Earthquake on Five-Story Building
**Outline**

1 **Description** 3

2 **Finite Element Model** 4
   2.1 Geometry 6
   2.2 Properties 8
   2.3 Boundary Conditions 11
   2.4 Loads 12
   2.5 Mesh 13

3 **Eigenvalue Analysis and Direct Response Analysis** 15
   3.1 Commands 15
   3.2 Results 21

4 **Modal Response Analysis** 41
   4.1 Commands 41
   4.2 Results 46

5 **Response Spectrum Analysis** 52
   5.1 Commands 53
   5.2 Results 60

**Appendix A Additional Information** 78
1 Description

The model of this tutorial represents a five-story building with a cross-section as shown in Figure 1. The dimensions of the cross-section are small in comparison to the length of the building. Therefore we may assume a plane strain situation. The building is subjected to a base acceleration $\ddot{u}_{\text{base}} = 1 \text{ m/s}^2$ in the horizontal $X$ direction. First we create the model. Then we perform an eigenvalue analysis, followed by two types of frequency response analyses: direct response (Analysis1) and modal response (Analysis2). Furthermore, we apply an earthquake spectrum in a response spectrum analysis (Analysis3).

Concrete:

\[ E = 2.2 \times 10^{10} \text{ N/m}^2 \]
\[ \nu = 0.2 \]
\[ \rho = 2400 \text{ kg/m}^3 \]
2 Finite Element Model

Since plane strain conditions can be applied, quadratic, three-noded, infinite shell elements are used to construct the model of the building frame. This is done in the two-dimensional \( XY \) coordinate space.
We select the units millimeter and ton.

**Geometry browser** → Reference system → Units  [Fig. 3]
**Property Panel**  [Fig. 4]

![Geometry browser](image1)

![Property panel - units](image2)

Figure 3: Geometry browser

Figure 4: Property panel - units
2.1 Geometry

Seven lines are required to build the model. We draw the line for the first floor and it copy it four times to create the other floors and the roof.

Main menu ➔ Geometry ➔ Create ➔ Add line [Fig. 5] [Fig. 6] [Fig. 7]
Next we create the two walls. We draw the line for the left wall and copy it to create the right wall.

**Main menu → Geometry → Create → Add line**  
**[Fig. 8]**

**Main menu → Geometry → Modify → Array copy**  
**[Fig. 9] [Fig. 10]**

---

*Figure 8: Add left wall*

*Figure 9: Copy wall*

*Figure 10: Model geometry*
2.2 Properties

Material and physical properties are assigned first to the floors and roof. We define an elastic isotropic material named Concrete with Young’s modulus $E = 22000 \text{ N/mm}^2$, Poisson’s ratio $\nu = 0.2$ and mass density $\rho = 2.4E - 09 \text{ T/mm}^3$ (2400 kg/m$^3$).

![Main menu](geometry.png)

**Main menu** ➔ Geometry ➔ Assign ➔ Shape Properties ➔ [Fig. 11]

**Shape Properties** ➔ Material ➔ Add material ➔ Edit material ➔ [Fig. 12][13]

---

**Figure 11:** Property assignment floors and roof

**Figure 12:** Add new material

**Figure 13:** Edit material
As a physical property, the thickness of the floors and roof are specified as *Thick* with $t = 400$ mm. The shape of the infinite shell elements are defined as flat.

![Figure 14: Floor and roof thickness *Thick*](https://dianafea.com)
The same operations are performed for the set of walls. However, since the material properties are the same, we do not need to define a new material. The thickness of the walls are specified as Thin with $t = 150$ mm. The shape of the infinite shell elements are defined as flat.

**Figure 15**: Property assignment walls

**Figure 16**: Wall thickness Thin
2.3 Boundary Conditions

The bottom end of the vertical lines are clamped, i.e. translations and rotations are restricted.

Figure 17: Attach supports

Figure 18: Clamped base
2.4 Loads

A base excitation with a horizontal acceleration of $\ddot{u}_{\text{base}} = 1000 \text{ mm/s}^2$ in global X direction is applied. Base excitation loads are applied in the direction of the corresponding supports, i.e. at the bottom end of the vertical lines in this model.

Main menu ➔ Geometry ➔ Assign ➔ Add global loads [Fig. 19]
2.5 Mesh

The mesh is defined by dividing the horizontal lines into ten equally sized line segments and the vertical lines into fifteen equally sized line segments [Fig. 20] [Fig. 21]. We then generate the mesh. We shrink the elements and present the node ids to get a better idea of the mesh [Fig. 22].

---

**Main menu ➔ Geometry ➔ Assign ➔ Mesh properties**  [Fig. 20]  [Fig. 21]

**Main menu ➔ Geometry ➔ Generate mesh**  

**Main menu ➔ Mesh ➔ Mesh ➔ Shrunken shading**  

**Main menu ➔ Viewer ➔ Selection mode ➔ Node selection**  

**Graphics window ➔ Select all ➔ Show ids**

---

![Figure 20: Set mesh properties - horizontal lines](image1)

![Figure 21: Set mesh properties - vertical lines](image2)
Figure 22: Mesh - shrunk elements and node labels
3 Eigenvalue Analysis and Direct Response Analysis

We perform a free vibration eigenvalue analysis in order to check the model, followed by a direct frequency response analysis, with which we determine the response of the model to a base excitation.

3.1 Commands

First we set up the commands for the free vibration eigenvalue analysis.
We ask for ten eigenmodes to be determined in the structural eigenvalue analysis. The default output settings will be used for the eigenmodes.

![Analysis browser](image)

**Analysis browser** ➔ Analysis1 ➔ Structural eigenvalue ➔ Execute eigenvalue analysis ➔ Edit properties [Fig. 25]

**Properties - EXECUT** ➔ Number of eigenfrequencies ➔ 10 [Fig. 26]

![Figure 25: Analysis browser - Execute eigenvalue analysis](image)

![Figure 26: Edit properties - Execute eigenvalue analysis](image)
Next, we define the commands for the structural direct frequency response.

**Analysis browser** ➔ **Analysis1** ➔ **Add command** ➔ **Structural direct response**  

![Analysis browser](Fig. 27)  
![Add command](Fig. 28)
A range of excitation frequencies is specified from 0 to 10 Hz in steps of 0.1 Hz for the direct frequency response analysis. This can be input as 0–10(0.1). In a direct frequency response analysis the complex system of equations needs to be solved for each excitation frequency, because the system is frequency dependent. Therefore, direct response spectrum analyses are computationally more heavy than modal response analyses, but are needed in case of frequency dependent properties or when there is a considerable amount of damping or discrete dampers in the model.
The last step in defining the analysis commands is specifying the output. We select the displacements, velocities and accelerations.

**Analysis browser** ➔ Analysis1 ➔ Structural direct response ➔ Output frequency response analysis ➔ Edit properties

Figure 31: Analysis browser - Output frequency response analysis

Figure 32: Edit properties - Output
We select the Amplitude/Phase angle representation for each output item, because, in this manner, the peaks in the displacement amplitudes can be shown and are expected to be in agreement with the eigenfrequencies found in the eigenvalue analysis.
3.2 Results

After termination of the analysis we assess the results. For an eigenvalue analysis these typically are the eigenfrequencies and the eigenmodes. The Messages dialog displays the eigenfrequencies and the corresponding relative errors. The small relative errors indicate that all eigenfrequencies could be determined with high accuracy.

![Eigenfrequencies](https://dianafea.com)

We see that eigenfrequencies are in a range from about 1 to 8 Hz, which is approximately within the frequencies of a typical earthquake spectrum.
Now the ten eigenmodes are displayed. Note that modes 1, 2, 3, 7, and 10 are dominated by the deformation of the walls, and modes 4, 5, 6, 8, and 9 by the deformation of the floors [Fig. 37 to 46].
Figure 37: Eigenmode 1 - 1.04 Hz

Analysis
Mode 1, Eigen frequency 1.043 Hz
Displacements ΔtX
min: 0.00mm max: 1.00mm
Figure 38: Eigenmode 2 - 3.12 Hz

Mode 2, Eigen frequency 3.1192 Hz
Displacements DtX
min: -0.94 mm max: 1.00 mm
Figure 39: Eigenmode 3 - 5.12 Hz

Analysis
Mode 3, Eigen frequency 5.1244 Hz
Displacements DTX
min: -0.92mm max: 1.00mm
Figure 40: Eigenmode 4 - 6.13 Hz

Analysis
Mode 4, Eigen frequency 6.1348 Hz
Displacements ΔtX
min: -0.14 mm max: 0.14 mm
Figure 41: Eigenmode 5 - 6.48 Hz
Figure 42: Eigenmode 6 - 6.79 Hz
Analysis
Mode 7, Eigen frequency 6.9058 Hz
Displacements ΔtX
min: -0.72 mm max: 1.00 mm

Figure 43: Eigenmode 7 - 6.90 Hz
Analysis
Mode 8, Eigen frequency 7.0469 Hz
Displacements ΔtX
min: 9.44e-02 mm max: 9.44e-02 mm

Figure 44: Eigenmode 8 - 7.05 Hz
Analysis
Mode 9, Eigen frequency 7.1874 Hz
Displacements ΔtX
min: -7.93e-02 mm max: 7.93e-02 mm

Figure 45: Eigenmode 9 - 7.19 Hz
Figure 46: Eigenmode 10 - 8.17 Hz
A graph of the horizontal displacement (mm) of node 44 (connection of third floor and left wall) and node 86 (connection of roof and left wall) as a function of the excitation frequency (Hz) is generated. The location of these nodes in the mesh is shown in Figure 22.

Figure 47: Results browser
The graphs show a maximum amplitude of 275 mm (node 44) and 350 mm (node 86), at an excitation frequency of approximately 1 Hz which corresponds to the first eigenfrequency. The other amplitudes also correspond with eigenfrequencies, all of them representing an eigenmode dominated by deformation of the walls: 1, 2, 3, 7, and 10.

Figure 48: Horizontal displacement response of left wall
A graph of the vertical displacement (mm) of node 49 (mid node of the third floor) and node 91 (mid node of the roof) as a function of the excitation frequency (Hz) is generated. See the location of the selected nodes in the mesh in Figure 22.

Figure 49: Results browser
Both graphs show very small displacements with a maximum amplitude in the order of $10^{-11}$ mm at an excitation frequency of approximately 6.8 Hz which presumably corresponds to the fifth and/or sixth eigenmode, dominated by floor deformation. Please note that the exact results of these graphs may vary on different machines. This is due to the small amplitudes of the displacements.

The conclusion of the direct response analysis is that the vertical displacements of the floors are very small compared to the horizontal displacements of the walls. The eigenmodes dominated by the deformation of the walls are, by far, the most significant ones for the horizontal base acceleration load.
A graph of the horizontal velocity (mm/s) of node 44 (connection of third floor and left wall) and node 86 (connection of roof and left wall) as a function of the excitation frequency (Hz) is generated. See the location of the selected nodes in the mesh in Figure 22.

Figure 51: Results browser
Figure 52: Horizontal velocity response of left wall

Note that peak amplitudes occur at the same frequencies as for the displacements (compare with Figure 48). Compared to the displacements the peaks are relatively higher for higher excitation frequencies.
A graph of the horizontal accelerations (mm/s²) of node 44 (connection of third floor and left wall) and node 86 (connection of roof and left wall) as a function of the excitation frequency (Hz) is generated. See the location of the selected nodes in the mesh in Figure 22.

Figure 53: Results browser
Again, note that peak amplitudes occur at the same frequencies as for the displacements [Fig. 48]. Compared to the displacements and velocities, the peaks are relatively higher for higher excitation frequencies.
4 Modal Response Analysis

For comparison with the direct response analysis, as performed in the previous section, we will now determine the response of the model to a base excitation via a modal response analysis. Compared to the direct response analysis, modal response analysis has the advantage of not requiring to set up the element matrices for the complex system for each excitation frequency; they are only set up once. The disadvantage is that a modal response analysis requires the preliminary determination of eigenvalues and eigenmodes and is therefore limited to modal damping and cannot be used for models with a considerable amount of damping or discrete dampers. Frequency dependent properties cannot be taken into account in a modal response analysis.

4.1 Commands

| Main menu | Analysis | Add analysis | Fig. 55 |
| Analysis browser | Analysis2 | Add command | Structural modal response | Fig. 56 | Fig. 57 |

Figure 55: Analysis browser

<table>
<thead>
<tr>
<th>Phased</th>
</tr>
</thead>
<tbody>
<tr>
<td>Structural linear static</td>
</tr>
<tr>
<td>Checking design</td>
</tr>
<tr>
<td>Stiffness adaptation</td>
</tr>
<tr>
<td>Structural eigenvalue</td>
</tr>
<tr>
<td>Structural modal response</td>
</tr>
<tr>
<td>Structural direct response</td>
</tr>
<tr>
<td>Structural response spectrum</td>
</tr>
<tr>
<td>Hybrid frequency-time domain</td>
</tr>
<tr>
<td>Structural nonlinear</td>
</tr>
<tr>
<td>Strength reduction</td>
</tr>
<tr>
<td>Engineering liquefaction</td>
</tr>
<tr>
<td>Engineering creep</td>
</tr>
<tr>
<td>Structural stability</td>
</tr>
<tr>
<td>Nonlinear vibration</td>
</tr>
<tr>
<td>Staged construction</td>
</tr>
</tbody>
</table>

Figure 56: Add command

Figure 57: Analysis browser
We toggle off the output of the eigenvalue analysis and ask for ten eigenmodes to be determined.

**Analysis browser** → Analysis2 → Structural modal response → Eigenvalue analysis → Output eigenvalue analysis → Toggle off

**Properties - EXECUT** → Number of eigenfrequencies → 10

---

Figure 58: Analysis browser - Output eigenvalue analysis
Figure 59: Analysis browser - Execute eigenvalue analysis
Figure 60: Edit properties - Execute eigenvalue analysis
We assign a damping coefficient of 0.01, specify a range of excitation frequencies from 0 to 10 Hz in steps of 0.1 Hz, which can be input again as 0–10(0.1), and ask to return the eigenmodes that are dominated by deformation of the walls, i.e., eigenmode 1, 2, 3, 7 and 10, because from the direct frequency analysis we know that the floors will not show large deformations due to the horizontal earthquake.

---

3Modal damping is required if any excitation frequency is equal to an eigenfrequency. The real damping factor of the structure is often not known. Normally, values are chosen between 0 and 10% of the critical damping factor. Because we do not want to overestimate the damping we choose for 1% of the critical damping.
The last step in defining the analysis commands is specifying the output: we select the displacements, velocities and accelerations.

Analysis browser ➔ Analysis2 ➔ Structural modal response ➔ Frequency response analysis ➔ Output frequency response analysis ➔ Edit properties

Figure 63: Analysis tree - Output frequency response analysis

Figure 64: Edit properties - Output frequency response analysis
We select the Amplitude/Phase angle representation for each output item, because, in this manner, the peaks in the displacement amplitudes can be shown and are expected to be in agreement with the eigenfrequencies found in the eigenvalue analysis, similar to the direct frequency response analysis.

DianaIE

Results Selection ➔ Add... DISPLA TOTAL TRANSL GLOBAL  [Fig. 65]
< Repeat for VELOCI TOTAL TRANSL GLOBAL and ACCELE TOTAL TRANSL GLOBAL >

Results Selection ➔ DISPLA TOTAL TRANSL GLOBAL ➔ Properties  [Fig. 66]
< Repeat for VELOCI TOTAL TRANSL GLOBAL and ACCELE TOTAL TRANSL GLOBAL >

Main menu ➔ Analysis ➔ Run selected analysis

Figure 65: Edit properties - Results Selection

Figure 66: Edit properties - Result Item Properties
4.2 Results

We plot graphs of the frequency response for displacements, velocities and accelerations of the nodes at the junction of the left wall with the third floor and the left wall with the roof. A graph of the horizontal displacement (mm) of node 44 (connection of third floor and left wall) and node 86 (connection of roof and left wall) as a function of the excitation frequency (Hz) is generated first. See the location of the selected nodes in the mesh in Figure 22.

Figure 67: Results browser
The maximum amplitudes near 1 Hz for the modal response analysis with 1% modal damping [Fig. 68] are considerably larger than those obtained via direct response analysis without damping [Fig. 48]. This may appear strange, but for the direct frequency response analysis there is an asymptote at the eigenfrequencies leading to very sharp spikes in the response in a small frequency range around these eigenfrequencies. While in the modal response analysis with modal damping, there are no longer asymptotes at the eigenfrequencies, i.e. the amplitudes are limited to a certain value, but the peak is spread over a larger frequency range around these eigenfrequencies. Since we execute at discrete excitation frequencies, we may observe, that we see larger amplitudes for the modal frequency response with damping than for the direct frequency response without damping at specific excitation frequencies as can be seen in this specific tutorial.

Figure 68: Horizontal displacement response of left wall
A graph of the horizontal velocity (mm/s) of node 44 and node 86 as a function of the excitation frequency (Hz) is generated. See the location of the selected nodes in the mesh in Figure 22.
The maximum amplitudes of the velocities [Fig. 70] are also a bit lower than those obtained via direct response analysis [Fig. 52], except for the peaks at around 1 (Hz) who reach a value of approximately 7200 and 9500 (mm/s).

Figure 70: Horizontal velocity response of left wall
A graph of the horizontal accelerations (mm/s²) of node 44 and node 86 as a function of the excitation frequency (Hz) is generated.

Figure 71: Results browser
The maximum amplitudes of the accelerations [Fig. 72] are considerably lower than those obtained via direct response analysis [Fig. 54] except for the peak near 1 Hz. Apparently, modal damping and/or floor modes affect the acceleration more than the displacement and the velocity. Only for the peak near 1 Hz we see a higher acceleration due to the larger frequency range around the peak of the damped first eigenfrequency for the modal frequency response with 1% modal damping as explained earlier.

Figure 72: Horizontal acceleration response of left wall
5 Response Spectrum Analysis

In this last section we demonstrate a response spectrum analysis of the frame. As is done for the modal response analysis, we only consider the eigenmodes dominated by deformation of the walls. Before the analysis commands are specified, an addition needs to be made to the model: a typical earthquake spectrum is applied which in DIANA relates frequencies to load amplification factors for the base excitation load.

---

Figure 73: Geometry browser

Figure 74: Edit frequency dependency factors
5.1 Commands

We add a new analysis.

Main menu → Analysis → Add analysis [Fig. 75]
Analysis browser → Analysis3 → Add command → Structural response spectrum [Fig. 76]
Analysis browser → Analysis3 → Structural response spectrum → Rename → Structural response spectrum ABS [Fig. 77]
Analysis browser → Analysis3 → Add command → Structural response spectrum [Fig. 76]
Analysis browser → Analysis3 → Structural response spectrum → Rename → Structural response spectrum SRSS [Fig. 77]
We ask for ten eigenmodes to be determined.

**Analysis browser** ➔ Analysis3 ➔ Structural response spectrum ➔ Eigenvalue analysis ➔ Execute eigenvalue analysis ➔ Edit properties 📜 [Fig. 78]

**Properties - EXECUT** ➔ Number of eigenfrequencies ➔ 10 [Fig. 79]

---

**Figure 78**: Analysis browser - Execute eigenvalue analysis

**Figure 79**: Edit properties - Execute eigenvalue analysis
For the output of the eigenvalue analysis, we specify the eigenmodes that are dominated by deformation of the walls, i.e. eigenmode 1, 2, 3, 7 and 10.

Figure 80: Analysis browser - Output eigenvalue analysis

Figure 81: Edit properties - Output
For the output of the response spectrum analysis, we select the displacements, residual forces and the Cauchy total stresses. The maximum modal quantity values are superposed according to the absolute rule, that is the sum of absolute maximum values obtained for each eigenmode.
Figure 82: Analysis browser - Output response spectrum analysis

Figure 83: Edit properties - Output

Figure 84: Edit properties - Results selection
Next to the absolute superposition, we also want to get the same results items where the maximum modal quantity values are superposed according to the Square Root of the Summed Squares (SRSS). These results will be defined in another output block.
Earthquake on Five-story Building | https://dianafea.com
5.2 Results

Eigenmodes 1, 2, 3, 7, and 10 are presented in Figure 89 to Figure 93.

Figure 88: Results browser
Figure 89: Displacement response individual modes - Eigenmode 1 - 1.04 Hz
Figure 90: Displacement response individual modes - Eigenmode 2 - 3.12 Hz
Figure 91: Displacement response individual modes - Eigenmode 3 - 5.12 Hz
Analysis3
Mode 7, Eigen frequency 6.8958 Hz
Displacements ΔX
min: -0.72mm max: 1.00mm

Figure 92: Displacement response individual modes - Eigenmode 7 - 6.90 Hz
Analysis 3
Mode 10, Eigen frequency 8.1690 Hz
Displacements DTX
min: -0.94mm max: 1.00mm

Figure 93: Displacement response individual modes - Eigenmode 10 - 8.17 Hz
We select the superposed modal displacements, starting with superposition type ‘ABS’ [Fig. 95]. The absolute maximum of the displacements are the sum of the maximum displacements of all contributing modes separately.
Figure 95: Displacement response superposed modes - ABS rule
In general it is not likely that all modes reach their maximum at the same moment. A more realistic approach to calculate the maximum displacements is using the SRSS rule which takes the Square Root of the Summed Squares [Fig. 97]. Therefore, lower amplitudes are obtained with the SRSS superposition compared to the absolute superposition.

Figure 96: Results browser
Figure 97: Displacement response superposed modes - SRSS rule
We select the superposed modal forces, starting with superposition type ‘ABS’ [Fig. 99]. The absolute maximum of the forces are the sum of the maximum forces of all contributing modes separately. This absolute maximum is calculated by the ABS rule.

Figure 98: Results browser
Figure 99: Force response - ABS rule
In general it is not likely that all modes reach their maximum at the same moment. A more realistic approach to calculate the maximum forces acting on the structure is using the SRSS rule which takes the Square Root of the Summed Squares [Fig. 101]. Therefore, lower amplitudes are obtained with the SRSS superposition compared to the absolute superposition.
Figure 101: Force response - SRSS rule
We select the equivalent (i.e. Von Mises) stresses in order to get an estimation of the damage inflicted by the earthquake on our building. Starting with superposition type ‘ABS’ [Fig. 103].

Figure 102: Results browser

Results browser ➔ Case ➔ Superposition type ABS [Fig. 102]
Figure 103: Equivalent stress response - ABS rule
In general it is not likely that all modes reach their maximum at the same moment. A more realistic approach to calculate the equivalent stresses is using the SRSS rule which takes the Square Root of the Summed Squares [Fig. 105]. Therefore, lower amplitudes are obtained with the SRSS superposition compared to the absolute superposition.

DianaIE

Results browser  ➔  Case  ➔  Superposition type SRSS  [Fig. 104]

Results browser  ➔  Analysis3  ➔  Output response spectrum analysis  ➔  Element results  ➔  Cauchy Total Stresses  ➔  SeqH  ➔  Show contours  [Fig. 104]

Figure 104: Results browser
Figure 105: Equivalent stress response - SRSS rule
Appendix A Additional Information

Folder: Tutorials/FiveStoryBuilding

Number of elements ≈ 80

Keywords:
- ANALYS: dynam response spectr.
- CONSTR: suppor.
- ELEMEN: cl8pe pstrai shell.
- LOAD: base freque.
- MATERI: elasti isotro.
- OPTION: direct units.
- POST: binary ndiana.
- PRE: diana.
- RESULT: cauchy displa force reacti residu stress total vonmis.

Earthquake on Five-story Building | https://dianafea.com
© DIANA FEA BV

Disclaimer: The aim of this technical tutorial is to illustrate various tools, modelling techniques and analysis workflows in DIANA. DIANA FEA BV does not accept any responsibility regarding the presented cases, used parameters, and presented results.