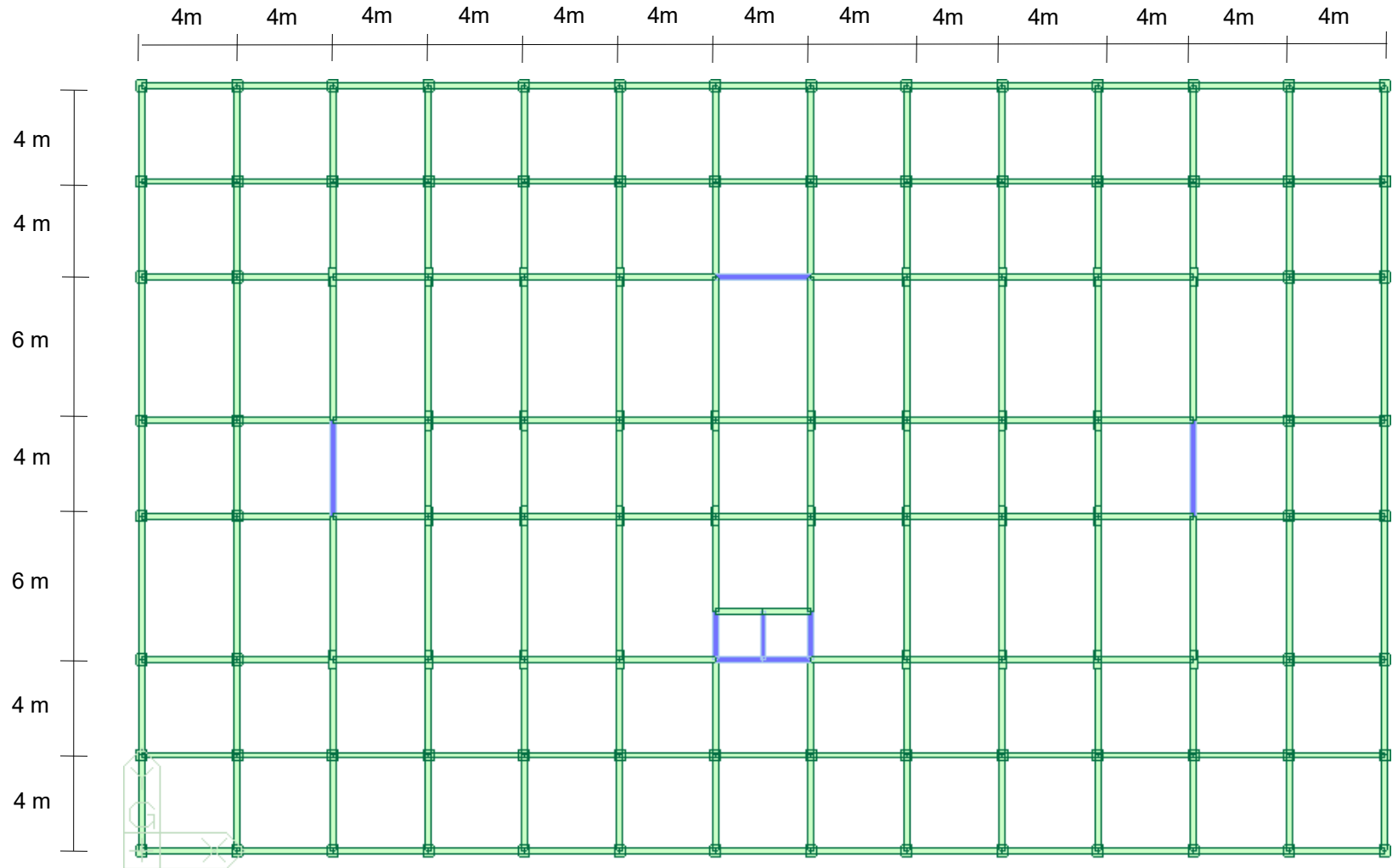
	Design Code
Wind Load	IS875(2015)
Static Seismic Load	IS1896(2016)
Response Spectrum	IS1896(2016)

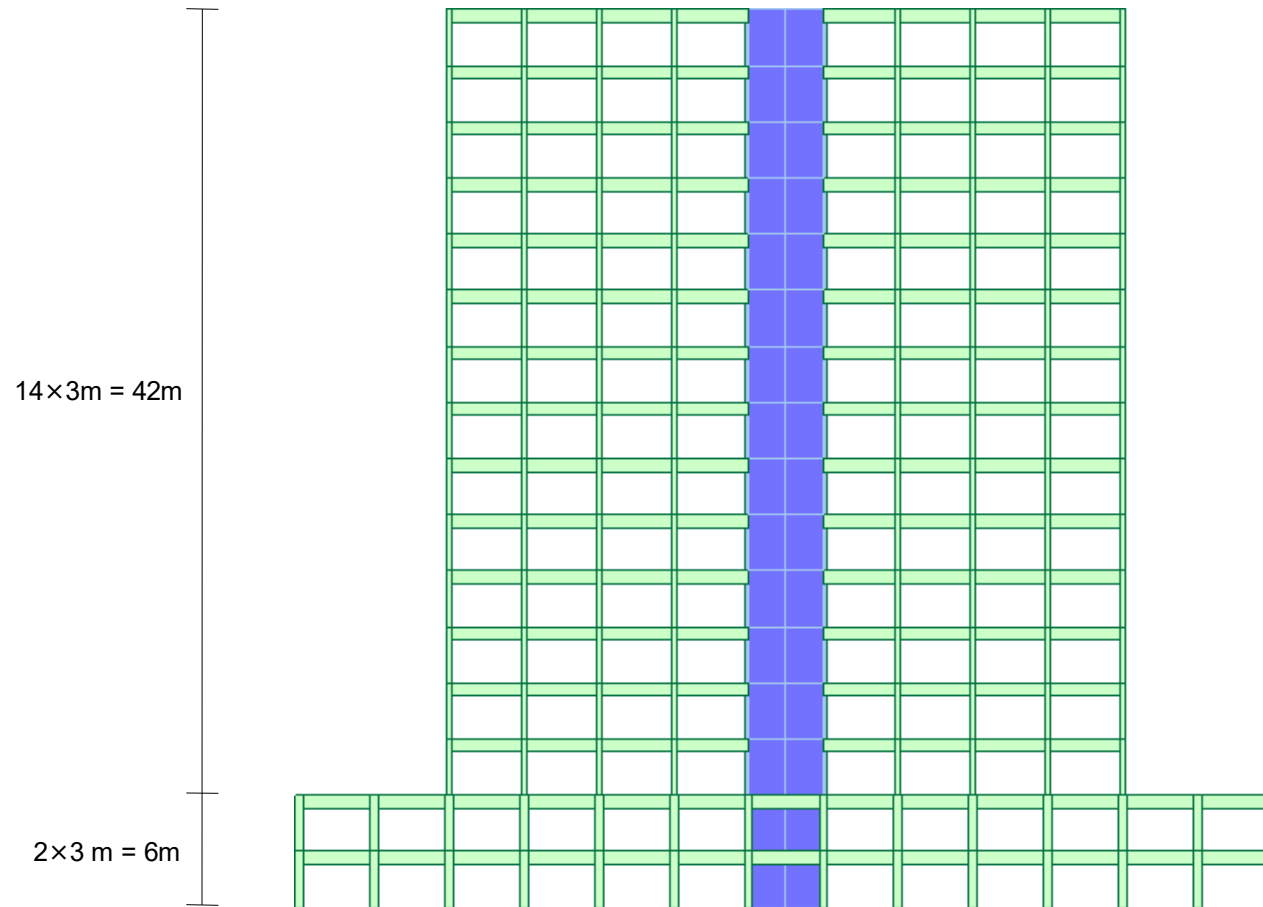
2. Check **Save Changes Upon OK**

3. Click **OK**









1. Design Codes

IS456:2000, IS1893:2016, IS875(2015),
IS13920:2016

2. Material Properties

1. Concrete Compressive Strength ($f_{c'}$)

1. Columns/Shear Walls
2. Beams

$= 40000 \text{ kN/m}^2$
 $= 30000 \text{ kN/m}^2$

2. Reinforcing Bar Yield Strength (f_y)

1. Main Reinforcing Bars
2. Ties/Stirrups

$= 415 \times 10^3 \text{ kN/m}^2$
 $= 415 \times 10^3 \text{ kN/m}^2$

3. Design Loads to be considered

Load	Description	Intensity
Dead Load	Self Weight	
Typical Floor Load	Super imposed Dead Load Live load	4 kN/m ² 2 kN/m ²
Podium Floor	Super imposed Dead Load Live load	4 kN/m ² 4 kN/m ²
Roof Floor	Super imposed Dead Load Live load	5.5 kN/m ² 1.5 kN/m ²
Wind Load	X, Y direction	IS 875 Part3
Earthquake Loads	X, Y direction	IS 1893-2016

NOTES



3.4 Wind Profile IS875(2015)

Basic Wind Speed, V	= 47 m/sec
Terrain Category	= III
Building Class	= A
Wind Directional Factor, K_d	= 1.0
Frictional Drag Coef. (C_f)	= 0.9
Combinational Factor, k_c	= 1.0

3.5 Seismic Profile (IS1893 : 2016)

Importance Factor, I	= 1.2
Soil Type Seismic	= II (Medium Soil)
Zone	= Zone III (0.16)
Seismic Zone Factor, z	= 0.16
Response Reduction Factor, R	= 5.0

NOTES



1. Material Property

IS (RC) M30

IS (RC) M40

2. Section Properties

No.	Name	Dimension (B x H) (mm)
1	Podium Column	450 x 450
2	Building Column	300 x 800
3	Beams	230 x 700
4	Walls	230 thick



The screenshot displays the Midas Gen 2024 interface. The 'File' menu is open, and the 'Import' option is highlighted. A red box labeled '1. Click Import' points to this menu item. The 'Import' submenu is also visible, with 'midas Gen MGT File...' highlighted. A red box labeled '2. Midas Gen MGT File...' points to this option. An 'Open' dialog box is open, showing the file '4 Storey Building' selected. A red box labeled '3. Click MGT File then Open' points to the 'Open' button in the dialog box. The background shows the main software interface with various toolbars and a tree menu on the right.

Gen 2024 - [C:\Users\MIDAS\Documents\1. WORK FILES\BUSINESS TRIPS\SEMINAR & WORKSHOPS\MIDAS GEN - FOUNDATION (INHOUSE TRAINING)\Foundation Analysis and Design]

Analysis Results Pushover Design Seismic Performance Query Tools

Active Inactive All Inverse Active

Grids Snap Display Window New Window Previous Next Close

Tile Horizontally Tile Vertically Cascade Window Tile

Tree Menu 2

Menu Tables Group Works Report Seismic

Structure Analysis

Configuration

Geometry

Static Loads

Response Spectrum Analysis

Time History Analysis

Moving Load Analysis

Settlement Analysis Data

Composite Section Analysis Data

Heat of Hydration Analysis Data

Non-Linear Analysis Data

Imperfection

Construction Stage Analysis Data

Results

Design

Query

Open

Organize New folder

Name Date modified Type

4 Storey Building 04/11/2024 9:35 am MIDAS/G

File name: 4 Storey Building MGT Files (*.mgt)

Open Cancel

Message Window

Your maintenance period expires in 50 day(s).

Command Message Analysis Message

For Help, press F1

None! U: 0, 0, 0 G: 0, 0, 0 kN m 1 2



The screenshot displays the Midas Gen software interface. The main window shows a 3D model of a building frame structure. The left sidebar contains a tree menu with various analysis and design options. The right sidebar contains a tree menu with various analysis and design options. The bottom status bar shows the current frame and unit information.

Tree Menu 1:

- Menu
- Tables
- Group
- Works
- Rep...
- Seis...
- Works
 - Analysis Control Data
 - Eigenvalue Analysis [Type=Eigenvalues]
 - Structures
 - Stories : 17
 - Nodes : 964
 - Elements : 2064
 - Properties
 - Material : 2
 - 1 : M30
 - 2 : M40
 - Section : 3
 - 1 : Podium Column
 - 2 : Building Column
 - 3 : Beam
 - Thickness : 1
 - Boundaries
 - Supports : 116
 - Masses
 - Loads to Masses : 3
 - Static Loads
 - Static Load Case 1 [DL ; Dead Load]
 - Static Load Case 2 [LL ; Live Load]
 - Static Load Case 3 [SIDL ; Super Imposed]
 - Static Load Case 4 [WX ; Wind Load in X]
 - Static Load Case 5 [WY ; Wind Load in Y]
 - Static Load Case 6 [EXP ; Earthquake Lo
 - Static Load Case 7 [EXN ; Earthquake Lo
 - Static Load Case 8 [EYP ; Earthquake Lo
 - Static Load Case 9 [EYN ; Earthquake Lo
 - Response Spectrum Analysis
 - Response Spectrum Functions : 1
 - Response Spectrum Load Cases : 2

Tree Menu 2:

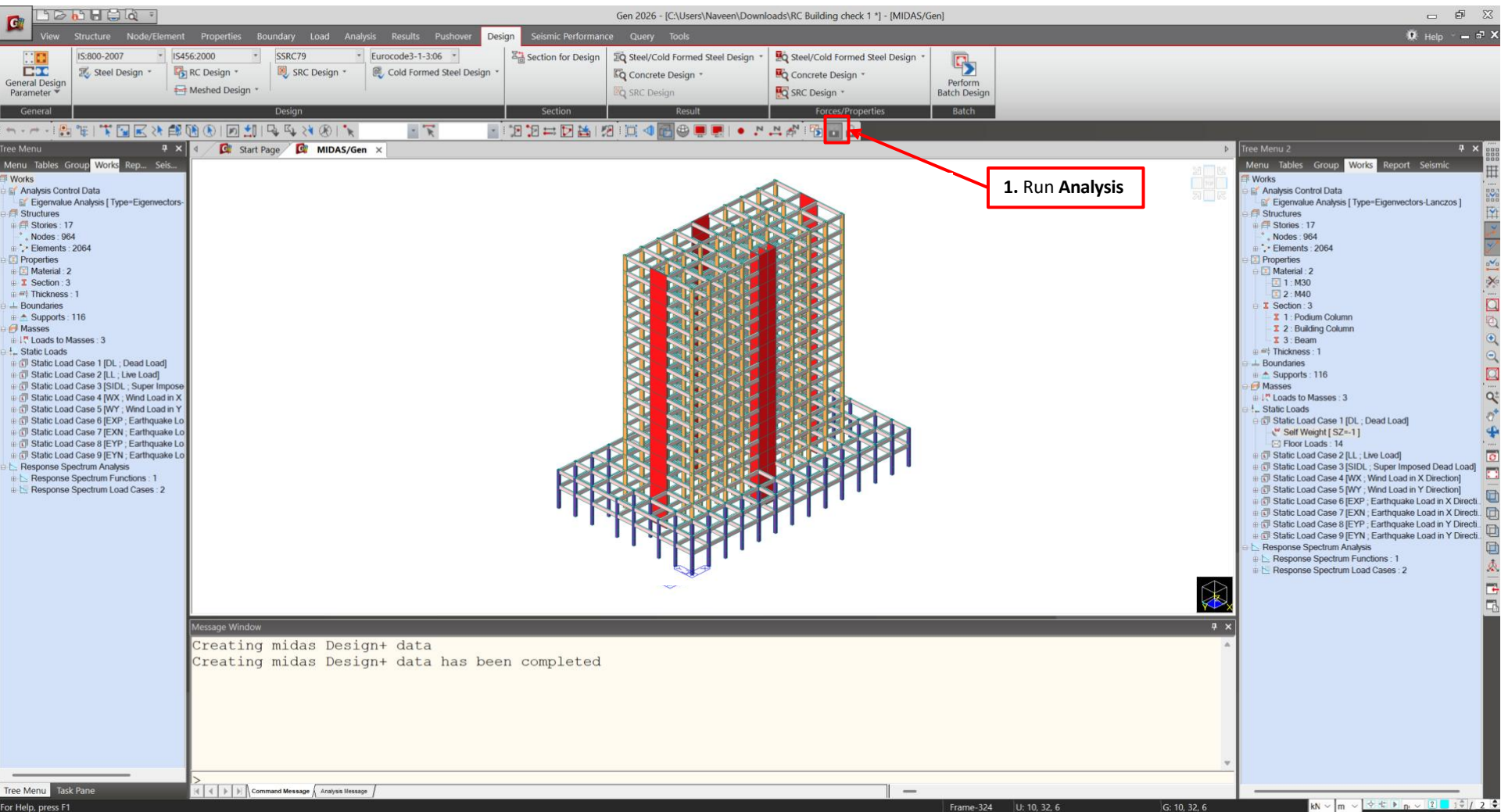
- Menu
- Tables
- Group
- Works
- Report
- Seismic
- Works
 - Analysis Control Data
 - Eigenvalue Analysis [Type=Eigenvalues-Lanczos]
 - Structures
 - Stories : 17
 - Nodes : 964
 - Elements : 2064
 - Properties
 - Material : 2
 - 1 : M30
 - 2 : M40
 - Section : 3
 - 1 : Podium Column
 - 2 : Building Column
 - 3 : Beam
 - Thickness : 1
 - Boundaries
 - Supports : 116
 - Masses
 - Loads to Masses : 3
 - Static Loads
 - Static Load Case 1 [DL ; Dead Load]
 - Self Weight [SZ=1]
 - Static Load Case 2 [LL ; Live Load]
 - Static Load Case 3 [SIDL ; Super Imposed Dead Load]
 - Static Load Case 4 [WX ; Wind Load in X Direction]
 - Static Load Case 5 [WY ; Wind Load in Y Direction]
 - Static Load Case 6 [EXP ; Earthquake Load in X Direct
 - Static Load Case 7 [EXN ; Earthquake Load in X Direct
 - Static Load Case 8 [EYP ; Earthquake Load in Y Direct
 - Static Load Case 9 [EYN ; Earthquake Load in Y Direct
 - Response Spectrum Analysis
 - Response Spectrum Functions : 1
 - Response Spectrum Load Cases : 2

Message Window:

```
Creating midas Design+ data
Creating midas Design+ data has been completed
```

Frame: 324 U: 10, 32, 6 G: 10, 32, 6 kN m





The screenshot displays the MIDAS/Gen software interface. The main window shows a 3D model of a building frame structure. A red arrow points to the 'Run Analysis' button (represented by a play icon) in the top toolbar. A red callout box with the text '1. Run Analysis' is positioned next to the button. The software title bar indicates the file path: 'Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]'. The left sidebar shows the 'Tree Menu' with various analysis and design options. The right sidebar shows the 'Tree Menu 2' with a detailed list of analysis parameters. The bottom status bar shows 'Frame-324 U: 10, 32, 6 G: 10, 32, 6'.

1. Run Analysis

Creating midas Design+ data
Creating midas Design+ data has been completed



1. Click Results

2. Click Reactions then select Reaction Forces/Moments

The screenshot displays the MIDAS/Gen software interface. The top menu bar includes View, Structure, Node/Element, Properties, Boundary, Load, Analysis, Results, Pushover, Design, Seismic Performance, Query, and Tools. The Results menu is open, showing options like Reactions, Stresses, Strain, Beam/Element, Local Direction, Displacement Participation Factor, Mode Shapes, Modal Damping Ratio, Nodal Results of RS, Mode shape, Influ. Lines, Influ. Surfaces, Moving Load, T.H. Results, T.H. Graph/Text, Stage/Step Graph, Time History, Column Shortening Graph for CS, Story Shear Force Ratio, Misc., Unknown Load Factor, Tendon Loss Graph, Text Output, Results Tables, and Tables. The Reactions option is highlighted, and its sub-menu is open, showing Reaction Forces/Moments, Search Reaction Forces/Moments, and Soil Pressure. The Reaction Forces/Moments option is selected. The main window shows a 3D model of a building structure. The left Tree Menu shows the Analysis Control Data, Eigenvalue Analysis [Type=Eigenvalues-Lanczos], Structures, Stories, Nodes, Elements, Properties, Material, Section, Thickness, Boundaries, Supports, Masses, Loads to Masses, Static Loads, and Response Spectrum Analysis. The right Tree Menu 2 shows the same structure, but with the Results option selected. The Message Window at the bottom displays the text: "Creating midas Design+ data" and "Creating midas Design+ data has been completed".

Message Window

Creating midas Design+ data
Creating midas Design+ data has been completed



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

Load Combination Reactions Stresses Strain Beam/Element Local Direction Displacement Participation Factor Mode Shapes Influ. Lines T.H. Results Column Shortening Graph for C.S. Story Shear Force Ratio Unknown Load Factor Tendon Loss Graph Text Output Results Tables

Tree Menu 1: Reaction Forces/Moments, Load Cases/Combinations (CBC: d.CB38), Components (FX, FY, FZ, FXYZ, MX, MY, MZ, MXYZ, Mb, Local (if defined)), Type of Display (Values, Legend), Arrow Scale Factor: 1.000000, Apply, Close.

Tree Menu 2: Works, Analysis Control Data, Eigenvalue Analysis [Type=Eigenvalues-Lanczos], Structures, Stories: 17, Nodes: 964, Elements: 2064, Properties, Material: 2 (M30, M40), Section: 3 (Podium Column, Building Column, Beam), Thickness: 1, Boundaries, Supports: 116, Masses, Loads to Masses: 3, Static Loads, Static Load Case 1 [DL; Dead Load], Self Weight [SZ=1], Floor Loads: 14, Static Load Case 2 [LL; Live Load], Static Load Case 3 [SIDL; Super Imposed Dead Load], Static Load Case 4 [WX; Wind Load in X Direction], Static Load Case 5 [WY; Wind Load in Y Direction], Static Load Case 6 [EXP; Earthquake Load in X Direct.], Static Load Case 7 [EXN; Earthquake Load in X Direct.], Static Load Case 8 [EYP; Earthquake Load in Y Direct.], Static Load Case 9 [EYN; Earthquake Load in Y Direct.], Response Spectrum Analysis, Response Spectrum Functions: 1, Response Spectrum Load Cases: 2.

Legend Details: Legend Position (Left, Right), Rank Value Type (Exponential, Fixed), Decimal Point (2), Apply upon OK, OK, Cancel.

Message Window: Creating midas Design+ data. Creating midas Design+ data has been completed. The project will be saved by the auto-save feature.

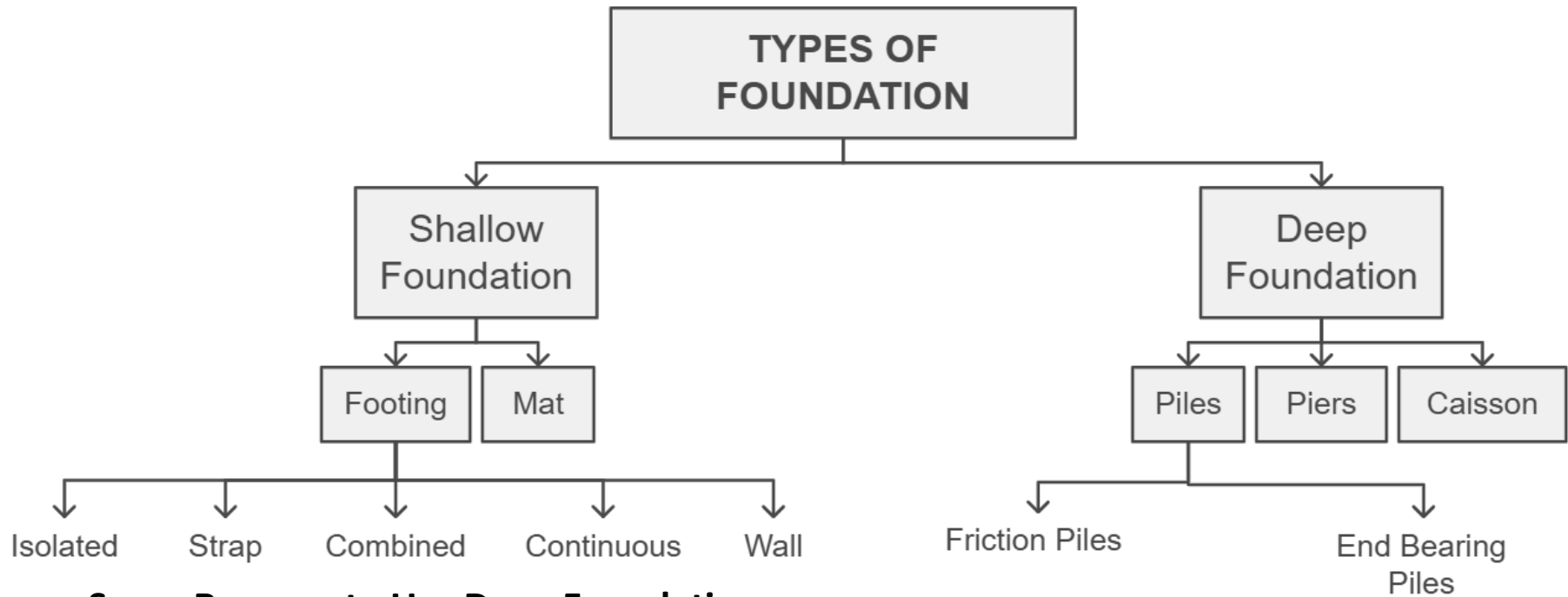
Frame: 1981 U: 44, 24, 48 G: 44, 24, 48

NOTES

Red Arrows signify the Max. Reactions

Clicking the button  beside **Type of Display** functions will open additional Display Settings dialog box

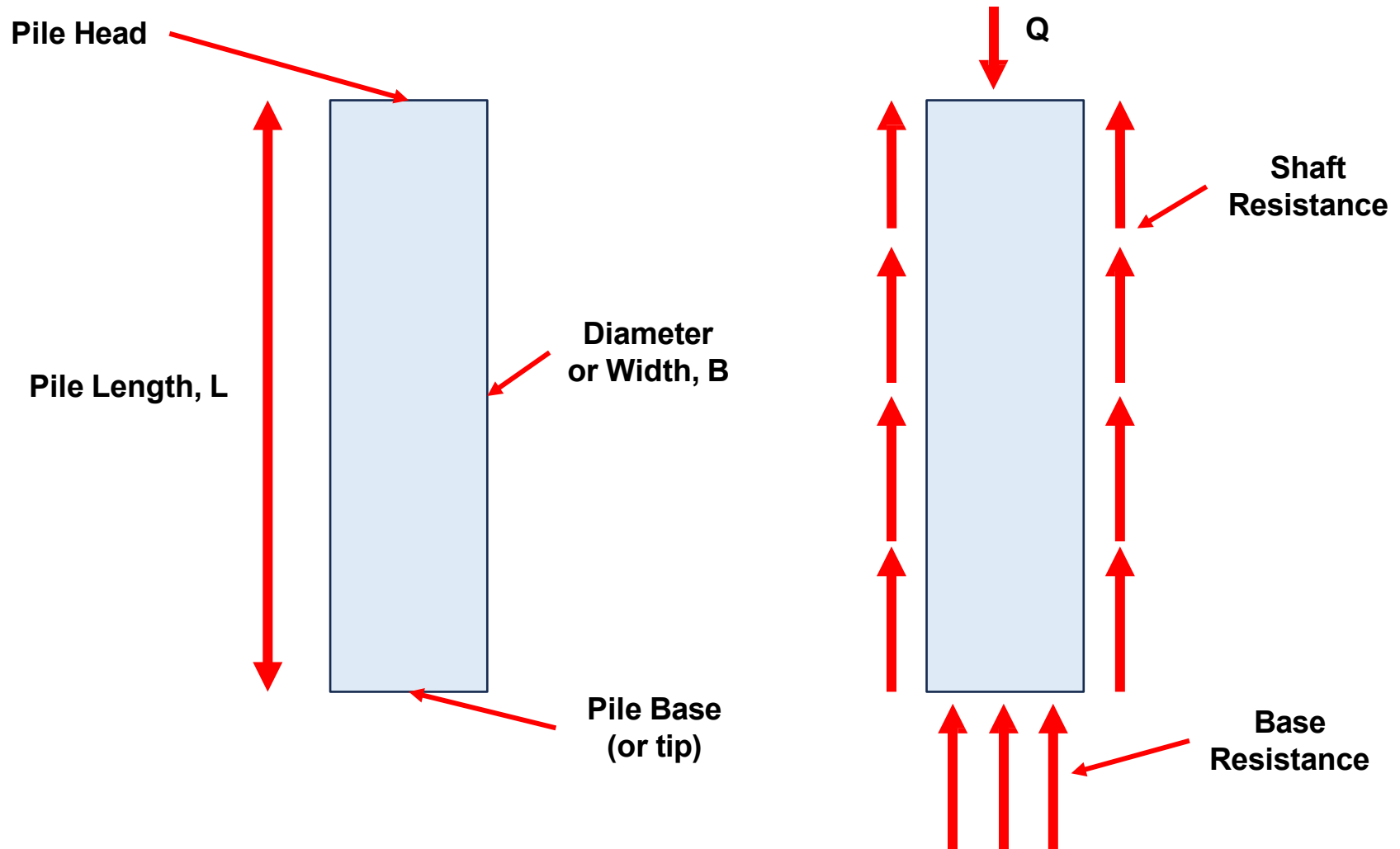




Some Reasons to Use Deep Foundations

1. Upper layer of Soil is weak while structural loads are high
2. Upper layer of Soil are subjected to scour and undermining
3. Foundation must penetrate through water
4. Need large uplift capacity
5. Need large lateral load capacity





The screenshot displays the MIDAS Design+ 2024 software interface. The title bar reads "midas Design+ 2024 - [Start Page]". The menu bar includes: Mode/Link, RC, Steel, SRC, Aluminum, Reinforce, Load, Option, Tool, View, and Help. The toolbar contains icons for Project Mode, Simple Mode, Check Mode, midas Link, Link Option, Member, Member Drawing, Quantity List, and Edit Mode. The WorkBar on the left shows the "Add New Member" section with dropdowns for System, Type, and Name, and a checkbox for "Keep Sect. & Bar Data". Below this is a tree view for "RC Design Procedure" with sub-items like Option, Design Code, Live Load, Rebar DB, Design Option, Drawing Option, Report Option, and Preference. The main window is titled "Start Page" and features a "Welcome to MIDAS" message. A "Recent" section lists "New Project" with a red box and an arrow pointing to it. A red box with text "1. Click **New Project** (Ctrl+N)" points to the "New Project" button. Another red box with text "2. Immediately save this file to a folder." is positioned below it. The right side of the interface shows a "Report" section with a 100% zoom level and buttons for Print, Save, Report, Option, and Summary Report. The bottom status bar shows "Done" and a "User" dropdown.

midas Design+ 2024 - [Start Page]

Mode/Link RC Steel SRC Aluminum Reinforce Load Option Tool View Help

Project Mode Simple Mode Check Mode midas Link Link Option Member Member Drawing Quantity List Edit Mode

WorkBar

Add New Member

System Type Name

... Add New Member

☐ Keep Sect. & Bar Data

RC | Steel | SRC | Aluminum | Reinforce |

RC Design Procedure

Option

Design Code : Live Load : Rebar DB : Design Option Drawing Option Report Option Preference

Slab Beam Column Column (General) Shear Wall Footing Basement Wall Buttrass Stair Corbel / Bracket Retaining Wall Anchor Bolt Beam Table Slab Table Batch Beam Batch Column Batch Wall

Start Page

Welcome to MIDAS

Contact us | MIDAS Account

Recent

New Project

1. Click **New Project** (Ctrl+N)

2. Immediately save this file to a folder.

Report

100% Print... Save... Report... Option... Summary Report

Message

Done

User



The screenshot shows the MIDAS Design+ 2025 software interface. The main window displays a "Server Maintenance Notice" dialog. Overlaid on this is the "RC Design Procedure" dialog box. The "RC" tab is selected in the left-hand "WorkBar". The "RC Design Procedure" dialog box has several sections: "Option" (with "Design Code : IS456:2000", "Live Load : IS875-1987", and "Rebar DB : IS" selected), "Design Option", "Drawing Option", "Report Option", and "Preference". The "Rebar Option" sub-dialog is open, showing "Rebar Code" set to "IS", "Rebar Option for Design" with "RC-1", "RC-2", and "SRC" tabs, and "Spacing List" with a table of values. The "Apply" button is highlighted in the "Rebar Option" dialog.

1. Go to RC tab

2. Design Code: IS456:2000
3. Double-click REBAR DB:

4. Set Rebar Code to IS

5. Adjust limiting rebar sizes as necessary

6. Click Apply

NOTES

When designing through Design+, it is recommended to use the same design code as used in Gen.



midas Design+ 2025 - [Untitled *] - [Start Page]

Mode/Link RC Steel SRC Aluminum Reinforce Load Option Tool View Help

Project Mode Simple Mode Check Mode midas Link Link Option Member Member Drawing Quantity List Edit Mode

WorkBar

Add New Member

System RC

Type Footing

Node

Import from Gen

Keep Sect. & Bar Data

RC | Steel | SRC | Aluminum | Reinforce

RC Design Procedure

Option

Design Code : IS456:2000

Live Load : IS875-1987

Rebar DB : IS

Design Option

Drawing Option

Report Option

Preference

Slab

Beam

Column

Column (General)

Shear Wall

Footing (1)

F01

Footing (Combined)

Basement Wall

Buttress

Stair

Corbel / Bracket

Retaining Wall

Anchor Bolt

Report Option

Global Data | Member Data | Number Format

Calculation Result Summary

☒ Include Visual Items

☒ Include Visual Items in Printed

☒ Print Result Table

☐ Print Bar Cluster

☐ Simplified Column Header

Detail Report

☒ Include Visual Items

☒ Include Visual Items in Printed

☒ Print Result Table

☒ Print Bar Cluster

☐ Simplified Column Header

Summary Report

☒ Include Visual Items

☒ Include Visual Items in Printed

☐ Print Result Table

☒ Print Bar Cluster

☐ Simplified Column Header

☐ Apply Option by Member

Unit System

☐ Report in Design Code Unit

☐ User Unit System

SI Unit

Default

OK

Cancel

1. Double-click **Report Option**

2. Uncheck **Report in Design Code Unit** and **User Unit System** and select **SI Unit**

3. Click **OK**

temporarily unavailable.

We appreciate your understanding as we work to improve your experience.

Message

Done

User US Unit English



1. Click *midas Link*

2. Select the specific *Gen* file you want to connect

3. Click *Connect*

4. Verify *Message Window* if successfully linked.

Server Maintenance Notice

To improve system stability and performance,
we will be performing scheduled server maintenance.

Link with : ☒ midas Gen
Gen 2025 - [F:\MIDAS\Anmol Sir Work\STEP FILES-20250702T100454Z-1]

Connect Disconnect Cancel

Message
Successfully linked with Gen 2025 - [F:\MIDAS\Anmol Sir Work\STEP FILES-20250702T100454Z-1-001\STEP FILES\Normal Building IS code] - [MIDAS/Gen].

NOTES

Gen file should be open before doing the midas Link in Design+



The screenshot displays the MIDAS/Gen software interface. The main window shows a 3D grid model of a building structure. Four red boxes with arrows point to specific UI elements, indicating the steps to select a node:

- 1. Click View**: Points to the 'View' button in the top toolbar.
- 2. Click Initial View**: Points to the 'Initial View' button in the 'Dynamic View' dropdown menu.
- 3. Select Single (Ctrl + Shift + S)**: Points to the 'Select Single' button in the 'Dynamic View' dropdown menu.
- 4. Select Node 999**: Points to a specific node in the 3D grid model.

The interface includes a 'Tree Menu' on the left and right sides, a 'Message Window' at the bottom, and a 'Command Message' bar at the bottom right. The 'Tree Menu' on the left lists various analysis and design options, while the 'Tree Menu' on the right lists the model's components and properties. The 'Message Window' shows the text 'Creating midas Design+ data'.

NOTES



The screenshot shows the MIDAS Design+ 2025 software interface. The 'Link Option' dialog box is open, and several steps are highlighted with red boxes and arrows:

- 1. Click RC Footing**: Points to the 'RC' option in the 'Add New Member' list.
- 2. Click (...)**: Points to the 'Import from Gen' button in the 'RC Design Procedure' list.
- 3. Check Link by Section**: Points to the 'Link by Section' radio button in the 'Link Type by Member' section.
- 4. Select your preferred importing of loads**: Points to the 'Load Combination Type' dropdown menu.
- 5. Select Reaction at the Support**: Points to the 'Force Type' dropdown menu.
- 6. Click OK**: Points to the 'OK' button in the 'Link Option' dialog box.
- 7. Click Import from Gen**: Points to the 'Import from Gen' button in the 'RC Design Procedure' list.

The 'Link Option' dialog box shows the following settings:

- Link Type by Member: ☒ Apply Link Option by Member
- Link by Section: ☒ Link by Section
- Link by Member: ☐ Link by Member
- Consider Material Additionally: ☒ Consider Material Additionally
- Load Combination Type: ☒ by All Load Combination
- Force Type: ☒ Reaction of Support

The 'RC Design Procedure' list shows the following options:

- Design Code: IS456:2000
- Live Load: IS875-1987
- Rebar DB: IS
- Design Option
- Drawing Option
- Report Option
- Preference
- Slab
- Beam
- Column
- Column (General)
- Shear Wall
- Footing (1)
- Footing (Combined)
- Basement Wall
- Buttress
- Stair
- Corbel / Bracket
- Retaining Wall
- Anchor Bolt
- Beam Table
- Slab Table

The 'Rebar' table shows the following data:

	Y-Direction (Mux)	X-Direction (Muy)
Moment (kN.m)		
1 (mm)	P22 300.00	P22 400.00
2 (mm)		
3 (mm)		

NOTES

Gen file should be open before doing the midas Link in Design+



Material | Section | Load |

Material

Concrete: 20.68 MPa

Main Bar: 227 MPa

☐ Light Weight Concrete

Factor: 1

Stress-Strain: Equivalent Rectangle

Column Section

☒ Rectangle ☐ Circle

Cx: 500.00 mm

Cy: 500.00 mm

Ex: 0.00 mm

Ey: 0.00 mm

Material

Concrete: Design compression strength of concrete

Main Bar: Design yield strength of rebar

Column Section

Cx,Cy: Column Width and Height

Ex,Ey: Eccentricity distance of column (x and y direction)

Material | Section | Load |

Footing

Type: Isolated (Mat)

Depth: 300.00 mm

Cover: 80.00 mm

Property

FOOTING SIZE		
Lx	3.00	m
Ly	3.00	m

SOIL BEARING		
Capacity(fe)	150.00	KPa

Footing

Depth: Thickness of Footing

Cover: Cover thickness of reinforcement

Footing Size

Lx,Ly: Size of footing (x and y direction)

Soil Bearing

Capacity(fe): Allowable bearing capacity of footing

Notes

Input the following data.



Material
Section
Load

Design Load

Ps
800
kN

Msx
0.00
kN.m

Msy
0.00
kN.m

☐
Load Combinations (1) ...

Pu
1200
kN

Mux
0.00
kN.m

Muy
0.00
kN.m

☐
Load Combinations (1) ...

☒ Include Self-Weight

Mx <-> My

☒ Surcharge Load

Surface Load
10
kN/m²

Weight Density
18.00
kN/m³

Height
1
m

Shear Force Option

☒ Include Self-Weight

☒ Include Surcharge Load

Design Load

Pa: Axial force(**working load**)

Msx,Msy: Bending Moment (x and y direction, **working load**)

Pu: Axial force(**factored load**)

Mux,Muy: Bending Moment (x and y direction, **factored load**)

Note: If checking 'Include Self-Weight', MIDAS checks the foundation considering self-weight.

Surcharge Load

Surface Load: Load applying to the surface

Weight Density: Specific gravity of soil

Height: Height of soil

Notes

Input the following data.



midas Design+ 2026 - [E:\1. MIDAS INTERN\23. Dev Bhaiya\Completed\Foundation *] - [Member]

Mode/Link RC Steel SRC Aluminum Reinforce Load Option Tool View Help

Project Mode Simple Mode Check Mode midas Link Link Option Link DataBase Member Member List Drawing Quantity UI Mode midas Link Edit Mode

WorkBar Add New Member System RC Type Footing Node 999 Import from Gen Keep Sect. & Bar Data RC | Steel | SRC | Aluminum | Reinforce

RC Design Procedure Option Design Code : IS456:2000 Live Load : IS875-1987 Rebar DB : IS Design Option Drawing Option Report Option Preference Slab Beam Column Column (General) Shear Wall Footing (1) 1Podium Column(115) Footing (Combined) Basement Wall Buttrass Stair Corbel / Bracket Retaining Wall Anchor Bolt Beam Table Slab Table

Start Page Member Member List Drawing Quantity

General Member Name 1Podium Column(115) Apply this Member to Dwn & Report

Material Section Load

Material Concrete 40 MPa Main Bar 415 MPa Light Weight Concrete Factor 1 Stress-Strain Equivalent Rectangle Column Section Rectangle Circle Cx 450.00 mm Cy 450.00 mm Ex 0.00 mm Ey 0.00 mm

2. Click Material

3. Verify data. Concrete: 40 MPa Main Bar: 415 Mpa Cx = 450mm; Cy =450mm Ex = 0mm; Ey = 0mm

1. Click Imported Node

Rebar

	Y-Direction (Mux)	X-Direction (Muy)
Moment (kN.m)		
Layer 1 (mm)	P25 @ 450.00	P25 @ 450.00
Layer 2 (mm)	@ 0.00	@ 0.00
Result		

Message Creating midas Design+ data has been completed

100% Print... Save... Report... Option... Summary Report Include

Done User SI Unit Enlish



Start Page

Member

Member List

Drawing

Quantity

General

Member Name1Podium Column(115)

Apply this Member toDwg & Report

MaterialSection

Footing

TypeIsolated (Mat)

Depth500.00 mm

Cover80.00 mm

Property

FOOTING SIZE

Lx	3.00	m
Ly	3.00	m

SOIL BEARING

Capacity(fe)	200.00	kPa
--------------	--------	-----

1. Click Section

2. Input Type: Isolated Mat
Depth: 500 mm
Cover: 80 mm

3. Input Lx: 3.00 m
Ly: 3.00 m
Capacity (fe): 200 kPa

Rebar

	Y-Direction (Mux)	X-Direction (Muy)
Moment (kN.m)		
Layer 1 (mm)	P25 @ 450.00	P25 @ 450.00
Layer 2 (mm)	@ 0.00	@ 0.00
Result		

Report

100%

Print...

Save...

Report...

Option...

Summary Report

Include



1. Click Load

2. Click Load Combinations

3. Verify Load Combinations

4. Input 1.5m Height

5. Click Design (F4)

The software interface shows the following details:

- Menu Bar:** Mode/Link, RC, Steel, SRC, Aluminum, Reinforce, Load, Option, Tool, View, Help.
- Toolbar:** Project Mode, Simple Mode, Check Mode, midas Link, Link DataBase, Member List, Member Drawing, Quantity.
- Left Sidebar:**
 - System: RC
 - Type: Footing
 - Node: 999
 - RC Design Procedure:
 - Option
 - Design Code: IS456:2000
 - Live Load: IS875-1987
 - Rebar DB: IS
 - Design Option
 - Drawing Option
 - Report Option
 - Preference
 - Slab
 - Beam
 - Column
 - Column (General)
 - Shear Wall
 - Footing (1)
 - 1Podium Column(999)
 - Footing (Combined)
 - Basement Wall
 - Buttress
 - Stair
 - Corbel / Bracket
 - Retaining Wall
 - Anchor Bolt
 - Beam Table
 - Slab Table

Design Load Table:

SN	CHK	NAME	1	2	3	4	5	6	Description
MAX	✓	CLCB6	165.2	-29.29	-2.60	SERV -D + 0.8(L) +			
1	✓	CLCB6	165.2	-29.29	-2.60	SERV -D + 0.8(L) +			
2	✓	CLCB6	33.18	37.65	5.14	SERV -D - EYN			
3	✓	CLCB6	36.39	40.20	0.79	SERV -D + EXP			
4	✓	CLCB6	150.2	-39.47	0.62	SERV -D - EXP			
5	✓	CLCB6	51.32	-1.88	25.27	SERV -D - EYN			
6	✓	CLCB6	135.2	2.61	-23.86	SERV -D + EYN			

Design Load Summary:

Material	Section	Load
Ps	165.28	kN
Mux	-29.29	kN.m
Muy	-2.60	kN.m
Pu	230.12	kN
Mux	-55.38	kN.m
Muy	-5.60	kN.m

Design Parameters:

- Include Self-Weigh: ☒
- Surcharge Load: ☒
- Surface Load: 0.00 kN/m²
- Weight Density: 18.00 kN/m³
- Height: 1.5 m
- Shear Force Option:
 - Include Self-Weigh: ☒
 - Include Surcharge Load: ☒

Buttons: Design(F4), Check(F5), Report..., Apply(F3)



midas Design+ 2026 - [E:\1. MIDAS INTERN\23. Dev Bhaliya\Finalised Updated\foundation *] - [Member]

Mode/Link RC Steel SRC Aluminum Reinforce Load Option Tool View Help

Project Mode Simple Mode Check Mode midas Link Link Option Link DataBase Member Member Drawing Quantity List Edit Mode

WorkBar Add New Member System RC Type Footing Node 999 Import from Gen Keep Sect. & Bar Data

RC Steel SRC Aluminum Reinforce

RC Design Procedure Option Design Code : IS456:2000 Live Load : IS875-1987 Rebar DB : IS Design Option Drawing Option Report Option Preference Slab Beam Column Column (General) Shear Wall Footing (1) 1Podium Column(999) Footing (Combined) Basement Wall Buttress Stair Corbel / Bracket Retaining Wall Anchor Bolt Beam Table Slab Table

Start Page Member Member List Drawing Quantity

General Member Name Apply this Member

Material Section Load

Design Load

Ps 150.21 kN
M_{ax} -39.47 kN.m
M_{ay} 0.62 kN.m

Load Combinations (6) ...

PU 230.12 kN
M_{ux} -55.38 kN.m
M_{uy} -5.60 kN.m

Load Combinations (6) ...

☒ Include Self-Weigh Mx <-> My

☒ Surcharge Load

Surface Load 0.00 kN/m²
Weight Density 18.00 kN/m³
Height 1.50 m

Shear Force Option

☒ Include Self-Weigh
☒ Include Surcharge Load

Rebar

	Y-Direction (Mux)	X-Direction (Muy)
Moment (kN.m)	27.95	21.51
Layer 1 (mm)	P22 @ 250.00	P22 @ 250.00
Layer 2 (mm)	@	@
Result	OK(0.137)	OK(0.100)

Design(F4) Check(F5) Report ... Apply(F3)

Message

Creating midas Design+ data has been completed

Report

Print... Save... Report... Option... Summary Report Include

Rectangle 450mm 450mm 0.000mm 0.000mm

1. Click Detail Report and Tick Include Input Data

Rebar

Layer-1 (Y)	Layer-2 (Y)	Layer-1 (X)	Layer-2 (X)
P22@250	-	P22@250	-

6. Check Capacity

Check Items	Calculated	Criteria	Ratio
Soil Capacity (kN/m ²)	65.10	200	0.325
q _{ult} (kN/m ²)	98.37	-	-
q _{ult} (kN/m ²)	71.27	-	-
One Way Shear-X (kN)	0.000	548	0.000
One Way Shear-Y (kN)	22.41	531	0.0422
Two Way Shear (kN)	667	2.222	0.296
Moment-X Direction(Mux, kN m/m)	21.51	216	0.0996
Moment-Y Direction(Muy, kN m/m)	27.95	204	0.137
Rebar Space-Y Direction(sx, mm)	250	300	0.833
Rebar Space-X Direction(sy, mm)	250	300	0.833



413.2.6 Design Criteria

413.2.6.1 Foundations shall be proportioned to resist factored loads and induced reactions.

413.2.6.2 Foundation systems shall be permitted to be designed by any procedure satisfying equilibrium and geometric compatibility.

413.4.3 Deep Foundation Members

413.4.3.1 Portions of deep foundation members in air, water, or soils not capable of providing adequate restraint throughout the member length to prevent lateral buckling shall be designed as columns in accordance with the applicable provisions of Section 410.

413.4 Deep Foundations

413.4.1 General

413.4.1.1 Number and arrangement of piles, drilled piers, and caissons shall be determined from unfactored forces and moments transmitted to these members and permissible member capacity selected through principles of soil or rock mechanics.

413.4.2 Pile Caps

413.4.2.1 Overall depth of pile cap shall be selected such that the effective depth of bottom reinforcement is at least 300 mm.

413.4.2.2 Factored moments and shears shall be permitted to be calculated with the reaction from any pile assumed to be concentrated at the centroid of the pile section.



Minimum No. of Piles = Unfactored Load
/ Allowable Bearing Capacity

$$N_{\text{piles}} = \frac{P}{Q_{\text{all}}}$$

Allowable Bearing Capacity = Ultimate
Bearing Capacity / Factor of Safety

$$Q_{\text{all}} = \frac{Q_u}{F.S.}$$

Ultimate Bearing Capacity = Shaft
Resistance + Tip Resistance

$$Q_u = Q_s + Q_t$$

SUGGESTED MINIMUM CENTER TO CENTER PILE SPACING			
Pile Type	BOCA, 1993 (Sec. 1013.8)	NBC, 1976 (Sec. 912.1I)	Chicago, 1994 (Sec. -132-120)
Friction	2D or 1.75H ≥ 760 mm	2D or 1.75H ≥ 760 mm	2D or 1.75H ≥ 760 mm
Point Bearing	2D or 1.75H ≥ 760 mm	2D or 1.75H ≥ 760 mm	



midas Design+ 2026 - [E:\1. MIDAS INTERN\23. Dev Bhaiya\Completed\Foundation *] - [Member]

Mode/Link RC Steel SRC Aluminum Reinforce Load Option Tool View Help

Project Mode Simple Mode Check Mode midas Link midas Link Link DataBase Member Member Drawing Quantity List Edit Mode

WorkBar

Add New Member

System RC

Type Footing

Node ... Import from Gen

☐ Keep Sect. & Bar Data

RC | Steel | SRC | Aluminum | Reinforce |

RC Design Procedure

Option

Design Code : IS456:2000

Live Load : IS875-1987

Rebar DB : IS

Design Option

Drawing Option

Report Option

Preference

Slab

Beam

Column

Column (General)

Shear Wall

Footing (1)

1Podium Column(999)

Footing (Combined)

Basement Wall

Buttress

Stair

Corbel / Bracket

Retaining Wall

Anchor Bolt

Beam Table

Slab Table

Start Page Member Member List Drawing Quantity

General

Member Name 1Podium Column(115)

Apply this Member to Dwg & Report

Material Section Load

Footing

Type Isolated (Pile)

Depth Isolated (Mat)

Cover Isolated (Pile)

80.00 mm

Property

PILE LAYOUT		
Quantity	4	EA
Diameter	400.00	mm
Space	3.00	D
Extension	1.25	D

PILE CAPACITY		
Compression	600.00	kN/EA
Tension	0.00	kN/EA

Double click to Zoom

1. Click Section and choose Isolated Pile type

Rebar

	Y-Direction (Mux)	X-Direction (Muy)
Moment (kN.m)	P22 @ 250.00	P22 @ 250.00
Layer 1 (mm)	@	@
Layer 2 (mm)	@	@
Result		

Design(F4) Check(F5) Report ... Apply(F3)

Report

125% Print... Save... Report... Option... Summary Report Include

Done Message



3. Click Detail Report and Tick Include Input Data

midas Design+ 2026 - [E:\1. MIDAS INTERN\23. Dev Bhaliya\Finalised Updated\foundation *] - [Member List]

Mode/Link RC Steel SRC Aluminum Reinforce Load Option Tool View Help

Project Mode Simple Mode Check Mode midas Link Link Option Link DataBase Member Member Drawing Quantity Edit Mode

UI Mode midas Link

Start Page Member Member List Drawing Quantity

General
Member Name 1Podium Column(999)
Apply this Member to Dwg & Report

Material Section Load
Footing
Type Isolated (Pile)
Depth 500.00 mm
Cover 80.00 mm

Property
PILE LAYOUT
Quantity 5 EA
Diameter 500.00 mm
Space 2.50 D
Extension 2.00 D
PILE CAPACITY
Compression 600.00 kN/EA
Tension 0.00 kN/EA

1. Input Quantity: 5 EA
Diameter: 500 mm
Space: 2.5D
Extension: 2D
Compression: 600 KN/EA

Design(F4) Check(F5) Report ...

2. Click Design

Report

125% Print... Save... Report... Option... Summary Report Include

Rectangle 450mm 450mm 0.000mm 0.000mm

3,800
3,800
1,500
500

5. Rebar

	Layer-1 (Y)	Layer-2 (Y)	Layer-1 (X)	Layer-2 (X)
	P22@250	-	P22@250	-

6. Check Pile Capacity

Index	X (mm)	Y (mm)	V _u (kN)	V _{u,net} (kN)	V _{u,comp} (kN)	V _{u,tens} (kN)	V _u (kN)	V _u / V _u
01	-900	900	200	1,847	2,217	1,755	1,755	0.114
02	900	900	197	1,847	2,217	1,755	1,755	0.112
03	0.000	0.000	214	1,847	3,360	2,690	1,947	0.116
04	-900	-900	231	1,847	2,217	1,755	1,755	0.132
05	900	-900	228	1,847	2,217	1,755	1,755	0.130

• V_u, V_u: Pile Punching

7. Check Capacity

Check Items	Calculated	Criteria	Ratio
Pile Capacity-Comp. (kN)	154	600	0.257
Pile Capacity-Tens. (kN)	0.000	0.000	0.000
q _{u,net} (kN)	231	-	-
q _{u,comp} (kN)	197	-	-

Message
Creating midas Design+ data has been completed



1. Choose 2 adjacent column nodes in Gen and Click Import in Design+ to design combined footing

2. Click Footing

3. Verify data. Concrete: 40 MPa
Main Bar: 415 MPa
Depth : 500 mm
Cover : 50 mm

Design Parameters:

- Concrete: 40 MPa
- Main Bar: 415 MPa
- Factor: 1
- Light Weight Concrete: ☐
- Footing Depth: 500.00 mm
- Cover: 50.00 mm

Property:

FOOTING SIZE	
Ext. (Left)	1.00 m
Ext. (Right)	1.00 m
Width (Ly)	3.00 m

SOIL BEARING	
Capacity (fc)	200.00 KPa

Surcharge Load:

- Surface Load: 0.00 KPa
- Weight Density: 18.00 kN/m³
- Height: 1.5 m
- ☒ Include Self-Weight
- ☒ Include Surcharge Load

Bar Arrangement (X-Dir.): Cantilever (L)

Items	
Moment (kN)	
Eff. Width (m)	
Bottom (mm)	P25 @ 450.0
Max. Spacing (mm)	@
p / pmin (%)	/
As / As,Req (%)	/
Result	

Bar Arrangement (Y-Dir.): Support (1)

Items	
Moment (kN)	
Eff. Width (m)	
Bottom (mm)	P25 @ 450.0
Max. Spacing (mm)	@
p / pmin (%)	/
As / As,Req (%)	/
Result	

Calculation Result:

Check Items	Result
Soil Bearing (KPa)	
1Way Shear (kN)	
2Way Shear (kN)	
Max. Cantilever (k)	
Max. Support (k)	
Max. Span (kN)	
Max. Y-Dir (kN)	

Design(F4) Check(F5) Report ... Apply(F3)

Message: Creating midas Design+ data has been completed



midas Design+ 2026 - [E:\1. MIDAS INTERN\23. Dev Bhaiya\Finalised Updated\c f *] - [Member]

Mode/Link RC Steel SRC Aluminum Reinforce Load Option Tool View Help

Project Mode Simple Mode Check Mode midas Link Link Option Link DataBase Member Member Drawing Quantity List Edit Mode

WorkBar Add New Member System RC Type Footing (Combined) Node ... Import from Gen Keep Sect. & Bar Data RC Steel SRC Aluminum Reinforce RC Design Procedure Option Design Code : IS456:2000 Live Load : IS875-1987 Rebar DB : IS Design Option Drawing Option Report Option Preference Slab Beam Column Column (General) Shear Wall Footing (1) 1Podium Column(999) Footing (Combined) (1) F01 Basement Wall Butress Stair Corbel / Bracket Retaining Wall Anchor Bolt Beam Table Slab Table

Start Page Member Member List Drawing Quantity Report

General Member Name F01 Apply this Member to Dwg & Report

Footing Column

Column Information

No.	Shape	Span (m)
1	Rectangle	3.00
2	Rectangle	4.00

Add Delete

Column Section ☒ Rectangle ☐ Circle

Cx 450.00 mm Cy 450.00 mm

Service Load Ps 123.16 kN Msy 0.58 kN.m Load Combinations (33) ...

Factored Load Pu 184.74 kN Mu 0.87 kN.m Load Combinations (49) ...

Design(F4) Check(F5) Report ... Apply(F3)

Message Creating midas Design+ data has been completed

2. Click Footing

3. Verify data. Cx = 450 mm Cy = 450 mm

Load Combinations (Service Load)

SN	CHK	NAME	Ps (kN)	Msy (kN.m)	Description
MAX		CLCB5	123.1	0.58	SERV :D + (L)
1		CLCB5	123.1	0.58	SERV :D + (L)
2		CLCB5	84.44	7.01	SERV :D + WX
3		CLCB5	72.70	0.36	SERV :D + WY
4		CLCB5	102.1	-6.29	SERV :D - WX
5		CLCB5	113.9	0.36	SERV :D - WY
6		CLCB5	110.1	5.85	SERV :D + 0.8(L) +
7		CLCB5	100.7	0.53	SERV :D + 0.8(L) +
8		CLCB5	124.2	-4.79	SERV :D + 0.8(L) -
9		CLCB5	133.6	0.53	SERV :D + 0.8(L) -
10		CLCB6	36.39	40.20	SERV :D + EXP
11		CLCB6	153.4	-36.92	SERV :D + EXN
12		CLCB6	56.96	2.61	SERV :D + EYP
13		CLCB6	135.2	2.61	SERV :D + EYN
14		CLCB6	150.2	-39.47	SERV :D - EXP
15		CLCB6	33.18	37.65	SERV :D - EXN
16		CLCB6	129.6	-1.88	SERV :D - EYP

All None Invert Add Insert Delete OK Cancel

3. Verify Load Combinations

2. Click Design



1. Click Material Properties

2. Click Add

3. Name: Dummy

4. Choose None

5. Modulus of Elasticity: 2.5126×10^7 KN/m²
Poisson's Ratio: 0.2
Thermal Coefficient: 5.0000×10^{-6} 1/[F]
Weight Density: 0 KN/m³

6. Click OK

NOTES



The screenshot displays the MIDAS/Gen software interface with several windows and toolbars. The main window shows a 3D model of a building frame. Overlaid on this are two dialog boxes: 'Properties' and 'Section Data'. Red boxes and arrows highlight the following steps:

- 1. Click Section Properties**: A red box highlights the 'Section Properties' icon in the 'Properties' toolbar.
- 2. Click Add**: A red box highlights the 'Add...' button in the 'Properties' dialog box.
- 3. Choose Solid Round**: A red box highlights the 'Solid Round' radio button in the 'Section Data' dialog box.
- 4. Name: Dummy 1 m Pile**: A red box highlights the 'Name' field in the 'Section Data' dialog box, which contains the text 'dummy 1 m pile'.
- 5. Tick User**: A red box highlights the 'User' radio button in the 'Section Data' dialog box.
- 6. Input 1 m**: A red box highlights the 'D' field in the 'Section Data' dialog box, which contains the value '1'.
- 7. Click OK**: A red box highlights the 'OK' button in the 'Section Data' dialog box.
- 8. Repeat for 1 m Pile**: A red box highlights the 'OK' button in the 'Section Data' dialog box.

NOTES



1. Click Properties

2. Click Thickness

3. Click Add

4. Name 0.6 m Pile Cap

5. Input 0.6 m

6. Click OK

1. Run Analysis

Thickness Data

Value | Stiffened |

Thickness ID 2 Name raft

☒ In-plane & Out-of-plane 1.2 m

☐ In-plane 0 m

☐ Out-of-plane 0 m

☐ Plate Offset

☒ Thickness Ratio

Local z 0

☐ Value

Local z 0 m

Offset Distance

Show Calculation Result... OK Cancel Apply

Message Window

Command Message Analysis Message

None! U: 0, 32, 0 G: 0, 32, 0 kN m



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

Material Properties Change Property Material Link Comp. Strength

Plastic Material Section Properties Scale Factor Tapered Group Thickness Inelastic Control Data Inelastic Hinge Inelastic Material Group Damping Property Tables

Material Time Dependent Material Plastic Section Inelastic Properties Damping Tables

Tree Menu

Menu Tables Group Works Rep... Seis...

Works

- Analysis Control Data
- Eigenvalue Analysis [Type=Eigenve...
- Structures
 - Stories : 17
 - Base.1F
 - Base.2F
 - Base.3F
 - Base.4F
 - Base.5F
 - Base.6F
 - Base.7F
 - Base.8F
 - Base.9F
 - Base.10F
 - Base.11F
 - Base.12F
 - Base.13F
 - Base.14F
 - Base.15F
 - Base.16F
 - Base.Roof
 - Nodes : 964
 - Elements : 2064
 - Beam : 1936
 - Wall : 128
- Properties
 - Material : 3
 - 1 : M30
 - 2 : M40
 - 3 : Dummy
 - Section : 5
 - 1 : Podium Column
 - 2 : Building Column
 - 3 : Beam
 - 4 : dummy 1 m pile
 - 5 : 1 m pile
 - Thickness : 1
 - 1 : 0.23 (Shear wall & slab)
- Boundaries
 - Supports : 116
- Masses
 - Loads to Masses : 3

Tree Menu 2

Menu Tables Group Works Report Seismic

- Structure Analysis
 - Configuration
 - Geometry
 - Static Loads
 - Response Spectrum Analysis
 - Time History Analysis
 - Moving Load Analysis
 - Settlement Analysis Data
 - Composite Section Analysis Data
 - Heat of Hydration Analysis Data
 - Non-Linear Analysis Data
 - Imperfection
 - Construction Stage Analysis Data
- Results
- Design
- Query

Message Window

The project will be saved by the auto-save feature.

Command Message Analysis Message

For Help, press F1

None! U: 0.32, 0 G: 0.32, 0 kN m

1. Click **Front View** (Ctrl + Shift + F)

2. Click **Select Single** (Ctrl + Shift + S)

3. **Select Nodes**

4. **Activate** (F2 on keyboard)

5. **Top View** (Ctrl + Shift + T)



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary

1. Click **Node/Element**

2. Select Single

3. Click Translate

4. Input coordinates per corner (See table)

5. Click Apply

Translation

Mode: ☒ Copy ☐ Move

Translation

☒ Equal Distance

dx,dy,dz: -1,1,0 m

Number of Times: 1

☐ Unequal Distance

Axis: ☒ x ☐ y ☐ z

☐ Arbitrary

Distance: m

(Example: 5, 3, 4.5, 3@5.0)

Direction Vector: 0, 0, 0

Corner	Coordinates (dx,dy,dz)
1	-1, 1, 0
2	1, 1, 0
3	-1, -1, 0
4	1, -1, 0

Tree Menu 2

- Structure Analysis
 - Configuration
 - Geometry
 - Static Loads
 - Response Spectrum Analysis
 - Time History Analysis
 - Moving Load Analysis
 - Settlement Analysis Data
 - Composite Section Analysis Data
 - Heat of Hydration Analysis Data
 - Non-Linear Analysis Data
 - Imperfection
 - Construction Stage Analysis Data
 - Results
 - Design
 - Query

Tree Menu Task Pane

Command Message Analysis Message

For Help, press F1

Node-172 U: 52, 18, 0 G: 52, 18, 0 kN m

1. Click Node/Element

2. Click Define Column Capital

3. Click Add

4. Name C1-500X500

5. Tick Bottom of Column

**6. Input B1 = 0.25 m
B2 = 0.25 m
H1 = 0.25 m
H2 = 0.25 m**

7. Tick Auto Rigid Link

8. Click Apply

Message Window
The project will be saved by the auto-save feature.



The screenshot displays the MIDAS/Gen software interface. The main window shows a 3D model of a building frame with a central column. Two dialog boxes are open: 'Assign Column Strip' on the left and 'Add/Modify Column Capital' on the right. The 'Add/Modify Column Capital' dialog box has several fields and options that are highlighted with red boxes and numbered 1 through 5.

Assign Column Strip Dialog:

- Column Strip Name: [Empty]
- Assign by Node: [Unselected]
- Assign by Element: [Selected]
- Option: [Add / Replace]

Add/Modify Column Capital Dialog:

- Name:** C 2 - 300 X 800
- Location:** ☒ Bottom of Column
- Shape & Rigid Link:**
 - B1: 0.15 m
 - B2: 0.15 m
 - H1: 0.4 m
 - H2: 0.4 m
 - ☒ Auto Rigid Link
- OK**

1. Name C2-450x450

2. Tick Bottom of Column

3. Input B1 = 0.15 m
B2 = 0.15 m
H1 = 0.40 m
H2 = 0.40 m

4. Tick Auto Rigid Link

5. Click OK



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

Create Nodes Translate Divide Merge Delete Mirror Scale Nodes Table Create Elements Translate Extrude Divide Merge Intersect Rotate Mirror Change Parameters Elements Table Effective Beam Column Capital Auto-mesh Define Sub-Domain Map-mesh Define Domain

Nodes Elements Flat/Plate Structure Mesh

Tree Menu

Assign Column Capital

Assign Column Capital

Option

☒ Add / Replace ☐ Delete

Column Capital Name

C1 - 500 X 500

Apply Close

Tree Menu 2

Menu Tables Group Works Report Seismic

Structure Analysis

Configuration

Geometry

Static Loads

Response Spectrum Analysis

Time History Analysis

Moving Load Analysis

Settlement Analysis Data

Composite Section Analysis Data

Heat of Hydration Analysis Data

Non-Linear Analysis Data

Imperfection

Construction Stage Analysis Data

Results

Design

Query

Message Window

Command Message Analysis Message

For Help, press F1

None! U: 0, 32, 6 G: 0, 32, 6 kN m

1. Click **Front View**
(Ctrl + Shift + F)

2. Click **Select Single**
(Ctrl + Shift + S)

3. **Select Elements**

4. **Activate** (F2 on keyboard)

5. **Top View** (Ctrl + Shift + T)



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

Create Nodes Translate Divide Merge Delete Mirror Scale Project Nodes Table Create Elements Translate Extrude Divide Merge Intersect Rotate Mirror Change Parameters Elements Table Effective Beam Column Capital Auto-mesh Define Sub-Domain Map-mesh Define Domain Flat/Plate Structure Mesh

115to130 1

1. Select Exterior Columns

2. Choose C1-450x450

3. Click Apply

Assign Column Capital

Option: Add / Replace Delete

Column Capital Name: C1 - 450 X 450

Apply Close

Tree Menu 2

Menu Tables Group Works Report Seismic

Works

- Analysis Control Data
- Eigenvalue Analysis [Type=Eigenvalues-Lanczos]
- Structures
 - Stories: 17
 - Base: 1F
 - Base: 2F
 - Base: 3F
 - Base: 4F
 - Base: 5F
 - Base: 6F
 - Base: 7F
 - Base: 8F
 - Base: 9F
 - Base: 10F
 - Base: 11F
 - Base: 12F
 - Base: 13F
 - Base: 14F
 - Base: 15F
 - Base: 16F
 - Base: Roof
 - Nodes: 968
 - Elements: 2064
 - Beam: 1936
 - Wall: 128
- Properties
 - Material: 3
 - M30
 - M40
 - Dummy
 - Section: 5
 - 1: Podium Column
 - 2: Building Column
 - 3: Beam
 - 4: dummy 1 m pile
 - 5: 1 m pile
 - Thickness: 1
 - 1: 0.23 (Shear wall & slab)
 - Column Capital: 2
 - C1 - 450 X 450 [Element, 0]
 - C2 - 300 X 800 [Element, 0]
 - Boundaries
 - Supports: 116
 - Masses

Message Window

Command Message Analysis Message

Node: 159



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

Create Nodes Translate Divide Merge Delete Mirror Scale Nodes Table Create Elements Translate Extrude Divide Merge Intersect Rotate Mirror Change Parameters Elements Table Effective Beam Column Capital Auto-mesh Define Sub-Domain Map-mesh Define Domain Flat/Plate Structure Mesh

1. Select Interior Columns

2. Choose C2-300x800

3. Click Apply

Assign Column Capital

Option

☒ Add / Replace ☐ Delete

Column Capital Name

C2 - 300 X 800

Apply Close

Tree Menu 2

Menu Tables Group Works Report Seismic

- Stories : 17
 - Base:1F
 - Base:2F
 - Base:3F
 - Base:4F
 - Base:5F
 - Base:6F
 - Base:7F
 - Base:8F
 - Base:9F
 - Base:10F
 - Base:11F
 - Base:12F
 - Base:13F
 - Base:14F
 - Base:15F
 - Base:16F
 - Base:Roof
- Nodes : 968
- Elements : 2064
 - Beam : 1936
 - Wall : 128
- Properties
 - Material : 4
 - 1 : M30
 - 2 : M40
 - 3 : Dummy
 - 4 : M35
 - Section : 5
 - 1 : Podium Column
 - 2 : Building Column
 - 3 : Beam
 - 4 : dummy 1 m pile
 - 5 : 1 m pile
 - Thickness : 1
 - 1 : 0.23 (Shear wall & slab)
- Column Capital : 2
- Boundaries
- Supports : 116
- Masses
- Loads to Masses : 3
- Static Loads
 - Static Load Case 1 [DL ; Dead Load]
 - Static Load Case 2 [LL ; Live Load]
 - Static Load Case 3 [SIDL ; Super Imposed Dead Lo.]

Message Window

For Help, press F1

None! U: -1, 33, 0 G: -1, 33, 0 kN m



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

Create Nodes Translate Divide Merge Delete Rotate Scale Project Merge Intersect Rotate Mirror Change Parameters Elements Table Effective Beam Column Capital Column Strip Drop Panel Auto-mesh Map-mesh Define Domain Define Sub-Domain

Nodes Elements Flat/Plate Structure Mesh

Tree Menu 1: MIDAS/Gen

Mesh

Auto-mesh Planar Area

Mesh Method Nodes

1083, 1084, 1086, 1085

Type Quad+Triangle

☒ Mesh Inner Domain

☒ Include Interior Nodes

☒ Include Interior Lines

☒ Include Boundary Connectivity

Mesh Size

☒ Length ☐ Div. 0.5 m

Property

Element Type Plate

Material 4: M 35

Thickness 1 2: 1.2000

Domain

Name MAT

☐ Delete Boundary Line Elem.

☒ Subdivide Boundary Line Elem.

Apply Close

Tree Menu 2

Menu Tables Group Works Report Seismic

Stones : 17

Base: 1F

Base: 2F

Base: 3F

Base: 4F

Base: 5F

Base: 6F

Base: 7F

Base: 8F

Base: 9F

Base: 10F

Base: 11F

Base: 12F

Base: 13F

Base: 14F

Base: 15F

Base: 16F

Base: Roof

Nodes : 968

Elements : 2064

Beam : 1936

Wall : 128

Properties

Material : 4

1: M30

2: M40

3: Dummy

4: M 35

Section : 5

1: Podium Column

2: Building Column

3: Beam

4: dummy 1 m pile

5: 1 m pile

Thickness : 1

1: 0.23 (Shear wall & slab)

Column Capital : 2

C1 - 450 X 450 [Element, 72]

C2 - 300 X 800 [Element, 32]

Boundaries

Supports : 116

Masses

Loads to Masses : 3

Static Loads

Static Load Case 1 [DL; Dead Load]

Message Window

For Help, press F1

Node-970 U: 53, -1, 0 G: 53, -1, 0 kN m



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

Create Nodes Translate Divide Merge Delete Rotate Scale Mirror Nodes Table Create Elements Translate Extrude Divide Merge Intersect Delete Rotate Mirror Change Parameters Elements Table Effective Beam Column Strip Drop Panel Auto-mesh Map-mesh Define Sub-Domain Define Domain

Nodes Elements Flat/Plate Structure Mesh

Tree Menu Start Page MIDAS/Gen

Mesh

Auto-mesh Planar Area

Mesh

Method Nodes

327

Type Quad+Triangle

☒ Mesh Inner Domain

☒ Include Interior Nodes

☒ Auto ☐ User

☒ Include Interior Lines

☒ Auto ☐ User

☒ Include Boundary Connectivity

Mesh Size

☒ Length ☐ Div. 1 m

Property

Element Type Plate

Material 4 4: M 35

Thickness 1 1: 0.2300

Domain

Name MAU

☐ Delete Boundary Line Elem.

☒ Subdivide Boundary Line Elem.

Apply Close

Message Window

Tree Menu 2

Menu Tables Group Works Report Seismic

Stories : 17

- Base:1F
- Base:2F
- Base:3F
- Base:4F
- Base:5F
- Base:6F
- Base:7F
- Base:8F
- Base:9F
- Base:10F
- Base:11F
- Base:12F
- Base:13F
- Base:14F
- Base:15F
- Base:16F
- Base:Roof

Nodes : 7386

Elements : 8792

- Beam : 1936
- Plate : 6728
- Wall : 128

Domain : 1

MAT [Elem=0]

- [1] [Elem=6312: Type=Slab]
- [2] [Elem=4: Type=Slab]
- [3] [Elem=4: Type=Slab]
- [4] [Elem=4: Type=Slab]
- [5] [Elem=4: Type=Slab]
- [6] [Elem=4: Type=Slab]
- [7] [Elem=4: Type=Slab]
- [8] [Elem=4: Type=Slab]
- [9] [Elem=4: Type=Slab]
- [10] [Elem=4: Type=Slab]
- [11] [Elem=4: Type=Slab]
- [12] [Elem=4: Type=Slab]
- [13] [Elem=4: Type=Slab]
- [14] [Elem=4: Type=Slab]
- [15] [Elem=4: Type=Slab]
- [16] [Elem=4: Type=Slab]
- [17] [Elem=4: Type=Slab]
- [18] [Elem=4: Type=Slab]
- [19] [Elem=4: Type=Slab]
- [20] [Elem=4: Type=Slab]

Node-238 U: 0, 32, 6 G: 0, 32, 6

kN m



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]

1. Click Load

2. Click Static Load Cases

3. Name Soil Load

4. Click Add

Static Load Cases

Name : **Soil Load**

Type : Dead Load (D)

Description :

No	Name	Type	Description
1	DL	Dead Load (D)	Dead Load
2	LL	Live Load (L)	Live Load
3	SIDL	Dead Load (D)	Super Imposed Dead
4	WX	Wind Load on Stru	Wind Load in X Directi
5	WY	Wind Load on Stru	Wind Load in Y Directi
6	EXP	Earthquake (E)	Earthquake Load in X
7	EXN	Earthquake (E)	Earthquake Load in X
8	EYP	Earthquake (E)	Earthquake Load in Y
9	EYN	Earthquake (E)	Earthquake Load in Y
10	Soil Loa	Dead Load (D)	

Subdivide Boundary Line Elem.

Apply Close

Message Window

Tree Menu 2

Menu Tables Group Works Report Seismic

Stories : 17

- Base:1F
- Base:2F
- Base:3F
- Base:4F
- Base:5F
- Base:6F
- Base:7F
- Base:8F
- Base:9F
- Base:10F
- Base:11F
- Base:12F
- Base:13F
- Base:14F
- Base:15F
- Base:16F
- Base:Roof

Nodes : 7386

Elements : 8792

- Beam : 1936
- Plate : 6728
- Wall : 128

Domain : 1

MAT [Elem=0]

- [1] [Elem=6312: Type=Slab]
- [2] [Elem=4: Type=Slab]
- [3] [Elem=4: Type=Slab]
- [4] [Elem=4: Type=Slab]
- [5] [Elem=4: Type=Slab]
- [6] [Elem=4: Type=Slab]
- [7] [Elem=4: Type=Slab]
- [8] [Elem=4: Type=Slab]
- [9] [Elem=4: Type=Slab]
- [10] [Elem=4: Type=Slab]
- [11] [Elem=4: Type=Slab]
- [12] [Elem=4: Type=Slab]
- [13] [Elem=4: Type=Slab]
- [14] [Elem=4: Type=Slab]
- [15] [Elem=4: Type=Slab]
- [16] [Elem=4: Type=Slab]
- [17] [Elem=4: Type=Slab]
- [18] [Elem=4: Type=Slab]
- [19] [Elem=4: Type=Slab]
- [20] [Elem=4: Type=Slab]

Node-536 U: 8, 24, 21 G: 8, 24, 21 kN m



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary **Load** Analysis Results Pushover Design Seismic Performance Query Tools

Static Loads Dynamic Loads Settlement/Misc. Temp./Prestress Construction Stage Load Tables Moving Load Heat of Hydration

Static Load Cases Using Load Combinations Create Load Cases

Self Weight Nodal Body Force Nodal Loads Loads to Masses Specified Displ. Structure Loads / Masses

Wind Loads Earth Pressure Wind Pressure Seismic Loads Lateral

Element Line Typical Beam Load

Pressure Loads Hydrostatic Pressure Assign Plane Loads Pressure Load

Initial Forces Assign Floor Loads Finishing Material Loads Initial Forces/Misc.

1. Click Pressure Loads

Load Type Name

Load Group Name Soil Load

Options Add Replace Delete

Element Types Plate/Plane Stress(Face)

Pressure on Plate

Selection: Node Element

Pressure Face: Face #1

Direction: Global Z

Vector: 0, 0, 0

Projection: Yes No

Loads Uniform Linear

P1 -1.5*17 kN/m

P2 0 kN/m²

P3 0 kN/m²

P4 0 kN/m²

2. Choose Soil Load

3. Choose Global Z

4. Input -1.5*17 kN/m²

5. Double Click Plate

6. Click Apply

Tree Menu 2

Menu Tables Group Works Report Seismic

Stories: 17

Base: 1F

Base: 2F

Base: 3F

Base: 4F

Base: 5F

Base: 6F

Base: 7F

Base: 8F

Base: 9F

Base: 10F

Base: 11F

Base: 12F

Base: 13F

Base: 14F

Base: 15F

Base: 16F

Base: Roof

Nodes: 7386

Elements: 8792

Beam: 1026

Wall: 128

Domain: 1

MAT [Elem=0]

[1] [Elem=6312: Type=Slab]

[2] [Elem=4: Type=Slab]

[3] [Elem=4: Type=Slab]

[4] [Elem=4: Type=Slab]

[5] [Elem=4: Type=Slab]

[6] [Elem=4: Type=Slab]

[7] [Elem=4: Type=Slab]

[8] [Elem=4: Type=Slab]

[9] [Elem=4: Type=Slab]

[10] [Elem=4: Type=Slab]

[11] [Elem=4: Type=Slab]

[12] [Elem=4: Type=Slab]

[13] [Elem=4: Type=Slab]

[14] [Elem=4: Type=Slab]

[15] [Elem=4: Type=Slab]

[16] [Elem=4: Type=Slab]

[17] [Elem=4: Type=Slab]

[18] [Elem=4: Type=Slab]

[19] [Elem=4: Type=Slab]

[20] [Elem=4: Type=Slab]

Message Window

Tree Menu Task Pane

For Help, press F1

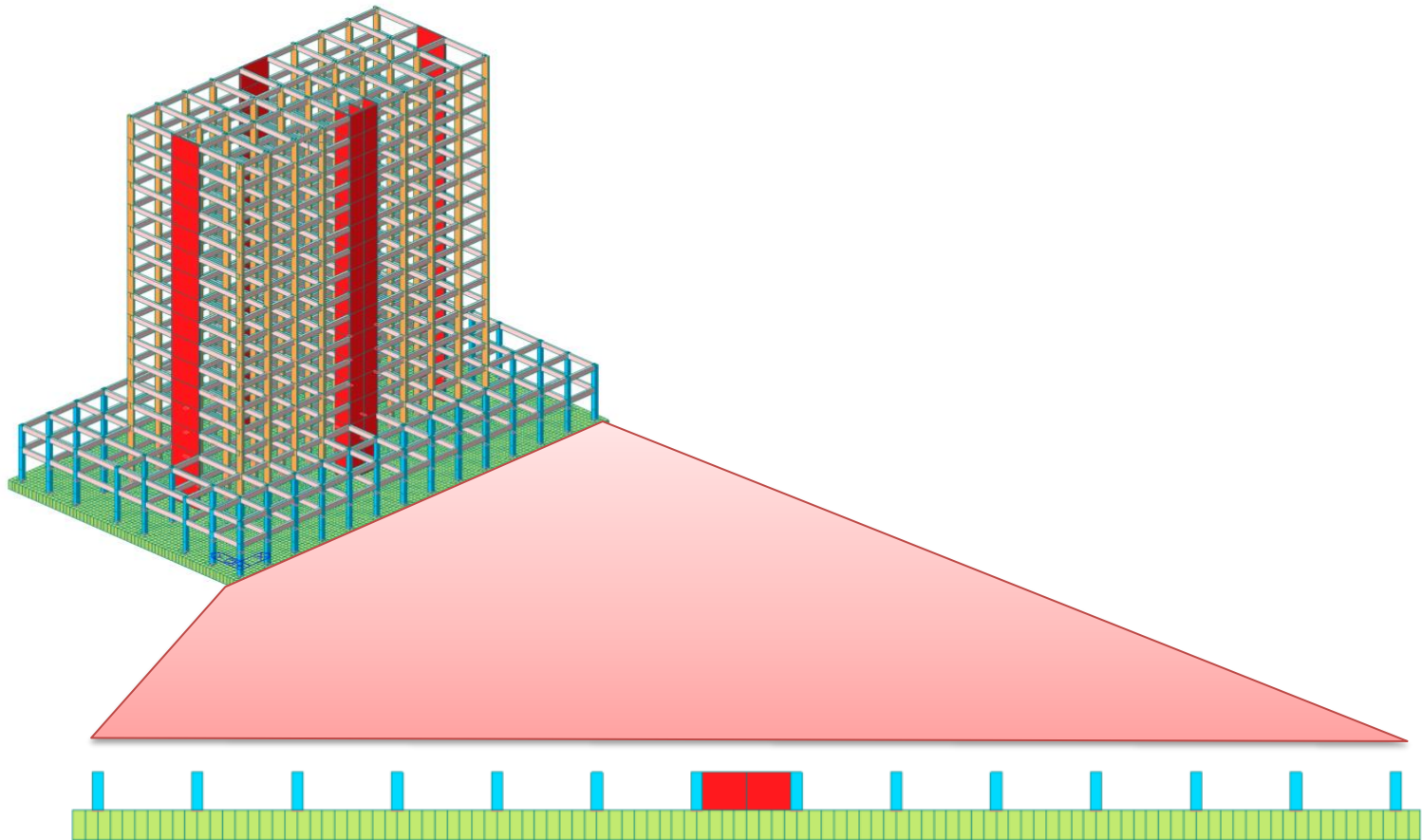
G: 8, 24, 28.5

kN m

NOTES

Sample geotechnical data. Approval by a geotechnical engineer is required for use.





NOTES

Representative elements and nodes are positioned at the center of the pile cap.



The screenshot shows the MIDAS/Gen software interface with several annotations indicating the steps to define a surface spring:

- 1. Click Boundary**: Points to the 'Boundary' button in the top toolbar.
- 2. Click Surface Spring**: Points to the 'Surface Spring' button in the 'Spring Supports' dropdown menu.
- 3. Tick Point Spring**: Points to the 'Point Spring' radio button in the 'Surface Spring' settings panel.
- 4. $K_x = 2400 \text{ kN/m}^3$
 $K_y = 2400 \text{ kN/m}^3$
 $K_z = 3300 \text{ kN/m}^3$** : Points to the input fields for the subgrade reaction modulus.
- 5. Double Click Plate**: Points to the 'Plate' element in the 'Tree Menu 2' on the right.
- 6. Click Apply**: Points to the 'Apply' button at the bottom of the settings panel.

The central window displays a 3D model of a mat foundation with a central column. The left panel shows the 'Surface Spring Supports' settings, and the right panel shows the 'Tree Menu 2' with the 'Plate' element selected.

NOTES

Sample geotechnical data. Approval by a geotechnical engineer is required for use.



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1] - [MIDAS/Gen]

View Structure Node/Element Properties **Boundary** Load Analysis Results Pushover Design Seismic Performance Query Tools

Define Supports Point Spring Surface Spring General Spring * Pile Spring Supports Elastic Link Rigid Link General Link Beam End Release Beam End Offsets Plate End Release Linear Constraints Define Label Dir Ignore Elements for Load Cases Panel Zone Effects Diaphragm Disconnect Node Local Axis Story Diaphragm Group Misc. Boundary Tables

Tree Menu Node Element Boundary Mass Load Default

Surface Spring

- Convert to Nodal Spring
 - Point Spring
 - Elastic Link
- Distributed Spring

Diagram: A rectangular area with a central vertical line and two horizontal lines, labeled with K and K_s .

Equations: $K_s \cdot A_{eff} = K_s$, A_{eff} = Effective Area per Node, K_s = Modulus of Subgrade Reaction

Element Type Planar Face #1 Width: 0 m

Spring Type Type Linear

Modulus of Subgrade Reaction: Node Local Axis (if defined)

Kx: 2400 kN/m³
Ky: 2400 kN/m³
Kz: 3300 kN/m³
Phu: 0 kN/m²

Damping Constant / Area
Cx: 0 kN·sec/m³
Cy: 0 kN·sec/m³
Cz: 0 kN·sec/m³

Apply Close

Tree Menu 2 Menu Tables Group Works Report Seismic

- Base:16F
 - Base:Roof
 - Nodes: 8792
 - Beam: 1936
 - Plate: 6728
 - Wall: 128
- Domain: 1
- MAT [Elem=0]
- Properties
- Material: 4
 - 1: M30
 - 2: M40
 - 3: Dummy
 - 4: M35
- Section: 5
 - 1: Podium Column
 - 2: Building Column
 - 3: Beam
 - 4: dummy 1 m pile
 - 5: 1 m pile
- Thickness: 1
- 1: 0.23 (Shear wall & slab)
- Column Capital: 2
- Boundaries
- Supports: 116
 - Type 1 [1111111]
- Point Spring Supports: 6538
- Linear: 6538
 - Type 1 [SSS000 000000]
 - Type 2 [SSS000 000000]
 - Type 3 [SSS000 000000]
 - Type 4 [SSS000 000000]
 - Type 5 [SSS000 000000]
 - Type 6 [SSS000 000000]
 - Type 7 [SSS000 000000]
 - Type 8 [SSS000 000000]
 - Type 9 [SSS000 000000]
 - Type 10 [SSS000 000000]
 - Type 11 [SSS000 000000]
 - Type 12 [SSS000 000000]
 - Type 13 [SSS000 000000]
 - Type 14 [SSS000 000000]
 - Type 15 [SSS000 000000]
 - Type 16 [SSS000 000000]

Message Window

The project will be saved by the auto-save feature.

Tree Menu Task Pane

For Help, press F1

None! U: -1, 33, 0 G: -1, 33, 0 kN m

1. Click Type 1 [1111111] and click Del on keyboard



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis **Results** Window Design Column Design

1. Click Results

2. Click Load Combination

Load Combination

Automatic Generation of Load Combinations

Option: ☒ Add ☐ Replace ☒ Add Envelope

Code Selection: ☐ Steel ☒ Concrete ☐ SRC ☐ Cold Formed Steel ☐ Footing ☐ Aluminum

Design Code: IS456:2000

Scale Up of Response Spectrum Load Cases: Scale Up Factor: 1 Spec X

Factor Load C... Add
2... Spec X Modify
2... Spec Y Delete

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

☐ Consider Orthogonal Effect: Set Load Cases for Orthogonal Effect...

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

Generate Additional Load Combinations: ☐ for Special Seismic Load ☐ for Vertical Seismic Forces Factors for Seismic Design...

☐ Will Execute Construction Stage Analysis ☐ Consider Losses for Prestress Load Cases

Transfer Stage: 1 Define Factors
Service Load Stage: 1

3. Click Auto Generation

File Name: C:\Users\Naveen\Downloads\RC Building check 1.kp Browse Make Load Combination Sheet Close

4. Click OK

Tree Menu 2: Menu Tables Group Works Report Seismic

- Base:16F
- Base:Roof
- Nodes: 7386
- Elements: 8792
- Beam: 1936
- Plate: 6728
- Wall: 128
- Domain: 1
- MAT [Elem=0]
- Properties
- Material: 4
- 1: M30
- 2: M40
- 3: Dummy
- 4: M35
- Section: 5
- 1: Podium Column
- 2: Building Column
- 3: Beam
- 4: dummy 1 m pile
- 5: 1 m pile
- Thickness: 1
- 1: 0.23 (Shear wall & slab)
- Column Capital: 2
- Boundaries
- Supports: 116
- Type 1 [11111111]
- Point Spring Supports: 6538
- Linear: 6538
- Type 1 [SSS000 000000]
- Type 2 [SSS000 000000]
- Type 3 [SSS000 000000]
- Type 4 [SSS000 000000]
- Type 5 [SSS000 000000]
- Type 6 [SSS000 000000]
- Type 7 [SSS000 000000]
- Type 8 [SSS000 000000]
- Type 9 [SSS000 000000]
- Type 10 [SSS000 000000]
- Type 11 [SSS000 000000]
- Type 12 [SSS000 000000]
- Type 13 [SSS000 000000]
- Type 14 [SSS000 000000]
- Type 15 [SSS000 000000]
- Type 16 [SSS000 000000]
- Type 17 [SSS000 000000]
- Type 18 [SSS000 000000]
- Type 19 [SSS000 000000]
- Type 20 [SSS000 000000]
- Type 21 [SSS000 000000]
- Type 22 [SSS000 000000]
- Type 23 [SSS000 000000]
- Type 24 [SSS000 000000]
- Type 25 [SSS000 000000]
- Type 26 [SSS000 000000]
- Type 27 [SSS000 000000]
- Type 28 [SSS000 000000]
- Type 29 [SSS000 000000]
- Type 30 [SSS000 000000]
- Type 31 [SSS000 000000]
- Type 32 [SSS000 000000]
- Type 33 [SSS000 000000]
- Type 34 [SSS000 000000]
- Type 35 [SSS000 000000]
- Type 36 [SSS000 000000]
- Type 37 [SSS000 000000]
- Type 38 [SSS000 000000]
- Type 39 [SSS000 000000]
- Type 40 [SSS000 000000]
- Type 41 [SSS000 000000]
- Type 42 [SSS000 000000]
- Type 43 [SSS000 000000]
- Type 44 [SSS000 000000]
- Type 45 [SSS000 000000]
- Type 46 [SSS000 000000]
- Type 47 [SSS000 000000]
- Type 48 [SSS000 000000]
- Type 49 [SSS000 000000]
- Type 50 [SSS000 000000]
- Type 51 [SSS000 000000]
- Type 52 [SSS000 000000]
- Type 53 [SSS000 000000]
- Type 54 [SSS000 000000]
- Type 55 [SSS000 000000]
- Type 56 [SSS000 000000]
- Type 57 [SSS000 000000]
- Type 58 [SSS000 000000]
- Type 59 [SSS000 000000]
- Type 60 [SSS000 000000]
- Type 61 [SSS000 000000]
- Type 62 [SSS000 000000]
- Type 63 [SSS000 000000]
- Type 64 [SSS000 000000]
- Type 65 [SSS000 000000]
- Type 66 [SSS000 000000]
- Type 67 [SSS000 000000]
- Type 68 [SSS000 000000]
- Type 69 [SSS000 000000]
- Type 70 [SSS000 000000]
- Type 71 [SSS000 000000]
- Type 72 [SSS000 000000]
- Type 73 [SSS000 000000]
- Type 74 [SSS000 000000]
- Type 75 [SSS000 000000]
- Type 76 [SSS000 000000]
- Type 77 [SSS000 000000]
- Type 78 [SSS000 000000]
- Type 79 [SSS000 000000]
- Type 80 [SSS000 000000]
- Type 81 [SSS000 000000]
- Type 82 [SSS000 000000]
- Type 83 [SSS000 000000]
- Type 84 [SSS000 000000]
- Type 85 [SSS000 000000]
- Type 86 [SSS000 000000]
- Type 87 [SSS000 000000]
- Type 88 [SSS000 000000]
- Type 89 [SSS000 000000]
- Type 90 [SSS000 000000]
- Type 91 [SSS000 000000]
- Type 92 [SSS000 000000]
- Type 93 [SSS000 000000]
- Type 94 [SSS000 000000]
- Type 95 [SSS000 000000]
- Type 96 [SSS000 000000]
- Type 97 [SSS000 000000]
- Type 98 [SSS000 000000]
- Type 99 [SSS000 000000]
- Type 100 [SSS000 000000]

Command Message: The project will be saved by the auto-save feature.

For Help, press F1

None! U: -1, 33, 0 G: -1, 33, 0 kN m



The screenshot displays the MIDAS/Gen software interface. The main window shows a 3D model of a building foundation with a central red column and surrounding blue columns. A dialog box titled "Check Changed Story Information" is open, displaying the message: "Entered story data does not match to the current model. Please check the story data if it has been correctly entered." The dialog box has three buttons: "Update Story Data...", "Continue", and "Cancel". A red arrow points to the "Continue" button. The software interface includes a menu bar at the top, a toolbar, and a left sidebar with various analysis and design options. The bottom status bar shows coordinates and units.

1. Run Analysis

2. Click Continue

Message Window

The process of Data Preparation for Analysis/Post-processing Completed.



The screenshot shows the MIDAS/Gen software interface with the following steps highlighted:

- 1. Click Results**: The 'Results' menu is selected in the top toolbar.
- 2. Click Plate Forces/Moments**: The 'Plate Forces/Moments' option is selected in the 'Results' dropdown menu.
- 3. Tick UCS**: The 'UCS' radio button is selected in the 'Plate Force Options' dialog box.
- 4. Tick Myy**: The 'Myy' radio button is selected in the 'Components' section of the 'Plate Force Options' dialog box.
- 5. Tick Contour and Legend**: The 'Contour' and 'Legend' checkboxes are selected in the 'Type of Display' section of the 'Plate Force Options' dialog box.
- 6. Click Apply**: The 'Apply' button is clicked at the bottom of the 'Plate Force Options' dialog box.

The background shows a 3D model of a mat foundation with a color-coded stress distribution. The right side of the interface displays a tree menu with various model components like 'Beam', 'Plate', 'Wall', 'Domain', 'Material', 'Properties', 'Section', 'Point Spring Supports', and 'Linear'.





1. Click Design

2. Click Meshed Design

3. Click Design Criteria for Rebars

4. Verify Design Criteria for Rebars

The screenshot shows the MIDAS/Gen software interface. The main workspace displays a 3D model of a slab and rebar. The left-hand panel shows the Tree Menu with options like 'Serviceability Load Combination Type...', 'Slab/Wall Load Combinations...', 'Design Criteria for Rebars...', 'Slab/Wall Rebars for Checking...', 'Slab Serviceability Parameters...', 'Meshed Design Tables', 'Slab Flexural Design...', 'Slab Flexural Checking...', 'Slab Shear Checking...', 'Slab Serviceability Checking...', 'Cracked Section Analysis Control...', 'Perform Cracked Section Analysis', 'Wall Design...', and 'Wall Checking...'. The right-hand panel shows the Tree Menu 2 with options like 'Menu', 'Tables', 'Group', 'Works', 'Report', 'Seismic', and a list of elements including 'Beam: 2040', 'Plate: 10695', 'Wall: 136', 'Domain: 1', 'MAT [Elem=0]', 'Properties', 'Material: 4', 'M30', 'M40', 'Dummy', 'm35', 'Section: 5', 'Podium Column', 'Building Column', 'Beam', 'Dummy 1 m pile', '5: 1 m pile', 'Thickness: 2', '1: 0.23 (Shear wall & slab)', '2: 1.2 (raft)', 'Boundaries', 'Point Spring Supports: 10640', 'Linear: 10640', and a list of types from Type 1 to Type 30. The bottom status bar shows 'Frame-2038', 'U: 24, 18, 46.5', 'G: 24, 18, 46.5', and 'mN m'.



The screenshot displays the MIDAS/Gen software interface for structural design. The main window shows a 3D model of a MAT foundation with a meshed design. The design process is guided by five numbered steps:

- 1. Click Design**: The 'Design' button in the top menu bar is highlighted.
- 2. Click Meshed Design**: The 'Meshed Design' option in the 'Design' dropdown menu is highlighted.
- 3. Click Slab Flexural Design**: The 'Slab Flexural Design...' option in the 'Design' tree is highlighted.
- 4. Click A_s_{req}** : The ' A_s_{req} ' option in the 'Design' tree is highlighted.
- 5. Click Apply**: The 'Apply' button in the 'Design' tree is highlighted.

The right-hand panel shows the 'Tree Menu 2' with various design options, including 'Base Roof', 'Nodes: 7386', 'Elements: 8792', 'Beam: 1936', 'Plate: 6728', 'Wall: 128', 'Material: 4', 'M30', 'M40', 'Dummy', 'M35', 'Section: 5', 'Section Stiffness Scale Factor', '5: 1 m pile [Group=Default, Ax=1, Ay=1, Az=1, I', 'Thickness: 1', '1: 0.23 (Shear wall & slab)', 'Boundaries', 'Masses', 'Loads to Masses: 3', 'Static Loads', 'Static Load Case 1 [DL; Dead Load]', 'Self Weight [SZ=1]', 'Floor Loads: 14', 'Floor Loads: 59', 'Static Load Case 3 [SIDL; Super Imposed Dead Lo.', 'Floor Loads: 45', 'Static Load Case 4 [WX; Wind Load in X Direction]', 'Wind Loads [IS875(2015)]', 'Static Load Case 5 [WY; Wind Load in Y Direction]', 'Wind Loads [IS875(2015)]', 'Static Load Case 6 [EXP; Earthquake Load in X Dire', 'Static Seismic Loads [IS1893(2016)]', 'Static Load Case 7 [EXN; Earthquake Load in X Dire', 'Static Seismic Loads [IS1893(2016)]', 'Static Load Case 8 [EYP; Earthquake Load in Y Dire', 'Static Seismic Loads [IS1893(2016)]', 'Static Load Case 9 [EYN; Earthquake Load in Y Dire', 'Static Seismic Loads [IS1893(2016)]', 'Static Load Case 10 [Sol Load;]', 'Pressure Loads: 6730', 'Response Spectrum Analysis', 'Response Spectrum Functions: 1', 'Response Spectrum Load Cases: 2', 'Domain: 1', 'MAT [Elem=0]', '[1] [Elem=6312 Type=Slab]'. The bottom status bar shows 'Frame: 324', 'U: 10, 32, 6', 'G: 10, 32, 6', and 'kN m'.



The screenshot displays the MIDAS GEN software interface. The main window shows the 'Design Results' for a 'Slab Flexural Design'. The results are presented in a table format, showing design parameters and calculated values.

Design Results Table:

Thk	Elem	POS	AsReq	AsUse	Mu (LCB)	M.Rd	Rat	CHK
1.2000	5341	BOT	0.0088	0.0024	3391.98	(1) 991.829	3.42	NG
5706	TOP	0.0017	0.0018	691.991	(1) 745.151	0.929	OK	

Below the table, the software provides detailed information about the design parameters and materials used.

Information of Parameters:

- Elem No. : 5341
- Node No. : 1073
- Thickness : 1.2000 m.
- Materials : fck = 35000.0000 KPa.
- fy = 415000.0000 KPa.
- Covering : dB = 0.0260 m.
- dI = 0.0260 m.
- LCB No. : 1

Information of Design:

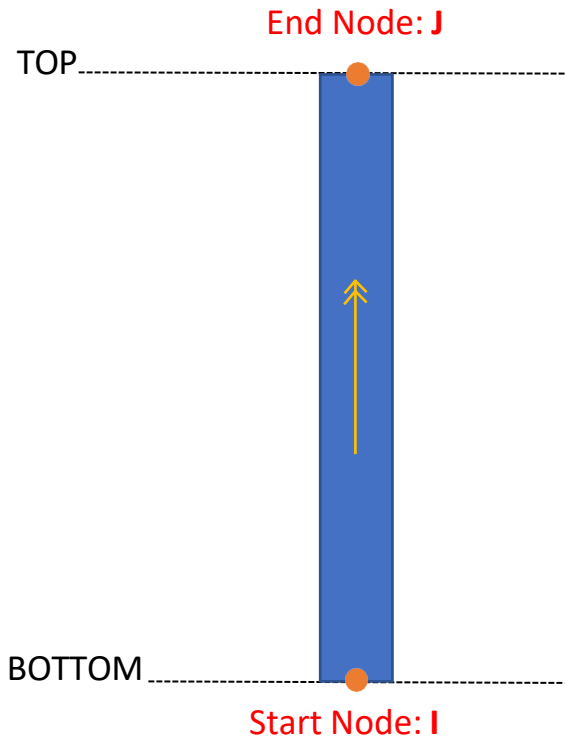
- b = 0.0010 m. (by Code Unit Length).
- d = 1.1740 m.

The 'Design Results' window is highlighted with a red box, and a red arrow points to the 'Design Result' button in the left sidebar. A message window at the bottom indicates the end of the design process.

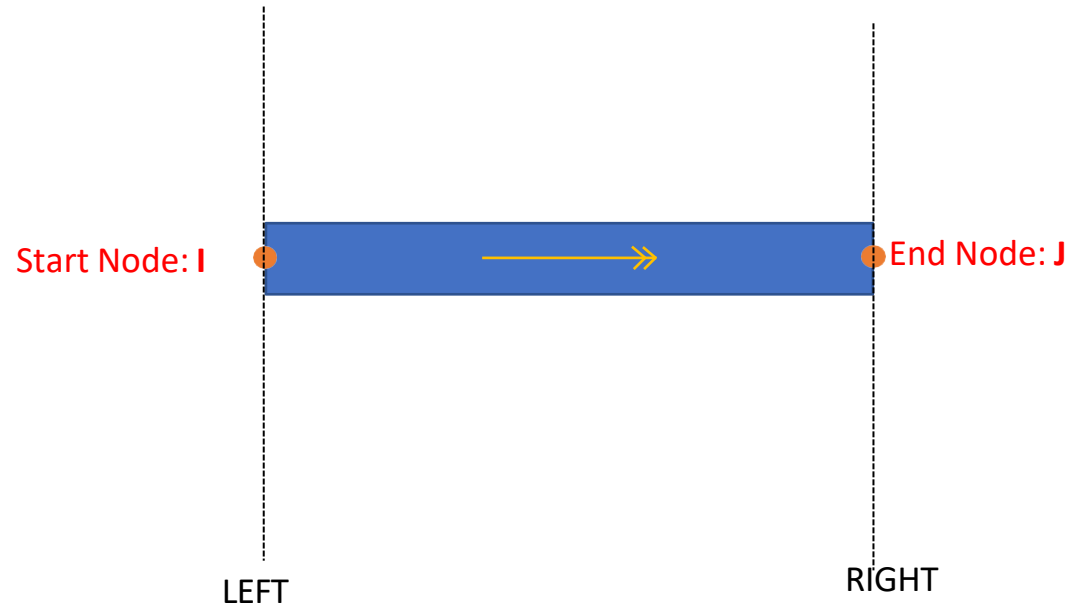
Message Window:

```
*** End Design By IS456:2000
Total Design/Checking Time... 24.50 [sec]
```





COLUMN



BEAM

1. Click Node/Element

2. Click Extrude

3. Choose Node - > Line Element

4. Tick Reverse I-J
Material: Dummy
Section: Dummy 1 m Pile

5. Input (0, 0, -0.3)

NOTES

Continue Next Slide...



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

Create Nodes Translate Divide Merge Delete Rotate Scale Nodes Table Create Elements Translate Extrude Divide Merge Intersect Delete Rotate Mirror Change Parameters Elements Table Effective Beam Column Capital Auto-mesh Define Sub-Domain Map-mesh Define Domain

Nodes Elements Flat/Plate Structure Mesh

Tree Menu 1: Node Element Boundary Mass Load

Extrude Elements

Start Number

Node Number : 7405

Element Number : 8793

Extrude Type

Node -> Line Element

Source ☐ Remove ☐ Move

☒ Reverse I-J

Element Attribute

Element Type: Beam

Material : 3 3: Dummy

Section : 4 4: dummy 1 m pile

Beta Angle : 0 [Deg]

Generation Type

☒ Translate ☐ Rotate ☐ Project

Translation

☒ Equal Distance ☐ Unequal Distance

dx,dy,dz: 0, 0, -0.6 m

Number of Times : 1

Tree Menu 2: Menu Tables Group Works Report Seismic

Base 6F

Base 7F

Base 8F

Base 9F

Base 10F

Base 11F

Base 12F

Base 13F

Base 14F

Base 15F

Base 16F

Base Roof

Nodes : 7386

Elements : 8792

Beam : 1936

Plate : 6728

Wall : 128

Properties

Material : 4

Section : 5

Thickness : 1

Boundaries

Supports

Point Spring Supports : 6538

Rigid Link : 104

Masses

Loads to Masses : 3

Static Loads

Static Load Case 1 [DL ; Dead Load]

Static Load Case 2 [LL ; Live Load]

Static Load Case 3 [SIDL ; Super Imposed Dead Lo.]

Static Load Case 4 [WX ; Wind Load in X Direction]

Static Load Case 5 [WY ; Wind Load in Y Direction]

Static Load Case 6 [EXP ; Earthquake Load in X Dire]

Static Load Case 7 [EXN ; Earthquake Load in X Dire]

Static Load Case 8 [EYP ; Earthquake Load in Y Dire]

Static Load Case 9 [EYN ; Earthquake Load in Y Dire]

Static Load Case 10 [Soil Load ;]

Response Spectrum Analysis

Response Spectrum Functions : 1

Response Spectrum Load Cases : 2

Domain : 1

Column Capital : 2

C1 - 450 X 450 [Element, 72]

C2 - 300 X 800 [Element, 32]

Message Window

*** End Design by 15456:2000

Total Design/Checking Time... 26.96 [sec]

Tree Menu Task Pane

For Help, press F1

None! U: 0, 24, 1.5 G: 0, 24, 1.5 kN m

NOTES

Continue Next Slide...



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

Create Nodes Translate Divide Merge Delete Rotate Scale Nodes Table Create Elements Translate Extrude Divide Merge Intersect Delete Rotate Mirror Change Parameters Elements Table Effective Beam Column Capital Auto-mesh Define Sub-Domain Column Strip Map-mesh Drop Panel Define Domain Mesh

Nodes Elements Flat/Plate Structure

Tree Menu Node Element Boundary Mass Load

Extrude Elements

Start Number

Node Number : 7405

Element Number : 8793

Extrude Type

Node -> Line Element

Source ☐ Remove ☐ Move

☒ Reverse I-J

Element Attribute

Element Type: Beam

Material : 3 3: Dummy

Section : 4 4: dummy 1 m pile

Beta Angle : 0 [Deg]

Generation Type

☒ Translate ☐ Rotate ☐ Project

Translation

☒ Equal Distance ☐ Unequal Distance

dx,dy,dz: 0, 0, -0.6 m

Number of Times : 1

Active [AA]

Active All [AA11]

Active Identity

Active Previous

456:2000

Checking Time... 26.96 [sec]

Tree Menu 2 Menu Tables Group Works Report Seismic

- Base: 6F
- Base: 7F
- Base: 8F
- Base: 9F
- Base: 10F
- Base: 11F
- Base: 12F
- Base: 13F
- Base: 14F
- Base: 15F
- Base: 16F
- Base: Roof
- Nodes: 7386
- Elements: 8792
 - Beam: 1936
 - Plate: 6728
 - Wall: 128
- Properties
 - Material: 4
 - Section: 5
 - Thickness: 1
- Boundaries
 - Supports
 - Point Spring Supports: 6538
 - Rigid Link: 104
- Masses
 - Loads to Masses: 3
- Static Loads
 - Static Load Case 1 [DL; Dead Load]
 - Static Load Case 2 [LL; Live Load]
 - Static Load Case 3 [SIDL; Super Imposed Dead Lo.]
 - Static Load Case 4 [WX; Wind Load in X Direction]
 - Static Load Case 5 [WY; Wind Load in Y Direction]
 - Static Load Case 6 [EXP; Earthquake Load in X Dire]
 - Static Load Case 7 [EXN; Earthquake Load in X Dire]
 - Static Load Case 8 [EYP; Earthquake Load in Y Dire]
 - Static Load Case 9 [EYN; Earthquake Load in Y Dire]
 - Static Load Case 10 [Soil Load;]
- Response Spectrum Analysis
 - Response Spectrum Functions: 1
 - Response Spectrum Load Cases: 2
- Domain: 1
- Column Capital: 2
 - C1 - 450 X 450 [Element, 72]
 - C2 - 300 X 800 [Element, 32]

Tree Menu Task Pane

For Help, press F1

None! U: 0, 24, 1.5 G: 0, 24, 1.5

kN m

NOTES

Continue Next Slide...



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS\Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

Create Nodes Translate Divide Merge Delete Rotate Scale Nodes Table Create Elements Translate Extrude Divide Merge Intersect Delete Rotate Mirror Change Parameters Elements Table Effective Beam Column Capital Auto-mesh Define Sub-Domain Column Strip Map-mesh Define Domain Drop Panel Flat/Plate Structure Mesh

Tree Menu 1: Node Element Boundary Mass Load

Extrude Elements

Start Number

Node Number : 7405

Element Number : 8793

Extrude Type

Node -> Line Element

Source ☐ Remove ☐ Move

☒ Reverse I-J

Element Attribute

Element Type: Beam

Material : 1 1: M35

Section : 9 9: 1 m Pile

Beta Angle : 0 [Deg]

Generation Type

☒ Translate ☐ Rotate ☐ Project

Translation

☒ Equal Distance ☐ Unequal Distance

dx,dy,dz: 0, 0, -1 m

Number of Times : 20

2. Select Bottom Nodes

3. Material: M35
Section: 1 m Pile

4. Input (0, 0, -1)
Number of Times: 20

5. Click Apply

Message Window

*** End Design ***

Total Design Time... 26.96 [sec]

Tree Menu 2: Menu Tables Group Works Report Seismic

Base 6F
Base 7F
Base 8F
Base 9F
Base 10F
Base 11F
Base 12F
Base 13F
Base 14F
Base 15F
Base 16F
Base Roof
Nodes : 7386
Elements : 8792
Beam : 1936
Plate : 6728
Wall : 128

Properties

Material : 4
Section : 5
Thickness : 1

Boundaries

Supports

Point Spring Supports : 6538
Rigid Link : 104

Masses

Loads to Masses : 3

Static Loads

Static Load Case 1 [DL ; Dead Load]
Static Load Case 2 [LL ; Live Load]
Static Load Case 3 [SIDL ; Super Imposed Dead Lo.]
Static Load Case 4 [WX ; Wind Load in X Direction]
Static Load Case 5 [WY ; Wind Load in Y Direction]
Static Load Case 6 [EXN ; Earthquake Load in X Dire]
Static Load Case 7 [EXN ; Earthquake Load in X Dire]
Static Load Case 8 [EYN ; Earthquake Load in Y Dire]
Static Load Case 9 [EYN ; Earthquake Load in Y Dire]
Static Load Case 10 [Soil Load ;]

Response Spectrum Analysis

Response Spectrum Functions : 1
Response Spectrum Load Cases : 2

Domain : 1

Column Capital : 2

C1 - 450 X 450 [Element, 72]
C2 - 300 X 800 [Element, 32]

For Help, press F1

None! | U: 0, 0, 0 | G: 0, 0, 0 | kN | m | 2

NOTES

Continue Next Slide...



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

Create Nodes Translate Divide Merge Delete Rotate Scale Nodes Table Create Elements Translate Extrude Divide Merge Intersect Delete Rotate Mirror Change Parameters Elements Table Effective Beam Column Capital Auto-mesh Define Sub-Domain Column Strip Map-mesh Drop Panel Define Domain

Nodes Elements Flat/Plate Structure Mesh

Tree Menu 1: Node Element Boundary Mass Load

Element Number: 10873

Extrude Type: Node -> Line Element

Source: ☐ Remove ☐ Move ☒ Reverse I-J

Element Attribute: Element Type: Beam

Material: 4 4: M 35

Section: 5 5: 1 m pile

Beta Angle: 0 [Deg]

Generation Type: ☒ Translate ☐ Rotate ☐ Project

Translation: ☒ Equal Distance ☐ Unequal Distance

dx,dy,dz: 0, 0, -1 m

Number of Times: 20

Merging Tolerance: Apply Close

Tree Menu 2: Menu Tables Group Works Report Seismic

- Base 6F
- Base 7F
- Base 8F
- Base 9F
- Base 10F
- Base 11F
- Base 12F
- Base 13F
- Base 14F
- Base 15F
- Base 16F
- Base Roof
- Nodes: 9466
- Elements: 10872
 - Beam: 4016
 - Plate: 6728
 - Wall: 128
- Properties
 - Material: 4
 - 1: M30
 - 2: M40
 - 3: Dummy
 - 4: M35
 - Section: 5
 - 1: Podium Column
 - 2: Building Column
 - 3: Beam
 - 4: dummy 1 m pile
 - 5: 1 m pile
 - Thickness: 1
 - 1: 0.23 (Shear wall & slab)
- Boundaries
 - Supports:
 - Point Spring Supports: 6538
 - Rigid Link: 104
 - Masses
 - Loads to Masses: 3
 - Static Loads
 - Static Load Case 1 [DL; Dead Load]
 - Static Load Case 2 [LL; Live Load]
 - Static Load Case 3 [SIDL; Super Imposed Dead Lo.]
 - Static Load Case 4 [WX; Wind Load in X Direction]
 - Static Load Case 5 [WY; Wind Load in Y Direction]
 - Static Load Case 6 [EXP; Earthquake Load in X Dire]
 - Static Load Case 7 [EXN; Earthquake Load in X Dire]

Active [AA]

Active All [AA11]

Active Identity

Active Previous

456:2000

Checking Time...: 26.96 [sec]

Tree Menu Task Pane

Command Message Analysis Message

For Help, press F1

Node-9034 U: 52, 14, -16 G: 52, 14, -16 kN m



The screenshot shows the MIDAS/Gen software interface with the 'Section Stiffness Scale Factor' dialog box open. The dialog box has a 'Section' list on the left and a 'Scale Factor' section on the right. The 'Section' list contains the following items:

N.	Name
1	Podium Column
2	Building Column
3	Beam
4	dummy 1 m pile
5	1 m pile

The 'Scale Factor' section contains the following values:

Scale Factor	Value
Area	1
Asy	1
Asz	1
Iyy	0.7
Izz	0.7
Weight	1

The 'Add / Replace' button is located at the bottom of the dialog box. The 'Close' button is also visible at the bottom right of the dialog box.

Annotations for steps 1 through 6 are as follows:

1. Click Properties
2. Click Section Stiffness Scale Factor
3. Click 1m Pile
4. Input Iyy = 0.7 Izz = 0.7
5. Click Add/Replace then Close
6. Activate All (Ctrl + A)



1. Click Boundary

2. Input Soil Type: Sand
 Ground Level: -3.2 m
 Pile Diameter: 1m
 Unit Weight of Soil: 19 kN/m³
 Earth Pressure Coeff. at rest (Ka): 0.4
 Coeff. Of Subgrade Reaction (Kh): 40000 kN/m³
 Internal Friction Angle: 30 deg
 Initial Soil Modulus: Dense

3. Using Select Single, Select nodes

4. Click Apply

Pile Spring Data
 Soil Type: Sand
 Ground Level: 1.5 m
 Pile Diameter (D): 1 m
 Unit Weight of Soil (γ): 19 kN/m³
 Earth Pressure Coeff. at rest (Ko): 0.4
 Coeff. of Subgrade Reaction (Kh): 40000 kN/m³
 Internal Friction Angle (φ): 30 (deg)
 Initial Soil Modulus (k1): Dense
 33930 kN/m³
 Apply Close

Tree Menu 2
 Menu Tables Group Works Report Seismic
 Domain: 1
 MAT [Elem=0]
 Material: 4
 1: M30
 2: M40
 3: Dummy
 4: m35
 Section: 5
 1: Podium Column
 2: Building Column
 3: Beam
 4: Dummy 1 m pile
 5: 1 m pile
 Section Stiffness Scale Factor
 5: 1 m pile [Group=Default; Ax=1; Ay=1; Az=1]
 Thickness: 2
 1: 0.23 (Shear wall & slab)
 2: 1.2 (raft)
 Boundaries
 Point Spring Supports: 10640
 Linear: 10640

Command Message
 Active Previous

None! U: -1, 33, -1.5 G: -1, 33, -1.5 kN m

NOTES

Direction: Enter the direction of a spring with respect to the node local coordinate system. Sample geotechnical data. Approval by a geotechnical engineer is required for use.



The screenshot shows the MIDAS/Gen software interface. The 'Boundary' menu is highlighted in the top toolbar. A red box and arrow point to the 'Boundary' menu with the text '1. Click Boundary'. The 'Add/Modify/Show Forces-Deformation Function' dialog box is open, showing a table of data and a graph of Force (F(y)) versus Displacement (d(x)).

Forces-Deformation Functions

Name	Type	Symm...
IbNsprFunc-1	Force	Symm.
IbNsprFunc-3	Force	Symm.
IbNsprFunc-5	Force	Symm.
IbNsprFunc-7	Force	Symm.
IbNsprFunc-9	Force	Symm.
IbNsprFunc-11	Force	Symm.
IbNsprFunc-13	Force	Symm.
IbNsprFunc-15	Force	Symm.

Add/Modify/Show Forces-Deformation Function

Name: IbNsprFunc-5

Type: ☒ Force ☐ Moment ☒ Symmetric ☐ Unsymmetric

d(x) /m	F(y) /kN
1	0.00000 0.00000
2	0.00069 97.1175
3	0.01666 454.902
4	0.03750 731.649
5	0.05833 731.649
6	0.07916 731.649
7	

The graph shows a symmetric, nonlinear force-displacement curve. The x-axis (d(x)) ranges from -0.09 to 0.085, and the y-axis (F(y)) ranges from -900 to 900. The curve is zero for negative displacements and increases non-linearly for positive displacements, reaching a plateau at approximately 730 kN for displacements greater than 0.04 m.

NOTES

Pile Spring Support will generate Force Deformation Function.

Lateral springs for the soils adjacent to piles are modeled as **symmetric nonlinear** elastic springs

Vertical springs for the soils adjacent to piles are modeled as **linear** elastic springs



The screenshot displays the MIDAS Gen 2026 software interface for a structural analysis project titled "Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS\Gen]". The main window shows a 3D model of a building frame with a pile foundation. The software is set to design using IS:800-2007, IS456:2000, SSRC79, and Eurocode3-1-3:06. The Design menu is open, showing options for Steel/Cold Formed Steel Design, Concrete Design, and SRC Design. The Member Assignment option is selected in the Design menu. The Tree Menu 2 on the right shows the model hierarchy, including Base, Nodes, Elements, Properties, Section, and Boundaries. The Properties section is expanded, showing the material (M30, M40, Dummy, M35) and section (Podium Column, Building Column, Beam, dummy 1 m pile, 1 m pile) for the selected member. The Message Window at the bottom shows the end of the design process by IS456:2000, with a total design/checking time of 26.96 seconds.

1. Click Design

2. Click Member Assignment

3. Double click 1 m Pile

Message Window

*** End Design by IS456:2000
Total Design/Checking Time... 26.96 [sec]



Gen 2026 - [C:\Users\Naveen\Downloads\RC Building check 1 *] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

IS:800:2007 IS456:2000 SSRC79 Eurocode3-1-3:06

Steel Design RC Design SRC Design Cold Formed Steel Design

Section for Design Steel/Cold Formed Steel Design Steel/Cold Formed Steel Design Concrete Design SRC Design Concrete Design SRC Design Perform Batch Design

General Design Parameter General Design Design Section Result Forces/Properties Batch

Tree Menu General Steel Concrete SRC Cold Fo...

Member Assignment

Option

☒ Add/Replace ☐ Delete

Member

☐ Allow Single Element Member

Assign Type

☒ Manual ☐ Automatic

Selection Type

☐ All ☒ By Selection

1. Tick Manual and By Selection

Idx Elem. List

107... 10769, 10665, 1056...

107... 10770, 10666, 1056...

107... 10771, 10667, 1056...

107... 10772, 10668, 1056...

107... 10773, 10669, 1056...

107... 10774, 10670, 1056...

107... 10775, 10671, 1056...

107... 10776, 10672, 1056...

107... 10777, 10673, 1056...

107... 10778, 10674, 1057...

107... 10779, 10675, 1057...

107... 10780, 10676, 1057...

107... 10781, 10677, 1057...

107... 10782, 10678, 1057...

107... 10783, 10679, 1057...

107... 10784, 10680, 1057...

107... 10785, 10681, 1057...

107... 10786, 10682, 1057...

107... 10787, 10683, 1057...

107... 10788, 10684, 1058...

107... 10789, 10685, 1058...

Apply

2. Click Apply

Message Window

*** End Design by IS456:2000

Total checking Time... 26.96 [sec]

Tree Menu 2 Menu Tables Group Works Report Seismic

Base:3F

Base:4F

Base:5F

Base:6F

Base:7F

Base:8F

Base:9F

Base:10F

Base:11F

Base:12F

Base:13F

Base:14F

Base:15F

Base:16F

Base:Roof

Nodes : 9466

Elements : 10872

Beam : 4016

Plate : 6728

Wall : 128

Properties

Material : 4

1 : M30

2 : M40

3 : Dummy

4 : M35

Section : 5

1 : Podium Column

2 : Building Column

3 : Beam

4 : dummy 1 m pile

5 : 1 m pile

Section Stiffness Scale Factor

5 : 1 m pile [Group=Default ; Ax=1 , Ay=1 , Az=1 ; I

Thickness : 1

1 : 0.23 (Shear wall & slab)

Boundaries

Supports :

Point Spring Supports : 12154

Linear : 8618

Type 1 [SSS000 000000]

Type 2 [SSS000 000000]

Type 3 [SSS000 000000]

Type 4 [SSS000 000000]

Command Message Analysis Message

For Help, press F1

Node-9381 U: 0, 0, -20 G: 0, 0, -20

kN m



1. Click RC Design -> Design Code

2. Pick IS456 : 2000

3. Tick Apply Special Provisions for Seismic Design

3. Click OK

Concrete Design Code

Design Code : IS456:2000

☒ Apply Special Provisions for Seismic Design

Ductile Design Code for Seismic Zone III, IV or V.

☐ Apply IS13920:1993.

☒ Apply IS13920:2016.

☒ Consider strong column-weak beam ratio

☐ Consider strong column-weak beam on last floor

☐ Consider Boundary Element design

☐ Method-A ☒ Method-B

Shear for Design

Method ☒ MAX(Vu1, Vu2) ☐ MIN(Vu1, Vu2) ☐ Vu1 ☐ Vu2

Vu1 , Vg + a1*SUM(Mu)/L , a1 = 1.4

Vu2 , Vg + a2*Veq , a2 = 1

MAT Properties

Factor of Safety for Soil : 3

Modulus of Subgrade Reaction : 15000 kN/m³

Moment Redistribution Factor for Beam : 1

☐ Torsion Design

P-M Curve Calculation Method

☐ Keep P Constant

☒ Keep M/P Constant

OK

Message Window

Command Message Analysis Message

Node:5810 U: 5.949, 24.581, 0 G: 5.949, 24.581, -1.5 kN m



1. Click Design Criteria for Rebar

2. Input desired criteria for Pile Design

3. Click OK

Design Criteria for Rebars

For Beam Design

Main Rebar : P20 Rebar...

Stirrups : P10 Arrangement : 2

Side Bar : P12

m dB : 0 m

☒ Consider Spacing Limit for Main Rebar

Spliced Bars : ☒ None ☐ 50% ☐ 100%

For Column Design

Main Rebar : P20 Rebar...

Ties/Spirals : P10 Arrangement : Y: 4

do : 0.062 m Z: 4

☒ Consider Spacing Limit for Main Rebar

Spliced Bars : ☒ None ☐ 50% ☐ 100%

For Brace Design

Main Rebar : P20 Rebar...

Ties/Spirals : P10 Arrangement : Y: 2

do : 0 m Z: 2

☒ Consider Spacing Limit for Main Rebar

Spliced Bars : ☒ None ☐ 50% ☐ 100%

For Shear Wall Design

Vertical Rebar : P12 Rebar...

Horizontal Rebar : P10 End Rebar From : P10

Boundary Element Horz. Rebar : P10

Boundary Element Horz. Rebar Space : 0.2 m

Boundary Element Vert. Rebar Space : 0.1 m

de : 0 m dw : 0 m

Input Additional Wall Data...

OK Close

Rebar Information

CHK	Name	Dia (mm)	Area (mm ²)	Dia(Out) (mm)	Weight (kg/m)
☐	P6	0.0060	0.0000	0.0060	0.0022
☐	P8	0.0080	0.0001	0.0080	0.0039
☐	P10	0.0100	0.0001	0.0100	0.0061
☐	P12	0.0120	0.0001	0.0120	0.0087
☐	P16	0.0160	0.0002	0.0160	0.0155
☒	P18	0.0180	0.0003	0.0180	0.0196
☒	P20	0.0200	0.0003	0.0200	0.0242
☒	P22	0.0220	0.0004	0.0220	0.0293
☐	P25	0.0250	0.0005	0.0250	0.0378
☐	P28	0.0280	0.0006	0.0280	0.0474
☐	P32	0.0320	0.0008	0.0320	0.0619
☐	P36	0.0360	0.0010	0.0360	0.0784
☐	P40	0.0400	0.0013	0.0400	0.0968

OK Close

gn By IS456:2000
sign/Checking Time... 26.96 [sec]

Tree Menu Task Pane Command Message Analysis Message

For Help, press F1

Node-8164 U: 8, 28, -8 G: 8, 28, -8

kn m



The screenshot displays the MIDAS/Gen software interface. The top menu bar includes View, Structure, Node/Element, Properties, Boundary, Load, Analysis, Results, Pushover, Design, Seismic Performance, Query, and Tools. The Design menu is currently active, showing options for Section, Steel/Cold Formed Steel Design, Concrete Design, and SRC Design. A red box highlights the 'Analysis' button in the Design menu. A red arrow points from the '1. Run Analysis' label to this button.

In the center of the screen, a 3D model of a building structure is shown. A dialog box titled 'Check Changed Story Information' is open, displaying the message: 'Entered story data does not match to the current model. Please check the story data if it has been correctly entered.' The dialog box has three buttons: 'Update Story Data ...', 'Continue', and 'Cancel'. A red box highlights the 'Continue' button, and a red arrow points from the '2. Click Continue' label to it.

On the left side, there is a 'Member Assignment' panel with options for 'Option' (Add/Replace, Delete), 'Member' (Allow Single Element Member), 'Assign Type' (Manual, Automatic), and 'Selection Type' (All, By Selection). Below this is a table with columns 'Idx' and 'Elem. List'.

On the right side, there is a 'Tree Menu 2' panel showing a hierarchical list of elements and properties, including 'Base 3F' through 'Base 16F', 'Base Roof', 'Nodes : 9466', 'Elements : 10872', 'Beam : 4016', 'Plate : 6728', 'Wall : 128', 'Properties', 'Material : 4', '1 : M30', '2 : M40', '3 : Dummy', '4 : M25', 'Section : 5', '1 : Podium Column', '2 : Building Column', '3 : Beam', '4 : dummy 1 m pile', '5 : 1 m pile', 'Section Stiffness Scale Factor', '5 : 1 m pile [Group=Default ; Ax=1 ; Ay=1 ; Az=1 ;', 'Thickness : 1', 'Boundaries', 'Supports', 'Point Spring Supports : 12154', 'Linear : 8618', 'Type 1 [SSS000 000000]', 'Type 2 [SSS000 000000]', 'Type 3 [SSS000 000000]', and 'Type 4 [SSS000 000000]'.

At the bottom, a 'Message Window' displays the text: 'The process of Data Preparation for Analysis/Post-processing Completed.'

Gen 2026 - [E:\1. MIDAS INTERN\23. Dev Bhaiya\RC Building check 1] - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

General Design Parameter Steel Design RC Design SRC Design Cold Formed Steel Design

Section Design Steel/Cold Formed Steel Design Concrete Design SRC Design

Result Forces/Properties Batch

Perform Batch Design

Tree Menu

Menu Tables Group Works Rep... Seis...

Works

Analysis Control Data

Eigenvalue Analysis [Type=Eigenvalue]

Structures

Stories : 17

Base: 1F

Base: 2F

Base: 3F

Base: 4F

Base: 5F

Base: 6F

Base: 7F

Base: 8F

Base: 9F

Base: 10F

Base: 11F

Base: 12F

Base: 13F

Base: 14F

Base: 15F

Base: 16F

Base: Roof

Nodes : 9466

Elements : 10872

Beam : 4016

Plate : 6728

Wall : 128

Properties

Material : 4

1 : M30

2 : M40

3 : Dummy

4 : M35

Section : 5

1 : Podium Column

2 : Building Column

3 : Beam

4 : dummy 1 m pile

5 : 1 m pile

Section Stiffness Scale Factor

5 : 1 m pile [Group=Default; Ax=1; Ay=1; Az=1; I

Thickness : 1

1 : 0.23 (Shear wall & slab)

Boundaries

Point Spring Supports : 12154

Linear : 8618

Type 1 [SSS000 000000]

Type 2 [SSS000 000000]

Type 3 [SSS000 000000]

Type 4 [SSS000 000000]

Batch Design Dialog

Analysis

Steel Code Checking

Concrete Beam Design

Concrete Column Design

Concrete Brace Design

Concrete Wall Design

Concrete PS-Beam Design

Concrete Ductile Design

SRC Beam Checking

Cold Formed Steel Code Checking

Concrete Beam Checking

Concrete Column Checking

Concrete Brace Checking

Concrete Wall Checking

Concrete PS-Beam Checking

SRC Column Checking

Status

Check options and Press 'Run' button !

Run Close

3. Click Run

2. Tick Concrete Column Design

1. Double click 1 m Pile

Message Window

TOTAL SOLUTION TIME... 106.68 [SEC]

Tree Menu 2

Menu Tables Group Works Report Seismic

Base: 3F

Base: 4F

Base: 5F

Base: 6F

Base: 7F

Base: 8F

Base: 9F

Base: 10F

Base: 11F

Base: 12F

Base: 13F

Base: 14F

Base: 15F

Base: 16F

Base: Roof

Nodes : 9466

Elements : 10872

Beam : 4016

Plate : 6728

Wall : 128

Properties

Material : 4

1 : M30

2 : M40

3 : Dummy

4 : M35

Section : 5

1 : Podium Column

2 : Building Column

3 : Beam

4 : dummy 1 m pile

5 : 1 m pile

Section Stiffness Scale Factor

5 : 1 m pile [Group=Default; Ax=1; Ay=1; Az=1; I

Thickness : 1

1 : 0.23 (Shear wall & slab)

Boundaries

Point Spring Supports : 12154

Linear : 8618

Type 1 [SSS000 000000]

Type 2 [SSS000 000000]

Type 3 [SSS000 000000]

Type 4 [SSS000 000000]

For Help, press F1

Node-940 U: 44, 24, 48 G: 44, 24, 48 kN m



Gen 2026 - [E:\1. MIDAS INTERN\23. Dev Bhaiya(RC Building check 1) - [MIDAS/Gen]

View Structure Node/Element Properties Boundary Load Analysis Results Pushover Design Seismic Performance Query Tools

IS-800-2007 IS456:2000 SSRC79 Eurocode3-1-3:06 Section for Design Steel/Cold Formed Steel Design Steel/Cold Formed Steel Design Concrete Design SRC Design Perform Batch Design

Tree Menu Menu Tables Group Works Rep... Sels

Works Analysis Control Data Eigenvalue Analysis [Type=Eigenvalue]

Batch Design Dialog

1. Tick Checkbox

Analysis

Steel Check Design

Concrete Beam Design Concrete Beam Checking

Concrete Column Design Concrete Column Checking

Concrete Brace Design Concrete Brace Checking

Concrete Wall Design Concrete Wall Checking

Concrete PS-Beam Design Concrete PS-Beam Checking

Concrete Ductile Design

Concrete SRC Beam Checking Concrete SRC Column Checking

Cold Formed Steel Check Design

Status

Successfully performed batch design !

2. Click Graphic

Code : IS456:2000 Unit : KN , m

Sorted by Member Property

MEMB	SEL	Section	Bc	Hc	fck	f _{yk}	CHK
0		1 m pile	35000.0	415000			
5		0.0000.1.0000.	20.000	415000			PM

Preview Window

No. 5

Print Print All Close Save

1. Design Condition

Design Code : IS456:2000 UNIT SYSTEM : KN, m

Member Number : 10973 (Pile, 10973 (Shear))

Material Data : f_{ck} = 35000, f_{yk} = 415000 KPa

Column Height : 20 m

Section Property : 1 m pile (No. 5)

Rebar Pattern : S2 - 0 - P22 Ast = 0.0197652 m² (pat = 0.025)

2. Axial and Moments Capacity

Load Combination : 22 (Pos. J)

Concentric Max. Axial Load : P (kN) = 16214.6 kN

Axial Load Ratio : Pu / P.Rat = 751.20 / 87.5702 = 85.775 > 1.000 N.O.

Moment Ratio : M / M.Rat = 213007 / 2485.94 = 85.685 > 1.000 N.O.

M_y / M.Rat_y = 150505 / 1756.44 = 85.688 > 1.000 N.O.

M_z / M.Rat_z = 150733 / 1759.21 = 85.682 > 1.000 N.O.

M-N Interaction Diagram

3. Shear Capacity

[END]

	Y (LCB : 42, POS : J)	X (LCB : 42, POS : J)
Applied Shear Force (Vu)	199.069 kN	199.069 kN
Design Shear Strength (V _o V _s)	1078.89 + 305.910 = 1388.80 kN	1078.89 + 305.910 = 1388.80 kN
Shear Ratio	0.143 < 1.000 - O.K.	0.143 < 1.000 - O.K.
Asv-H req	0.00097 m/m, 2-P10 @160	0.00097 m/m, 2-P10 @160

[MODULE]

	Y (LCB : 30, POS : I2)	X (LCB : 30, POS : I2)
Applied Shear Force (Vu)	49.5064 kN	49.5064 kN
Design Shear Strength (V _o V _s)	744.849 + 305.910 = 1054.76 kN	744.849 + 305.910 = 1054.76 kN
Shear Ratio	0.047 < 1.000 - O.K.	0.047 < 1.000 - O.K.
Asv-H req	0.00097 m/m, 2-P10 @160	0.00097 m/m, 2-P10 @160

Tree Menu 2

Menu Tables Group Works Report Seismic

2 : Building Column

3 : Beam

4 : dummy 1 m pile

5 : 1 m pile

Section Stiffness Scale Factor

5 : 1 m pile [Group=Default, Ax=1, Ay=1, Az=1;]

Thickness : 1

1 : 0.23 (Shear wall & slab)

Boundaries

Point Spring Supports : 13153

Linear : 8743

Type 1 [SSS000 000000]

Type 2 [SSS000 000000]

Type 3 [SSS000 000000]

Type 4 [SSS000 000000]

Type 5 [SSS000 000000]

Type 6 [SSS000 000000]

Type 7 [SSS000 000000]

Type 8 [SSS000 000000]

Type 9 [SSS000 000000]

Type 10 [SSS000 000000]

Type 11 [SSS000 000000]

Type 12 [SSS000 000000]

Type 13 [SSS000 000000]

Type 14 [SSS000 000000]

Type 15 [SSS000 000000]

Type 16 [SSS000 000000]

Type 17 [SSS000 000000]

Type 18 [SSS000 000000]

Type 19 [SSS000 000000]

Type 20 [SSS000 000000]

Type 21 [SSS000 000000]

Type 22 [SSS000 000000]

Type 23 [SSS000 000000]

Type 24 [SSS000 000000]

Type 25 [SSS000 000000]

Type 26 [SSS000 000000]

Type 27 [SSS000 000000]

Type 28 [SSS000 000000]

Type 29 [SSS000 000000]

Type 30 [SSS000 000000]

Type 31 [SSS000 000000]

Type 32 [SSS000 000000]

Type 33 [SSS000 000000]

Type 34 [SSS000 000000]

Type 35 [SSS000 000000]

Type 36 [SSS000 000000]

Type 37 [SSS000 000000]

Type 38 [SSS000 000000]

Type 39 [SSS000 000000]

Type 40 [SSS000 000000]

Type 41 [SSS000 000000]

Type 42 [SSS000 000000]

Message Window

Reading Frame Element Result - Concrete Design

Tree Menu Task Pane

For Help, press F1

Node-1 U: 0.0, 3 G: 0.0, 3

KN m

