Dynamic - Seismic analysis

The aim of seismic analysis is to calculate the reaction of the model to earthquakes. This example describes the method to analyse the building using Modal Analysis. Modal Analysis is the accepted method and is recommended by mode design codes. The reaction of the model for each mode shape is calculated according to the response spectrum given by the seismic code.

Geometry definition

Because the aim of this example is to describe the method to do Modal Analysis, we will use a simple six-story structure, with a storey height of 3 meters. The seismic load carrying system consists of four shear walls.

We will define the model geometry and part of the loads using the AutoSTRAP program.

The DXF file can be downloaded from [http://atirftp.com/junk/ex4.dxf](http://atirftp.com/junk/ex4.dxf)

**AutoSTRAP:**

- click the AutoSTRAP icon on the desktop.
- click the "Open" icon.
- select the DXF file that you downloaded and click Open.
- the program displays the following menu. The lines representing the contours are found in Layer #4 and the lines forming the walls are in Layer #43 and these layers must be identified and assigned:

- the elements identified by the program are displayed on the screen. To display the slab element grid click on the icon.
- click on in the side menu:
Click in the lower side menu, then select in the upper side menu.
- select , click twice on Space #1 and define the loads:

Create the STRAP model: click in the side menu -

STRAP:
- click the STRAP icon on your desktop and double-click on the line with the "Modal analysis" model.
- click in the icon bar to display an isometric view.
- display the submodel:
- click in the icon bar to display the walls.
- define "dummy beams" on the slab perimeter (for applying line loads to the slab): click in the lower side menu, then click in the upper side menu.
- Select the start and end nodes of each beam.
- select in the upper side menu, click on the tab, select and 
- display the Main model:
**Loads**

Add the slab self-weight and the line loads along the slab perimeter:

- Click the **Loads** tab.
- Select the **Existing load** in the side menu, select the "Dead" load case and click **Define**.
- Select the **Element loads** in the lower side menu and select **Define** in the upper side menu.
- Click on **softw.**, select the local x3 direction and define a factor = -1.0.
- Click on **Select all elements**, the load is applied to all elements (the walls) in the main model (the slab self-weight was defined as part of the slab dead load).

- Select the first instance of the submodel:
- Select the **Beam loads** in the lower side menu and select **Define** in the upper side menu.
- Click on **OK**, then **Select all beams** and define the loads:

- Return to the **Main model**: Click **End load case**.
- Click **Solve** to solve the model for static loads.

**Dynamic analysis**

- Click the **Weights** tab. The masses are defined from the static loads.
- Select the **Static load** option:
• select the **Static load** option again:

• select **Mode shapes** in the side menu:

• click **Solve** to calculate the mode shapes.

• display the results graphically: click **Draw modes** in the side menu:
Seismic analysis

- Select the method for combining the modes:

  - **SRSS**: square root of sum of squares
    The estimated response $R$ (force, displacement, etc) at a specified coordinate is expressed as:
    $$ R = \sqrt{\sum R_i^2} $$
    where $R_i$ is the corresponding maximum response of the $i$th mode at the coordinate.

  - **CQC**: complete quadratic combination
    The estimated response is expressed as:
    $$ R = \sqrt{\sum R_i^2 \rho_i} $$
    where $\rho_i$ is the cross-modal damping coefficient

  Note:
  - When some of the modes are closely spaced, the SRSS method may grossly underestimate or overestimate the maximum response. Large errors have been found in particular in space models in which the torsional effects are significant. The term "closely spaced" may be arbitrarily defined as the case where the difference between two natural frequencies is less than 10% of the smaller frequency.
  - The CQC method is a more precise method of combining the maximum values of modal response.
  - The two methods are identical for undamped models ($\xi = 0$).

The seismic analysis for this is done according to the ASCE/SEI 7-05 Code; you may select other Codes, e.g. Eurocode 8, NBC-Canada, etc.

- select **Parameters** in the side menu:
Notes:
1. Specify the direction that the earthquake is applied. Select one of the global directions or define a vector as a combination of the three global directions. All mode shapes are used no matter in which direction the earthquake is applied. However, the modes which have deflections in the applied direction will dominate.
2. Specify the soil type as per Table 20.3.1.
3. Define the Importance factor as per Table 11.5.1.
4. Define the "Response modification coefficient" as per Table 12.2.1.
5. Specify a value of $T_u$, calculated according to Section 11.4.5 and the Figures in Chapter 22.
6. Specify the mapped maximum considered earthquake spectral response acceleration at short periods ($S_s$) and at 1s ($S_1$) as detailed in Section 11.4 and the Maps in Section 20.
7. Referring to Section 12.9.4, when the base shear calculated from the modal shape analysis is different than the base shear calculated according to the Equivalent lateral force procedure of Section 12.8, all corresponding responses, including moments and forces are adjusted accordingly.

- to display the modal results, click [Display modal results] and select [Display modal results].

**MDLAL RESULTS**

<table>
<thead>
<tr>
<th>Mode</th>
<th>T</th>
<th>$\psi \psi W_{tot}$</th>
<th>$F_n$ [kN]</th>
<th>$Q_n$ [m]</th>
<th>$V_n$ [m/s]</th>
<th>$A_n$ [m/s^2]</th>
<th>$F_n'/\psi W_{tot}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.6746</td>
<td>0.665</td>
<td>54.57</td>
<td>0.0230</td>
<td>0.0684</td>
<td>0.2033</td>
<td>0.116</td>
</tr>
<tr>
<td>2</td>
<td>0.6123</td>
<td>0.006</td>
<td>0.53</td>
<td>0.0025</td>
<td>0.0003</td>
<td>0.0272</td>
<td>0.116</td>
</tr>
<tr>
<td>3</td>
<td>0.5526</td>
<td>0.029</td>
<td>2.39</td>
<td>-0.0043</td>
<td>-0.0157</td>
<td>-0.0569</td>
<td>0.116</td>
</tr>
<tr>
<td>4</td>
<td>0.1451</td>
<td>0.176</td>
<td>12.09</td>
<td>0.0004</td>
<td>0.0053</td>
<td>0.0730</td>
<td>0.097</td>
</tr>
<tr>
<td>5</td>
<td>0.1321</td>
<td>0.000</td>
<td>0.00</td>
<td>0.0000</td>
<td>0.0001</td>
<td>0.0013</td>
<td>0.092</td>
</tr>
<tr>
<td>6</td>
<td>0.1111</td>
<td>0.000</td>
<td>0.02</td>
<td>0.0000</td>
<td>-0.0002</td>
<td>-0.0031</td>
<td>0.085</td>
</tr>
<tr>
<td>7</td>
<td>0.0560</td>
<td>0.067</td>
<td>3.13</td>
<td>0.0000</td>
<td>-0.0014</td>
<td>-0.0503</td>
<td>0.066</td>
</tr>
</tbody>
</table>

Total sum: 0.941, 72.72
RMS results: 56.03, 0.0235, 0.0708, 0.2306

The ASCE Standard (ASCE/SEI 7-05)
Direction: X1
$S=1$, $I=1.00$, $R=1.50$, $S_s=0.260$, $S_1=0.260$, $F_a=1.000$, $F_v=1.000$
85% Scaling: 1.2455, $T=0.7400$

Notes:
1. period (seconds)
2. Participation factor: a factor reflecting the relative influence of the mode shape. The sum should be greater than 90%.
3. sum of external forces in all global directions.
4. the root of the sum of the squares of the horizontal forces.

- click [Story data], select [RSS over modes 1 to 7], then select [Add nodes] and [Select all nodes].
How to .......

Click on the Stiffness and mass centers button in the above table:

Click on the Story shear forces button in the above table to display the shear force and the cumulative shear force at each level:

Create static load cases from the modal results and append them to the regular load cases:

Click in the side menu:
The program creates a fictitious load case representing an earthquake acting in the X1 direction, created by combining the mode shapes according to the code. The step should be repeated for the X2 direction, but the following steps assume that only this one case was created. In addition, we have not considered the minimum eccentricity required by the code.

**Results**

- click the **Results** tab.
- display only the main model: select **Display, Submodel instances** and **Remove all**; click **OK**.
- click **Draw result** and select the options in the following menu:
create combinations: select \textbf{Combinations} in the lower side menu, \textbf{Live/Def./Rev.} in the upper side menu and create the following combinations:

<table>
<thead>
<tr>
<th>No.</th>
<th>Title</th>
<th>1: Dead</th>
<th>2: Live</th>
<th>3: RSS _DIRECTION _X1</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11 1:40 _2:1:100</td>
<td>1.4</td>
<td>1.6</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>11 1:00 _3:2.20 _3:1.100</td>
<td>1</td>
<td>0.2</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>11 1:00 _2:1:100 _3:1.100</td>
<td>1</td>
<td>0.2</td>
<td>-1</td>
</tr>
</tbody>
</table>

\textbf{Concrete postprocessor}

- click the \textbf{Concrete} tab.
- click \textbf{Walls} in the lower side menu and \textbf{Seismic} in the upper side menu.
- specify the following parameters in the three menu tabs:
• click in the side menu. The program calculates the reinforcement for each wall segment according to the code provisions and displays the result summary:

- right-click on any wall segment and select **Display detailed results for wall**. For example
Steel connections

The steel connection design module is part of the STRAP package and cannot run as a stand-alone program.

Prior to designing the connections:
- define the model geometry and loads in STRAP; solve the model
- complete the design of the structural steel beams and columns in the STRAP Steel design module; a connection cannot be designed if the connected members have not been 'Computed'.

To start the connection design module:
- Steel design module: select File and Design connections. - or - STRAP main menu: Utilities and Connection design.

Select or in the bottom side menu.

Define general parameters:

- Define the height axis and the axis of the main beams (girders). This is required by the program to identify the "supporting" member and the "supported" member at each connection: To define the "supporting" and "supported" members for individual connections, select in the side menu.

Define the connection default parameters:

- click to define default parameters for all connections in the model:
  - in the Default connections tab, specify the default connection type for the three connections configurations:
  - in the Connection parameters tab, specify the design code and steel, bolt and weld types.
- click to define different parameters for specific connections in the model.

Design the connections and display the results:

- click in the side menu.
- select the two connected members.
- the program displays the connection and the design calculations.
- click to display a rendered view of the connection:

Refine default parameters or specify individual connection parameters:
• click to refine parameters for the entire model or for specific connections.
• to specify exact plate dimensions, number of bolts, etc., click and select the tab.

Example:

This example will design the following connections for a **steel portal frame**:
• beam to column
• beam to beam at apex
• column base plate

Note that the beam-beam and the beam-column connections will be designed with haunches.

This example is a guide for the design of the connections only; the geometry and the load cases have been simplified and are not an example of a proper design for this type of structure.

**Geometry**

Create the following model:

• click the new model icon and enter a title.
• select **Space Frame**, click and select **Portal frame**
• define the dimensions:
Dimensions H2, H3, W2 and W3 define the haunch dimensions.

- the haunches may be added in CONNECT even if they are not defined here.
- If haunches are defined here, STRAP will modify the beam stiffnesses accordingly.

select IPE500 an IPE600 for the beams and columns, respectively.

**Loads**

Continuing in the Wizard, define three load cases - Dead, Live and Wind - by entering the following values:

- click [OK] to exit the wizard.
- click [Load] in the tab bar.
- click [**Solve**] to solve the model.

**Results**

- click on the [Combinations] icon and define the following two combinations:
Steel Postprocessor

Check that the assumed sections are adequate:

• click and select Check section from geometry

• click

• select Design connections in the File menu.

Note:

• connections cannot be designed unless Compute is selected.

• the results summary table will show that the sections are not adequate. This is because lateral supports have not been defined for the beams, however this will not affect the connection design which is dependent only on the reactions.

Connection design

Height/main axes:

The first step is to define the "height axis" and "main axis" directions:

• Height axis:
  The program assumes that the columns are parallel to the height axis and that all other members are beams supported by the columns.

• Main axis:
  The program assumes that the members parallel to the main axis are the supporting (primary) beams and the members not parallel to this axis are the supported (secondary) beams.

In this model the height axis = X2 and the main axis = X1.

To define the axis directions:
Default parameters:

Select the default deflection type for beam-column connections and beam-beam connections:

- Click the icon in the side menu.
- Select the type as follows:

Design & display:

- to design the connection and display the detailed drawing, click select the connection:
the program displays the results:

CONNECTING ELEMENTS:

<table>
<thead>
<tr>
<th></th>
<th>Steel</th>
<th>Steel</th>
<th>Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$f_p = 275$</td>
<td>$f_p = 430$</td>
<td></td>
</tr>
<tr>
<td>Beam</td>
<td>$b = 500.00$</td>
<td>$t = 200.00$</td>
<td>$t_b = 16.00$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Shape</th>
<th>$b = 265$</th>
<th>$t_p = 430$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plate</td>
<td>$H = 1194.0$</td>
<td>$W = 200.00$</td>
<td></td>
</tr>
<tr>
<td>Steel</td>
<td>$t = 18.00$</td>
<td>Offset from top of beam $= 56.00$</td>
<td></td>
</tr>
<tr>
<td>Welds</td>
<td>(web) fillet</td>
<td>$Y_f = 355.0$</td>
<td>$U_f = 440.0$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Load Case</th>
<th>Shear $F_1$</th>
<th>Axial $F_2$</th>
<th>Moment $M_1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>158.19</td>
<td>-135.48</td>
<td>732.06</td>
</tr>
<tr>
<td>2</td>
<td>-79.56</td>
<td>54.89</td>
<td>-413.08</td>
</tr>
</tbody>
</table>

Axial $F_2$ = Compressive

DESIGN CHECKS

MOMENT CAPACITY (positive moment)

<table>
<thead>
<tr>
<th>Moment Resistance</th>
<th>$M_{Rd}$</th>
<th>$M_{Ed}$</th>
<th>$M_{Ed}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$F_1$ 200.4</td>
<td>$F_2$ 311.7</td>
<td>$F_3$ 220.4</td>
<td></td>
</tr>
<tr>
<td>$h_1$ 1043.9</td>
<td>$h_2$ 227.9</td>
<td>$h_3$ 857.9</td>
<td></td>
</tr>
<tr>
<td>$M_{Ed}$ $&lt; 1.00$</td>
<td>$F_1 = 135.48$</td>
<td>$h_{Ed} = 742.91$</td>
<td></td>
</tr>
<tr>
<td>$M_{Ed} = 631.41$</td>
<td>$M = 732.06$</td>
<td>$M_{Ed} = 695.0$</td>
<td>$0.91$</td>
</tr>
</tbody>
</table>

Base plate design:

Base plate design works on the same principle as beam-column and beam-beam connection design:

- the program retrieves and uses the STRAP reaction results
- the footings should be designed before the base plate design (footing dimensions, concrete type, etc., are required to design the base plate.)
• specify default parameters for all supports  
• specify different parameters for selected supports  
• compute and display the baseplate

Design the right support:  
• click the  icon at the bottom of the side menu.  
• specify default parameters for all supports:

![Default parameters window](image1)

• design the base plate and display the detailed drawing, click  
• select the connection:  

![Base plate and support node](image2)

• the program displays the results:
Plate: Plate 800x320x20, S 275
Welds: M16 E35 (Fillet)

Concrete: C30/37

Bolts: 4 M20 4.6, Anchor Plate = 100X100X12

<table>
<thead>
<tr>
<th>Plate</th>
<th>Shape</th>
<th>PL 800x320x20 - S 275</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>$f_y$</td>
<td>265</td>
</tr>
<tr>
<td></td>
<td>$f_u$</td>
<td>430</td>
</tr>
<tr>
<td>Length</td>
<td></td>
<td>800.00</td>
</tr>
<tr>
<td>Width</td>
<td></td>
<td>320.00</td>
</tr>
<tr>
<td>Thickness</td>
<td></td>
<td>20.00</td>
</tr>
</tbody>
</table>

WELDS: Fillet

Electrode = E35

Size: 16.00 $Y_a = 355.0$ $U_a = 440.0$

BOLTS: 4.6

Holes: Standard

Diameter: 20.0 $f_{th} = 240.00$ $f_{uh} = 400.00$

Tensile area: 245.0

Columns of bolts: 2

Rows of bolts: 2

Top Edge: 50.00

Side Edge: 50.00

Load Case | Shear $F_v$ | Axial $F_t$ | Moment $M$
---|---|---|---
1 | 116.61 | -170.71 | 0.00
2 | -19.39 | 54.58 | 0.00

Axial: (-) = Compression
### BASE PLATE:

<table>
<thead>
<tr>
<th></th>
<th>Expression</th>
<th>Value</th>
<th>Value</th>
<th>α</th>
<th>f_{cd}</th>
<th>( \gamma_d )</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Compression</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>resistance (6.2.8.3)</td>
<td>( N_{i,Rd} = \beta_i f_{cd} \alpha A_{off} )</td>
<td>( \beta_i = 0.67 )</td>
<td>( f_{cd} = 20.00 )</td>
<td>1.88</td>
<td>0.07</td>
<td></td>
</tr>
<tr>
<td></td>
<td>( \frac{F_t}{N_{i,Rd}} &lt; 1.00 )</td>
<td>( F_t = 170.71 )</td>
<td>( N_{i,Rd} = 2340.51 )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Tension</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>resistance (6.2.4)</td>
<td>( N_{i,Rd} = 2 F_{t,Rd} )</td>
<td>( F_t = 54.58 )</td>
<td>( N_{i,Rd} = 93940.25 )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( \frac{F_t}{N_{i,Rd}} &lt; 1.00 )</td>
<td>( F_t = 98.00 )</td>
<td>( N_{i,Rd} = 196.00 )</td>
<td></td>
<td></td>
<td>0.28</td>
</tr>
</tbody>
</table>
**Dynamic - Time-History**

This module calculates the transient (history) response of a model subject to dynamic loads in which viscous damping is present. It enables the dynamic analysis of models subject to impact, impulse or cyclic loads or any other type of load that varies with time.

The stages in solving a model for this type of dynamic loading are:

- geometry definition
- definition of masses
- calculation of natural frequency
- definition of the time-history function and associated loads
- display of results and transfer to STRAP

**Geometry definition**

Because the aim of this example is to describe the method to do Time-History Analysis, we will use a simple frame structure with a height of 3 meters. A ZIP file containing model files may be downloaded by clicking on this link: [http://tinyurl.com/qgoqcqv](http://tinyurl.com/qgoqcqv)

In the STRAP main menu:

- click on **Files** and select **Unzip a model**, select the downloaded file.
- click and highlight the model and click **Copy**

**Mode shape analysis**

- click the **Weights** tab
- select the **Add weights** option in the side menu
- define a weight = 50 kN on node 5:

```
Weight = 50
OK Cancel
Advanced...
```
- select node 5
- select **Mode shape...** in the side menu and specify the following options:
select **solve** in the side menu to calculate the mode shapes.

**Time-History Analysis**

**Load suddenly released:**
Calculate the cycles of vibration if a horizontal load is applied to the top of the frame and then suddenly released. Assume 4% damping.
- click on **TimeHist** in the menu bar.
- click on **New load** in the side menu
- type in the name of the load case, e.g. "Load suddenly removed"
- click on **Add** in the side menu
- Define a horizontal load of 100 kN:

There are several ways to apply the load. We will apply it gradually and linearly so that the load is applied fully at t=30 sec, then reduce it to a zero load at t=30.01 sec.
- click on **History** in the side menu
- define the first segment of the load - 0 to 30 sec - as follows:
• define the second segment of load - 30.0 to 30.01 sec - similar to above:

- click to continue.

Define the damping:

- click on in the side menu
Display the results:

1. Natural frequency and period:

   - click on in the side menu.
   - select in the menu
   - the following table is displayed:

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>Eigenvalue ($\Omega_m^2$)</th>
<th>Natural Frequency</th>
<th>Period</th>
<th>Damping Coeff. (%)</th>
<th>Max translation Node-DOF</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>344.672</td>
<td>2.9548</td>
<td>0.33844</td>
<td>4.60</td>
<td>5.1</td>
</tr>
</tbody>
</table>

2. Displacements (graphic):

   A time span must be defined before displaying the displacements. For example, display the deflections from 0 to 35 sec:

   - click on in the side menu.
   - specify the result type, direction and time span:

   and select node 2. The program displays the time-deflection diagram:
The rate at which the displacements decrease is a function of the damping value.

3. Displacements (tables):
A time span must be defined before displaying the displacements. For example, display the deflections from 0 to 35 sec at node 2:
- select "Time tables" in the menu bar:

Repeat to add $t = 35$ sec, then delete $t = 0$ sec:
click in the side menu.
• select the result type, etc:

The program displays the table:

**Periodic forcing function:**
The following motor is located at node 5:
• weight = 40 kN
horizontal period force = 8.5 kN at a frequency = 1.75 Hz.
damping ratio = 4%

Similar to the previous example:

- select the option in the side menu and define an additional weight = 40 kN on node 5.
- select in the side menu to calculate the mode shapes.
- click on in the menu bar.
- click on in the side menu
- type in the name of the load case, e.g. "Periodic force"
- Define a horizontal load of 8.5 kN at node 2.
- click on in the side menu
- Enter the following history function (1.75 Hz = 0.5714 sec)

![History function f(t)](image)

- click and display the results as described in the previous example, e.g. X1 deflections (graphic):

![X1 DEFLECTIONS](image)

The deflections become stable after 7 sec.

and the natural frequency:

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>Eigenvalue (Omega^2)</th>
<th>Natural Frequency</th>
<th>Period</th>
<th>Damping Coeff. (%)</th>
<th>Max translation Node-DOF</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>191.485</td>
<td>2.2024</td>
<td>0.45406</td>
<td>4.00</td>
<td>5-1</td>
</tr>
</tbody>
</table>

- click on in the side menu
• click on **History** in the side menu and revise the "Period T" to 0.454 sec (the natural frequency).
• The X1 deflections are:

![X1 Deflections](image)

- The steady-state deflection is 59.4 mm.
- The horizontal deflection for a model with a horizontal static load = 8.5 kn is 4.83 mm.
- The Dynamic Load Factor (DLF) = 59.4/4.83 = 12.3

**Blast load:**

Calculate the deflections for a blast load of 30 kN that is applied at t =0 and decreases linearly to 0 kN ant t = 0.16 sec.

Similar to the previous example:

• click on **New load** in the side menu
• type in the name of the load case, e.g. "Blast load"
• Define a horizontal load of 30 kN at node 2.
• click on **History** in the side menu
• Enter the following history function (as described in the first example) by defining **two** points: t=0., f(t)=1.0 and t=0.16, f(t)=0.

![History function f(t)](image)

• click **Main menu**
• click on **Damping** in the side menu and define 5% damping.
• display the results as described in the previous examples, e.g. X1 deflections (graphic):
How to .....28

Ground motion:

Calculate the deflections for a horizontal ground acceleration that increases linearly from 0 to 1.962 m/sec\(^2\) at 0.04 sec, then subsides linearly to 0 m/sec\(^2\) at 0.08 sec.

Similar to the previous example:

- click on \(\text{New load}\) in the side menu
- type in the name of the load case, e.g. "Ground motion"
- click on \(\text{Base accel.}\) in the side menu and define a magnitude = 0.2g (= 0.2x9.981 = 1.962 m/sec\(^2\))
- click on \(\text{History}\) in the side menu
- Enter the following history function (as described in the first example):

![History function f(t)](image)

- click \(\text{Main menu}\) and display the results as described in the previous examples, e.g. X1 deflections (graphic):

![X1 Nodal Forces](image)

Display the forces and the moments at the top end of the left column as they vary in time:
• click on Display gr... in the side menu and select the following options:

The program displays the results:

These results can be transferred to STRAP and added as load case results:

• click on Update res... in the main side menu
• select the following options:
• select the "Ground motion" load case.

Return to the STRAP results module and display the graphic results for this load case. For example = moments:

![Diagram showing graphic results for moments]
Concrete slab deflection

STRAP calculates the linear elastic deflection of a concrete slab based on the gross cross-section moment-of-inertia. However the actual slab deflections are much greater due to several important factors:

- cracking
- reinforcement ratio
- time-dependent non-linear factors, such as creep and shrinkage.

The STRAP results module has an option to calculate the deflection using a method which takes into account these factors. The method is an empirical one based on an "effective" moment-of-inertia approach and it important to understand that this method is not an exact one.

The method calculates an "effective" (reduced) moment-of-inertia that is a function of the ratio of the actual moment to the cracking moment of the element.

**Eurocode 2:**

\[ I_e = 0.5 \left( \frac{M_{cr}}{M} \right)^2 I_g + \left( 1 - 0.5 \left( \frac{M_{cr}}{M} \right)^2 \right) I_{cr} \leq I_g \]

**ACI 318:**

\[ I_e = \left[ \frac{M_{cr}}{M_{cr}} \right]^4 I_g \left[ 1 - \left( \frac{M_{cr}}{M} \right)^4 \right] I_{cr} \leq I_g \]

where the fourth power is used as suggested by Branson for continuous integration.

for both codes:

- \( I_e \) = effective moment-of-inertia
- \( I_g \) = gross moment-of-inertia, including reinforcement
- \( I_{cr} \) = cracked moment-of-inertia
- \( M \) = service moment
- \( M_{cr} \) = cracking moment

STRAP calculates the effective moment-of-inertia and for each element in both direction and then solves the model again using the reduced stiffness values.

The total deflection \( a_t \) is the sum of the immediate deflection \( a_i \) from all service loads and the long-term deflection \( a_l \) from the sustained service loads, therefore different stiffness values are used for immediate and long-term deflection calculations based on the value of \( M \) derived from the loads applied; the user must define different load combinations for immediate and long-term loads.

**Geometry**

- click the new model icon
- select and click defined
**Nodes:**
- define the eleven corner nodes that form the slab contour:

**Elements:**
- **click** and then select **Mesh** in the side menu.
- click **OK** in the following menu.
- select the eleven corner nodes (in the order they are numbered) to define the floor contour and close by selecting the first node again.
- select **End contour definition**.
- click **OK** in the following menu to accept the default mesh parameters; the program creates and displays the floor slab.
- **click** in the side menu and click and highlight Property 1 in the table.
- Define thickness = 200 mm and $E = 30,000 \text{ mPa (30 x 10}^6 \text{ kN/m}^2)$

**Restraints:**
Define pinned supports at the nodes as shown in the following drawing:

**Loads**
Define dead and live service loads in separate load cases:
- **click** at the top of the screen.
- **click** and type in "Dead" as the load case title.
- select **Element loads** in the side menu.
- select **Define** and **and** define a loads = 10 kN/m²:
• repeat for a second load case titled "Live" with a uniform load = -3 kn/m² applied to all elements.

• click to solve the model.

**Results**

The slab deflections will be calculated according to Eurocode 2.

First we will check the STRAP uncracked elastic deflection:

**Combinations:**

Three combinations are required
• ultimate loads - total - to calculate the reinforcement
• service loads - total - to calculate the immediate deflection \( a_i \)
• service loads - sustained - to calculate the long-term deflections \( a_t \); assume that 30% of the live load is sustained.

To define the combinations:

• select in the side menu and 
• define the following combinations:

<table>
<thead>
<tr>
<th>No</th>
<th>Title</th>
<th>1: Dead</th>
<th>2: Live</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Ultimate</td>
<td>1.5</td>
<td>1.5</td>
</tr>
<tr>
<td>2</td>
<td>Service</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>Sustained</td>
<td>1</td>
<td>0.3</td>
</tr>
</tbody>
</table>

**Elastic deflections:**

• click in the side menu and 
• arrange the menu as follows, click ; the program displays the deflection contour map:

The maximum elastic deflection is **5.97 mm**.
Deflections - cracked section:

- click in the side menu and specify the deflection parameters:

![Slab deflections parameters](image)

Notes:
1. The "creep factor" is used to calculate the total long-term deflection. The deflections calculated from the long-term combination using the effective moment-of-inertia are multiplied by this factor. The factor corresponds to:
   - Eurocode 2: Equation (7.20)
   - ACI 318: Equation (9-11)

2. The reinforcement values used to calculate the effective moments-of-inertia are determined as follows:
   - **Reinf. required for moments/forces**
     The program calculates the area required and then selects actual reinforcement according to the specified range of diameters and spacings. This actual area is used to calculate the effective moments-of-inertia.

   - **User defined reinforcement**
     The program uses the spacing and diameter specified in the reinforcement option in this dialog box for all elements, top and bottom, both directions. However, different reinforcement area may be defined for selected elements, as follows:
     - select in the side menu
     - select in the side menu

- click to calculate the reinforcement, the effective moments-of-inertia and to solve the model again with the reduced stiffnesses.

- click
The maximum deflection is 19.8 mm, (19.8/5.97) = 3.3 times greater than the elastic deflection.

Estimate the deflection at node 93 in terms of L/x, relative to support nodes 289 and 4:

- click << Display
- select @ Draw deflected shape and click OK; the program superimposes the deflected shape and deflection values:

- click relative at the bottom of the display.
- click on nodes 93, 289 and 4 (in that order). The program displays the relative deflection:

Display a table of the cracked section properties:

- click Display

Copyright © ATIR Engineering Software Ltd.
select Cracked sections table and click OK; the program displays the following table:

| Elem | Comb | Dir | Mcr  | M   | F   | As  | As' | x   | Ir/Ig | Ie/Ig |
|------|--|-----|------|-----|-----|-----|-----|-----|-----|------|-------|
| 89   | 2  | X   | 29.32| 33.00| 0.00| 10  | 0   | 4.1 | 0.183| 0.336 |
|      |    | Y   | 19.92| 18.09| 0.00| 6   | 0   | 3.3 | 0.120| 1.000 |
| 90   | 2  | X   | 20.23| 32.30| 0.00| 9   | 0   | 4.0 | 0.170| 0.332 |
|      |    | Y   | 19.85| 18.47| 0.00| 5   | 0   | 3.1 | 0.109| 1.000 |

where:
- **Elem** - element number
- **Comb** - combination used for deflection calculation
- **Dir** - direction; properties are calculated in both reinforcement directions
- **Mcr** - the cracking moment
- **M** - the moment at the element center
- **F** - the axial force in the element
- **As** - the tension reinforcement (calculated, minimum or user-defined)
- **As'** - the compression reinforcement (calculated, minimum or user-defined)
- **x** - height of the compression block in the section
- **Ir/Ig** - ratio between the cracked and uncracked moments-of-inertia.
- **Ie/Ig** - ratio between the effective and uncracked moments-of-inertia.

For example, in element 90
- the moment in the X-direction = 32.3 kN-m is greater than the cracking moment = 20.23 kN-m
- the section is cracked, hence the effective moment-of-inertia is 33.2% of the uncracked moment-of-inertia.
- in the Y-direction, the moment = 18.47 is less than the cracking moment, hence the program uses the uncracked section (Ie/Ig = 1.000)
Submodel

This example demonstrates how to define a submodel and add it to the main model.

Submodels are efficient when there is a portion of the structure that is repeated more than once in the model, e.g. the typical floor in a high rise building. The advantages of the submodel are:
- each submodel may contain up to 32,000 nodes, i.e. the total model sizes - comprised of the "main model" and all the submodels - is virtually unlimited.
- When a submodel is revised each "instance" (occurrence) of the submodel in the structure is automatically revised.
- The solution time is much faster than that required for an identical structure without submodels.

We will define the following structure:

The structure consists of:
- the "Main model" - the columns.
- Three different submodels:
  - the two lower floors
  - the two upper floors (with the balconies)
  - the roof

Each of the first two submodels has two "instances"; the roof submodel has one instance.

Each submodel instance is joined to the Main model at "connection points".
Main model

- click the new model icon

- select Space Frame and click

- click Nodes in the side menu and define the following nodes:
Copy the nodes to the other levels and create the columns:

- select in the bottom side menu and then click on .
- select all of the nodes.
- select Node 1 as the reference node.
- specify the new location of the reference node by coordinates;

and enter the new coordinate value:

[An image showing the coordinate input interface]

- click in the icon bar to display an isometric view of the model.

Erase the four intermediate columns at the top floor:

- click [Beams] and then select [Delete] in the side menu.
- select the four columns.
Define the floor slab at level +3.00:

- display only the level at +3.00:
  - select "+3.00" in the list and click OK.
  - click in the icon bar to cancel the isometric view.
  - click Elements and then select Mesh in the side menu.
  - click OK in the following menu.
  - select the four corner nodes to define the floor contour and close by selecting the first node again.
  - select End contour definition.
  - click OK in the following menu to accept the default mesh parameters (check that Property 3 is specified); the program creates and displays the floor slab.

Define the perimeter beams:

- click Beams and then select Define in the side menu.

- Define the beams along the perimeter.
  - Again select Remove and Display selected levels in the menu bar, but this time remove +3.00 from the list.
  - click in the icon bar to display the isometric view of the model.
Define the beam and column sections:

- click **Beams** and then select **Properties** in the side menu.

- click and highlight Property 1 in the table (the columns); click **Define/revise**, select **dimen** and **** and define the section dimensions:

```
Rectangular section - property no. 1

- B = 30
- H = 30
- Specify 30 x 30
```

- repeat for Property 2 (the beams) - 30 x 60 cm.

- click **Elements** and then select **Properties** in the side menu.

- click and highlight Property 3 in the table (the floor slab); click **Define/revise**.

```
Element property no. 3

- Thickness = 25
```

Define the supports:

- click **Restrains** and then select **Pinned** in the side menu.
How to .....42

• select all nodes at level +0.00.

**Submodels**

Create the first submodel from the floor slab at +3.00:

• display the X1-X3 plane on the screen: click in the icon bar, click on X1-X3 plane and End
• click in the side menu:

![Submodels window]

• Select the all the nodes at the floor level (use the Select by window option):

![Node selection]

The program now opens a small window that lists all of the submodel instances in the structure: window is displayed in a program modules (geometry, loads, results, etc) and is used to go from one submodel to another.
• select the first submodel from the list; the program displays it on the screen:
Create the second instance of the submodel:

- select the main model in the submodel window:

- and click in the icon bar to restore the isometric view of the model.

- click in the side menu:

- Select the lower-left corner of the submodel as the "reference point"
- Select the location of the reference point in the Main model:
Create the submodel at +9.00:

The second submodel is identical to the first one with the addition of the balconies. We will create it by copying and modifying the first submodel.

- click **Submodels** in the side menu:
- click **New** (submodel) in the following menu and -
- select *Use a copy of an existing submodel* and define the submodel name as "**Slab +9.00**".
- select **Slab +3.00** from the list of existing submodels.
- the program displays the submodel; add the balcony elements as shown:

Add both instances of the second submodel:

- click **Submodels** in the side menu:
- Select **2 Slab +9.00** in the submodel list and click **Add** (instance).
• select the lower left corner of the submodel as the "reference point"
• the program then displays the Main model; select the two reference points defining the location of the two instances of the submodel:

Define the third submodel at +15.00:

The roof slab is identical to the first submodel but with different slab and beam dimensions.
• click [Submodels] in the side menu:
• click [New] (submodel) in the following menu and -
• select Use a copy of an existing submodel and define the submodel name as "Slab +15.00".
• select Slab +3.00 from the list of existing submodels.
• the program creates and displays the submodel.

Revise the geometry as follows:
• click [Beams] and then select [Properties] in the side menu.
• click and highlight Property 4 in the table; click [Define/revise], select and and define the section dimensions: 30 x 100
• assign this property to all beams in the submodel.
• click [Elements] and then select [Properties] in the side menu.
• click and highlight Property 5 in the table; click [Define/revise] and define the slab thickness: 30 cm.
• assign this property to all elements in the submodel.

Remove the unnecessary connection points (note that connection points without an exact corresponding Main model node are automatically connected by Rigid links to the closest Main model node).
• select [Connections] in the side menu and click on [Delete]
• select the four intermediate connection points.

The program assumes by default rigid connections between the submodel and the main model. Assume that the roof beams are pinned to the columns, i.e. release the rotational degrees-of-freedom at the connection points:
• select [Connections] in the side menu and click on [Pinned]
• select the four corner connection points.

Add the third submodel to the structure:
How to

click in the side menu:
Select 3 Slab +15.00 in the submodel list and click Add (instance).
Specify 1 copy and define the instance name.
select the lower left corner of the submodel as the "reference point"
the program then displays the Main model; select the top of the lower-left column as the location of the reference point in the Main model.

Connection points - additional options:

There is an option to automatically define the connection point for a submodel to the main model. The program searches for Main model nodes that are within a "tolerance" distance of the selected submodel nodes.

Note:
• this option does not revise existing connection points
• connection points without an exact corresponding Main model node are automatically connected by Rigid links to the closest Main model node.

For example, use this option to redefine the connection points for the submodel Slab +3.00:
• select the submodel in the small window:

Delete the existing connection points because the program will not revise them.

• select Connections in the side menu and click on

• select the four connection points.

• return to the Main model

Submodels - tables

Display a table with the submodel data:
Submodels - display options

Selected submodels may be displayed/removed from the main model display. This option is useful when creating the geometry or displaying the results.
Loads applied to a submodel may be applied to all instances or different loads may be defined for each instance of the submodel:

- click the **Loads** tab.
- select **New load** in the side menu and enter a name for the load case.
- select the **Slab +3.00** in the small window.
- select **Element loads** in the side menu.

- select **Define** and **and define the loads:**

- select the submodel instance:
• apply the load to selected elements.

Continue to define loads, solve the model and display the results.
Column - solid section

This example demonstrates how to design a column with an arbitrary cross-section defined by the user.

The model is a simple one member column, loaded with an axial load and moment; the column has the following cross-section:

The example shows how to -
- create the section in the CROSEC section generator
- copy the section to STRAP
- arrange the column reinforcement temple (corner bars and groups) in the concrete design module.

The section will be imported into the program from a DXF file. Please download the file from www.goo.gl/5dWnu (note the upper-case "W")

Geometry - general

- click the new model icon
- select Space Frame and click
- rotate the model to the X1-X3 plane:
  - click the Dynamic rotate icon
  - click the X1-X3 plane button
  - click End
- click Nodes in the side menu and define the following two nodes:
  - X1=0 ; X2=0 ; X3=0
  - X1=0 ; X2=0 ; X3=5
- click Beams and define a beam connecting the two nodes.
- click Restraints and define the following supports at the two nodes:
  - Bottom node: Fixed
  - Top node: restrain the node against horizontal movement and allow vertical deflection; select Other

The defined geometry is displayed as:
How to ....

Geometry - section

The section is defined in the CROSEC section generator program:

- select File in the menu bar and "Section generator" in the menu:

There are three ways to define a section in CROSEC:

- define the lines
- select a standard section from the library
- import a DXF format file

We will use the third option.

If you have not downloaded the DXF file as explained at the beginning of this example, please do so now.

- select File in the menu bar and "Import DXF file" in the menu (or click in the icon bar):

- select the file (in the folder where you saved the file) and click Open

- Select the layers to import:

![DXF Import dialog box]

Copyright © ATIR Engineering Software Ltd.
• the program displays the section. Select -

<table>
<thead>
<tr>
<th>Output</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Display Properties</td>
<td>Print section</td>
</tr>
<tr>
<td>Copy to clipboard</td>
<td></td>
</tr>
</tbody>
</table>

   to display the section properties:

• copy the section to the computer's "clipboard"; select -

<table>
<thead>
<tr>
<th>Output</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Display Properties</td>
<td>Print section</td>
</tr>
<tr>
<td>Copy to clipboard</td>
<td></td>
</tr>
</tbody>
</table>

• select File in the menu bar and "Exit" in the menu
**Geometry - continued ...**

- click **Beams** and then select **Properties** in the side menu.
- click and highlight the first row in the table ("- Not used -") and click **Define/revise**.

![Select section type](image1)

- check the section orientation and material:

![Definition of property no. 1](image2)

- click **OK** to display the rendered drawing:
- click **Loads** in the tab bar.
**Loads**

Define joint loads at the top of the column:

- click **New load** in the side menu and enter a title.
- click **Joint load** in the side menu and define the following axial load and moment:

![Joint load interface](image)

- click **End load case** and **Solve**
- click the **Concrete** tab

**Concrete design module**

- click **Columns**
- click **Define**
- click **Automatic definition of all columns** and click **End**
- click **Defaults** and specify various design parameters - design code, concrete type, reinforcement grade, cover, etc.

To arrange the reinforcement template for the solid section:

- click **Properties**
- click **Edit STRAP solid section**

- The program displays the default reinforcement arrangement:
  - a "corner bar" and every perimeter corner
  - a reinforcement line between every pair of adjacent corner bars:
The following changes will be made:
- lines 8 to 15 will be deleted as the corner bars along the arc are sufficient.
- the following symmetric line pairs will be specified as identical: 7-16, 3-5 and 2-6.

- click **Delete lines**: highlight line 8 and click the mouse. Repeat for line 9 to 15

- click **Assign lines to group**: click on line 7 and then on line 16; line 16 is renumbered "7". Repeat for lines 3-5 and 2-6.
When completed the section should appear as:

- click OK
- click End
- click in the side menu; the column result summary is displayed.

To display the detailed results, right-click on the column:

and select Design combination only

<table>
<thead>
<tr>
<th>Column: C1</th>
<th>STRAP bm no. 1</th>
<th>Design combination = 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>f_c = 4000</td>
<td>f_y = 350 MPa</td>
<td></td>
</tr>
<tr>
<td>la = k * lu</td>
<td>r</td>
<td></td>
</tr>
<tr>
<td>M3: 5.00 = 1.00 * 5.00</td>
<td>0.17</td>
<td>20.9</td>
</tr>
<tr>
<td>M2: 5.00 = 1.00 * 5.00</td>
<td>0.18</td>
<td>27.8</td>
</tr>
</tbody>
</table>

Reinforcement:
- 22@12
  - A_s = 24.9 cm²
  - % = 0.74
  - Reduced Effective area

Design loads [kN m]:

- P | M2 | M3

- Input: 220.0 | 40.0 | 0.0
- Design: 220.0 | 40.0 | 0.0
- Min: 0.0 | 1.20 | 7.3

Capacity Factor = 0.99
Finally, you may modify the diameter of quantity of bars in any reinforcement group:

and select the column. The program displays the following screen:

Note: The "solid section" option can also be used to calculate a beam with any section by defining it as a "column". However, it is the user’s responsibility to ensure that all Code requirements for beams are satisfied.