Dynamic Analysis Of The Milad Tower

Edwin Wilhelm, Mitchell Ford, Darren Coelho, Lachlan Lawler, Peter Ansourian, Fernando Alonso-Marroquin, and Faham Tahmasebinia

School of Civil Engineering, The University of Sydney, Sydney NSW 2006, Australia

a)Corresponding email: faham.tahmasebinia@sydney.edu.au

Abstract. This report involves the modelling of the Milad Tower using the finite element analysis program Strand7. A dynamic analysis was performed on the structure in order to understand the deflections and stresses as a result of earthquake and wind loading. In particular, Linear Static as well as Natural Frequency and Spectral Response solvers were used to determine the behaviour of the structure under loading. The findings of the report highlight that the structure was modelled accurately with the outputs representing realistic values. The report suggests that the design of the beams, columns, slabs and all structural members was sufficient enough to support the tower during maximum loading cases. The governing load case was earthquake loading.
INTRODUCTION

The Milad tower is a multipurpose tower located in Tehran, Iran. Since Tehran is located close to two tectonic plate fault lines, seismic behaviour is of particular interest. The Milad Tower is the 6th tallest tower in the world, at 436 metres high and is also the 4th tallest telecommunications tower. The tower includes many stories of shops and restaurants as well as being used for telecommunications, with highly sensitive equipment.

The tower, as shown by the photo of the tower in Fig 1, is made up of five main components; the antenna mast, the head structure, the concrete tower shaft, the lobby and the foundation. The antenna mast is a slender concrete section that stretches over 100m, composed of four different sections. The head structure sits around the main concrete shaft and makes up a 12 storey structure. This structure is a space basket and consists of radial and peripheral beams and columns. The radial and peripheral beams transfers the loads directly to the columns. The loads from the columns are transferred directly to the steel basket and then to the concrete shaft.

The concrete shaft carries most of the gravitational and lateral loads of the structure. It is 315 metres high and consists of four main tapered trapezoidal walls and two octagonal shapes connected by several walls. The octagons are post tensioned in order to increase the bending capacity and stiffness of the structure to reduce the deflections. The foundation and lobby area consists of a circular foundation which is a transition structure (Yahyai et al. 2009). The circular mat foundation is approximately 66 metres in diameter and sits directly beneath the transition structure. The transition structure is a pyramid shape and is made up of a central core, inclined walls and triangular-shaped walls. The structure is also post-tensioned in order resist the high stresses and punching shear that the foundation will experience.

The Milad Tower was proposed as part of the ‘Shahestan Pahlavi’ project, which was planned to encompass five million square metres, accommodating 50,000 residents, government ministries, commercial offices and a variety of cultural centers (libraries and museums). However this project was cancelled following the 1979 Revolution. The tower was brought back and construction began in 1997, taking 11 years to complete (2008). The structure is multipurpose, consisting of 63 trade units, many food courts, observation decks, telecommunication services and a carpark of 27,000 m².

<table>
<thead>
<tr>
<th>Location: Tehran, Iran</th>
</tr>
</thead>
<tbody>
<tr>
<td>Architects: Mohammad Reza Hafezi</td>
</tr>
<tr>
<td>Structural Engineers: Mohammad Reza Hafezi</td>
</tr>
<tr>
<td>Function: Shops, Restaurants, Hotel and Conferences, telecommunications tower</td>
</tr>
<tr>
<td>The Year of Built: 2009</td>
</tr>
<tr>
<td>Approximate Cost: $5 billion</td>
</tr>
<tr>
<td>Overall Height: 436m</td>
</tr>
<tr>
<td>Floor Area: approximately 154,000m²</td>
</tr>
<tr>
<td>The Structure of the Plan: Space structure around the concrete core.</td>
</tr>
<tr>
<td>Number of Floors: 145 floors above ground, a 6 storey lobby and 12 levels in the head structure</td>
</tr>
</tbody>
</table>

**TABLE 1. Structure Details**

*FIGURE 1. The Milad Tower in Tehran*
Below is the description of the members of the structure:

Floor system: Concrete slab (220mm deep with N16 bars at 300mm centres) sitting on steel beams, acting compositely (the slab provides lateral restraint to the top flanges of the beams).

Beams: Steel beams (460UB82.1) are in the space truss and extended radially from the core, the beam span is 10m at the perimeter and 5m at the core.

Columns: Steel columns (350WC158) are in the space truss and the largest column spacing is 10m at the perimeter. The column spacing closer to the core is 5m. Floor to floor height is 4m.

Core: The core is 40MPa concrete and consists of four main tapered trapezoidal walls and two octagonal shapes connected by several walls, as shown by Fig. 1. The octagons are post tensioned in order to increase the bending capacity and stiffness of the structure to reduce the deflections, in particular for the wind loading.

Foundation: The foundation design was outside the scope of this project and the boundary conditions. For this reason, the substructure and associated structure at the base of the tower have not been analyzed. The bottom of the tower is fully fixed to prevent translational and rotational movement. This aims to provide a similar representation of the foundation in real life.

Table 2 shows the Structural elements of the tower and the comparison between the suggested members from the Australian Design Codes and the members chosen for the design.

<table>
<thead>
<tr>
<th>Details of the Structural Elements</th>
<th>Suggested Structural Element Sizes</th>
<th>Suggested Standard Designs (e.g. AS3600, AS4100)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Head structure columns</td>
<td>350WC230</td>
<td>310UC158</td>
</tr>
<tr>
<td>Head structure beams</td>
<td>460UB82.1</td>
<td>460UB82.1</td>
</tr>
<tr>
<td>Slabs</td>
<td>40MPa 220mm deep</td>
<td>210mm</td>
</tr>
<tr>
<td>Foundation</td>
<td>40MPa 1.5m thick</td>
<td>1.25m</td>
</tr>
<tr>
<td>Core Shaft</td>
<td>40MPa 1.5m thick</td>
<td>1.25m</td>
</tr>
<tr>
<td>Spire</td>
<td>40MPa 0.5m thick tapered</td>
<td>1.25m</td>
</tr>
<tr>
<td>Steel Basket</td>
<td>400WC270</td>
<td>500WC440</td>
</tr>
</tbody>
</table>

**STRUCTURAL SYSTEM**

Figure 2 shows the key plan of the core of the structure at ground level. The central concrete core extends through the entire structure. The hollow concrete structure has a large amount of area away from the centroid and thus has a large second moment of area, which contributes greatly to increasing the stiffness for lateral loading. The core has varying degrees of post-tensioning, depending on the height, which increases the bending capacity of the structure as well as the stiffness. By increasing the stiffness of the structure, the mast will also deflect less.

The lateral loads from wind that act on the concrete core will place one side of the structure in tension and the other in compression. The post-tensioning within the structure will aid in resisting the tensile forces in the concrete and reduce potential cracking. The tower can be treated as a cantilevered beam structure, where lateral loading will induce a bending moment through the system. This bending moment will travel through the structure and into the foundation. The design of the foundation is outside the scope of this analysis and it is assumed that the base of the structures acts as a fixed support.
The main components of the head structure are the radial and peripheral beams, columns, space basket and concrete shaft, this can be seen in Fig. 3. Figure 3a shows an isometric view of the head structure, which includes the 12 floors of restaurants and shops. Figure 3b shows a cross section of the head structure and the critical structural members, namely the slabs, steel space basket, radial beams and the peripheral beams and the core shaft. The structure firstly transfers loads to both the radial and peripheral beams. The radial beams are between columns or between the columns and the concrete shaft, working to transfer the loads from the slab to the columns or to the shaft. The peripheral beams are on the perimeter of the structure and work to transfer the loads to the columns. The columns will transfer the loads through columns or to the space basket, which is finally transmitted to the concrete shaft. Figure 4 shows a plan view of level 7 of the head structure, which is the biggest floor. As shown by the layout, the I-beam columns are orientated outwards to ensure the major axis bending is in the radial direction. This is done to increase the stiffness and reduce deflections in the radial direction. However, the members for the columns are steel welded columns, which are also quite stiff in minor axis bending, meaning that the capacity is also quite high in the minor direction. The layout for each of the other floors is very similar to the level 7 layout shown in Fig. 4.

The beams and columns were designed in accordance with Australian Design Codes to be 460UB82.1 and 310UC158 respectively and the concrete slabs were initially designed to be at a depth of 210 mm. The members had been designed in accordance with strength and the members were put into the Strand7 model and the linear static analyses were performed. It was found that the deflections of the members were too great as they were exceeding design deflection limits hence the members were changed to 460UB82.1 and 350WC230 for beams and columns respectively and 220 mm slab depth. This member choice is shown in Table 1 and discussed further in the results section. Figure 5a. shows an elevation of the structure, with a total height of 436m, which highlights the slenderness of the tower and hence the sensitivity to lateral loading. Figure 5b. shows a typical detail of the slab to beam connection for the floors of the head structure. It should be noted that full lateral restraint is provided to the top flange of the beam, which reduces the tendency of the beam to flexurally torsionally buckle, when the top flange is in compression. This is the case for most of the beams on the floors. However, at the same time, when the beams are cantilevered (such as at the outer edge of each floor), the bottom flange will be in compression. Therefore, the slab will provide no restraint to prevent the torsional buckling mode. Therefore, a deep UB section was chosen to provide sufficient bending capacity.
FIGURE 3. a) Isometric view of head structure b) Head structure elevation

FIGURE 4. Typical plan view of the head structure
The following describes the methodologies involved to determine the different loads which would be imposed on the structure to run in Strand7.

The Dead Load of the Floor

The dead weight of the floor was calculated over the 12 floors of the head structure. The slab depth was 220mm thick. An additional dead load of 1kPa to account for services and furnishing was also added. Thus, the dead load of the floor is:

\[ P = 0.22 \times 25 + 1 = 6.5kPa \]  

The Dead Weight of the Beams and Columns

The dead weight for the beams were calculated by finding the mass per metre of both the columns and beams (provided by the One Steel Manufacturing Catalogue). The weight was found multiplying the mass per length by gravity. The beam chosen was a 460UB82.1 which has a mass per metre as 82.1kg/m:

\[ W_{beams} = \frac{mass}{length} \times g = 82.1 \times 9.81 = 0.81kN / m \]  

Similarly, the columns are calculated using a 350WC230 section, which has a mass per metre of 158kg/m.
The Live Load on the Floor

The live load was found from AS1170.1 (2007) to be 2 kPa for restaurants. Therefore,

\[ Q = 2kPa \]  \hspace{1cm} (4)

The Maximum Design Load on the Floor

The maximum design load for the floor can be found using the load combinations in AS/NZS1170.0. The worst case was given by a combination of the pressure from the floor and live load as well as the dead load from the beams and columns.

\[ \text{Pressure} = 1.2G + 1.5Q = 1.2(6.5) + 1.5(2) = 10.8kPa \]  \hspace{1cm} (5)

\[ \text{Line..Load} = 1.2(0.81 + 2.26) = 3.68kN/m \]  \hspace{1cm} (6)

Wind Loading on the Tower

The wind load was applied as a pressure to the plates on one side of the building. The velocity profile, varying with height was found in accordance with AS/NZS1170.2 (2011). The terrain category was taken to be 2 as Tehran’s skyline is similar to the profile described by TC2. The velocity of the wind flow was established from the mathematical expressions developed from the Deaves and Harris model (D&H Model, 1978). This model is based on full scale data and the mean velocity flow profile in strong winds and is derived using the logarithmic law. Equation 7 below can be found in the supplementary document for AS/NZS1170.2

\[
V_z \approx \frac{u^*}{0.4} \left[ \log_e \left( \frac{z}{z_0} \right) + 5.75 \left( \frac{z}{z_g} \right) - 1.88 \left( \frac{z}{z_g} \right)^2 - 1.33 \left( \frac{z}{z_g} \right)^3 + 0.25 \left( \frac{z}{z_g} \right)^4 \right]
\]  \hspace{1cm} (7)

Where the mean velocity is based on a mean gradient wind speed of 50 m/s. \( \bar{V}_z \) is the design hourly wind speed at height \( z \), \( u^* \) is the friction velocity which is a non-dimensional measure of the surface shear stress, \( z_0 \) is the roughness length and \( z_g \) is the gradient height which are experimentally found parameters which vary with the terrain category. Once the mean velocity has been found, it is used to find the gust wind speed which is given by Eq. 8.

\[
V = V_z \left[ 1 + 3.7 \left( \frac{\sigma_v}{V_z} \right) \right]
\]  \hspace{1cm} (8)

Where \( \sigma_v \) is the fluctuating wind speed (standard deviation) of the flow and is given by Eq. 9.

\[
\sigma_v = 2.63\eta u^* \left[ 0.538 + 0.09\log_e \left( \frac{z}{z_g} \right) \right]^{\eta/6}
\]  \hspace{1cm} (9)

Where \( \eta \) is defined by Eq. 10 below.

\[
\eta = 1.0 - \left( \frac{z}{z_g} \right)
\]  \hspace{1cm} (10)

The velocity profile was then converted to a pressure using Eq. 11 found in AS/NZS1170.2.

\[
P = \frac{1}{2} V_{des}^2 C_{fig} C_{dyn}
\]  \hspace{1cm} (11)
C_{fig} was taken as 0.7 (for a cylindrical cross section) and C_{dyn} was taken as 1. The velocity and pressure profiles are shown below in Fig. 6a and 6b.

![Velocity Profile](image)

![Pressure Profile](image)

**FIGURE 6.** (a) Wind velocity profile (b) Pressure profiles acting on the structure

It can be seen in Fig. 6b that the pressure profile cannot be accurately modelled by a single straight line equation. Therefore, the curve was separated into four separate approximate linear sections. The pressure equations were applied to the model in Strand7 as plate loads in the global horizontal direction varying with height z. To increase accuracy, the head structure was modelled with 30mm glass panels which the load was applied to.

**Earthquake loads on the fixities of the building**

The earthquake loads were applied by first conducting the natural frequency analysis to determine the natural vibrations of the building. The natural frequency analysis was run for 20 modes, 17 of which were converged. From this, a load factor vs. period table taken from AS/NZS1170.4 for soil class C, as discussed in the earthquake analysis section.

**Load Paths**

The floor loading is transmitted through the slab to the beams, which is then transferred to the columns. The load then moves down to either the space basket, or the columns below which is transferred to the concrete core shaft, which is finally taken to the foundation. This can be seen in red in Fig. 7. The wind loading acting on the structure will act on the glass panels which will be transmitted to the beams and columns. The loads will then move through the beams, columns and space basket until it reaches the concrete core, which transmits the loads to the foundation. The wind acting on one panel can be seen in Fig. 7 in blue, this shows how the load moves through the structure when acting at that particular level.

![Load Paths](image)

**FIGURE 7.** Wind and gravity load paths through the structure
NUMERICAL ANALYSIS

The model was created by modelling one quarter of the tower and then mirroring it about the vertical axis to create the full model. The first section was to model the concrete core, shown by the yellow plates in Fig. 8a. Using the cylindrical coordinate system, the nodes of the concrete shaft were input and connected with beam2 elements. The beams were extruded up a height of 250m, the height at which the head structure begins, creating plates members (plate members will be subdivided later to create square mesh).

The core shaft for the head structure, shown as the green plates below in Fig. 8a was modelled. The nodes for the coordinates of the first floor were drawn. These nodes were linked with beam2 elements, shown in red in Fig. 8a. Quad4 plate elements were drawn between the beams. The plates and beams for the first floor were subdivided to create squarish elements which can be seen in Fig. 8b. It is to be noted to ensure that the nodes for the beams and the plates line up, to guarantee that the mesh is compatible. The columns are modelled by extruding the according nodes upwards by a height of 4m. Lastly, the nodes on the outermost part of the floor were adjusted to an absolute radial value. Therefore, a curvilinear approach to the mesh was utilized. The final mesh for the first floor is shown by Fig. 8b for one quarter of the building.

FIGURE 8. (a) Beam layout for first floor, (b) First floor mesh, (c) Second floor mesh, (d) Spire mesh
The second floor was generated by copying all plates and beams for the first floor to a height of 4m above the first floor. From this, the extra nodes, beams and plates were input in the radial direction, shown by Fig. 8c. Again, this process was repeated, adding the necessary plates and beams as the floor number increased, in order to match the structural design of the head structure. Once all twelve floors were completed, the spire was added to the top of the building. This involved drawing elements similar to how the core was modelled. From this, these beams were extruded upwards to create plates and subdivided accordingly, as shown by Fig. 8d. This was then done again at a smaller diameter, to represent the spire getting narrower as the height increases.

Lastly, the yellow plates shown in Fig. 8a were subdivided to match the mesh for the green plates. The quarter section of the model is now ready to be copied and mirrored about the vertical axis in order to create the full model. The total model involved roughly 50000 plates and 4000 beams, with the mesh size equating to roughly 1m x 1m squares. Although this mesh will lead to a substantial computational time for the different solvers in Strand7, it assists in providing a more realistic stress distribution, especially in high gradient stress zones.

With regards to the restraints for the model, the only restraints provided were fully fixed restraints to the bottom nodes. All other nodes in the model were left as free. It should be noted that the vast majority of the modelling was performed using the cylindrical coordinate system, to allow the use of radial measurements. This assisted in ensuring an appropriate mesh for the model, to increase the accuracy of the tower design.

Once the model was created with the associated boundary conditions, the properties were allocated for each member type. This involved using the Structural Steel library available in Strand7 to allocate all steel sections for beams and columns. The concrete members including the slabs and the core shaft were then chosen as the appropriate sizings and characteristic strengths.

Figure 9a shows an elevation view of the full mesh, along with close-up views of the meshing at the connections of the slab and the core shaft and a view of the full head structure. Figure 9b shows an isometric view of the concrete core shaft and the mesh used. As mentioned earlier, it is critical that the mesh used has no incompatibilities, as this will lead to inaccurate results for the stress distributions.

**FIGURE 9.** (a) Modelling the structure in Strand7, (b) Meshing of the concrete core

Member sizes of the main elements of the structure are displayed below in Table 3.
TABLE 3. Member sizes and modelling technique for the elements used in the model

<table>
<thead>
<tr>
<th>Structural Element</th>
<th>Material/Size</th>
<th>Strand7 Modelling Technique</th>
</tr>
</thead>
<tbody>
<tr>
<td>Head structure columns</td>
<td>Steel – 350WC230</td>
<td>Beam 2</td>
</tr>
<tr>
<td>Head structure beams</td>
<td>Steel – 460UB82.1</td>
<td>Beam 2</td>
</tr>
<tr>
<td>Slabs</td>
<td>40MPa 220mm deep</td>
<td>Quad 4 plate</td>
</tr>
<tr>
<td>Foundation</td>
<td>40MPa 1.5mm thick</td>
<td>Quad 4 plate</td>
</tr>
<tr>
<td>Core Shaft</td>
<td>40MPa 1.5mm thick</td>
<td>Quad 4 plate</td>
</tr>
<tr>
<td>Spire</td>
<td>40MPa 0.5m thick tapered</td>
<td>Quad 4 plate</td>
</tr>
<tr>
<td>Steel basket</td>
<td>500WC440</td>
<td>Beam 2</td>
</tr>
</tbody>
</table>

Linear Static Analysis

The linear static analysis involved two loading cases; dead load and live load. The dead load was performed by applying gravity to the structure as well as the additional 1kPa imposed load. The live load involved applying a 2kPa face pressure to the slab plates in the head structure (which is the live loading for restaurants according to AS1170.1, 2002). A loading combination of 1.2G + 1.5Q was then applied. The deflection contours are shown by Fig. 10.

FIGURE 10. Linear Static Analysis contour for load case 1.2G+1.5Q

<table>
<thead>
<tr>
<th>Load Case (plate analysis)</th>
<th>Maximum Deflection (mm)</th>
<th>Maximum Stress (MPa)</th>
<th>Maximum Bending Moment (kNm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>G</td>
<td>52</td>
<td>5.8</td>
<td>76</td>
</tr>
<tr>
<td>Q</td>
<td>8.8</td>
<td>2.0</td>
<td>29</td>
</tr>
<tr>
<td>1.2G+1.5Q</td>
<td>75</td>
<td>9.9</td>
<td>141</td>
</tr>
</tbody>
</table>

As shown by Table 4, the maximum deflection is 75mm at the cantilevered sections of the slabs. It should be noted that this is a global deflection of 75mm. That is, the deflection of the core section is 45mm. The local deflection of the slab was found to be 30mm, which is within the deflection limits stated by AS1170.1. The stresses in the concrete slab, particularly for the combined case are significant. This could cause cracking of the section, meaning that slab depth could arguably be increased to gain a higher capacity. However, this does create issues with having excessive masses high up in the building and thus increases the moment effects in an earthquake.
Static Wind Analysis

The static wind analysis involved applying the global pressure that was discussed previously. The load was applied to the plates as pressures, in the global x direction, varying with height \( z \) (m). These results are shown by Fig. 11a and 11b. As shown by the results, the largest deflection for the plates is 873mm.

![Image](image1.png)

**FIGURE 11.** (a) Wind Loading applied to head structure, (b) Deflection results

Natural Frequency Solver

In real situations, structures are subject to random fluctuations in wind speed that will interact with the vibrational modes. This results in a dynamic wind load which varies in time acting on the structure. Therefore, it is not realistic to solely base a tall building design on a static analysis. In particular for high-rise buildings, the wind loads and earthquake loads activate the structures natural frequencies which in turn produce oscillations which must be damped. Instead, the use of a dynamic analysis provides a closer representation to how wind acts on structures, particularly for those outside the scope of the Australian Design Codes.

For any multistory structure the matrix element equation becomes (Alonso-Marroquin, 2016).

\[
M\ddot{U} + KU = F(t) \tag{12}
\]

Where \( M \) is the matrix equation, \( \ddot{U} \) is the acceleration vector, \( K \) is the stiffness matrix, \( U \) is the displacement vector and \( F(t) \) is the time dependent force acting on the structure. Normally, a transient solver will provide a solution to Eq. 12. However, due to the enormity of the Milad Tower an alternate method can be used which is known as the Spectral Response Solver. The theory and methodology for this is explained in detail below.

A Natural Frequency solver is first used to determine frequencies, modes and mode shapes which can later be utilized in a Dynamic Wind and Earthquake Analysis. To do so, an oscillatory function with a frequency \( \omega \) is substituted into Eq 12. This takes the form of:

\[
U(t) = U_0 e^{i\omega t} \tag{13}
\]

In terms of the mathematical process of finding the natural frequencies, Eq. 12 is used and the external forces and damping forces acting on the structure are taken as 0:

\[
\det(K - \omega^2 M) = 0 \tag{14}
\]
Solving this equation yields a polynomial function of $\omega^2$, which is the natural frequency of a mode. The order of this equation is equal to the number of degrees of freedom for mass displacements in the structure. This will then provide the natural frequencies in the structure. The natural frequencies of the structure as calculated in Strand7 can be seen in Table 5.

**TABLE 5.** The converged modes from natural frequency of the structure

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency (Hz)</th>
<th>Period (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.151</td>
<td>6.617</td>
</tr>
<tr>
<td>2</td>
<td>0.151</td>
<td>6.617</td>
</tr>
<tr>
<td>3</td>
<td>0.383</td>
<td>2.612</td>
</tr>
<tr>
<td>4</td>
<td>0.383</td>
<td>2.612</td>
</tr>
<tr>
<td>5</td>
<td>0.807</td>
<td>1.240</td>
</tr>
<tr>
<td>6</td>
<td>0.807</td>
<td>1.240</td>
</tr>
<tr>
<td>7</td>
<td>1.238</td>
<td>0.808</td>
</tr>
<tr>
<td>8</td>
<td>1.242</td>
<td>0.805</td>
</tr>
<tr>
<td>9</td>
<td>1.242</td>
<td>0.805</td>
</tr>
<tr>
<td>10</td>
<td>2.054</td>
<td>0.487</td>
</tr>
<tr>
<td>11</td>
<td>2.054</td>
<td>0.487</td>
</tr>
<tr>
<td>12</td>
<td>2.601</td>
<td>0.385</td>
</tr>
<tr>
<td>13</td>
<td>2.601</td>
<td>0.385</td>
</tr>
<tr>
<td>14</td>
<td>3.070</td>
<td>0.326</td>
</tr>
<tr>
<td>15</td>
<td>3.554</td>
<td>0.281</td>
</tr>
<tr>
<td>16</td>
<td>4.053</td>
<td>0.247</td>
</tr>
<tr>
<td>17</td>
<td>4.201</td>
<td>0.238</td>
</tr>
</tbody>
</table>

By finding the natural frequencies $\omega$, the different mode shapes can be found. By replacing Eq. 13 above into Eq. 12, we obtain Eq. 15 below. This equation is solved for $U_i$ for each mode of the structure.

$$ (K - \omega^2 M)U_i = 0 \quad (15) $$

$$ U_i M U_i = 1 \quad (16) $$

$U_i$ is solved utilizing the relationship shown in Eq. 16: This is the simplified process for how Strand7 solves natural frequencies of the structure.

**Dynamic Wind Analysis**

The second step of the analysis involves generating a Power Spectral Density (PSD) curve for the wind force. A PSD curve gives a measure of the winds intensity within the frequency domain. Utilising the Strand7 webnotes, it was possible to develop an equation for the curve based off a number of variables specific to the Milad Tower, as shown by Eq. 17.

$$ S_p(n) = 4(C_d \rho)^2 \frac{x^2}{\bar{V}^4} k \frac{x^2}{n(1 + x^2)^{3/2}} \quad (17) $$

Where: $S_p(n)$=wind pressure factor, $C_d$=drag coefficient, $\rho$=air density, $\bar{V}$=wind velocity, $k$=surrounding surface roughness parameter, $x=\frac{1200h}{\bar{V}}$ and $n$=frequency. The resulting equation yields the Wind Force PSD as shown in Fig. 12.
This graph is then included in the Strand7 spectral response analysis as a factor vs frequency table. The Wind Force PSD has the effect of factoring the wind effects at certain frequencies. Therefore, when the building’s natural frequencies coincide with the frequencies that are heavily factored, the effect of the wind loading will be significantly greater. The peak shown by the graph occurs at approximately 0.1Hz, which coincides with the Fundamental Mode of frequency of the tower, as shown by the natural frequency solver. Therefore, the response of the building to dynamic wind will be more intense. It is important to note that a number of different factor vs frequency graphs exist, and they are greatly dependent on the location of the wind. Therefore, there is a degree of uncertainty in this calculation, however it provides a platform for further research into the area of wind dynamics. Other effects of wind that should be considered is the crosswind excitation which occurs due to a process called vortex shedding along the height of the building.

Utilising the Strand7 spectral response solver with the included natural frequencies, factor vs frequency graph, and the applied wind pressures (as opposed to base acceleration for earthquake) we are able to calculate the displacements and stresses of the various modes of vibration. Utilising the combined SRSS case it is then possible to determine the peak displacements, stresses and their locations within the structure as shown in Fig. 13. As shown by the results, the largest deflection for the plates is 2.74m.

**FIGURE 12. Factor vs. Frequency graph for dynamic wind analysis**

**FIGURE 13. Deflection and peak stress locations of spectral Analysis case SRSS**
Earthquake Analysis

The earthquake design involved using the previously determined natural frequency vibrational modes. From this, a spectral analysis was performed. The spectral analysis solves ordinary differential equations by utilizing harmonic oscillator with the natural frequency and damping ratio to an imposed acceleration. Both SRSS and Modal analysis were used for the earthquake analysis. Spectral analysis involved inputting a relationship between spectral ordinates $C_h(T)$ and period (T) (factor vs. frequency). The soil category was chosen to be Class C as this is the most similar geological conditions, in accordance with AS1170.4 (2007). This is shown by Fig. 14.

![Figure 14. Spectral Response curve (AS1170.4, 2007)](image1)

The $C_h(T)$ factor is the spectral shape factor for the period of T seconds. This varies with the type of soil class that the structure sit on. The earthquake was then factored by 0.22 of the value of gravity and applied in the x direction to simulate an applied earthquake along a single axis. Since the model is axisymmetric, the effect would be the same from the y direction. The earthquake was applied as a base acceleration, to represent the ground moving. The results for the plate deflection in the x direction are shown by Fig 15.

![Figure 15. Earthquake Deflection Contour Plot](image2)
Figure 16 shows the deflected shape for a number of different natural frequencies. Due to the height of the tower, there are many different failure modes, all of which need to be considered during the design process.

The modal analysis finds the natural frequencies of the structure and the shape of each mode at that frequency. As the mode shapes are orthogonal, they can be expressed as a linear combination of these eigenvectors. Using Eq. 13 and 14, the mode shapes $U_i$ can be found and the mass participation factor for each node subsequently calculated.

$$\Gamma = \frac{U_i^T M r}{U_i^T M U_i}$$

Where $r$ is taken as an arbitrary vector in the number of degrees of freedoms dimensional space. At every mode, mass participation will take place and will be associated with a frequency. The mass participation is the amount of mass which has contributed in that mode. All modes with a low mass participation can be ignored. The accumulated mass participation is expected to be greater than 90%. A value of 84.31% was obtained through the calculations on Strand7 which for the design of a tall structure is relatively close to the expected outcome. The results of mass participation can be seen below in Table 6.

**TABLE 6. Mass participation of the structure following earthquake analysis**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Participation (%)</th>
<th>Mode</th>
<th>Participation (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>54.854</td>
<td>10</td>
<td>7.221</td>
</tr>
<tr>
<td>2</td>
<td>0.012</td>
<td>11</td>
<td>0.001</td>
</tr>
<tr>
<td>3</td>
<td>2.317</td>
<td>12</td>
<td>0.27</td>
</tr>
<tr>
<td>4</td>
<td>0</td>
<td>13</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>18.321</td>
<td>14</td>
<td>0</td>
</tr>
<tr>
<td>6</td>
<td>0.004</td>
<td>15</td>
<td>0.001</td>
</tr>
<tr>
<td>7</td>
<td>0</td>
<td>18</td>
<td>0</td>
</tr>
<tr>
<td>8</td>
<td>1.302</td>
<td>19</td>
<td>0.003</td>
</tr>
<tr>
<td>9</td>
<td>0</td>
<td>20</td>
<td>0.003</td>
</tr>
<tr>
<td></td>
<td>TOTAL MASS PARTICIPATION</td>
<td>84.31%</td>
<td></td>
</tr>
</tbody>
</table>
STRUCTURAL DESIGN

Maximum distributed load

Dead Load and Live Load:
The dead load is calculated from the self-weight of the concrete slab. An additional 1 kPa is conservatively added for services and furnishing.

\[ q_D = 25 \times 0.22 + 1 = 6.5kPa \]  \hspace{0.5cm} (19)

\[ q_L = 2kPa \]  \hspace{0.5cm} (20)

Since it is a 2 way slab, tributary areas are chosen appropriately equating to one quarter of the span. The tributary area is therefore equal to 2.25 metres. This is multiplied with the loading factors to convert the pressure into a distributed load.

\[ UDL_D = 6.5 \times 2.25 = 14.6kN/m \]  \hspace{0.5cm} (21)

\[ UDL_L = 2 \times 2.25 = 4.5kN/m \]  \hspace{0.5cm} (22)

The maximum loading combination for the distributed load was found in accordance with AS1170.0. This was considered as 1.2G + 1.5Q.

\[ Q_{1.2G+1.5Q} = 1.2 \times 14.6 + 1.5 \times 4.5 = 24.27kN/m \]  \hspace{0.5cm} (23)

Maximum Axial Load

The maximum axial load found to be acting on a member is the column situated at the bottom of the steel basket. The tower head is 12 stories tall and therefore there are 11 stories acting on the column. The slab, beam and predicted column size are to be accounted for when considering the axial load.

Column:
Assuming column is a 310UC158 and the storey height is 4 metres:

\[ AF_{column} = 158 \times 4 = 6.32kN \]  \hspace{0.5cm} (24)

\[ AF_{C11} = 11 \times 6.32 = 69.52kN \]  \hspace{0.5cm} (25)

Beam:
As calculated below, the chosen beam is a 610UB125, and is located twice along a 9 metre span.

\[ AF_B = 125 \times 9 = 11.25kN \]  \hspace{0.5cm} (26)

\[ AF_{B11} = 11 \times 11.25 \times 2 = 247.5kN \]  \hspace{0.5cm} (27)

Slab:
For the slab calculated below, the depth corresponded to 220 mm for an area of 91 m².

\[ AF_s = 0.22 \times 91 \times 25 = 400kN \]  \hspace{0.5cm} (28)

\[ AF_{s11} = 11 \times 400 = 4400kN \]  \hspace{0.5cm} (29)

Therefore, the maximum total axial force acting on a column is given by:

\[ AF_{max} = 69.5 + 247.5 + 4400 = 4717kN \]  \hspace{0.5cm} (30)

Beam Design

Using the maximum distributed load calculated previously, the formula for the maximum bending moment for a continuous support over the span of 3 columns is:

\[ M^* = 0.2wl^2 = 0.2 \times 24.3 \times 9^2 = 393.66kNm \]  \hspace{0.5cm} (31)

As the top flange of the beam is fully restrained by the concrete slab, \( M_b \) shall be taken as \( M_s \). This is in accordance with clause 5.3 of AS4100. The formula for section capacity is given by:

\[ M_s = f'_s Z_e \]  \hspace{0.5cm} (32)

Rearranging this for the section modulus
\[ Z_e = \frac{M_s}{f_y} = \frac{393660000}{300} = 1312.2 \times 10^3 \text{mm}^4 \]  

Therefore, a member with a greater section modulus than 1312.2 must be chosen. The most economic beam to be chosen in accordance with section modulus would be 460UB82.1.

**Column Design**

The maximum axial force acting on any member was calculated to be 4717 kN. For strength design, this must be less than the nominal section capacity as well as be less than the nominal member capacity. All values and equations were extracted from AS4100.

**Nominal section capacity:**

\[ N^* \leq \phi N_s \]  
\[ N_s = k_f A_y f_y = 1.0 \times 20100 \times 300 = 6030kN \]  

These factors were obtained from onesteel for a 310UC158, as an initial estimate.

**Nominal Member Capacity:**

\[ N_c = \alpha_c N_s \]  
\[ \lambda_n = \left( \frac{l_e}{r} \right) \left( \sqrt{k_f} \right) \left( \frac{f_y}{250} \right) = 40.33 \]  

The effective length was taken as the 4 metre height multiplied by a k value of 1. The radius and yield strength was obtained from onesteel.

\[ \alpha_b = 0 \]  

Taken as 0 for a hot rolled UC.

Therefore, from table 6.3.3(3):

\[ \alpha_c = 0.905 \]  
\[ N_c = 0.905 \times 6030 = 5457.15kN \]  
\[ \phi = 0.9 \]  

Hence,

\[ \phi N_s = 5427kN \]  
\[ \phi N_c = 4911.435kN \]  

Since the maximum axial force is less than both the section and member capacity, the chosen structural member for the column is 310UC158. However, although the column was designed for strength, the suggested column caused large deflections in STRAND7. For a high profile tower an extra safety factor was considered and the selected column which led to a reduction in the total deflections was a 350WC230.

**Slab Design**

The deemed to comply approach for the design of a slab designs for both serviceability and for strength in accordance with clause 9.3.4.1, if the below equation is satisfied.

Initially, a slab depth of 210 mm was selected to determine the loading combinations to apply to the slab. The dead load had an additional 1 kPa added to it for services and furnishing.

\[ q_g = 0.21 \times 25 + 1 = 6.25kPa \]  
\[ q_l = 2kPa \]
The 2 kPa was chosen in accordance with AS1170.1, table 3.1, for a typical restaurant loading.

The actual slab area is a complicated shape, however for simplicity, the tributary area for the slab was assumed to be a 9 by 9 metre square.

The method used was a deemed-to-comply approach:

\[
\frac{L_{ef}}{d} \leq k_3 k_4 \left[ \frac{\left( \frac{\Delta}{L_{ef}} \right) 1000 E_c}{F_{d,ef}} \right]^\frac{1}{3}
\]

(46)

\( \frac{\Delta}{L_{ef}} \) is limited to 1/500 where provisions are made to minimise the effect of movement.

\( k_3 \) is taken as 1 given by 9.3.4.2(a)

\( k_4 \) is taken as 3.6 for all four edges of the slab being continuous (Table 9.3.4.2)

\( E_c \) is taken as 32,800 MPa given by Table 3.1.2

\( F_{d,ef} = (1.0 + k_{cs})g + (\psi_s + k_{cs}\psi_l)q \)

And, \( k_{cs} = [2 - 1.2(A_{ac}/A_{st})] \geq 0.8 \)

For a conservative approach, \( k_{cs} \) is assumed to be 2.0 (i.e. no tensile reinforcement).

\( \psi_s \) is taken as 0.7 and \( \psi_l \) is taken as 0.4 for retail and office space (AS/NZS1170.0 Table 4.1)

Therefore,

\[ F_{d,ef} = (1.0 + 2.0)6.25 + (0.7 + 2.0 \times 0.4)2 = 21.75 \]

\( L_{ef} = 9000 \text{mm} \)

Therefore,

\[
d \geq \frac{9000}{1 \times 3.6 \times \left[ \frac{1}{500} \cdot 1000 \times 32800}{21.75} \right]^\frac{1}{3} = 173 \text{mm}
\]

(47)

Assuming N16 bars will be used, with a 30 mm cover for a conservative approach:

\[
D \geq 173 + \frac{16}{2} + 30 = 211 \text{mm}
\]

(48)

The final depth of the slab will therefore be taken as 220 mm.

Shrinkage effects govern the design, and for moderate crack control, \( p_{min} = 0.0035 \) (AS/NZS3600 9.4.3.4). Therefore the minimum reinforcement spacing required is:

\[
p_{min} = \frac{A_{st}}{s} = \frac{\pi \times 8^2}{0.0035 \times 173} = 332 \text{mm}
\]

(49)

(50)

Taking N16 bars as reinforcement, spaced at 300 mm centres, this corresponds to

\[
\frac{\pi \times 8^2 \text{mm}^2}{0.3 \text{m}} = 670 \text{mm}^2 / \text{m}
\]

(51)

Therefore, the final design of the slab is 220 mm depth with N16 bars at 300 mm spacing’s.
CONCLUSIONS

The results from the wind analysis and linear static analysis of the tower were as hypothesized, however the excitation due to earthquake loading was the governing loading case. The structural systems found in the structure are tapered shear walls and a steel basket head structure. These systems had realistic results in terms of stress, deflection and bending moments. The mesh size and quality that was generated allowed for a more accurate stress distribution for each loading case. The selected members that were designed such as beams, columns and slabs were done with a degree of accuracy and minimized deflections in the tower.

The beam and column members in the head structure were derived by hand calculations to be 310UC158 and 460UB82.1 respectively (in accordance with AS/NZS 4100). However due to excessive deflections encountered in the analysis of the structure, the column members were slightly modified to be 350WC230. The slab depth was initially designed to be 220mm and this depth was found to be sufficient after running the analysis.

The wind loads were derived in accordance with AS/NZS1170.2. The velocity at each storey were found employing the Deaves & Harris model that was based on full scale data. These velocities were then converted to a corresponding pressure utilizing the quasi-static assumption.

The greatest local slab deflection was found to be 69mm in the largest span. The deflection at the top of the tower from the static wind analysis was found to be 1m. The deflection at the top of the tower due to the dynamic wind analysis was found to be 2.75m. And the total deflection found from the greatest mass participation mode in earthquake analysis was found to be 5m. The governing loading case is therefore the earthquake loading as this yields the greatest deflection of the structure.

Further analysis could be undertaken to refine the accuracy of this analysis and yield more accurate results by using a denser mesh and higher order elements. However due to the scope of this analysis, the model has been simplified.

REFERENCES

AS/NZS1170.2 – Australian/New Zealand Standards for Structural Design Actions Commentary (Supplement to AS/NZS 1170.2), Part 2: Wind Actions, 2002
Holmes JD. Wind loading of structures. CRC Press; 2015 Jan 27.