Numerical Methods in Civil Engineering

Dynamics of Structures 2016

Fernando Alonso-Marroquin

THE UNIVERSITY OF SYDNEY

Strand7 Software

ISBN 978-0-9944287-3-8

Content

Preface

We proudly present the new edition of the work by our Civil Engineering students at The University of Sydney on finite element modelling using Strand7 software. Our focus this year is the modelling of the response of landmark buildings under dynamics actions.

Since 1988, the Strand7-Finite Element Analysis Software has been developed by Strand7 Pty Ltd in Sydney, Australia. Early development on a predecessor of Strand7 was undertaken by academics of the University of Sydney and University of New South Wales in the mid 80's. Today Strand7 is an internationally renowned, well-established, finite element software. Strand7 has strong and versatile capabilities for dynamics analysis. Extremely efficient in-build solvers in Strand7 allows linear and non-linear transient dynamics, natural frequency, spectral response, quasistatic analysis, and linear and non-linear transient heat transfer analysis.

Here we present a selection of the Strand7 dynamic analyses performed on highly complex civil engineering structures. We start with a dynamics analysis of the Bao'an stadium in Shenzhen, China. The transient dynamics solver allows calculations of the earthquake response in this particular structure where standard spectral analysis shows slow convergence. We perform a fully dynamics analysis of the BMW Tower. This emblematic suspending structure in Munich, Germany proved the capabilities of Strand7 to model highly complex geometries in buildings under fire and wind loads. The Golden Gates Bridge's paper provides an excellent investigation of the bridge response to dead load, static load, wind load, and spectral seismic loads. The response of the cables of this enormous structure is investigated in detail using dedicated cable elements available in Strand7. The project on the Milad Tower in Tehran, Iran, provides the dynamics response of this tall building under wind load. The analysis is based on the Davenport spectral response formulation, which is much less expensive than computational fluid dynamics calculations.

Some contributions aim to face current challenges in our society: we present a blasting analysis on the iconic Eiffel Tower in Paris, France. A superb investigation based on original construction drawings largely replicates the existing tower. Aiming to bring wilderness to large cities, we explore the use of prestresses concrete slabs in vertical forests such as the Green Façade in Milan, Italy. Finally, the energy balance analysis of the collapse of the Twin Towers in New York, USA, confirms that the energy released by the aircraft fuel was sufficient to produce a localised buckling collapse after 2hrs of heat exposure.

Special thanks to Strand7 Pty Ltd and The University of Sydney to sponsor the production of this volume.

Contributors

Student's contributors:

David Knight
Rowan Whitefield
Eric Nhieu
Joaquin Indacochea-Beltran
Pearl Elgindy
Elaine Lee
Thiviya Vignesh
Lachlan Horlyck
Kieran Hayes
Ryan Caetano
Wenning Sun
Mingxin Li
Yinong Han
Moqi Wang
Thomas Game
Cameron Vos
Rafid Morshedi
Rebecca Gratton
Edwin Wilhelm
Mitchell Ford
Darren Coelho
Lachlan Lawler
Kaiqi Zhu
Kang Xu

Supervisors:

Peter Ansourian
Faham Tahmasebinia
Fernando Alonso-Marroquin

Designers:

Coraline Chiew
Mandana Abbasi

Bao’An Stadium, Shenzhen

Transient Dynamic Analysis

Description

Bao’An stadium is an iconic structure that has structural columns inspired by local bamboo forests, intended to create a prominent feature in Shenzhen China. The structure has a circular shape with an outer diameter of 236m and an inner diameter of 128m. A lightweight roof structure with a polytetrafluoroethylene membrane is cantilevered 54m to provide shelter for spectators. The stadium is a complex structure due to the inclusion of the inner ring cables that have a pre tension of 3600 kN in the top cable, and 1800 kN in the bottom cable. These cables are supported through a lightweight truss system comprising of thin cables that extend to the external columns which also support a large compression ring. The columns are large circular hollow sections ranging in sizes from 550 mm to 800 mm. The stadium behaves similar to a bicycle wheel, the trusses equivalent to tension spokes of the wheel and the rim representing the outer compression ring.

Source:

Knight, D., Whitefield, R., Nhieu, E., Tahmasebinia, F., Ansourian, P., & Alonso-Marroquin, F. (2016, August). Transient dynamic analysis of the Bao’An Stadium. In NUMERICAL METHODS IN CIVIL ENGINEERING: Dynamics of Structures 2016 (Vol. 1762, No. 1, p. 020001). AIP Publishing.

Aim

This study involves a simplified finite element model of Bao’An stadium using Strand7 to analyse the effects of deflections, buckling and earthquake loading. Modelling the cantilevers of the original structure with a double curvature was problematic due to unrealistic deflections and no total mass participation using the Spectral Response Solver. To rectify this, a simplified symmetrical stadium was created and the cable free length attribute was used to induce tension in the inner ring and bottom chord members to create upwards deflection. Further, in place of the Spectral Response Solver, the Transient Linear Dynamic Solver was inputted with an El-Centro earthquake.

BAMBOO FOREST: BAO'AN STADIUM

THE UNIVERSITY OF SYDNEY

David Knight, Eric Nhieu, Rowan Whitefield

Fernando Alonso-Marroquin, Peter Ansourian, Faham Tahmasebinia
School of Civil Engineering

AIMS

- Run dynamic earthquake analysis on Bao'An Stadium as per the Chinese Code for Seismic Design of Buildings GB 50011-2010 using Strand7
- Reduce deflections of the long span cantilever supporting the roof by using tensioned cables
- Analyse deflection, buckling and earthquake using Linear Static, Linear Buckling and Linear Transient Dynamic Solvers
- Create simplified symmetric 3D model with online editing tool

BACKGROUND

The design for Bao'An Stadium was inspired by the local bamboo forests in southern China. The exterior double row column layout also supports the cantilevered roof substructure. The roof is comprised of a structural membrane fabric connected to tensioned cable elements.

Figure 1:Boa'An Stadium (GMP Architekten)

MODELLING

All structural components and their property type are listed in Table 1

Structural Component	Size	Property Type	Element
External Columns	800x50 CHS	Beam	Beam2
Box Girder	200x100x2.4 RHS		
Top Chord of Truss	250x30 CHS		
Vertical Truss Members	400x10 CHS		
Bottom Chord of Truss	200x5 CHS	Cable	
Ring (Top and Bottom)	10 mm Cable		
Roof Membrane	1mm Teflon	Plate	Quad4

Table 1 : Structural Components

Cable Elements

- The bottom chord is pre-tensioned to prevent compressive stresses while the inner ring is in tension to limit deflections
- Pre-tensioning is achieved by selecting Attributes > Beam > Cable Free Length

Dynamic Earthquake Load

- Structures in the Shenzhen region are designed for 0.20g earthquakes as per GB 50011-2010
- Acceleration vs. Time data for a 0.20g El-Centro Earthquake applied in up-down, east-west, and north-south directions

RESULTS

The design capacity for all critical members was calculated as per AS4100 for flexure, axial compression/tension and buckling. The Strand7 results in Table 2 indicate all members were satisfactory.

Loading Type	Critical Member	Size	AS4100 Maximum	Strand7 Results
Flexure	Top Chord	250x30 CHS	M_i = 284 kNm	M^* = 226 kNm
Axial Compression	Column	800x50 CHS	φN_c = 15624 kN	N_c^* = 490 kN
Axial Tension (Cables)	Inner Ring	10 mm Cable	φN_t = 2142 kN	N_t^* = 1551 kN
Axial Tension (Beams)	Top chord	250x30 CHS	φN_t = 5971 kN	N_t^* = 225 kN
Deflection	Top Chord	250x30 CHS	δ = 450 mm	δ^* = 133 mm
Critical Buckling Case	Column	800x50 CHS	P_{crit} = 3894 kN	P^* = 710 kN

Table 2 : Comparison of AS4100 and Strand7 Results

Linear Buckling

Buckling	Force (kN)
Perfect Euler	4010
Critical (Non-Gravity)	4039
Critical (Gravity)	3894

Table 3 : Buckling Loads

Linear Static

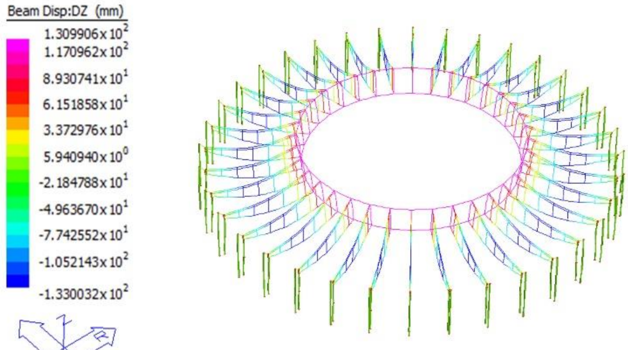


Figure 1 : Deflection for Self-weight

Linear Transient Dynamic

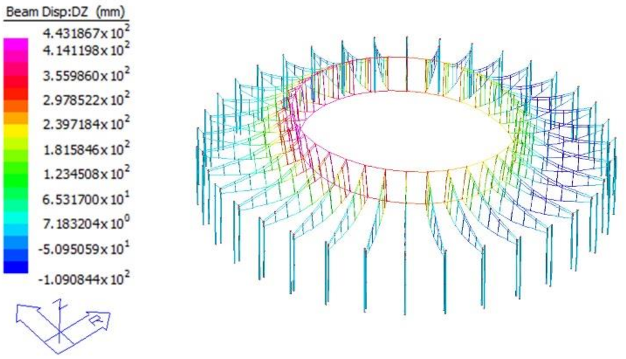


Figure 2 : Deflection for up-down El-Centro Earthquake

CONCLUSIONS

- The simplified symmetric 3D model was successfully created with the online editing tool
- Downward deflection of 133mm in Figure 1 was below AS1170 L/120 deflection limit of 450mm
- Upward deflection of 443mm in the dynamic earthquake in Figure 2 was achieved because of tensioned cables and unique tapered truss geometry
- The axial load in the columns under Linear Transient Dynamic Earthquake is significantly less than critical buckling load. 701 kN < 3894 kN
- Columns do not buckle under a gravity load or combination of gravity and earthquake
- The response of Boa'An Stadium to a 0.20g dynamic earthquake satisfied deflection requirements for AS1170 and strength requirements for AS4100

REFERENCES

CMC (2010). *Code for Seismic Design of Buildings (GB50011-2010)*. China Ministry of Construction, China Architecture and Building Press: Beijing, China. (in Chinese)

Guo, Y., Wang, K., Sun, W., Tian, G., & Zhang, B. (2013). Research on key structural design problems of the Bao'An Stadium. *Journal of Building Structures*, 5 (1) 11-20.

BMW Tower, Munich

Dynamic Analysis

Description

In the 1970s, world famous Austrian architect Karl Schwanzer designed an avant-garde suspended skyscraper for the new BMW headquarters. The BMW Tower was envisioned to resemble a four-cylinder motor and become a symbol for the recent flourishing success of BMW. The building consists of four pre-stressed concrete cylindrical structures suspended off the ground through the use of cables connected to enormous cross-shaped steel hangers that lie on top of a shear core. The hangers are connected through the central shear core. There is also a technical floor mid-structure, containing trusses which transfer loads between the columns and the cables. With 18 suspended floors, a total of 53000 m^2 of office space is available. This abundant space allowed the recentralization of staff previously dispersed around the region.

Source:

Indacochea-Beltran, J., Elgindy, P., Lee, E., Vignesh, T., Ansourian, P., Tahmasebinia, F., & Marroquín, F. A. (2016, August). Dynamic analysis of the BMW tower in Munich. In NUMERICAL METHODS IN CIVIL ENGINEERING: Dynamics of Structures 2016 (Vol. 1762, No. 1, p. 020002). AIP Publishing.

Aim

The aim of this project was to determine the stresses and deflections of the BMW Tower under static and dynamic loadings by using a 3-D model in Strand7. Ultimately, this analysis helps to understand the nature of suspended structures in relation to the Eurocode building standards. Thermal resistance has also been analysed using Strand7 to simulate a fire scenario and analyse the behaviour of the cable structure, which is the most critical building component.

BMW Tower

Beltran Joaquin, Elgindy Pearl, Lee Elaine, Vignesh Thiviya
Fernando Alonso-Marroquin, Peter Ansourian, Faham Tahmasebinia
School of Civil Engineering
THE UNIVERSITY OF SYDNEY

Background

The BMW Tower is located in Munich Germany and serves as the headquarters for BMW. This structure was created to recentralize staff. Designed by Karl Schwanzer, an Austrian architect, the structure is 99.5m tall with a total useable floor space of 53,000m². It is a suspended structure, with the design resembling the four cylinders of a car engine.

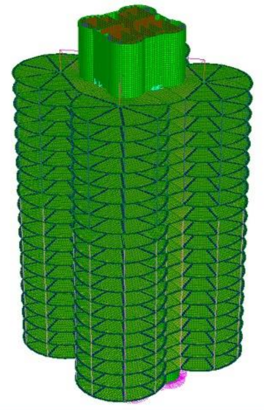
Aim

To undertake static and dynamic analysis of the innovative suspended design , and incorporate effective mesh generation techniques in Strand7. To analyse the influence of load distributions on the stresses and deflection results of the suspended elements and trusses in the structure.

FEM Modelling

Geometrical and Material Properties

	Strand7 Element	Material	Section Geometry	Element Sizes (mm)
Hanger	Beam	Structural Steel	Square	1500 x 1500
Cable	Truss	Structural Steel	Circular Diameter	250
Exterior Column Plate	Plate	Concrete 40MPa	Depth	600
Exterior Column	Truss	Concrete 40MPa	Square	300 x 300
Floor Beam	Plate	Concrete 40MPa	Square	600 x 600
Truss	Beam	Concrete 40MPa	Rectangular	300 x 600
Concrete Slab	Plate	Concrete 40MPa	Depth	150
Outer Shear Wall	Plate	Concrete 40MPa	Thickness	350
Inner Shear Wall 1	Plate	Concrete 40MPa	Thickness	200
Inner Shear Wall 2	Plate	Concrete 40MPa	Thickness	175



Method

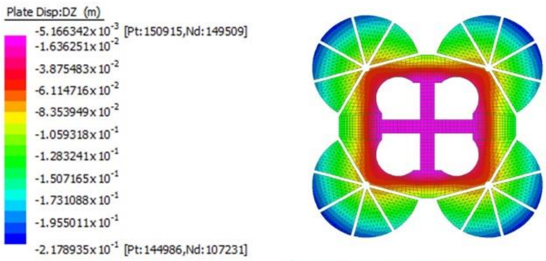
- The structure's nodes and elements were created on AutoCAD based on the BMW Tower floor plan
- These layers were put into Strand7. Columns, cables, shear walls, slabs and the steel hangers were created and grouped. Element properties and attributes were assigned
- The mesh in Strand7 was graded with Quad 4 and Tri 3 elements

Results

Serviceability

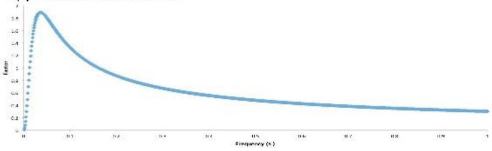
Static deflections were determined using the serviceability load combination: G + Q +W (Dead + Live + Wind)

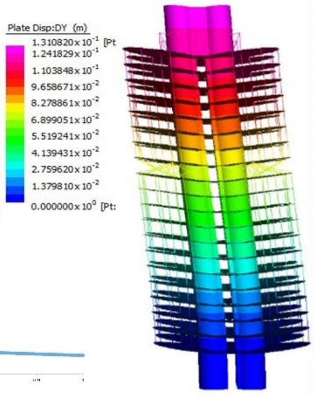
Element	Position	DZ (m)
Column	Base of Column	-0.204
Cable	Base of Cable	-0.174
Beam	Top edge of Beam	-0.174



Dynamic Wind Analysis

A spectral response was derived using vibration modes from a natural frequency analysis.
These modes are used in the spectral response analysis to derive the dynamic response the structure experiences under wind loading.
The normalised Davenport PSD curve (as seen in figure below) is used in conjunction with the static wind load to determine the dynamic response of the structure due to wind pressure loading in the along wind direction (DY). Additionally, 5% damping was applied to the structure



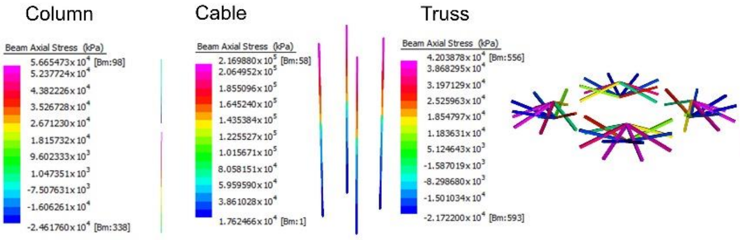


Stresses

Stresses are taken as axial for beam elements and ZZ for plate elements. Worst case load combination 1.35G + 1.35Q + 1.35 W was used.

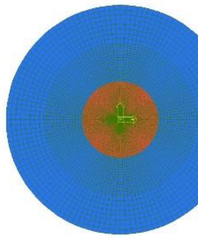
Element	Axial Tension (MPa)	Axial Compression (MPa)
Column	56.65	24.62
Cable	217	-
Truss	42.04	21.72

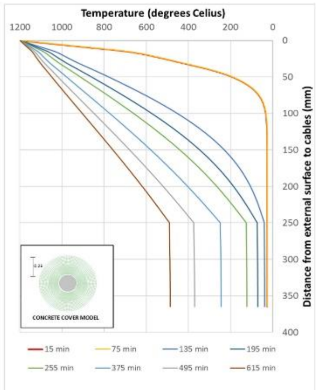
Column Cable Truss



Thermal Loading

A transient heat analysis of the steel cables with 15 minute intervals and initial ambient temperature of 25ºC was analysed. There was a 1200ºC fire loading. The Strand7 model of the steel cable and concrete cover is seen below.





Findings

Static

- Deflections increase down the structure. The truss reduces stress in the lower cable, reducing the deflection at the base of the structure
- The structure as a whole experiences a maximum lateral 37mm deflection in beam and plate elements at the connection between the angled hanger and the shear wall.
- Columns experience compression above the truss and tension beneath
- Cables experience greatest axial tension due to the suspension. This reduces tension further down the structure.
- Trusses successfully redistribute loads through a combination of tension and compression members.

Thermal

- The thermal resistance of the columns is sufficient, withholding loads up to 10.25 hours prior to failure.

Dynamic

- The displacement of the shear core at the height of the building is 13 cm in the along wind direction (DY) while static wind was 2.8cm.

Conclusion

BMW tower's innovative design showcases improvements in engineering capabilities. The member sizes were approximated, however the results shown are reasonable in magnitude. The BMW Tower modeled successfully redistributed the loads from the suspension into the foundation and support as expected. Further analysis is required with realistic data to provide results that are more representative of the physical structure.

References

- 7 Forum, (2016), BMW Hochhaus – BMW Meilenstein, retrieved from https://www.youtube.com/watch?time_continue=555&v=nVDkzqAsRd0
- P Cheimets, (1990), Results of Wind Spectrum Literature Search, retrieved from https://www.cfa.harvard.edu/sma/memos/20.pdf.
- European Committee for Standardisation (2002), BS EN 1991-1-1-2002, *Eurocode 1 - Actions On Structures (Part 1-1 General Actions - Densities, Self-weight, Imposed Loads For Buildings)*, European Commission, Brussels.
- European Committee for Standardisation (2005), BS EN 1991-1-4-2005, *Eurocode 1 - Actions On Structures (Part 1-4 General Actions - Wind Actions)*, European Commission, Brussels
- D Kiley, *Driven*, (John Wiley, Hoboken New Jersey, 2004)
- Ronstan (2013) *Ronstan Structural Cables*, retrieved from http://www.ronstantensilearch.com/wp-content/uploads/2013/11/RPA112_Structural_Cable_Catalogue_72dpi_0.pdf, pp 9.
- Strand7 Webnotes (2015). "Response Spectrum and Power Spectral Density", Strand7 Pty Limited, Sydney, Australia

Eiffel Tower, Paris

Blast Analysis

Description

The Eiffel Tower was constructed as the leading monument of the Universal Exposition to mark the centenary of the French Revolution of 1789. The large iron archways served as the gateway to Champ-de-Mars, with the Tower itself acting as the key attraction of the Universal Exposition. It was the first structure to reach the coveted 1000 feet, and maintained its status as the world's tallest structure until 1930. The tower consists of four separate curved legs that meet to form a single point at the peak. Each of these legs are approximately 15m by 15m and consist of four columns that are connected by an iron lattice, allowing the columns to behave like a truss. The four columns contained in each of the legs is assembled from a series of angles and flat sections that form a roughly square hollow cross section.

Source:

Horlyck, L., Hayes, K., Caetano, R., Tahmasebinia, F., Ansourian, P., & Alonso-Marroquin, F. (2016, August). Blasting response of the Eiffel Tower. In NUMERICAL METHODS IN CIVIL ENGINEERING: Dynamics of Structures 2016 (Vol. 1762, No. 1, p. 020003). AIP Publishing.

Aim

The overall objective of the project was to construct a detailed finite element model of the Eiffel Tower for the purpose of performing numerical analysis and determining its capacity to resist a number of loading conditions. As part of this investigation permanent and imposed wind loads were examined by performing a static analysis with the aim of determining the extent to which the structure was overdesigned. The investigation also aimed to analyse the resistance of the tower to a dynamic blast loading adjacent the base of one of the tower legs. Particular regard was paid towards the ability of the weakened structure to remain standing under static loads despite the missing or damaged members.

Eiffel Tower – Blast Analysis

Lachlan Horlyck, Kieran Hayes, Ryan Caetano
Fernando Alonso-Marroquin, Peter Ansourian, Faham Tahmasebinia
School of Civil Engineering
THE UNIVERSITY OF SYDNEY

AIMS

- This project aimed to analyse the Eiffel Tower under a number of loading types using Strand7.
- The analysis will model the structural and blast pressure due to an explosion at ground level. components, permanent load, imposed load, wind load, as well as seismic load
- Linear static, natural frequency, spectral response, and Nonlinear Transient Dynamic solvers are used.

MODELLING

Geometry

- Dimensioning has been determined from original design drawings.
- A quarter of the structure was modelled and mirrored in two axes of symmetry to form the entire model.

Lattice

- All members were modelled as Beam2 elements with sectional properties determined from original drawings.

Platforms

- QUAD4 elements were connected at all nodes to transfer stresses and imposed loads and estimate platform surface.

Loads

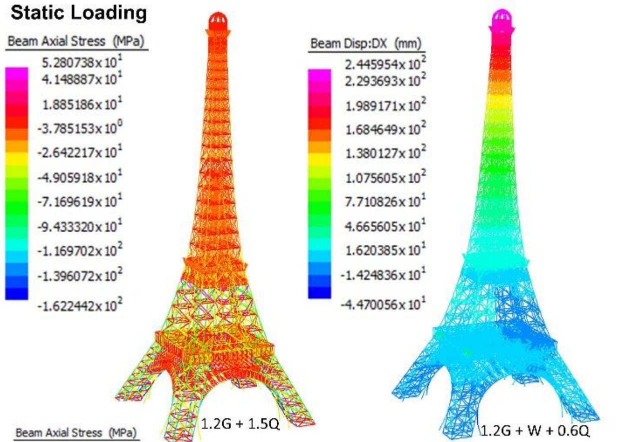
- European codes were not established at the time of construction, and so the tower was not designed to conventional loading, but instead the Eiffel Company's own estimates. As such, the Eiffel Tower has been modelled to Australian Standards and verified against fluid mechanics such as the Bernoulli principle.

BACKGROUND

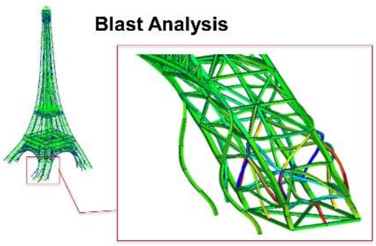
Despite significant protest and criticism, with initial intentions to dismantle the structure 20 years after construction, the Eiffel Tower is a global icon and the tallest building in Paris, France. It was constructed as the entrance to the 1889 World's Fair, and as such was designed using pre-computational methods. Regardless, the wrought iron lattice tower and geometric shape have been designed to efficiently withstand loads, perhaps inline with modern techniques.

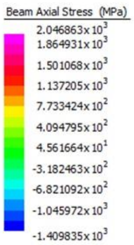
STRAND7 ANALYSIS

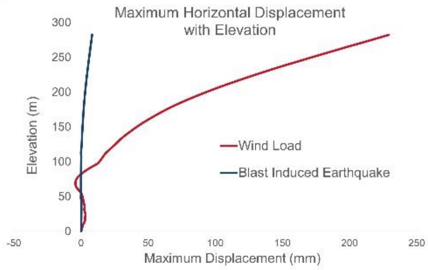
Static Loading



Blast Analysis



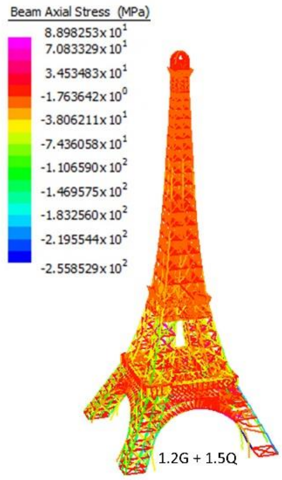




RESULTS

Static Loading

- Strength of the tower was assessed, returning a maximum axial stress of 162MPa. Compared to the tensile strength of wrought iron, under modern loading conditions and a load combination of 1.2G + 1.5Q, the safety factor for strength was assessed as 2.27.
- Serviceability was analysed, returning a maximum lateral deflection of 245mm under the load combination 1.2G + W + 0.6Q. Compared to serviceability limits from AS1170.0-2002, the safety factor for serviceability was assessed as 2.31.



Blast Analysis

- Blast pressure was applied radially from a ground location adjacent to one of the tower's main columns.
- Member stresses exceeded yield adjacent to the blast. Ground vibrations from the resulting 2.1 magnitude earthquake caused little disturbance to the structure compared to wind loading deflections.
- A linear static analysis of the structure after the blast, with beyond yield stress members (i.e. failed members) in the column being removed, revealed the tower could theoretically remain structurally sound.

Optimisation

- By comparing member capacities to maximum axial stress per section type, the Eiffel Tower design was optimised.
- The size of the members were reduced such that the member capacities still exceeded the maximum axial stress under static loading. The weight savings as a result were approximately 46% of the weight of the metal tower.

CONCLUSIONS

- A calculated maximum deflection of 245mm under wind loading is well within the serviceability limit (565mm).
- The structure has a safety factor of 2.27 according to axial stress, which is deemed acceptable by modern Australian standards.
- Blast pressures caused excessive stress in column members.
- The weakened tower remained standing under acceptable stresses below yield.
- The design was optimised by reducing member sizing by 46%.

REFERENCES

- Australian Standards AS1170
- B. Lemoine (2006). Gustave Eiffel: The Eiffel Tower, Hamburg Taschen GmBH
- P. Weidman, I. Pinelis (2004). Model equations for the Eiffel Tower profile: historical perspective and new results, C. R. Mecanique 332 (2004) 571–584.

Façade Greening of Bosco Verticale Building, Milan

Static Analysis

Description

Bosco Verticale (In Italian, "vertical forest") is characterized as the sustainable residential building located in central Milan designed by the famous architect Stefano Boeri. The building introduces the philosophy of the balance of nature in the polluted mega-cities. The Bosco Verticale consists of two rectangular shaped high-rise residential towers of 117m (26 floors) and 76m (18 floors) high that covers a total surface area of 30501m^2. Both towers are featured by the dense vegetation on the outer edges. The vegetation was planted in the pre-fabricated boxes and located at the periphery of the balcony. This building hosts more than 900 trees and over 2000 plants (shrubs and floral) on balconies in different façades. Façade greening in high rise residential buildings would effectively balance the limited land areas and the needs of green areas of urban people.

Source:

Sun, W., Li, M., Han, Y., Wang, M., & Ansourian, P. (2016, August). Façade Greening: High-rise apartment building in Milan using pre-stressed concrete slab. In NUMERICAL METHODS IN CIVIL ENGINEERING: Dynamics of Structures 2016 (Vol. 1762, No. 1, p. 020004). AIP Publishing.

Aim

The aim of this study was to model the overweighted balcony using both ordinary reinforced concrete slab and pre-stressed concrete slab and to compare the results of tensile stress and deflection. To this end, one single level of the Façade Greening was designed and modelled using finite element method in Strand7. A static analysis was performed in order to understand the deflection and the stress due to the extra loads imposed by the soil and plants. The results produced by the linear static solver are compared with the strength of the materials and the European limitations.

Façade Greening: High-Rise Apartment Building in Milan Using Pre-Stressed Concrete Slab

THE UNIVERSITY OF SYDNEY

Wenning Sun, Mingxin Li, Yinong Han, Moqi Wang

Fernando Alonso-Marroquin, Peter Ansourian, Faham Tahmasebinia
School of Civil Engineering

INTRODUCTION

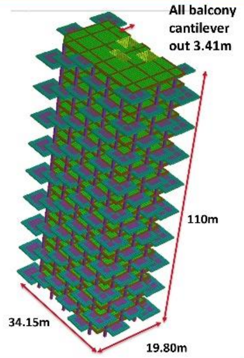
Background

Bocso Verticale also named as Vertical Forest, consists of two residential buildings in Milan, Italy. The taller building is 110 meters high and has 27 floors while the shorter building is 76 meters high and has 18 floors. The buildings are characterized as sustainable building by the various plants along the perimeter of each balcony and there are about 900 trees and over 2000 plants such as shrub and florals on both buildings.

Aims of the Project

The concept of sustainable building is getting more and more popular in urban architectural designs. The vertical forest implemented as green façade in high rise residential buildings would effectively balance the limited land areas and the needs of green areas of urban people. Therefore, the stability and serviceability of the high rise buildings with green façade under different load combinations need to be investigated. A more effective design should be developed to deal with the over-weighted cantilevered balcony. A model is going to be created using strand7 in order to achieve the following objectives:

- To model the balcony slab using ordinary reinforced concrete in Strand7
- To simulate pre-stressed concrete for balcony slab of one level using Strand7.
- To perform a static and dynamic analysis of the high rise apartment.



METHODOLOGY AND RESULTS

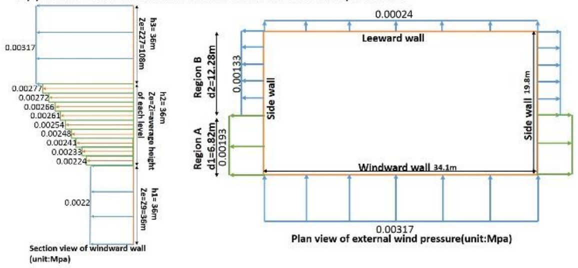
Material properties and loads of structural members

Member	Element type	Element thickness (mm)	Material selection	Dead load (kPa)	Live load (kPa)
Column	Beam2	800*800	f'c = 40 Mpa*	SW+Imposed load[a]	2[c]
Internal slab	Plate(Quad4)	150	f'c = 32 Mpa*	SW+Imposed load[a]	2[c]
Internal beam	Plate(Quad4)	600	f'c = 40 Mpa*	SW+Imposed load[a]	2[c]
Shear wall	Plate(Quad4)	500	f'c = 40 Mpa*	SW	N/A
Balcony slab	Plate(Quad4)	150	f'c = 40 Mpa*	SW	3[d]
Balcony beam	Plate(Quad4)	600	f'c = 40 Mpa*	SW+Tree and soil load[b]	3[d]

*. The materials used are AS3600 Concrete with the compressive strength indicated .
a. The imposed load (including ceiling, decorations, ventilation and lightings) is equal to 1.5 kPa (EN1991-1-1-2002 Table 6.2).
b. The tree and soil load is approximately 7kPa by calculation.
c. Live load for internal column, beam and slab is 2kPa (EN1991-1-1-2002 Table 6.2).
d. Live load for balcony is 3kPa (EN1991-1-1-2002 Table 6.2).

Wind loads

- ➢ The design wind probability is 0.02 (for 50yrs recurrence interval) and the reginal wind speed is taken as 27m/s (EN1994.1.4.2005 Clause 4.2).
- ➢ A complete wind analysis of windward, leeward and side walls were directly applied to the columns with the calculated wind pressure.

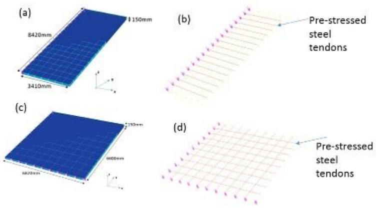


Methods

- **Strand7 model**
- ➢ The structure was modelled using strand7 with beam2, quad4 and tri3 elements as they are relative accurate elements.
- ➢ Some modified section dimensions were used due to the compatibility of mesh in strand 7.
- **General approach**
- ➢ 1. Beam(600mm) and slab(150mm)system with modified calculated reinforcement according to AS3600
- ➢ 2.Add pre-stress slab to the balcony slab

- **Concrete slab with reinforcement**
- ➢ The reinforcement scheduling of beam & slab system is determined by hand calculation according to AS3600. Then, input into strand7 properties

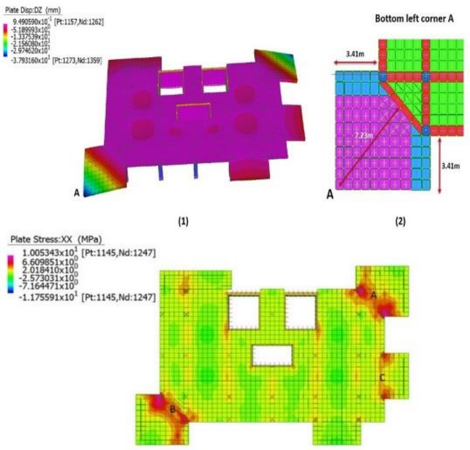
- **Pre-stressed concrete slab**
- ➢ Separated one-way and two-way balcony model were designed
- ➢ Brick elements were used to create the eccentricity of layers
- ➢ Steel tendons with 10mm diameter steel bars were embedded in the top layer of the balcony slab
- ➢ The pre-load of 3Mpa is applied to each tendon



Pre-stressed concrete balcony slab

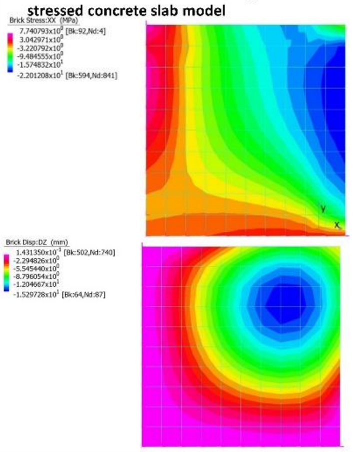
Results

Deflection and stress of one level of structure under ordinary reinforced concrete

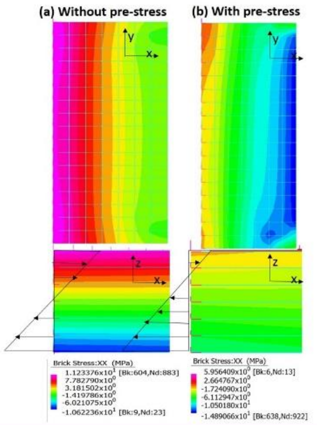


Improved using pre-stressed concrete

- ➢ **Stress and deflection in two-way pre-stressed concrete slab model**



- ➢ **Comparison of Stress in one-way slab model with and without pre-stressed concrete slab**



CONCLUSIONS

In conclusion, the high value of tension stress is identifying at the cantilever roots of the balcony. It will cause cracks in the concrete and in both long term and short term it will affect the serviceability and stability of the balcony. Therefore, a separate model of balcony with pre-stressed concrete is created. Improvement has been observed with pre-stressed concrete in tension stress in concrete. Due to some strand7 limitations, the model cannot fully simulate the pre-stressed concrete balcony with appropriate reinforcement. However, with reasonable assumptions, the effectiveness of pre-stressed concrete in dealing with large service load is justified.

REFERENCES

Giacomello, E. (2015). A New Urban Forest Rises in Milan. *CTBUH Journal*, [online] 2015(1), pp.12-18. Available at: http://global.ctbuh.org/resources/papers/download/2099-a-new-urban-forest-rises-in-milan.pdf [Accessed 15 May 2016].

Giacomello, E. and Valagussa, M. (2015). *Evaluating the High-Rise Vegetation of the Bosco Verticale, Milan.* 1st ed. [ebook] Chicago: Council on Tall Buildings and Urban Habitat, pp.1-16. Available at: https://store.ctbuh.org/PDF_Previews/Books/2015_VerticalGreenery_Preview.pdf [Accessed 17 May 2016].

Golden Gate Bridge, San Francisco

Dynamic Analysis

Description

The Golden Gate Bridge is one of the world's most spectacular and well known bridges that crosses the Golden Gate Channel, connecting the San Francisco Peninsula to Marin County. The bridge was opened in 1937 and held the title for being the longest main span suspension bridge for 27 years. The bridge consists of a bridge deck, supported on a system of beams and trusses. This truss system spans between the bridge pylons and is hung from vertical cables at 15m intervals. These vertical cables are supported by two major catenary suspension cables which pass over the pylons and into anchors at either ends of the bridge. The 2332 m long catenary cables are the longest bridge cables ever made. These cables used an innovative process to bind thinner wires together to make one large cable which allowed for the construction of the record breaking main span.

Source:

Game, T., Vos, C., Morshedi, R., Gratton, R., Alonso-Marroquin, F., & Tahmasebinia, F. (2016, August). Full dynamic model of Golden Gate Bridge. In NUMERICAL METHODS IN CIVIL ENGINEERING: Dynamics of Structures 2016 (Vol. 1762, No. 1, p. 020005). AIP Publishing.

Aim

An investigation into the structural systems of the Golden Gate Bridge when subject to dead, live, wind and earthquake loading was carried out using finite element modelling. This investigation was carried out using Strand7 and was verified through analytical calculations. This study outlines the modelling techniques, element types and analysis solvers used in modelling and analysing the structure. Finally, this study discusses this results produces by the analysis and verifies the results through simple hand calculations.

THE GOLDEN GATE BRIDGE

Cameron Vos, Rafid Morshedi, Rebecca Gratton and Thomas Game
Fernando Alonso-Marroquin, Peter Ansourian, Faham Tahmasebinia
School of Civil Engineering

BACKGROUND

Complete a full static and dynamic analysis of the structure under dead, live, wind and earthquake loads.

Motivations/why this project is interesting
Determining the Deflections of the structure under the above loading conditions for serviceability requirements.

San Andreas Fault creates a sizeable seismic risk for the site.

Set new standards for the engineering of long span bridges (2)

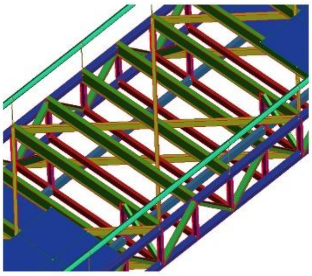
KEY FEATURES OF STRUCTURE

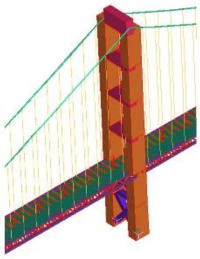
- 3D Deck Truss System
- Concrete Road Deck
- Vertical Cables at 15m Spacing
- Main Catenary Cables
- Pylons – Deep Earth Embedment.

MODELLING TECHNIQUE

- Member Types
  - Cables – B2 Truss Elements
  - Truss Elements- B2 Beam Elements
- Boundary Conditions
  - Fixed at the base of Pylon
  - Fixed in rotation and vertically at the abutments
- Loading
  - Dead Load – Factored Densities
  - Live Load - SM1600
  - Wind Load – Static based on local wind speed
  - Earthquake – Spectrum derived from USGS Data
- Catenary Cables

MEMBER SELECTION

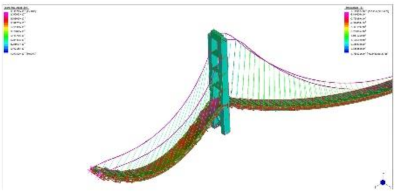




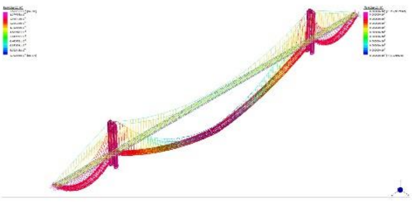
Details of the Structural Elements	Available Structural Element Sizes	Revised Member sizes for Modelling
Green Cross Girder	2500UB3650	2500UB3650
Red Cross Girder	1200x500x50 RHS	2000x2000x200 SHS
Blue Top and Bottom Chords	1400x700x75 RHS	2000x2000x200 SHS
Yellow Cross Bracing	1200x500x50 RHS	1200x500x50 RHS
Green Cross Bracing	500x250x50 RHS	1000x500x50 RHS
Blue Main Girders	1000x1000x100 SHS	1400x700x75SHS
Pink Vertical Members	500UB667	1500UB1590
Vertical Cables	0.126m Diameter	0.3m Diameter
Caternary Cables	0.92m Diameter	1.2m Diameter
Pylons	16000x10000	16000x10000
Pylon Diagonal Bracing	2500x2500	5000x5000
Pylon Cross Bracing	8000x4000	16000x8000
Bridge Deck Slab	0.5m Thick	0.5m Thick

MODEL BEHAVIOUR

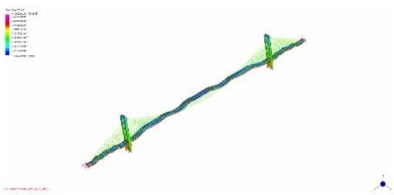
Vertical Loading
Worst case Load Combination: (1.2G +1.8Q)



Lateral Loading



Dynamic Loading



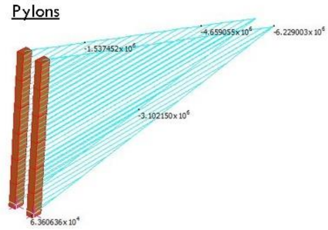
IMPLICATIONS AND DESIGN

Structural Design

- Bridge Deck

Truss Members				
Geometry		Loading	Capacity	% Cap
Member	A	Max Load	A_{yield}	
Bot Cross Girder	1.44	24000	0.0889	6%
Top/ Bot Chords	1.44	288000	1.0667	74%
Diag Top/Bot Cross Bracing	0.16	15200	0.0563	35%
Side Cross Bracing	0.14	57000	0.2111	151%
Blue longitudinal	0.293	36400	0.1348	46%
Vert Side Bracing	0.203	114000	0.4222	209%
Flexural Members				
Geometry		Loading	Capacity	
Member	Z	Max BM	Z_{yield}	
Top Cross Girder	0.2621	19000	0.0704	27%
Top/ Bot Chords	0.7872	738800	2.7363	348%
Bot Cross Girder	0.0460	1422	0.0053	11%

- Pylons



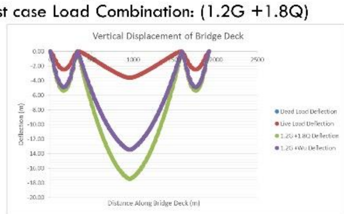
Checked for Axial Load and Flexural load and an area reduction of 88% for the pylon could be made. This assumes a solid steel column as the pylon.

- Cables

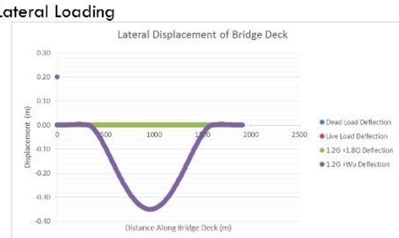
Member Type	Member Dimensions	Maximum Stress.	Redesigned Dimensions
Vertical Cables	0.3m Diameter	3.28×10^5 kPa.	0.172m Diameter
Main Cables	1.2m Diameter	1.14×10^6 kPa.	1.288m Diameter

DEFLECTIONS

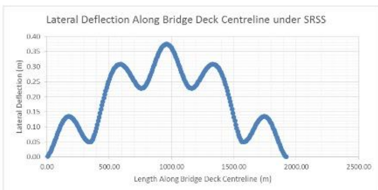
Vertical Loading
Worst case Load Combination: (1.2G +1.8Q)



Lateral Loading



Dynamic Loading



Milad Tower, Tehran

Dynamic Analysis

Description

The Milad tower is a multipurpose tower located in Tehran, Iran, which is the 6th tallest tower in the world, at 436 metres high. The tower 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 that transfer the loads directly to the columns. The loads from the columns are transferred to the concrete shaft which 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.

Source:

Wilhelm, E., Ford, M., Coelho, D., Lawler, L., Ansourian, P., Alonso-Marroquin, F., & Tahmasebinia, F. (2016, August). Dynamic analysis of the Milad Tower. In NUMERICAL METHODS IN CIVIL ENGINEERING: Dynamics of Structures 2016 (Vol. 1762, No. 1, p. 020006). AIP Publishing.

Aim

The aim of this study was 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.

Milad Tower

Mitchell Ford, Ed Wilhelm, Lachlan Lawler, Darren Coelho

Fernando Alonso-Marroquin, Peter Ansourian, Faham Tahmasebinia

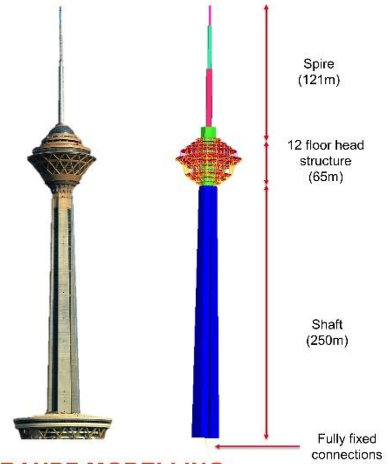
School of Civil Engineering

BACKGROUND

The Milad Tower in Tehran, Iran is the 6th largest building in the world, reaching a height of 436m. The building consists of a foundation, lobby, core shaft, head structure and an antenna mast. Due to the slenderness of the tower and the close proximity to tectonic plate fault lines, both wind and earthquake designs are critical.

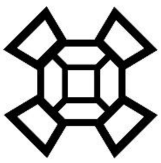
AIM

- To accurately model and design the Milad Tower using the Strand7 software
- To perform a static analysis of dead and live loading combinations
- To complete a full dynamic analysis that determines the stability of the building under wind and earthquake loading

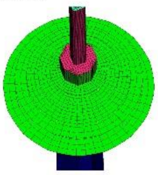


STRAND7 MODELLING

The tower was modelled using nodes, beams and plates elements. The dimensions of members were scaled off drawings provided in Yahyai et al. (2009). Cartesian and cylindrical coordinate systems were utilized to input the geometry. The model was generated by creating a quarter of the structure then mirroring about the z-axis to create the full structure.



1.5m concrete core walls



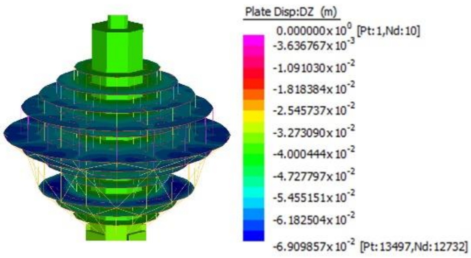
Curvilinear plate mesh

Section	Structural Member	Element Type	Property
Foundation	Central Core	Quad 4 Plates	40 MPa concrete
Core Shaft	Tapered Walls	Quad 4 Plates	40 MPa concrete
Head Structure	Beams	Beam 2	610UB123
	Columns	Beam 2	350WC230
	Slab	Quad 4 Plates	40 MPa concrete
	Steel Basket	Beam 2	500WC440
Antenna	Mast	Quad 4 Plates	40 MPa concrete

RESULTS

1 Linear Static Analysis (LSA)

The linear static analysis involved two loading cases; dead load and live load. The dead load was performed by applying gravity to the structure. The live load involved applying a 2kPa face pressure to the slab plates in the head structure. A loading combination of 1.2G + 1.5Q was applied. The deflection contours are shown below:



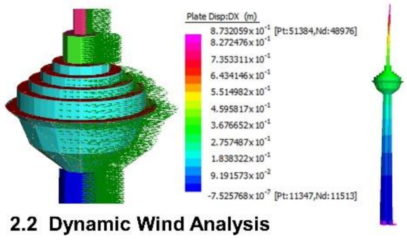
Load Case (plate analysis)	Maximum Deflection (mm)	Maximum Stress (MPa)	Maximum Bending Moment (kNm)
G	48	5.8	76
Q	7.3	2.0	29
1.2G+1.5Q	69	9.9	141

2.1 Static Wind Analysis

Due to the slenderness of the tower, static and dynamic wind analysis is essential. A global pressure to one side the tower. The velocity profile of the wind was developed and this was converted to a pressure by utilizing the quasi-steady assumption;

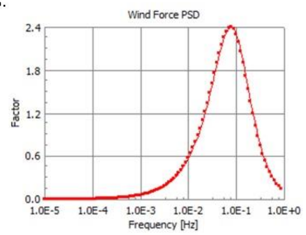
$$P = \frac{1}{2}\rho_{air}V_{des}C_{fig}C_{dyn}$$

To increase accuracy, the head structure was re-modelled with 40mm glass panels (shown in light blue) around the exterior. This allowed the pressure to be applied to the plates and for the load to be transferred through the structural elements.



2.2 Dynamic Wind Analysis

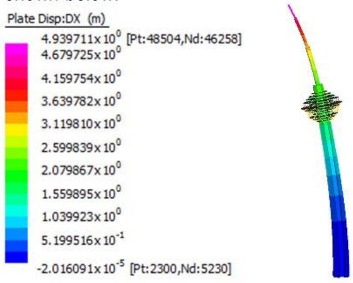
In a real life situation, structures are subject to random fluctuations in wind speed that will interact with the vibrational modes. A factor vs frequency graph allows the simulation of wind gusts.



Natural Frequency Analysis (NFA) of the structure resulted in 17 converged modes. Spectral response analysis (SRA) could then be run which indicated a maximum deflection of 2.74 meters, which occurred at the peak of the tower. This is a more rigorous approach that allows for further understanding of the wind loading, accounting for the non uniform gusty nature of wind.

3 Dynamic Earthquake Analysis

The earthquake design involved using the previously determined natural frequency vibrational modes. From this, a spectral analysis was performed. This involved inputting a table of values for the relationship between spectral ordinates and period for the chosen soil class, in accordance with AS1170.4 (2007). The earthquake was then factored by 0.22 of the value of gravity and applied in the x direction. The displacement for Mode 1 is shown below:



The mass participation rate for the structure was 88.46%.

The base reactions and overturning moments are summarized below:

	Force (kN)	Moment (kNm)
X direction	-120,470	48,582
Y direction	-2,448	2,390,824
Z direction	-923	50.29

CONCLUSION

- There is a degree of inaccuracy in the model as the main beam, slab and column sizes were estimated by hand calculations.
- The linear static analysis found allowable slab and beam deflections (69mm and 40mm respectively) in accordance with Australian Design Codes.
- The static wind analysis was effective with a deflection of approximately 870mm at the top of the mast
- The dynamic wind analysis was modelled and the deflection was approximately 1100 mm
- The earthquake analysis requires further investigation and design changes, due to the deflections of 5m at the antenna mast.

REFERENCES

- Alonso-Marroquin, F, 2016. *Finite Element Modelling*. 1st ed. Sydney: University of Sydney.
- Strand7, 2015. PSD Analysis of a Wind Load. Strand7 Webnotes - Linear/Dynamics, 1.10.20.3, pp.1-11
- Yahyai, M., Daryan, A.S., Ziaei, M. and Mirtaheri, S.M., 2011. Wind effect on milad tower using computational fluid dynamics. *The Structural Design of Tall and Special Buildings*, *20*(2), pp.177-189.
- Yahyai, M., Rezayibana, B. and Daryan, A.S., 2009. Nonlinear seismic response of Milad Tower using finite element model. *The Structural Design of Tall and Special Buildings*, *18*(8), pp.877-890.

Twin Towers, New York City

Thermal Analysis

Description

Twin Towers were components of the World Trade Centre located in Lower Manhattan, New York City. They consisted of two 110-story with 6-level basement commercial office buildings: the 417 m North Tower, and 415 m South Tower. At 9:03 a.m. on September 11 in 2011, the Twin Tower collapsed after attacked by two jet aircrafts. As a framed tube structure, it occupied approximately 63m x 63m with core of roughly 27m x 41 m. There were 59 external columns on each side of the structure and 4 columns on the four corners, thus there was a total number of 240 external columns. The structure core consisted of 47 steel columns running from the bedrock to the top of the tower. The large, column-free space between the perimeter and core was bridged by prefabricated floor trusses.

Source:

Zhu, K., Xu, K., Ansourian, P., Tahmasebinia, F., & Alonso-Marroquin, F. (2016, August). Energy balance in the WTC collapse. In NUMERICAL METHODS IN CIVIL ENGINEERING: Dynamics of Structures 2016 (Vol. 1762, No. 1, p. 020007). AIP Publishing.

Aim

The aim of this study is to analyse the collapse of the Twin Towers of the New York City's World Trade Centre after attacked by two jet aircrafts. The approach mainly focused on the effect of temperature on mechanical properties of the building, by modelling heat energy in the south tower. Energy balance during the collapse between the energy inputs by aircraft petrol and the transient heat to the towers was conducted. Both the overall structure between 80 to 83 stories and individual elements were modelled. The main elements exposed to the thermal load includes external and internal columns. Heat applied in 2D and 3D models for single elements was through convection and conduction. Analysis of transient heat was done using Strand7.

Collapse of the Twin Towers (Energy Balance)

Kaiqi Zhu, Kang Xu, Qianqian Yin, Yue Zhu

Fernando Alonso-Marroquin, Peter Ansourian, Faham Tahmasebinia

School of Civil Engineering

THE UNIVERSITY OF SYDNEY

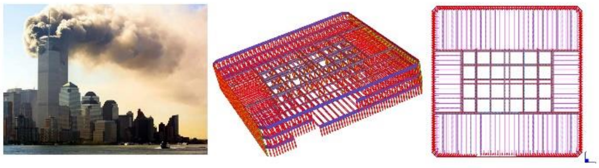
INTRODUCTION:

Project Brief

The World Trade Centre (WTC) buildings were attacked by the airplanes. The South Tower which collapsed after 1.5 hours was modelled and investigated. The energy of the fire and combustion was gained by fuel leaking from the aircraft after collision. It was assumed that fire temperature (800℃) at directly impacted area had transit the heat through the building elements. The building mainly consisted of steel structure, and the mechanical properties would be significantly affected by the extreme temperature, thus the increased temperature would result in loss of strength of columns and trusses and finally cause the failure of the structure. Instead of 110 levels, only 3 levels severely under attract (FL80-82) were constructed in the model. Heat transfer of individual element was analysed in 2D and 3D modelling with considering convection and conduction.

Aim

The project was conducted to determine whether the balance can be obtained between heat energy and the energy resource based on energy flow among the affected levels. Besides, it was also aimed to check whether the fire did result in the failure of the building by reducing buckling loads of columns.



Actual building and Strand7 model (3D & plan view)

Numerical Modelling

Models of individual element were created in Strand7 based on dimensions in table below:

Details of the Structural Elements	Structural Element Sizes
Columns - Structural Systems	External:365*365*9.5
	Internal:300*900
Trusses - Structural Systems	1800*740

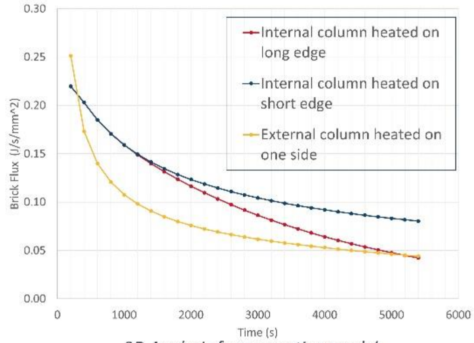
Buckling loads and Loads applied to each column for modelling were calculated as shown in table below:

	Load Capacity (KN)	Applied Load (KN)			
	Buckling Load Capacity	Dead Load G	Live Load Q	G + Q	1.2G + 1.5Q
External Column	2817	912	327	1239	1585
Internal Column	57079	3059	645	3705	4639

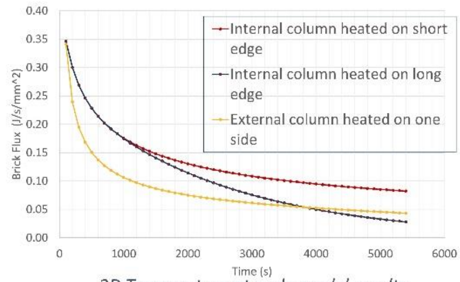
In this project the solver applied to models for analysis was Transient Heat solver.

RESULTS

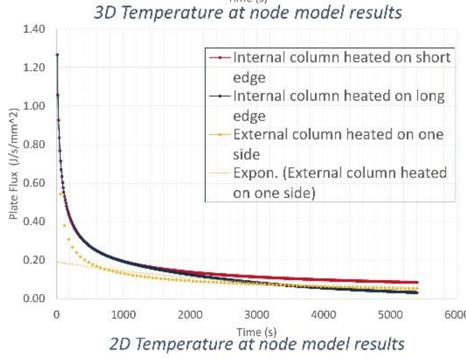
Flux of three sets of models' columns were achieved as shown in figures below.



3D Analysis for convection model



3D Temperature at node model results

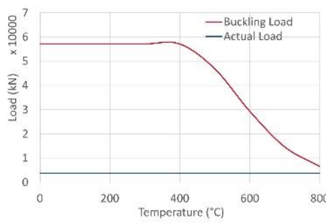


2D Temperature at node model results

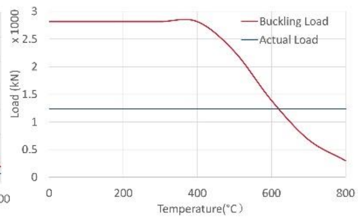
With three sets of flux shown above, total energy of three series of models were calculated as shown in table. The energy from aircraft petrol was calculated for comparison.

	Energy from convection	Energy from temperature		Energy from aircraft petrol
	3D	3D	2D	
Value (J)	1.17E+12	1.25E+12	1.58E+12	2.86E+12

Results below describe how buckling load capacity of internal and external columns decreased with growing temperature. It could be determined whether and when the columns buckled.



Buckling load and actual load of internal column



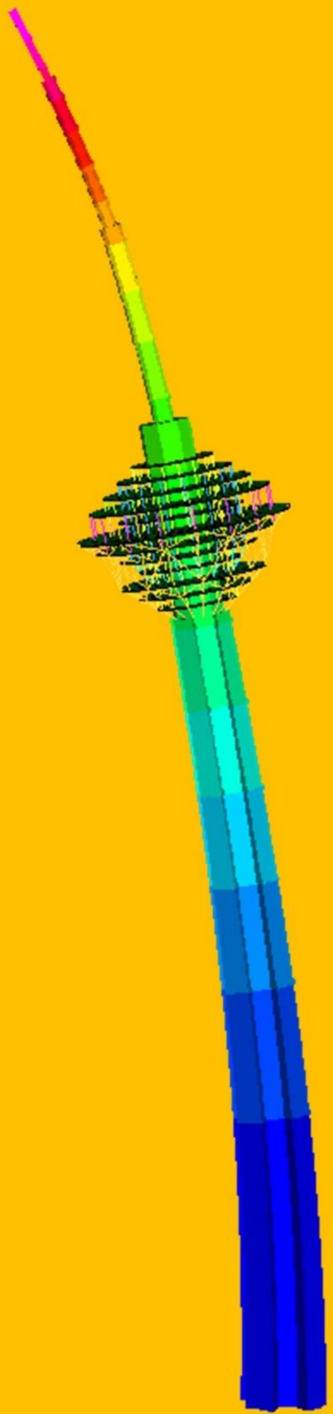
Buckling load and actual load for external column

DISCUSSION

- 3D models have smaller flux than 2D models, while the result from 2D model were closer to the flue energy.
- The convection analysis provided a more realistic result
- Compared with the overall energy from the plane fuel (90000L), the energy values based on numerical analysis in those three types of models were all a bit smaller than half of the fuel energy.
- Before temperature increased to 800 °C, external columns collapsed when actual load exceeded buckling load capacity at around 700°C.
- Although internal columns did not fail, actual load approached to buckling load capacity rapidly and finally the two quantities were very close. Additionally, the actual temperature might be higher than 800°C assumed. Therefore it was possible that internal columns failed as well.

CONCLUSION

In this project, the energy transferred from fuel leaked mainly into structural elements in contact with fire, and energy could be balanced generally. Also, the energy from aircraft fuel could cause a significant reduction in buckling load capacity of columns, therefore led to the buckling of external columns and the failure of the structure.



THE UNIVERSITY OF SYDNEY

Strand7® Software