PSZ 19:16 (Pind. 1/97)

UNIVERSITI TEKNOLOGI MALAYSIA

BORANG PENGESAHAN STATUS TESIS♦

JUDUL: BUILDING INFORMATION MODELLING AND STRUCTURAL ANALYSIS OF PRECAST HIGH-RISE BUILDING

SESI PENGAJIAN: 20062007

Saya LEE JIANG JUN
(HURUF BESAR)

mengaku membenarkan tesis (PSM/~~Sarjana/Doktor Falsafah~~)* ini disimpan di Perpustakaan Universiti Teknologi Malaysia dengan syarat-syarat seperti berikut:

1. Tesis adalah hakmilik Universiti Teknologi Malaysia.
2. Perpustakaan Universiti Teknologi Malaysia dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (✓)

☐	SULIT	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
☐	TERHAD	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
✓	TIDAK TERHAD	

Disahkan oleh

(TANDATANGAN PENULIS)	(TANDATANGAN PENYELIA)
Alamat Tetap:	
28, KAMPUNG ASAM KUMBANG,	ASSOC. PROF. DR. ABD. LATIF SALEH
44000 KUALA KUBU BHARU,	Nama Penyelia
SELANGOR DARUL EHSAN	
Tarikh: 18 April 2007	Tarikh: 23 April 2007

CATATAN
* Potong jika tidak berkenaan.
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD.
♦ Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara penyelidikan atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

"I/~~We~~* hereby declare that I/~~we~~* have read this thesis and in my/~~our~~* opinion this thesis is sufficient in terms of scope and quality for the award of the degree of Bachelor Of Civil Engineering"

Signature	:	
Name of Supervisor	:	ASSOC. PROF. DR. ABD. LATIF SALEH
Date	:	23 April 2007

* *Delete as necessary*

BUILDING INFORMATION MODELLING AND STRUCTURAL ANALYSIS OF PRECAST HIGH-RISE BUILDING

LEE JIANG JUN

A report submitted in partial fulfilment of the
requirements for the award of the degree of
Bachelor of Civil Engineering

Faculty of Civil Engineering
Universiti Teknologi Malaysia

APRIL 2007

I declare that this thesis entitled "*Building Information Modelling and Structural Analysis of Precast High-rise Building*" is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature : ....

Name : LEE JIANG JUN

Date : 18 April 2007

DEDICATION

To my beloved grandpa, grandma,
father, mother,
sister and brother.

ACKNOWLEDGEMENT

In completing this undergraduate dissertation, I would like to express my heartiest appreciation to my supervisor, Associate Professor Dr. Abd. Latif Saleh for his guidance, advices and encouragement. Without his motivation and support, this study would not be able to complete in time. I would also like to thank my fellow classmates for helping me in providing information and sharing of experience.

Not forgetting the helpful PSZ librarians who are always willing to assist me in acquiring information regarding to the preparation of my dissertation.

Lastly, to my parents who are always giving me full moral and financial support.

ABSTRACT

Loss of information and miscommunication in the construction industry are often being related to project completion delays and low quality final products. Therefore, with the advancement of Information Technology (IT), these problems can be reduced by implementing the Building Information Modelling (BIM) approach in the construction industry. Autodesk Revit Structure has the BIM feature implemented in the software, which allows information within an interface to be shared by many parties. With the Autodesk Revit Structure, a 3D model can be developed easily from the 2D architectural drawings. Besides, Autodesk Revit Structure acts as an interface to bridge with third party analysis software where the 3D model can be exported to the analysis software directly for structural analysis. With the 3D high-rise building model, the effect of wall thickness on the behaviour of the structure is observed. Besides, the study also includes the behaviour of structure with various types of joints. From the study, the information transferred between the Autodesk Revit Structure and ETABS is verified. In addition, the deflection of the structure with various wall thickness and joint types are presented. Lastly, the beam and column that are resisting the maximum shear force and maximum bending moment is identified. With the BIM featured software, model for structural analysis can be generated easily. The analysis procedure for precast concrete structure is much easier as the element that requires the largest capacity will be identified and mass produced to be used for the structure.

ABSTRAK

Maklumat yang tidak sempurna dan masalah komunikasi merupakan salah satu sebab yang sering dikaitkan dengan penangguhan tarikh penyiapan projek dan hasil pembinaan yang tidak berkualiti. Dengan kecanggihan Teknologi Maklumat (IT), masalah ini boleh dikurangkan dengan mengaplikasikan *Building Information Modelling* (BIM) dalam industri pembinaan. *Autodesk Revit Structure* merupakan perisian komputer yang mempunyai fungsi BIM yang membolehkan maklumat dikongsi oleh parti-parti yang terlibat dalam pembinaan. Model 3D dapat dibina menggunakan pelan 2D dengan menggunakan perisian *Autodesk Revit Structure*, dan model ini dapat dieksport ke perisian analisis untuk mengendalikan analisis struktur. Dengan model 3D yang telah dibina, analisis kesan ketebalan dinding ke atas perlakuan struktur dilakukan. Di samping itu, analisis perlakuan struktur dengan jenis sambungan yang berbeza turut dilakukan. Dalam kajian ini, kesahihan maklumat yang dipindahkan di antara perisian *Autodesk Revit Structure* dan *ETABS* turut dikaji. Keputusan kajian untuk pesongan struktur dengan ketebalan dinding serta jenis sambungan yang berbeza turut dipersembahkan. Rasuk dan tiang yang menanggung daya ricih serta momen lentur yang maksimum turut dikenalpasti. Dengan adanya perisian BIM, model untuk analisis struktur dapat dibina dengan mudah. Prosedur analisis untuk bangunan pasang siap adalah lebih mudah, iaitu dengan mengenalpasti anggota yang memerlukan kapasiti yang tertinggi dan seterusnya anggota itu akan digunakan untuk keseluruhan struktur berkenaan.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

a	-	Joint stiffness
k	-	Spring constant
E	-	Young's Modulus
I	-	Second moment of area
L	-	Length
$\lambda_I H$	-	Relative in-fill-to-frame stiffness
H	-	Storey height
E_m	-	Young's Modulus of the infill panel
E_c	-	Young's Modulus of the frame member
t	-	Panel thickness
θ	-	Angle between the strut and the horizontal frame member
I_c, I_{col}	-	Second moment of area of the column
H_m	-	Clear height of the particular storey
a	-	Width of the equivalent strut
D	-	Diagonal length
G_k	-	Dead load
Q_k	-	Imposed load
W_k	-	Wind load
d	-	Effective depth of a beam
h	-	Height of a beam
E_{conc}	-	Young's Modulus of concrete
b	-	Breadth
w	-	Uniformly distributed load
R	-	Reaction force

C_p	-	Pressure coefficient
h	-	Height of building (Wind load analysis)
w	-	Lesser horizontal dimension of building (Wind load analysis)
l	-	Greater horizontal dimension of building (Wind load analysis)
V	-	Basic wind speed
V_s	-	Design wind speed
S_1	-	Topography factor (Wind load analysis)
S_2	-	Ground roughness, building size and height above ground factor (Wind load analysis)
S_3	-	Factor S_3
q	-	Dynamic pressure
k	-	Unit conversion coefficient (Wind load analysis)
F	-	Wind load (Wind load analysis)

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Introduction

With the advancement of Information Technology (IT) now days, all construction information is digitized; the architectural plans, structural plans are drawn using computer. To reduce time wastage, software is developed to transfer information directly from designing software (AutoCAD) to analysis software. In addition, sharing of information between different parties within the software is getting more popular as this reduces the loss of information and saves time. However, transferred information need to be ensured its accuracy and consistency to make the analysis results reliable. With the introduction of Building Information Modelling (BIM) concept, works can be done faster with errors minimized. Software such as Autodesk Revit Structure provides users with the BIM facility.

Structural analysis is a basic and early stage procedure to be carried out as to analyze and design each section within a structure. The analysis is crucial as it determines the performance of a structure. Nevertheless, structural analysis is also needed after the construction for periodic maintenance. Some common analysis software available in market is StaadPro, ETABS and SAP2000.

A well designed structure will be sound in resisting all loadings. Besides the design of each section, the connections and joints within the structure—the structural integrity—is another important aspect that will influence the behaviour of a structure. A structure having high integrity is a structure that has good performance and high stability. For a precast and high-rise building, the structural integrity will need to be emphasized more as compared to other types of building.

In addition, high-rise structures will be subjected to more additional loads as compared to low-rise building. Seismic and wind load, for example, will have more effect on the performance and stability of high-rise buildings.

1.2 Problem Statement

The advancement of IT has made all documents digitized. Plans are drawn in computers and shared by many parties. Information loss and time wastage has been a critical issue as these will reduce the efficiency of the construction works and indirectly increase the cost of construction. Currently, software allows transferring of information from design and drafting software to analysis software. However, the transferring process must be ensured that information remains the same in order to obtain a reliable result.

AutoCAD, Autodesk Revit Structure and ETABS are some of the software used in structural design, drawing and analysis. AutoCAD is well known with the ability in drafting. Meanwhile, Autodesk Revit Structure allows structural drawing to be exported to ETABS for analyzing. In addition, this software allows bridging where updates are done automatically between the software. However, there are several aspects to be considered while applying this software in structural design and analysis such as the properties of the elements. Most of the element properties for instance types of material, height and dimensions, are predefined in the software. Hence, checking shall always be carried out to ensure the properties are identical to the structural properties in order to produce an accurate and reliable analysis results.

1.3 Research Objective

This study is carried out based on the following objectives:

i. To study the implementation of BIM in structural analysis;
ii. To develop from 2D digital architectural drawing into 3D model and determine the structural information/parameters required to transform 2D digital architectural drawing into 3D structural model for structural analysis software;
iii. To study the types of connection and its behaviour in precast building;
iv. To study the behaviour of precast high-rise building subjected to horizontal and vertical load.

In the application of software, information transfer between interfaces is common. However, to ensure the results are reliable, all structural information must remain consistent in all interfaces. Hence, a checklist is to be produced to highlight those aspects that require attention while sharing structural information between design and analysis software.

Respond of building to loading is essential, especially in high-rise building. Therefore, in studying precast high-rise building, both vertical and horizontal loading will be taken into consideration to evaluate the overall performance of the building.

1.4 Scope of Study

Throughout the study, two computer software: Autodesk Revit Structure and ETABS will be used to evaluate the overall performance of a self developed precast high-rise building model. The 2D digital architectural drawing will be used to develop a 3D structural model which represents the actual structure to be built, and the model will be exported to structural analysis software for analysis. The overall performance of the structure, including the behaviour of structure under loading will be within the scope of study. However, the modelled structure will only be subjected to vertical load and wind load as horizontal load. Seismic load will not be considered in this study.

1.5 Significance of Study

BIM has undergone rapid development in the engineering field. BIM is a very powerful tool which can help to reduce the problems that occur between the architect and engineer. In this study, the study of BIM is carried out using two computer software—Autodesk Revit Structure and ETABS. The computer application, Autodesk Revit Structure used in this study has the ability to import 2D structural drawings from AutoCAD. The drawings will then be used as a basis for generating a 3D model. The generated model can be viewed as 3D or 2D on the same interface depending to the user. The ETABS, as the third party software is used to analyze the modelled structure.

Besides, the Autodesk Revit Structure and ETABS can be bridged and all information can be transferred in both ways. Hence, unnecessary updating work, which often leads to information lost, can be eliminated.

The main purpose of introducing precast high-rise building in this study is that Malaysia is currently undergoing rapid development. To optimize the space available, high-rise building will usually be the choice of the client. However, with the traditional approach in construction, which is the cast *in-situ* concrete construction, delays and poor quality products often occur. Therefore, by introducing the precast elements in construction, the overall construction duration can be speed up. Besides, better quality control can be achieved with the use of precast elements.

CHAPTER 2

LITERATURE REVIEW

2.1 Building Information Modelling

Autodesk first introduced the Building Information Modelling (BIM) approach in the year of 2002. The BIM approach has improve the building design, construction and management in terms of time, cost and quality. The BIM approach increases the delivery time of a particular project which helps reduce the overall project duration. Besides, the BIM approach also provides a better coordination platform, results in fewer errors during the design, construction and management phases. Increase productivity and higher quality works are among other advantages of the BIM approach.

The BIM approach was developed based on the intersection of two critical ideas—to keep critical design information in a digital form to make updating and sharing process easier and more valuable; and to create a real-time and consistent relationship between the digital design data.

There are three characteristics of BIM approach. First, all building information are created and operated in a digital database to allow collaboration. Secondly, the database is well managed as a change to any part of the database is coordinated to the other parts in the database. Lastly, all the information are being captured and preserved for reuse by additional industry specific application.

2.2 Stability

Stability is the performance of a structure under the influence of lateral load. Structure subjected to lateral load will deform laterally, which is known as sway. The design of a structure, especially for high-rise structure, stability is a very important aspect to be considered. The sway of the structure must be limited.

There are several definitions for stability of a structure:

> "The building must not topple over, blow away or collapse due to lack of stiffness. The building must have sufficient bracing so that the walls remain at right angles to the floor (Gauld, 1995)."

> "Stability is the ability of a structure to support load while undergoing limited deformation and displacements (Hanaor, 1998)."

> "All buildings must be provided with the ability to resist wind loads which is known as stability (Dutt, 1984)."

"…relatively small changes in the system parameters and/or in the environmental conditions would bring about relatively small changes in the existing state of the system (structure) (Farshad, 1994)."

2.3 Progressive Collapse

Progressive collapse in the event of abnormal loadings—bomb explosion, vehicular collision, impact forces—in which the failure of one element leads to the collapse of another, then another, which can produce catastrophic results (Figure 2.1) (Nilson and Darwin, 1997).

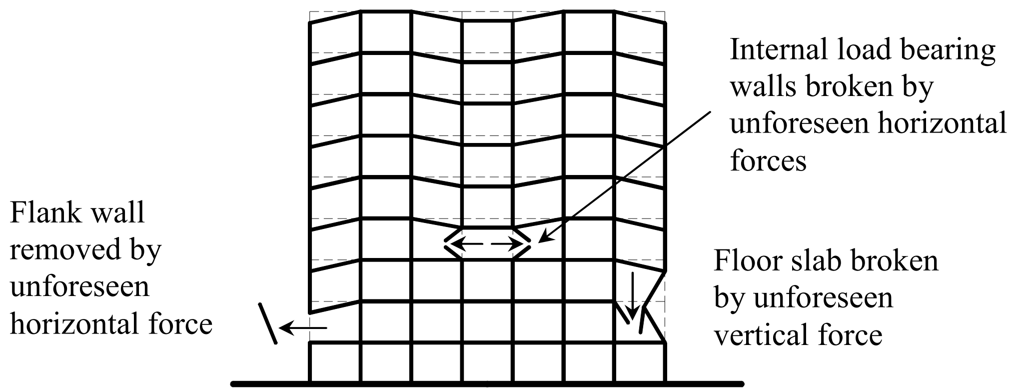


Figure 2.1: Progressive failure

The collapse of elements where the structure is entirely disproportionate to the significance of the initiating cause can progress in the horizontal or vertical directions (Schultz et al., 1977). The main cause of this is due to the insufficient continuity between members and/or insufficient bearing area, where the provision can jeopardize the stability of the structure as a whole (Hartland, 1975). However, determination of the initial point that triggers the collapse is difficult (Bell, 1989). The failure of the structure is usually initiated locally but collapse occurs under general stability. A typical example of progressive collapse is the Ronan Point apartment building, London, 16 May 1968.

The three most common occurrences of progressive collapse are high-rise concrete flat-plate structure (during construction or earthquake); formwork for concrete structure; and high-rise structure constructed with precast concrete elements. However, redundant structures—to shed load through alternative paths—are remarkable for their ability to overcome initial gross over stresses (Bell, 1989 and Beall, et al., 2003). Besides, protected members and the use of ties can eliminate progressive collapse.

2.4 Precast Concrete

There are several definitions of precast concrete as shown below:

"Concrete unit cast and cured in a place other than the final location in the works (BS EN 13369:2004)."

"The term 'precast concrete' is used to describe products made of concrete under factory conditions either in a permanent factory or in a temporary casting yard on a construction site and erected on site as finished articles (Hartland, 1975)."

"Structural precast concrete elements—slabs, beams, girders, columns, and wall panels—are cast and cured in industrial plants, transported to the construction job sites, and erected as rigid components (Allen, 1999)."

"Prefabrication or precasting in reinforced concrete involves a mould shaped to the required form, in which reinforcement is placed and concrete is then cast. The essential features are that the same mould is used many times, without any modification. Such casting is done either in a factory or at a fixed location on the site. The completed elements are finally transported to the erection area (Haas, 1983)."

Generally, precast concrete can be defined as mass-produced reinforced concrete elements that are prefabricated at a prefabrication yard, transported to the site and erected at the site to form a structure.

Precast concrete structures can be classified into three categories (Elliott, 1996):

a) wall frame – consisting of solid or voided vertical wall and horizontal slab unit only, and used extensively for multi-storey hotels, retail units, hospitals, housing and offices. Structural walls serve as acoustic and thermal partitioning;
b) portal frame – consisting of columns and roof rafters or beams, and used for single-storey retail warehousing and industrial manufacturing facilities;
c) skeletal structure – consisting of columns, beams and slabs for low- to medium-rise buildings, with a small number of walls for high-rise. Skeletal frames are used chiefly for commercial offices and car parks.

Precast concrete concept has been widely used in civil engineering works, to name a few, curbs, paving slabs, lampposts and pipes. However, in building works, the usage has not gain the popularity. Most of the building constructions are still using the conventional cast *in-situ* reinforced concrete.

2.4.1 Precast Concrete Elements

The production of precast elements is carried out conveniently at ground level (Allen, 1999). This allows the mixing and pouring operations to be highly mechanized. In addition, precast concrete can be cast in different climates and they are carried out under shelter.

Concrete and steel used in precast are usually Type III (high early strength) 35 MPa concrete and 1860 MPa steel. Precast concrete elements are usually steam cured. Steam curing utilizes the heat to accelerate the curing process and moisture for full hydration. The used of the high early strength cement and the application of steam curing shortens the producing process cycle of the plant to a 24-hour cycle.

2.4.1.1 Slab

The most fully standardized precast concrete elements are those used for making floor and roof slabs. There are four types of commonly produced precast slab elements: solid slab, hollow core slab, double-tee and single-tee.

The manufacturing of precast slab elements is usually with a rough top surface. A concrete topping will be poured over them and finished to a smooth surface. The topping usually 2 in. (50 mm) in thickness, bonds the precast elements and become a working part of their structural action. It also helps the precast elements to act together in resisting concentrated loads and diaphragm loads. Structural continuity across a number of spans can be achieved by casting reinforcing bars into the topping over the supporting beams or walls (Allen, 1999). Under floor electrical conduits may also be embedded in the topping.

Flat-slab precast floors are a system of precast slabs directly supported by the capitals of columns (Murashev et al., 1971). Flat slabs are suitable for short span and slab with minimum depth requirement. The main purpose of the capitals is to create a rigid connection between the floor and the column. In addition, capitals, which act as load transmitting mechanism to transmit load to the column, can reduce the effective span length. Flat slabs are usually 4 in. (100 mm) thick, although they are used as 2½ in. (65 mm)when continuous over several spans and are produced in widths of 4 to 8 ft. (1.2 to 2.4 m) and in lengths up to 36 ft. (11 m) (Nilson and Darwin, 1997). The design of slab depends highly on the magnitude of loads and on deflection limitations.

Hollow planks (also known as hollow-core slabs) provide better insulation and to cover longer spans. The internal longitudinal voids replace much or the nonworking concrete results in a lighter weight structure. The depths range from 4 to about 8 in. (100 to 200 mm), width from 2 to 4 ft. (0.6 m to 1.2 m) (Nilson and Darwin, 1997). Depending on load and deflection requirements, hollow planks are used on roof spans from about 16 to 34 ft. (5 to 10 m) and on 12 to 26 ft. (3.6 to 8 m) floor spans, which can be augmented to about 30 ft.(9 m), if a 2 in. (50 mm) topping is applied to act monolithically with the hollow plank (Nilson and Darwin, 1997).

Double-tee slabs are also suitable for long span. The 'tees' eliminates nonworking concrete. The usual depths for double-tee members are from 14 to 22 in. (350 to 550 mm). They are generally used on roof spans up to about 60 ft. (18 m) (Nilson and Darwin, 1997). When used as floor members, a concrete topping of at least 2 in. (50 mm) is usually applied to act monolithically with the precast member for spans up to about 50 ft. (15 m), depending on load and deflection requirements (Nilson and Darwin, 1997).

Meanwhile, single-tee slabs are suitable for the longest span. However, the use of single-tee slabs is less popular as they require temporary support against tipping until they are permanently fastened in place. Roof spans up to 100 ft. (30 m) often uses single-tee members.

2.4.1.2 Beam

There are several standard shapes for precast concrete beams and girders. The shape of precast beams depends chiefly on the manner of framing (Nilson and Darwin, 1997). Some commonly found precast concrete beams and girders are the L-shaped beams (also known as ledger beams) and inverted tees. These shapes provide direct support for precast slab elements as compared to slab elements resting on top of rectangular beams. For beams supporting floor and roof members, they are mostly rectangular. Meanwhile, L-beams are use to provide bearing.

2.4.1.3 Column

Precast columns are usually reinforced, and with square or rectangular in section. Simple prismatic precast columns are employed in single-storey height building so that beams rest on top of the columns. Alternatively, in multi-storey building construction, the columns can be made continuous for up to about six stories (Nilson and Darwin, 1997). Frequently, integral brackets provide bearing for the beams in continuous columns.

2.4.1.4 Wall

Load bearing wall panels in any types of low-rise and high-rise buildings usually use precast solid slabs (Allen, 1999). In addition, rigid foam insulation can be cast into wall panels for thermal insulation.

Wall panels can be produced in a variety of shapes depending to the architectural requirement. An example of precast wall elements is the panel walls. They are usually faced with finishes material and backed by insulating material. They are supported at each floor by the skeleton of the framework. Panel walls, however, do not transfer load from the upper floor to the lower floor. They only function to enclose the building.

Curtain wall is another category of wall panels that are similar to panel walls. However, curtain walls do not support each storey by the frame of the building, i.e. they are non-load bearing wall. To improve thermal insulation, sandwich panels are used that consist of an insulation core (e.g. foam glass, glass fibre or expanded plastics) between two layers of normal or lightweight concrete (Nilson and Darwin, 1997).

Bearing walls transfer vertical load from the upper storey to the lower storey in addition to its own weight. In precast concrete construction, bearing wall is rather popular as it helps to relieve the load burden of other structural elements.

Stresses in wall panels are frequently more severe in handling and during erection; therefore, the design of each elements must ensure that these temporary conditions are taken into consideration (Nilson and Darwin, 1997).

In nominal design process, the pressure of wind is usually the only load that is considered in determining the structural thickness of a wall panel, although in some cases exterior walls are used as diaphragms to transmit forces caused by horizontal loads down to the building foundations (Nilson and Darwin, 1997). Nevertheless, the designer should always ensure that all critical aspects, such as wind, function of wall and additional stresses in handling are considered.

2.4.1.5 Large Panel Structure

The term 'Large Panel Structure' is used to describe a building in which the structural framework is composed of precast concrete floors and load-bearing walls in panel sizes as large as possible within the constraints imposed by handling, transportation and erection consideration (Hartland, 1975 and Schultz, 1977). In most large panel systems, the walls transfer their loads directly to the substructure without an intermediate frame (Schultz et al., 1977). In addition, this construction restricts opening on the building at any level. Whereby, it is most suitable for high-rise residence building where all units are permanently partitioned.

2.5 Advantages and Disadvantages of Precast Concrete Construction

Precast concrete elements have stringent quality control at the prefabrication yard resulting in structures that are more reliable. In addition, the casting and curing process are done under controlled conditions, where the concrete is well compacted and well cured to achieve the optimum design strength. Moreover, workers working at prefabrication yard are usually well trained and specialized in prefabrication work.

Prior to construction, engineers have the opportunity to check and inspect the precast elements to reject substandard works. Thus, the final structure will be higher in quality. Well quality control, too, results in high quality finished surface, where plastering can be obviated and labour cost can be saved.

Having to construct a precast structure, weather will not be an obstruction in the construction progress. For instance, there will be no interruption in construction in countries with four seasons, especially during winter. Often, construction works stop during winter as the concrete requires additional attention during the mixing and curing process.

Another advantage of precast construction is less labour per unit. This is because of the mechanized series of production. Standardization and mass production of units make the mould available for re-use. Moulds are often very costly in terms of the material and construction process.

Precast construction usually consume shorter construction time because the works at site are merely involving foundations and connecting of precast units. Besides, labours at site can be reduced, which directly reduce the capital cost of the project. Precast concrete construction is able to reduce the amount of concrete mixing job at the site. Subsequently, the cost for skilled batching and concreting workers, and delivering of raw material to the site can be reduced.

Another advantage of precast concrete construction compared to cast *in-situ* concrete construction is the elements can be dismantled and re-erected elsewhere. However, this should be avoided at site, as it is a very tedious procedure in handling those heavy and large elements.

Incorporating non-structural elements, such as electrical conduits and other services into precast elements reduces the work at site; that eventually reduce the amount of material stocked at site.

The construction of precast concrete structures, however, requires standard plans and standard structural elements. Besides, the cost of transporting prefabricated elements to the site is very much higher as compared to transferring raw construction materials to the site. The tendency of technical problem in connecting precast elements at the site is relatively high too.

In a precast structure, the joints or connections are the most crucial problem to the engineer. Connections are points where load changes in direction, therefore, the connection must be sound in order to transfer load from elements to elements efficiently and effectively. During the design stage, the connection may appear to be perfect on paper, however, when comes to site work, the designed connection might be difficult for the engineer and construction workers to implement it. Thus, a small amount of *in-situ* concrete or non-shrinking grout is often used to substitute the complicated connection details to provide a monolithic connection.

Precast concrete construction helps builders to save time in construction. Hence, all shop drawings will be produced and verified before the construction. Prefabrication works are also been carried out ahead of construction schedule to minimize delays. As a result, last minute changes in the design will not be accommodated. Service ducting and wire conduits must be clarified at the very beginning of the construction to be embedded in the precast elements.

Design of precast elements should include the stress induced during transportation, handling and erection, besides the self weight and service load, resulting to additional reinforcement bars usage. Transportation would be another problem for precast elements especially for those large elements, such as wall panels and slabs. Besides, the design should also take into consideration the capacity of the machineries at site. If the weight of the designed elements falls beyond the capability of the machineries, additional cost would be needed in order to obtain higher capacity machineries.

Precast construction is unlikely for buildings with irregular shapes. Mass production will not be able to carry out if elements are of different dimensions where precast construction would no longer a wise choice in this situation.

2.6 Connections and Joints

The term 'connection' refers to major structural connections between precast components, whereas the term 'joint' is used to describe a more simplified jointing between components. For example, junction between beam and column is a connection; and between the landing and stair flight is a joint.

Precast members are prefabricated and assembled at site to form a structure. Therefore, precast concrete connections are often non-monolithic, which differ with cast *in-situ* concrete. Monolithic connections are very important for a structure as they are able to transfer moments and hold the entire structure in place effectively. Extra attention must be paid to the design, analysis and fabrication of connections (Murashev et al., 1971) as connection is one of the critical elements in a precast structure.

Connections of precast members can be classified based on their function: column-to-footing (also known as column-to-base or column-to-foundation) connection; column-to-column connection; column-to-beam connection, etc; and the kind of stresses acting in them: axial load or eccentric tension. For cast *in-situ* concrete, stresses are transmitted through the main reinforcement bar; meanwhile, for precast concrete, stresses are transmitted through the welding and embedded steel details, or directly through concrete for compression stresses. Connection between column and beam often induce eccentricity, where the load does not act directly through the centre of gravity of the column.

In addition, according to Nilson and Darwin (1997), and Holmes and Martin (1983), connections can further be classified into two categories: hard and soft connections.

> "…connections that achieve such continuity (continuous structure) by appropriate use of special hardware, rebars, and concrete to transmit all tension, compression, and shear stresses are sometimes called hard connections."

> "…connections that transmit reactions in one direction only, analogous to rockers or rollers in steel structures, but permit a limited amount of motion to relieve other forces, such as horizontal reactions components, are sometimes known as soft connections."

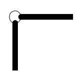
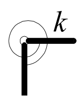
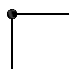
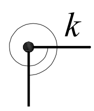
Initially in precast concrete construction, soft connections are used in order to permit dimensional changes from creep and shrinkage without causing addition internal stress within a member and in the connections, which solely depends to the gravity force. However, recent experience shows, "extensive use of soft connections indicated that the resulting structures tent to show insufficient stability against lateral forces such as high wind and particularly earthquake effects" (Nilson and Darwin, 1997). Therefore, precast concrete connections are later designed as hard connections to provide a higher degree of continuity.

In precast concrete construction, individual members are brought to the job site and joined in the field; connections are often designed to provide for the transfer of moment as well as shear and thrust, producing at least partial continuity (Nilson and Darwin, 1997). Grouting with concrete or mortar is carried out at connections in order to make the connection of continuous and monolithic. If compressive stresses are to be transmitted through the concrete at a connection, with the concrete subjected to design loads, the grade of concrete used to grout the connection should be the same as that employed in the elements being joined (Murashev et al., 1971).

There are generally three types of connections, namely pin-connected connection, roller-connected connection and rigid connection. A pinned connection and roller support allow some freedom for slight rotation, whereas the fixed connection allows no relative rotation between the connected members (Hibbeler, 2003).

Connections are often idealised in structural analysis. The torsional spring model is used to represent the connection types for pinned connection and fixed connection (Table 2.1). A pinned connection is assumed to have a spring of zero spring stiffness; on the other hand, the spring stiffness for rigid connection is assumed very large (approaching infinity).

Table 2.1: Types of connections (Hibbeler, 2003)

Type of Connection	**Idealized Connection Symbol**	**Torsional Spring Model**	**Spring Constant**
Pinned			$k = 0$
Fixed			$k \to \infty$

However, semi-rigid connections do exits where there is a certain value of spring constant. Semi-rigid connections allow partial moment transfer from beams to columns. In actual structure, the connections do not behave in either a perfectly rigid of a perfectly hinged (pinned) manner (McGuire, 1995). Semi-rigid connections are also known as flexible connections, spring hinged joints or sticky hinges (McGuire, 1995).

The behaviour of a connection depends highly to the joint stiffness. Knowing the Young’s modulus, moment area of inertia and the length of beam allow users to determine the joint stiffness by referring to the chart shown in Figure 2.2.

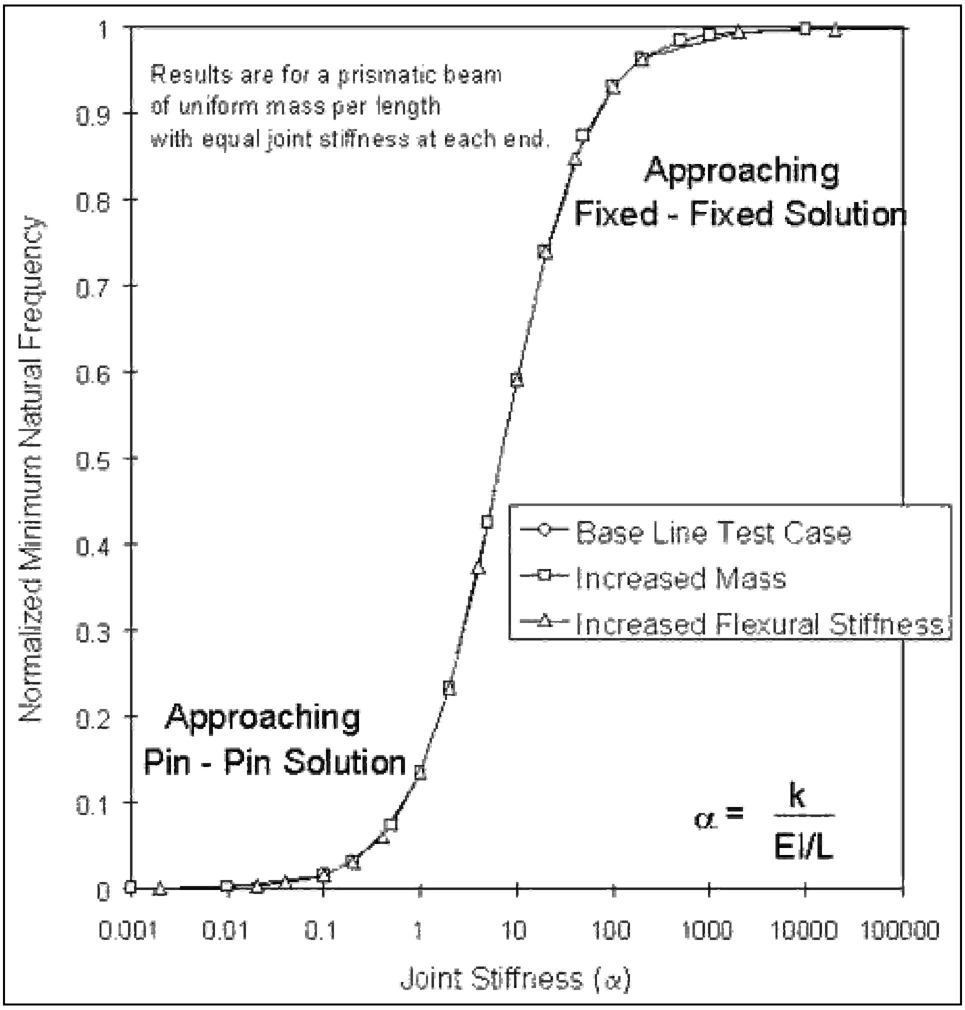


Figure 2.2: Graph of normalized frequency versus joint stiffness (McGuire, 1995)

Knowing the joints stiffness (α), the stiffness of the spring at the connection (k) can be determined using Equation (2.1).

$$k = \alpha \frac{EI}{L} \text{ kNm} \tag{2.1}$$

2.6.1 Column-to-footing Connection

Column-to-footing connections are usually constructed using steel base plate that is anchored into the precast column. Three typical types of column-to-footing connections that are commonly applied in construction are shown in Figure 2.3.

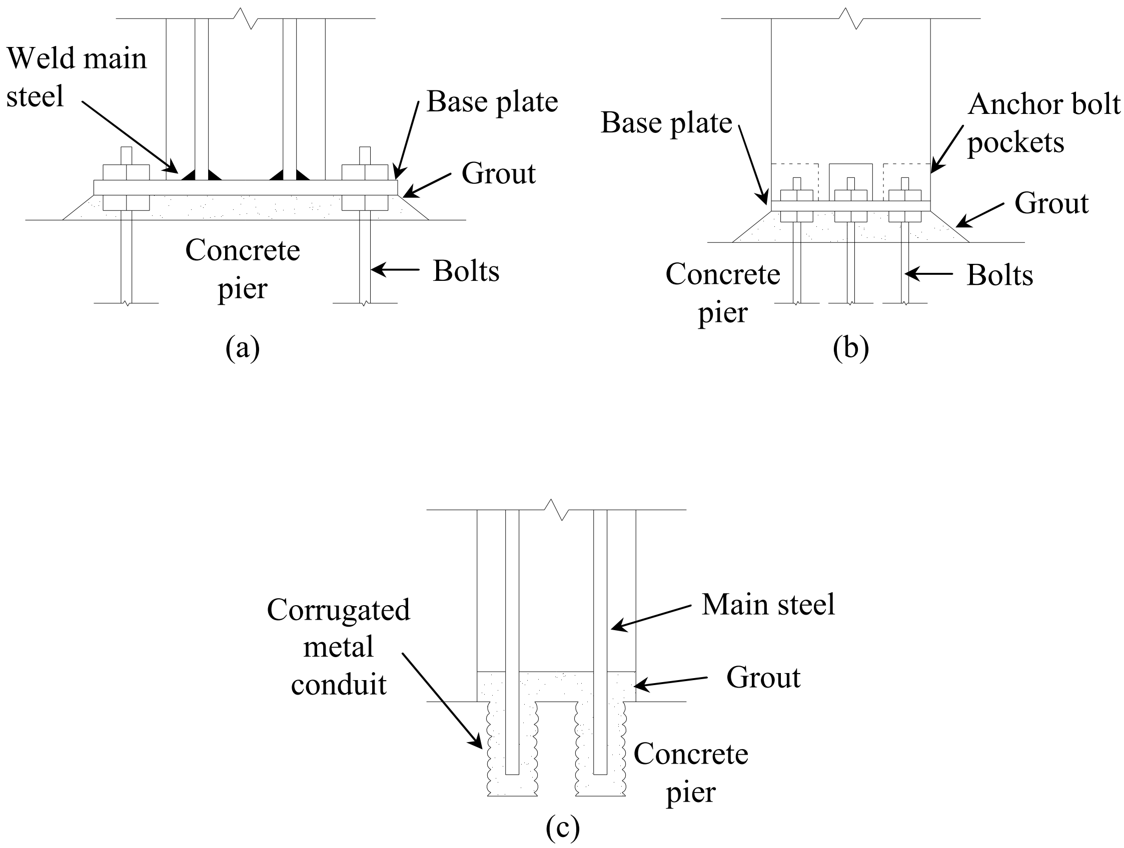


Figure 2.3: Column-to-footing connections (Nilson and Darwin, 1997)

Connection shown in Figure 2.3a has a projecting base plate. Tests have confirmed that such column connections can transmit full moment for which the column is designed (Nilson and Darwin, 1997). Meanwhile connection in Figure 2.3b has a base plate that is the same size or slightly smaller compared to the column. A non-shrinking grout is used between the base plate and concrete pier. Besides, the main bar from the column will be welded to the base plate (not shown in figure) as in

Figure 2.3a. Connection shown in Figure 2.3c is by projecting the main bar from the column into grout-filled holes without using base plate.

2.6.2 Column-to-column Connection

Column-to-column connections can be designed as pin-connected or rigid connections. Pin-connected columns can be designed to have a spherical surface at the joint of two columns. Meanwhile, rigid connected columns are usually welded and the load is transmitted through the embedded steel angles and strips, and the concrete. However, the columns of multi-storey buildings should never be designed as pin connections (Murashev et al., 1971). Figure 2.4 shows several types of column-to-column connections.

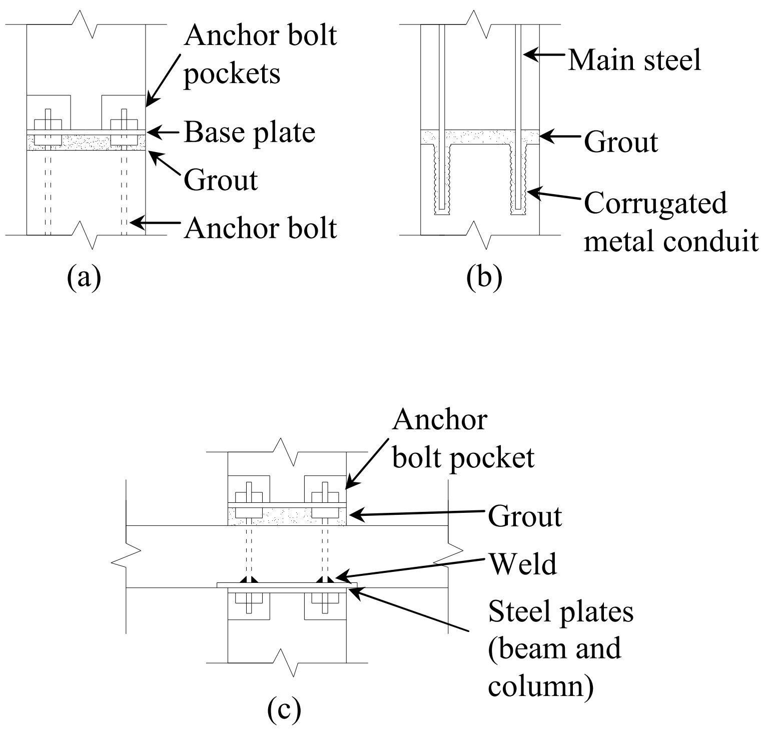


Figure 2.4: Column-to-column connections

Figure 2.4a is a type of connection that uses an anchor bolt pocket and double-nut system to level the column of the above floor. Meanwhile for connection shown in Figure 2.4b, it allows main steel from the upper column to lap-splice with the bar in the lower column. In Figure 2.4c, main bars in upper and lower column are welded to the steel cap and base plate in order to allow load transfer. In addition, closely spaced ties must be provided in both columns and beams to transfer the load between columns.

2.6.3 Beam-to-column Connection

Beams are generally connected near supports and directly placed at the side face of the columns. Transfer of forces in this region involves bending, torsion, shear, and thrust (Nilson and Darwin, 1997). Some common beam-to-column connections are shown in Figure 2.5.

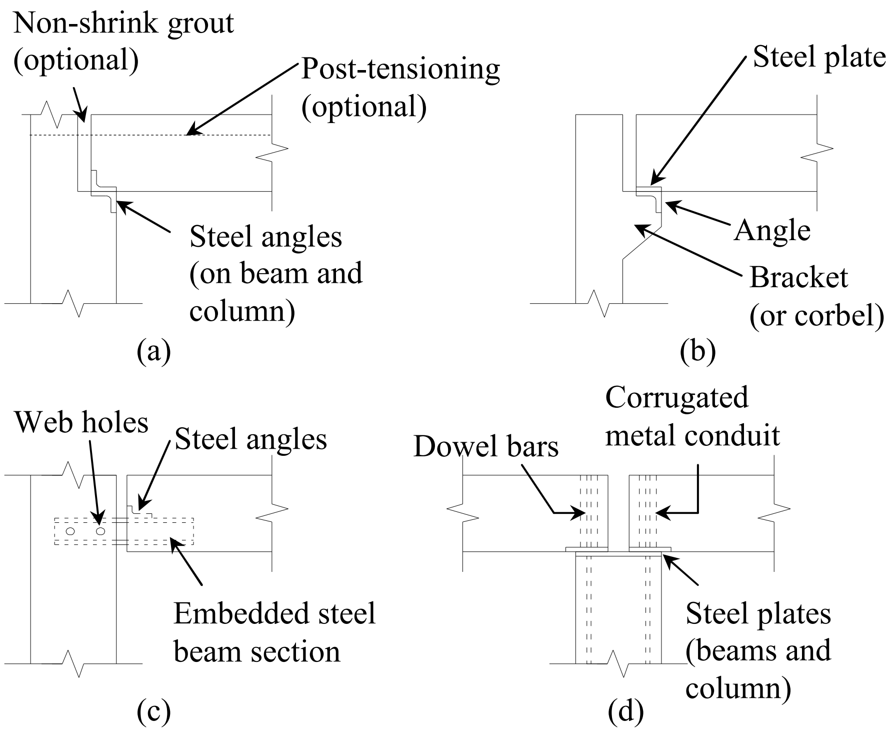


Figure 2.5: Beam-to-column connections

Figure 2.5a shows a connection detail with a concealed haunch (Nilson and Darwin, 1997). Steel angles provide a flat surface at the contact point. This type of connection is suitable to provide both vertical and horizontal reaction components. In addition, if the connection is post-tensioned, it will be able to provide moment resistance as well. Figure 2.5b is a common method in beam-to-column connection – by casting a bracket at the column and the beam will rest on the bracket.

Figure 2.5c is used if it is to avoid projections from the column surface. A socket is cast in the beam with a steel angle to receive the beam stub. Figure 2.5d is a connection with bars projecting from column in the holes in the beam-ends. These holes are normally grouted to ensure the connection is sound. In all designs of beam-to-column connections, embedded steel details and clearances are grouted over a screen with cement mortar (Murashev et al., 1971). The beam-to-column connections should always be designed as rigid connection as the connections help to increase the rigidity of the building.

2.6.4 Slab-to-beam Connection

Frequently in precast concrete construction, an L or inverted-T girder is used to provide a seat, or ledge, to support precast beams framing (or slab) into the carrying girder from the perpendicular direction (Nilson and Darwin, 1997). There are two types of girders: spandrel girder that provides exterior support and interior girder that carries reactions from two slabs. Typical ledge girder cross sections are shown in Figure 2.6.

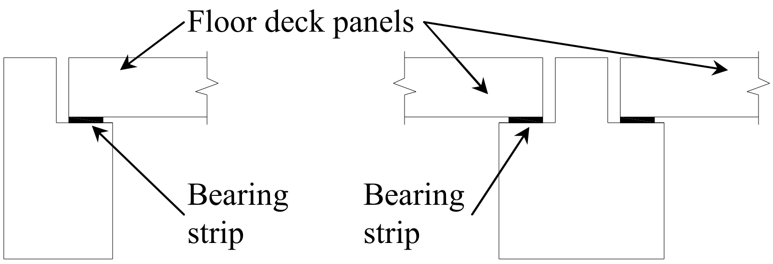


Figure 2.6: Slab-to-beam connections

2.6.5 Wall-to-slab Connection

Connection between wall and slab is another essential aspect to be well considered in design. The connection needs additional attention in detailing if the floor units are supported within the breath of the walls and the imposed load from the wall is high (Elliot, 1996). Two alternatives can be applied to this connection, the reinforced *in-situ* connections and welded connections. The reinforced *in-situ* connections provide high structural integrity and ductility; while the welded connections provide immediate stability to the element. Hence, a combination of both is usually used, where the welded connection or bolted connection is to provide temporary stability to the connection.

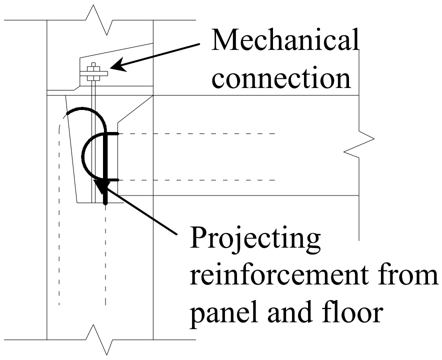


Figure 2.7: Floor slab to wall connection

2.7 Equivalent Diagonal Strut

Wall, or in-filled frames strength prediction, are often a complex and statically indeterminate problem. The strength and properties of the composite structure cannot be determined by summing the properties of both. Therefore, many researches have been carried out to further study the behaviour of in-filled frames. Polyakov (1960), Stafford-Smith (1962, 1966, 1969), just to name a few, are among researchers that have formed the basis for understanding and predicting the behaviour of the in-filled frame.

The research resulted with an equivalent diagonal strut based on the deformation shape of an in-filled frame under lateral load. This method is to simplify the in-filled frame analysis. Complex equations have been derived based on the length of contact between the column/beam and the in-fill, as well as the relative stiffness of the in-fill to the frame.

The assumption made to the equivalent diagonal strut is the strut is assumed to be pinned at both ends to the confining frame (Al-Chaar and Geogory, 2002).

The relative in-fill-to-frame stiffness is calculated using Equation (2.2), derived empirically by Stafford-Smith and Carter in 1969:

$$\lambda_I H = H\left[\frac{E_m t \sin 2\theta}{4E_c I_{col} h_m}\right]^{\frac{1}{4}} \qquad (2.2)$$

where H is the storey height; E_m is the Young's Modulus of the infill panel; t is the panel thickness, θ is the angle between the strut and the horizontal frame member; E_c

is the Young's Modulus of the frame member; I_c is the second moment of area of the column; and h_m is the clear height of the particular storey. Figure 2.8 shows the parameter used in Equation (2.2).

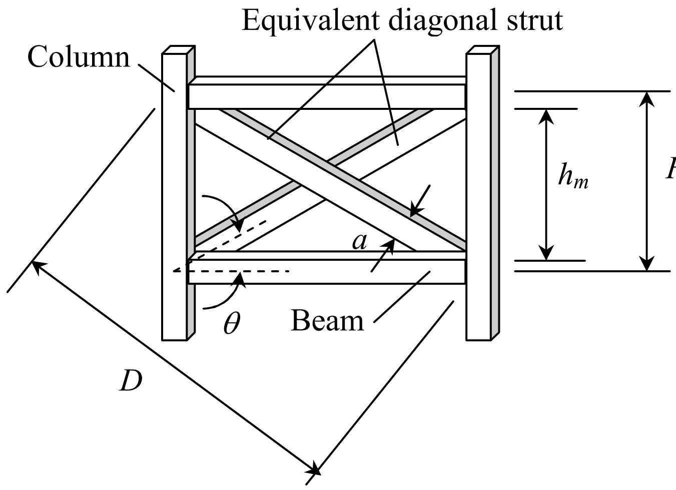


Figure 2.8: Equivalent diagonal strut

Based on the calculated value of ($\lambda_I H$) from Equation (2.2), the width of the equivalent strut, *a,* is calculated using Equation (2.3).

$$a = 0.175D(\lambda_I H)^{-0.4} \tag{2.3}$$

2.8 Standardization

In precast concrete construction, standardization is often the most important consideration. In standardization, the type of member that has proved most rational in service is selected for each constructional element of a building (Murashev et al., 1971). As compared to the other sizes, the standardized dimension must be the most economical in terms of materials, weight and cost. In addition, those elements are usually the simplest to fabricate and erect. Once the element is selected, it will be sent for mass prefabrication.

Quality can be highly assured through mass prefabrication. Besides, standardized elements are usually designed to be the largest size possible. Hence, during construction, the lifting and placing operation and connections can be reduced.

In standardization, elements are correlated based on a modular system in three categories: nominal dimension, design dimension and natural dimension. Nominal dimension is the distance between the centrelines found on the building plan. Meanwhile, the design dimension considered the clearances required during the erection process. The design dimensions usually consider a minimum clearance of 30 mm. The nominal dimensions and design dimensions can be visualized in Figure 2.9.

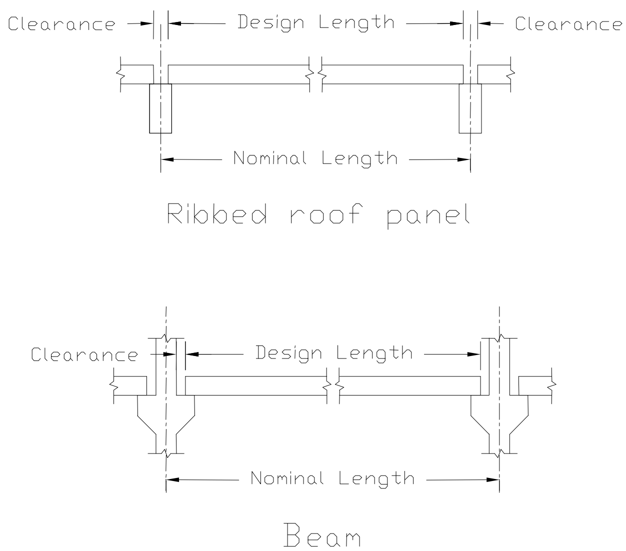


Figure 2.9: Nominal and design dimensions of precast elements (Murashev et al., 1971)

The natural dimension is the actual dimension of the elements after the prefabrication. The difference between the design dimension and the actual dimension after prefabrication is known as deviation.

2.9 High-rise Building

There is no specific definition to define the minimum limit for a structure to be classified as high-rise. However, several characterizations by researchers can be used as references for the study.

> "There is no point at which a building can be considered as tall, and the term is usually given to one the design of which is affected more by lateral than by gravity forces (Smolira, 1975)."

> "A high building is a building whose most important dimension is that of height and which dominates its environment (Bor, 1974)."

> "…'tallness' is a relative term (Amos, 1974)."

Hence, the term 'high-rise building' could be summarized as building which lateral load (wind) dominates vertical load (gravity).

There are several aspects to be considered in designing and analyzing high-rise buildings. Besides vertical load (gravity), lateral load (wind) has to be put into consideration. The effect of wind load on building can be of static load or dynamic load. Static load from wind causes swaying of structure. Dynamic load from wind, depending to several aspects, has a possibility of causing oscillating to the structure, which is rather critical regardless to the structure itself or the occupants.

2.10 Loading

Buildings are subjected loads, which can be classified into two categories: vertical load and lateral load.

2.10.1 Vertical Load

Examples of vertical loads are dead load and live load.

Dead load as defined in BS 6399-1:1996:

> "The load due to the weight of all walls, permanent partitions, floors, roofs, finishes and all other permanent construction including services of a permanent nature."

Meanwhile Nilson and Darwin (1997) defined dead load as those that are constant in magnitude and fixed in location throughout the lifetime of the structure. The main fraction that made up the dead load for a building is the weight of the structure itself. Hence, it can be calculated accurately from the dimension and material of the building. The other minor fraction is from the finish floors, plastered ceiling and may include wiring, piping and lighting fixtures.

Live load—also known as imposed load—as in BS 6399-1:1996:

"The load assumed to be produced by the intended occupancy or use, including the weight of movable partitions, distributed, concentrated, impact and inertia loads, but excluding wind loads."

During the operation period of a structure, live load may either be fully or partially in place or not present at all. Tabulated live loads should not be fully dependant on; they should only be used as references. The type of occupancy should be considered and the probable loads should be computed as accurately as possible (Nilson and Darwin, 1997).

2.10.2 Lateral Load

Generally, there are two types of lateral loads, which often influence the performance of a building: wind load and earthquake load. Wind load is often taken into consideration for all building, regardless of the height; while earthquake load is considered for building located within the seismic zone. The wind particularly constitutes one of the major forms of structural loading and even moderate winds are capable of imposing high forces on structures (Macdonald, 1975).

Wind is a natural occurring phenomenon due to the difference of pressure in the atmosphere. Nevertheless, the surface roughness has the most influence on the wind velocity. Area with high surface roughness (buildings and structures) will have lower surface wind velocity (as shown in Figure 2.10). However, the velocity increases as the height increase.

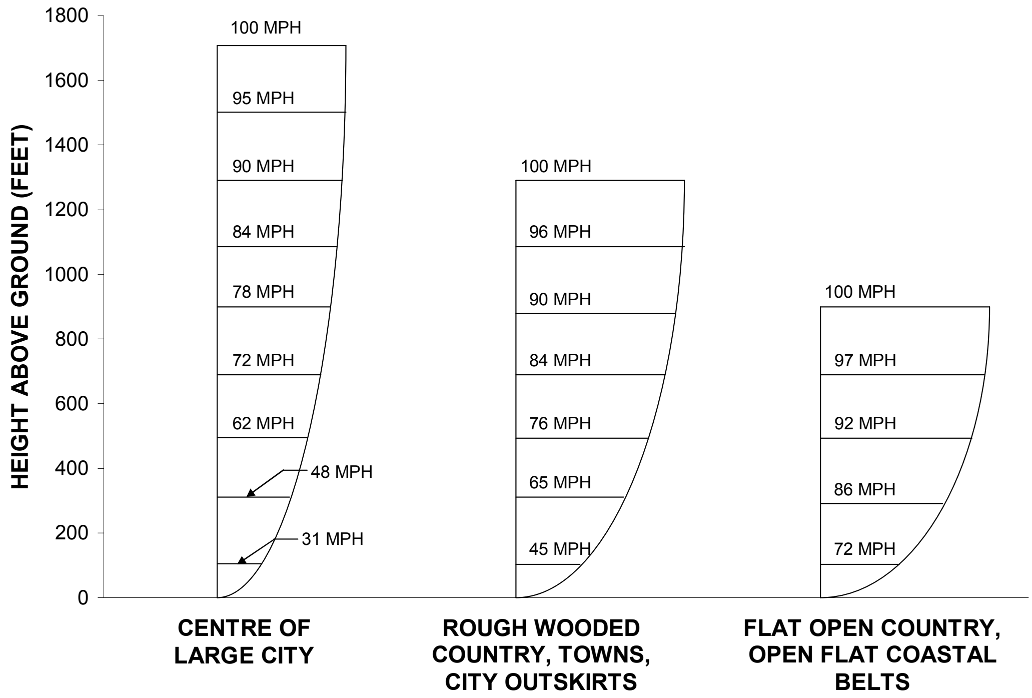


Figure 2.10: Wind velocity distribution of different surface roughness

Hence, wind load will have a very important role to the performance of a high-rise building. There are several reasons where wind effects on tall buildings are important (Chew et al., 1984):

a) The wind speed increases with height. Hence, the taller the building, the larger shear force and bending moment is induced at the foundation;

b) Buildings are seldom symmetrical. The wind load centre seldom coincides with the centre of the structural stiffness of a building, resulting in an additional torsional load to the building;

c) Advancement in construction technology produces lighter, taller and more slender building. The natural period of vibration and damping coefficient subsequently increases, where the building has higher possibility to have resonance with the wind force.

The effect of wind on structures can be classified into two: static and dynamic. Static pressure of wind acts on the structure when the airflow is constant in velocity and direction, besides it does not vary with time. On the other hand, when the air has horizontal and vertical fluctuations, and sudden and relatively brief motion, therefore, it will create a dynamic effect on the building. The sudden and relatively brief motion of wind is called gust and pulse respectively.

Alternatively, wind does not only create pressure on a building. It creates suction to a building too, depending to the wind direction relative to the building surface. The surface of the structure facing the wind direction—called windward—will experience pressure, whilst structure back facing the wind—called leeward—will experience suction.

The respond spectrum for wind is referred in order to determine the effect, static or dynamic of a wind to a specific structure. Wind spectra usually has long period where the dynamic amplification is significant. Meanwhile, for most civil engineering structure, the natural period is low. Hence, the wind will have a static effect on the structure.

Conversely, for very tall building, long span suspension bridge and electricity transmission line, they usually have long natural period, when compared to the natural period of wind, resonance will occur. Thus, these structures will experience the dynamic effect of wind.

Therefore, in tall building design and analysis, both static and dynamic effects should be taken into consideration. Static force, introduce a pressure on the building causing the building to sway; while dynamic force, will introduce oscillation to the building. The oscillation motion is in fact, the most significant aspect of the behaviour, which determines whether a tall building performs satisfactorily or not in service (Dutt, 1984). Oscillation directly determines the comfort of the occupants in the building as well. Controlling the proportion of total deflection of a building is the main key in controlling cracking of walls and external cladding caused by sway.

The designed structural system is often capable of carrying vertical load, but not horizontal load. Hence, additional system has to be introduced into the system to resist the horizontal load. Besides additional system, the cladding elements (windows, non-structural face panel) of the structure must be capable of safely transmitting the wind load to the structural elements (floor slabs, shear wall) (Hartland, 1975).

2.10.2.1Lateral Load Resistant

Lateral load resistant system are structural components in additional to those required to resist gravity loads (Dutt, 1984). Diaphragms and bracing are usually used to provide lateral load resistance to a building, especially high-rise building. The floors between storeys are considered as rigid horizontal bracing diaphragms that do not deform upon bending in their plane (Murashev et al., 1971).

There are several structural systems that provide lateral stability to a high-rise structure: interaction between flat slabs and columns; shear wall; rigid frame; perimeter tube (widely or closely spaced); core supported structures; and modular tubes. The selection of system depends to the types of structure and the height of structure. Each system has its advantages and disadvantages, which should be considered when making a choice of a system.

Shear wall is a popular element used in buildings to resist lateral load. Shear walls share the wind force at each floor, where the amount of force shared is direct proportion to the relative stiffness of each shear wall. Interaction between adjacent panels in shear wall determines the effectiveness of the shear wall in resisting wind force (Figure 2.11).

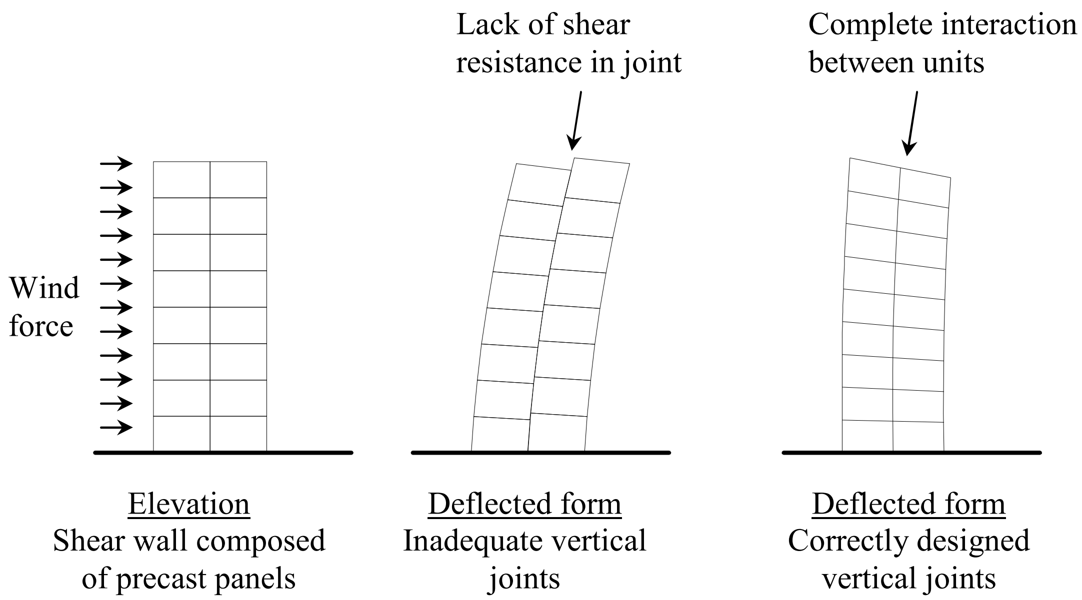


Figure 2.11: Joint performance of shear wall (Hartland, 1975)

2.11 Tributary Load

Tributary load is a term used in idealization of a panel structure. Tributary load can be defined as the necessity in determining the load transfer direction on panel elements such as walls, floors or roofs supported by a structural frame

(Hibbeler, 2005). In another word, tributary load is to classify the walls, floors or roofs as one-way system or two-way system. The classification depends on several aspects: the geometry of the structural system, the material from which it is made and method of its construction.

The one-way slab is a slab that delivers the load acting on the slab to the supporting member in one-way action. In terms of geometry, when the ratio of the longer support to the shorter support is greater than 2, the slab can be assumed as one-way slab. Besides, one-way acting slab is usually reinforced in one direction. Concrete poured on a corrugated metal deck or precast slab panel are classified under the one-way slab system.

Two-way slab is slab that delivers the load on the slab to the supporting members in both directions. A ratio of the longer support to the shorter support less than or equal to 2 will result in a two-way slab system. Reinforcements in a two-way slab is usually spanning in two directions.

2.12 Material Property

Material property is a very important aspect to be considered in structural analysis. The properties of a material will have a strong influence on the behaviour of a structure in terms of stress distribution.

Homogeneous material has the same physical and mechanical properties throughout its volume. Isotropic material has the same material properties in all local coordinate direction. An example of homogeneous and isotropic material is steel (Hibbeler, 2003)

Anisotropic and orthotropic materials both have different properties in different directions. However, anisotropic materials have shearing behaviour which is fully coupled with extensional behaviour and can be affected by temperature change. In contrast, orthotropic materials have shearing behaviour which is uncoupled with the extensional behaviour and is not affected by temperature change. An example of anisotropic material is timber, where the grain controls the behaviour of timber.

2.13 Stress

> "When a body that is subjected to an external load is sectioned, there is a distribution of force acting over the sectional area which holds each segment of the body in equilibrium. The intensity of this internal force at a point in the body is referred to as stress (Hibbeler, 2003)"

Stress can be defined as the value of force over a unit area, as the area approaches zero. The material at this point is assumed to be continuous and cohesive.

CHAPTER 3

RESEARCH METHODOLOGY

3.1 Introduction

This study is carried out based on computer software aided analysis. The software involved is the Autodesk Revit Structure and Extended Three Dimensional Analysis of Building Systems (ETABS). The main advantage of the software is that they allow bridging in order to perform a wider function. Generally, Autodesk Revit Structure provides the BIM facility and it is used to develop a digital model by assigning elements and defining elements properties. Meanwhile, ETABS can be used to analyze the structure and finally suggest the most suitable design for the elements based on the Code of Practice of interest.

3.2 Modelling Software

In Autodesk Revit Structure, 2D structural plans from AutoCAD software can be imported. Based on the plan, elements can be assigned accordingly (i.e. beams, columns and slabs). Before the assigning of elements can proceed to the higher level (for high-rise structure), the elevation of each floor should be defined. In Autodesk Revit Structure, the lowest level (0m) is named Level 1.

When assigning the elements, the properties of the elements must be first determined. Autodesk Revit Structure provides a prescribed library which consist a wide variety of elements for the users. This includes steel elements, concrete elements and timber elements.

However, if the elements properties required by the user cannot be found in the library, user can create a new element based on the required properties. For example, concrete with a specific strength which is not found in the software library can be self-defined by users, by providing required data such as the strength, modulus of elasticity, Poisson's ratio, unit weight and damping ratio. After assigning the required element, users should ensure the dimensions of the element are correct. Dimensions such as height of element (for column), width and breadth of element are predefined by the software and should be altered to the desired value by the users.

Upon completion of the elements assigning, users have two options to define the load cases that act on the structure. Load cases can be defined in this interface (Autodesk Revit Structure), or define later in ETABS, before the analysis. If users wish to define load cases in this interface, users can use the load button and choose the desired load cases (point load, uniformly distributed line load or uniformly distributed area load).

When the structure is properly defined, then the structure can be exported to ETABS for analysis. In order to export the structure, an option named 'External Tools' is used. This option allows users to select the software of interest to analyze the structure. There are three software that bridge with Autodesk Revit Structure – SAP2000, SAFE and ETABS. For the ETABS option, users can choose to save as an ETABS accessible file for later use; or directly transfer the structure into ETABS for analysis.

An advantage of Autodesk Revit Structure is that the interface provides users with several view points. The constructed structure can be viewed in 3D form and free rotation to allow users to look at the structure from any angle. In a 2D view, users can view the structure from four sides, relatively North, South, East and West point. Elevation of the overall structure can also be view in the elevation view. With the several view points available in Autodesk Revit Structure, users can easily visualize the to-be-constructed structure from different angle. Cross sectional plans, elevation plans and floor plans are ready at the same moment the model is fully developed.

3.3 Analysis Software

ETABS is a powerful software used to analyze structures, regardless 2D or 3D. As the model from Autodesk Revit Structure is imported, the model will be modelled as lines in ETABS. Several loading cases, such as dead load, live load and wind load can be assigned to the structure.

After the load cases are assigned, the model is checked using the Check Model function. If the software does not detect any problems from the structure, then the analysis can be carried out. The analysis result can be displayed in the 2D or 3D window. The results are shown in terms of deflection, reaction, shear force and bending moment diagram, and stress distribution. Results for deflection can be animated, in addition for different mode shape.

3.3.1 Elements

3.3.1.1 Frame Elements

In ETABS, all frame elements—beams and columns—are modelled as line elements. The frame element properties can be reassign or modify according to the suitability of the structural element use.

3.3.1.2 Area/Shell Elements

Generally, there are three categories of shell or area elements available in ETABS, namely membrane, plate (thin or thick plate) and shell. These elements are categorized according to the capability in resisting forces and moments. A membrane element is capable in resisting the in-plane forces and normal (drilling) moment; a plate element is capable in resisting the bending moment and transverse force, whilst the shell element is a combination of membrane and plate behaviour, which is capable in resisting all forces and moments acting on the element. In addition, a thin plate element neglects the transverse shearing deformation; whereas thick plate element includes the transverse shearing deformation.

A membrane element is usually used to model a shear wall; a plate element used in modelling the floor slab; whilst the shell element is usually used in 3D modelling, for example water tanks and domes.

3.3.2 Axis in ETABS

3.3.2.1 Global Axis

The global axis is used to define the overall structure. There are three component—the X-axis, Y-axis and Z-axis (Figure 3.1). The software defines the X-Y plane as the horizontal plane and the Z-axis is always the vertical axis, which is used to define the height of the structure.

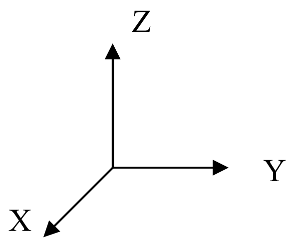


Figure 3.1: Global Axis

3.3.2.2 Local Axis

Three basic local axes (1-axis, 2-axis and 3-axis) are used in ETABS as the reference for each element. The orientation of the axes is shown in Figure 3.2.

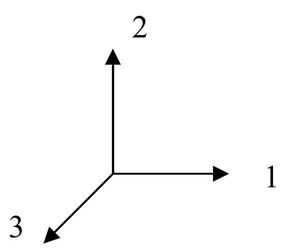


Figure 3.2: Local axis orientation

The local 1-axis is always the longitudinal direction of the reference element, and the positive direction is to be directed from the end *i* to the end *j*. The 2-axis is the vertical axis from the 1-axis, in another word; the local 1-2 plane is always vertical and parallel to the Z-axis (global axis).

3.4 Research Procedure

Throughout the study, a self-modelled structure will be used for analysis. All properties of elements and load cases will be predetermined and fixed for all analysis.

The plan of the model will be first drafted using the AutoCAD. The plan will then be imported to Autodesk Revit Structure. Elements will be assigned based on the plan and a 3D model will be generated. The properties of each element will then be assigned. The model will then be exported to ETABS through the 'External Tools' for analysis.

In ETABS, the load case combinations will be assigned during the analysis. Properties of the elements will be rechecked to ensure the accuracy. Constraints and types of restraint of each element are checked. Analysis will be carried out if no errors are detected in the model. The results of the structural performance generated by the software will be reported.

The overall analysis procedure is summarized in the following flow chart (Figure 3.3).

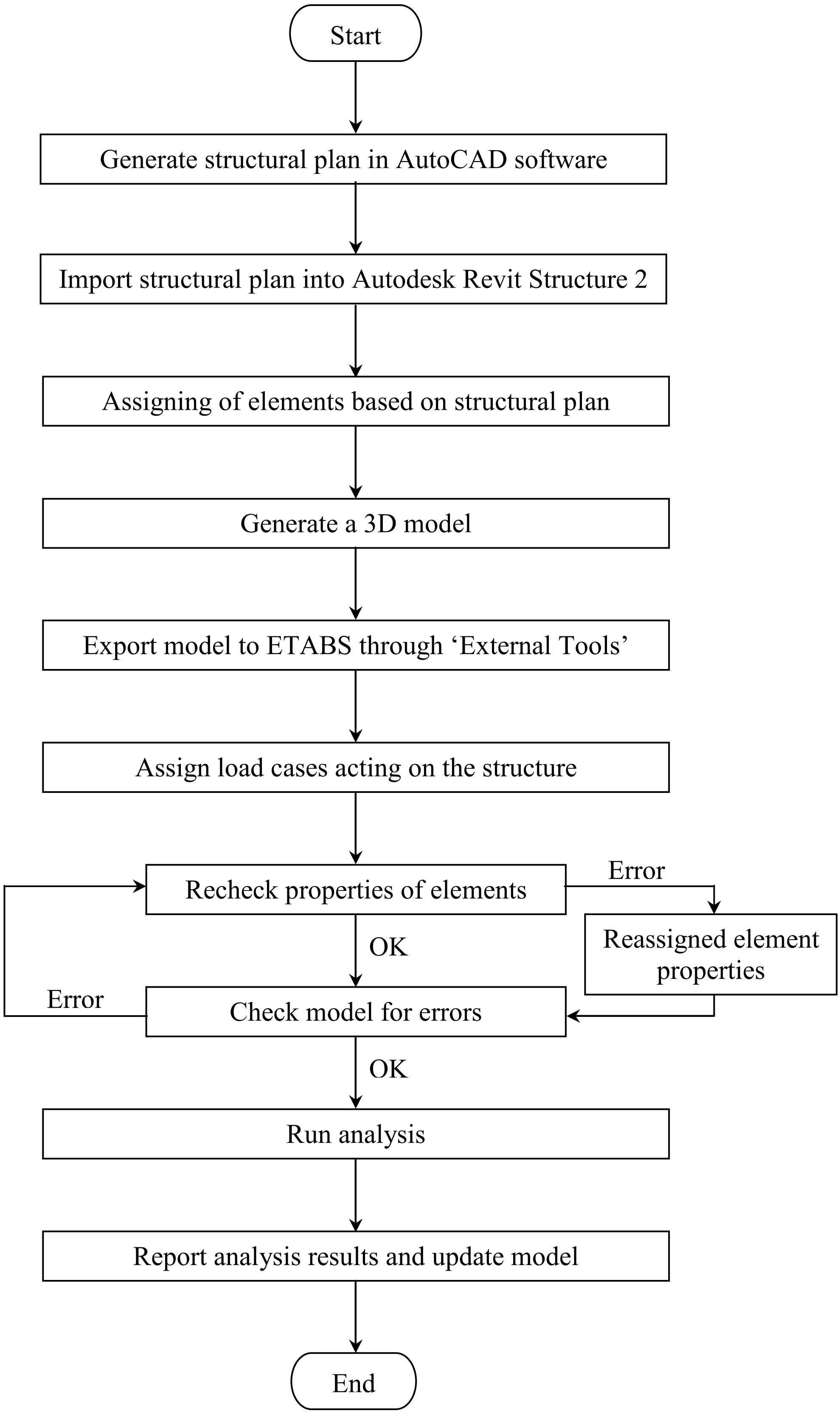


Figure 3.3: Analysis procedure

3.5 Assumptions

There are two assumptions made in this study. Firstly, the connections in the model are assumed as semi-rigid connections although theoretically the connections between precast elements for high-rise building should be rigid. This assumption is necessary as the connections for precast building are only welded or grouted. Therefore, the connections cannot be considered as monolithic connections as the cast *in-situ* construction. In addition, the connections are assumed as semi-rigid to take into consideration of the workmanship. This assumption also provides a larger range of factor of safety (FOS) to the structure.

The second assumption made is the design recommendation that will be provided by the analysis software is assumed to have considered the effect of additional stress within the member during the transportation and erection process. These additional stresses would probably initiate during the hoisting and deflection of member before it is fully supported. For example, an erected column without the support of beams will induce additional stress as the deflection will be large. The consideration of the additional stresses is very important as it might cause preliminary damage before the member would be put in place.

CHAPTER 4

RESULTS AND ANALYSIS

4.1 Introduction

This study is carried out to study the implementation of the Building Information Modelling (BIM) on the development of a 3D model on precast high-rise building to be used in structural analysis.

The floor plans (architectural drawings) for the model were first created in the AutoCAD interface (Figure 4.1). The 2D architectural drawing from AutoCAD is imported to the Autodesk Revit Structure 2 to develop the 3D model and finally exported to ETABS for structural analysis. In ETABS, the structure will be analyzed under the vertical load cases (dead load and imposed load) and horizontal load case (wind load).

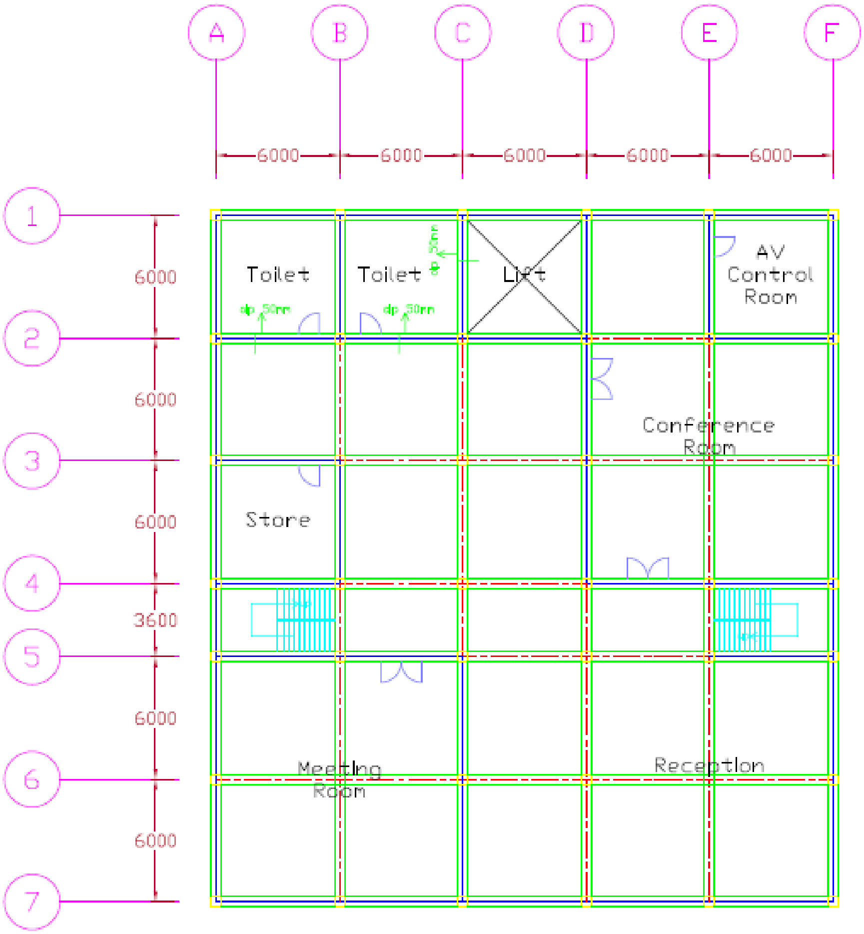


Figure 4.1: Layout plan in AutoCAD (Ground Floor)

At the end of the analysis, the deflection, shear force diagram and the bending moment diagram will be developed. The element that is resisting the maximum shear force and bending moment will be identified as it will be used as the reference value for element selection.

4.2 Building Information Modelling

BIM is a very powerful and useful tool in engineering modelling. This tool, however, has been incorporated into the Autodesk Revit Structure 2 software.

Generally, the BIM process starts with creating the model in the Autodesk Revit Structure 2 interface. The Autodesk Revit Structure 2 software interface provides a very user-friendly interface.

The modelling process is started with importing the 2D architectural drawing into the interface. As the 2D architectural file is in AutoCAD (.dwg) format, therefore, the import with the .dwg option from the File menu bar is selected (Figure 4.2).

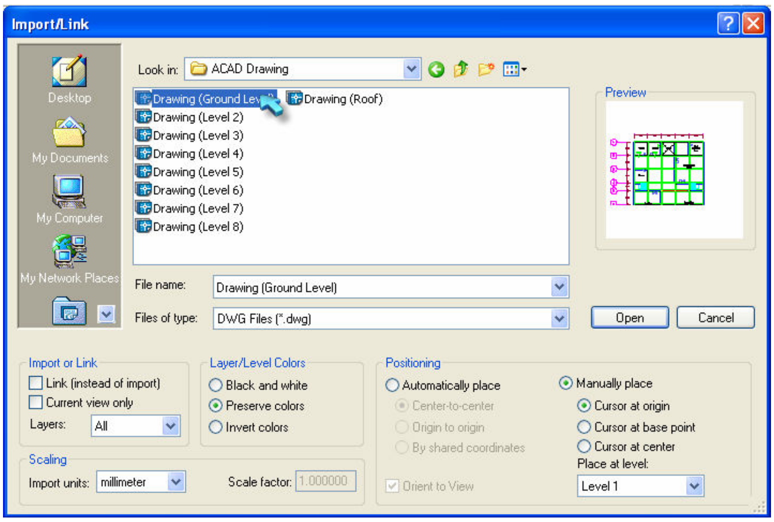


Figure 4.2: Import file from AutoCAD to Autodesk Revit Structure interface

The imported file will be treated as a block in the interface, therefore, the explode option is selected to enable each element to be selected individually (Figure 4.3).

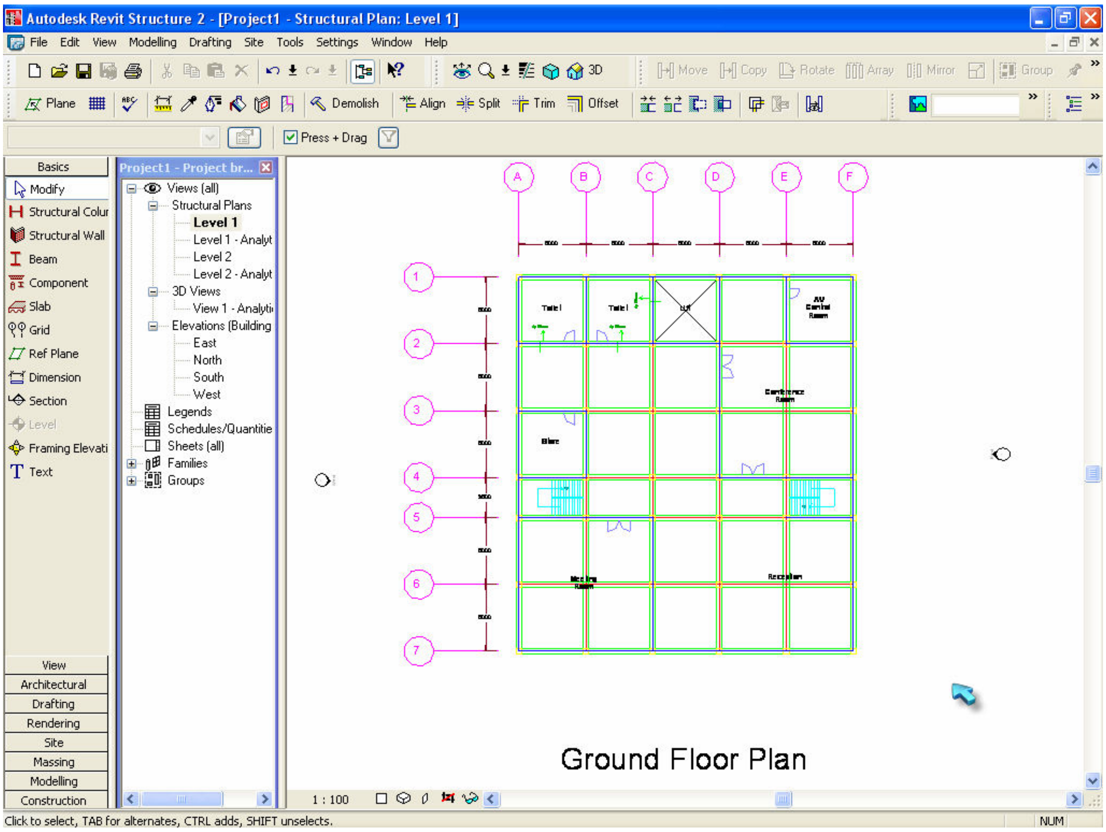


Figure 4.3: Imported AutoCAD drawing

The assignment of element based on the drawing should be done according to the construction sequence. First, to place the column, the suitable column section is loaded from the library that has been provided with the software. The suitability of the predefined column properties, such as the dimensions is checked, before the column is assigned. The cursor will then be brought to the column location on the plan to create a column at the appropriate place. The Autodesk Revit Structure 2 provides the snapping function as available in AutoCAD. The element will be snapped to the column point as the cursor moves near to it. After all the columns have been assigned, the beam will be assigned by first selecting the suitable beam section from the section library. The dimension is ensured correct before the assigning is done.

The initial sizes of the beams are determined using Clause 3.4.6.3 and Table 3.9 BS 8110-1:1997 (Appendix A).

The process will then proceed with the assignment of slab. The type of slab is selected from the section library. Slabs can be assigned in two ways, by selecting the supports (beams) or wall. As the walls have not been assigned, the slab will be assigned based on the available beams. Subsequently, the walls will be assigned. Opening for windows and doors will be created according to the architectural drawing. Upon completion for the ground floor modelling, users can view the created model from different views—plan view (Figure 4.4), elevation view (Figure 4.5) and 3D view (Figure 4.6).

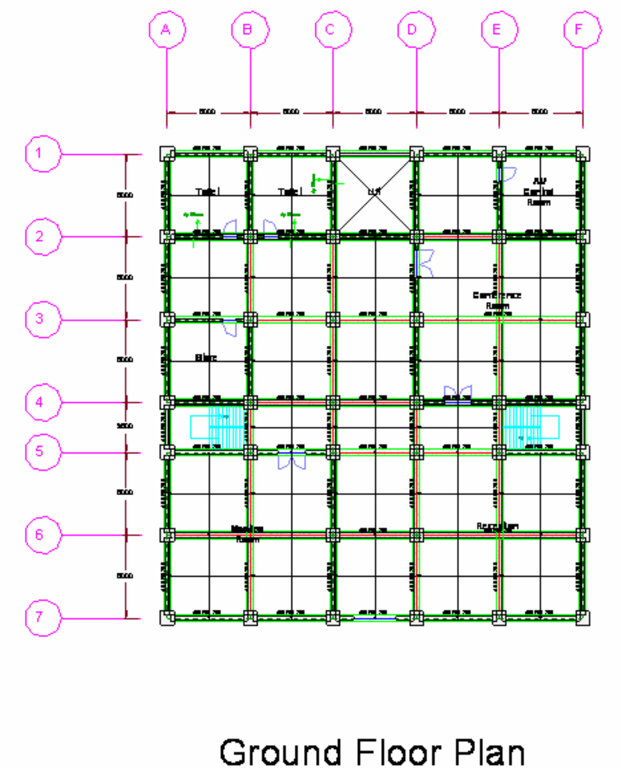


Figure 4.4: Plan view in Autodesk Revit Structure

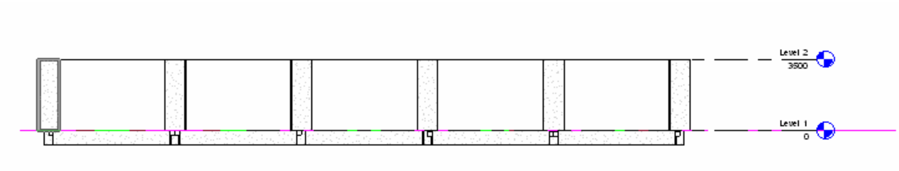


Figure 4.5: Elevation view (North view)

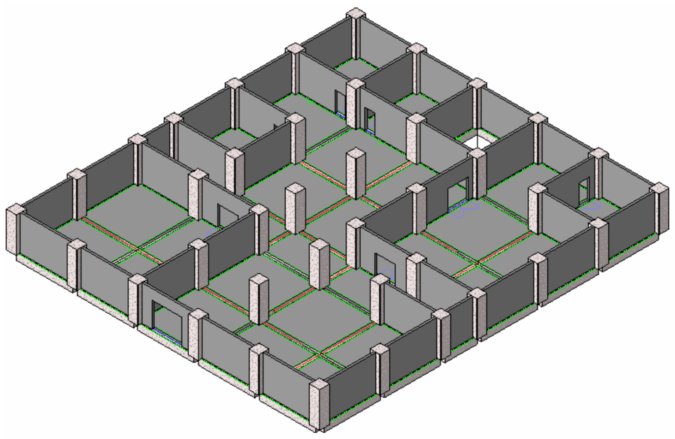


Figure 4.6: 3D view

In 3D view, there are several options, for instance, wireframe, hidden line, shading and shading with edges (Figure 4.6). The software, too, provides the external lighting option where users can model the sunlight. Based on the openings provided to the building, the brightness in the building can be determined. This allows users to verify the sufficient amount of openings on the building for energy saving.

Besides the above mentioned views, there are other available views such as cross section view, software predefined view, specifically defined view and the dynamic walk through view. These views allow users to view the arrangement of the structural elements in the building, in addition to identify any mistakes in the modelling.

Once the model accuracy is ensured correct, loading will be assigned. Loading on the building (live load or imposed load) can be assigned in two ways—in Autodesk Revit Structure 2 interface during the modelling or in ETABS, before the analysis is carried out. However, assigning the loading in Autodesk Revit Structure 2 is more convenient as the user can assign the appropriate imposed load based on the label on the architectural plan. If the load were assigned in ETABS interface, the users would have to refer to the initial architectural plans for the function of the particular floor space.

Once the loading is assigned, the modelling process can proceed to the following level. Before proceeding to the following level, the height of the following level should be defined in the elevation interface (Figure 4.7).

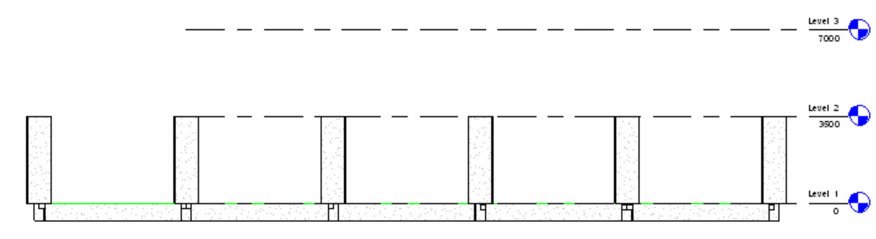


Figure 4.7: Create new elevation

After the new elevation has been created, the assigning process of columns, beams, slabs, walls, openings and loadings is repeated. The process is repeated until the entire structure has been fully modelled.

Once the modelling process is done, the model is again view in the 3D view (Figure 4.8) to check for any mistakes, such as accidentally left out member.

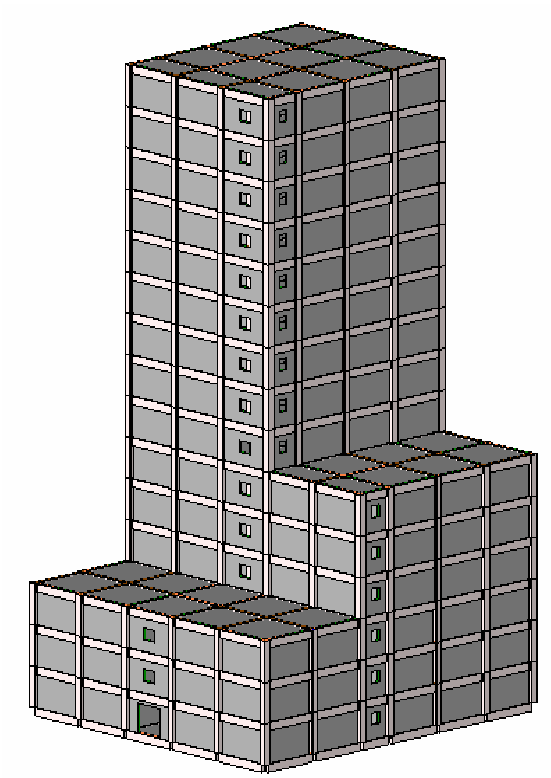


Figure 4.8: 3D view of the completed model

Once the model has been completed, there is plenty of information that can be shared by different parties. The 3D model will be able to give the client a better view of the structure that is going to be built. With the model, the client can view clearly the entire structure including the partitions within the building and the openings. The floor plans and elevation drawing can be used by the architect for further alteration. In addition, customized section view can be generated from the model to be used by the designing engineer.

With this modelling application in-use, any alterations made to the model will directly change the related information in the other views, i.e. plans view, elevation view and sectional view. Thus, eliminates the redundant works of correcting every drawing with respective to the changes, and the most important, the lost of information and miscommunication is minimized.

By implementing the BIM in construction, the construction data can be kept in the digitized format. Moreover, all parties involved in construction in addition to client are able to obtain all the information of the construction project just within one digital file. Updating works can be done periodically as the construction works is taking place. As the project completes, all the information will be stored for further action and reference in the future. BIM also helps in eliminating paper storage where all the plans are digitized. Information searching is made easier with BIM.

Besides providing complete information of a project for almost all the parties involved in the construction, BIM also provide analysis tools in aiding the designing engineer to obtain the design data with the same 3D model generated with the Autodesk Revit Structure 2. However, the analysis tools are considered third party software as they are not packaged with Autodesk Revit Structure 2.

To proceed with the structural analysis procedure, the model will need to be first exported to the analysis software. The Autodesk Revit Structure 2 is bridged with three powerful Computers and Structures Inc. (CSi) software, namely ETABS, SAFE and SAP2000.

To export the model, an add-in tools—CSiXRevit V9.0.1 – CSi Revit Data Interchange Program—need to be installed to enable the function. The installation of the CSiXRevit will provide an additional option in the Tools toolbar where exporting of model to third party analysis software is enabled (Figure 4.9).

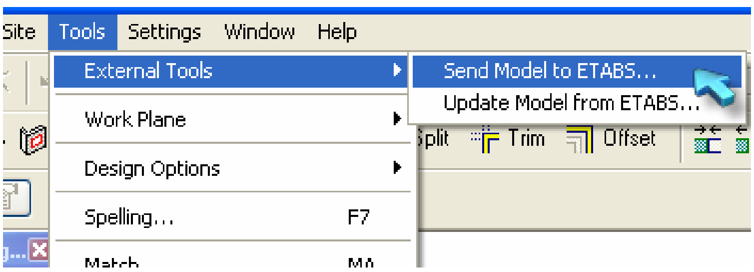


Figure 4.9: External tools to export model for structural analysis

Upon clicking on the option, the software will process the model by calculating the total numbers of each types of element and the load cases assigned on the model to be transferred to the analysis software (Figure 4.10).

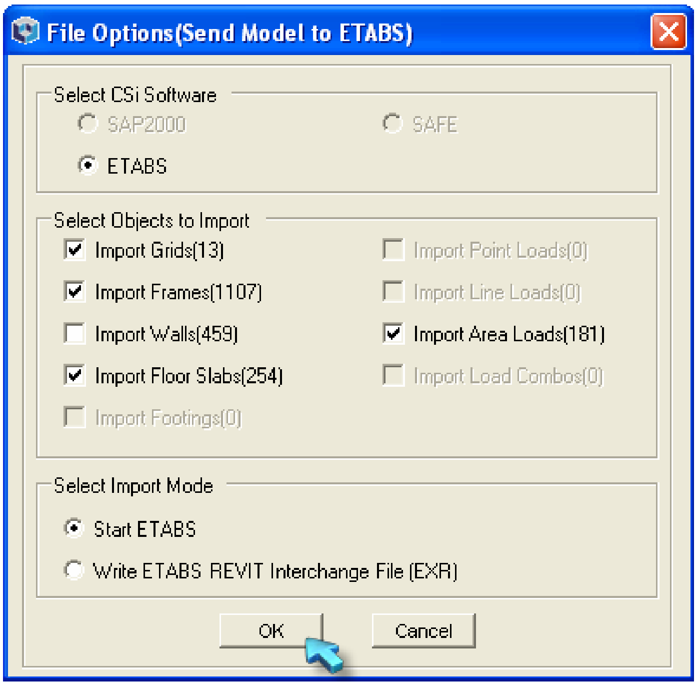


Figure 4.10: Exporting model to ETABS

This interface allows users to select criteria to be exported to ETABS for analysis besides selecting the other available analysis software and whether to start with the analysis software or just update the information to the file.

At this stage, however, there is a problem during this information transfer between Autodesk Revit Structure 2 and ETABS. Wall elements cannot be transferred from Autodesk Revit Structure 2 to ETABS. The transferring process will be terminated when the ETABS application is executed, resulting model transfer failure.

To enable the transferring process, wall elements will not be selected as items to be transferred to ETABS; however, they will be reassigned in the ETABS interface.

4.3 Information Transfer between the Software

The information is transferred from Autodesk Revit Structure 2 to the ETABS for structural analysis. The information transferred should remain consistent to avoid information lost and to ensure the analysis results is reliable. However, two problems occur during the information transfer process.

When the model with the wall element is selected to be exported to ETABS from Autodesk Revit Structure 2, the model will not be able to export. The exporting process will be terminated once the model reached the ETABS interface.

On the other hand, the frame information is not fully transferred into the ETABS model. The sectional dimension for the beams and columns are not transferred as the precast concrete section library is not found in the ETABS database.

The information that is successfully transferred from Autodesk Revit Structure 2 to ETABS model is summarized in Table 4.1. Besides, Table 4.1 shows the terms used in both Autodesk Revit Structure 2 and ETABS.

Table 4.1: Information transfer from Autodesk Revit Structure 2 to ETABS

Classification	**Autodesk Revit Structure 2**	**Information Transferred**	**ETABS**
Material	Concrete – Precast Concrete 35 MPa	Behaviour Unit weight Young's Modulus Poisson's ratio Shear modulus Thermal expansion coefficient Concrete compression strength	Concrete – Precast Concrete 35 MPa
Frame	Column	Column height Material properties	Frame/Line element (AUTO#)
	Beam	Beam length Material properties	Frame/Line element (AUTO#)
Floor	Floor: Generic 200 mm	Dimension Material properties	Rvt-Floor1
Load case	Area Loads : Area Load 1 DL1 (1)	Load magnitude Load position	Rvt-DL1
	Area Loads : Area Load 1 LL1 (2)	Load magnitude Load position	Rvt-LL1
	Area Loads : Area Load 1 WIND1 (3)	Load magnitude Load position	Rvt-WIND1
	Area Loads : Area Load 1 SNOW1 (4)	Load magnitude Load position	Rvt-SNOW1
	Area Loads : Area Load 1 LR1 (5)	Load magnitude Load position	Rvt-LR1
	Area Loads : Area Load 1 ACC1 (6)	Load magnitude Load position	Rvt-ACC1
	Area Loads : Area Load 1 TEMP1 (7)	Load magnitude Load position	Rvt-TEMP1
	Area Loads : Area Load 1 SEIS1 (8)	Load magnitude Load position	Rvt-SEIS1
Grid	Grid	Grid position Grid data	Grid

Information that has failed to be transferred from Autodesk Revit Structure 2 is shown in Table 4.2.

Table 4.2: Information transfer failed

Classification	Information transfer failed	Note
Wall	Wall element	Model will not be transferred if wall element is included in export option
Frame	Column section Beam section	Frame sections are automatically assigned as AUTO# which does not have the sectional properties as assigned in Autodesk Revit Structure 2

4.4 ETABS

The Extended Three Dimensional Analysis of Building Systems, Nonlinear Version 9.0.0 (ETABS) will be used as the third party structural analysis software for Autodesk Revit Structure 2. When the model is exported to ETABS, the software will be executed while the Autodesk Revit Structure 2 will be in a freeze mode till the ETABS interface is exited. The exported model will be displayed in two views in the ETABS interface, tiled vertically. However, there are several view options for the users.

In ETABS, similar to Autodesk Revit Structure 2, there are different views—object shrink, object fill, object edge (Figure 4.11) and extrusion—opt to the selection of users. However, the object edge (similar to wireframe in Autodesk Revit Structure 2) is often used as the windows can be refreshed and updated at a faster rate.

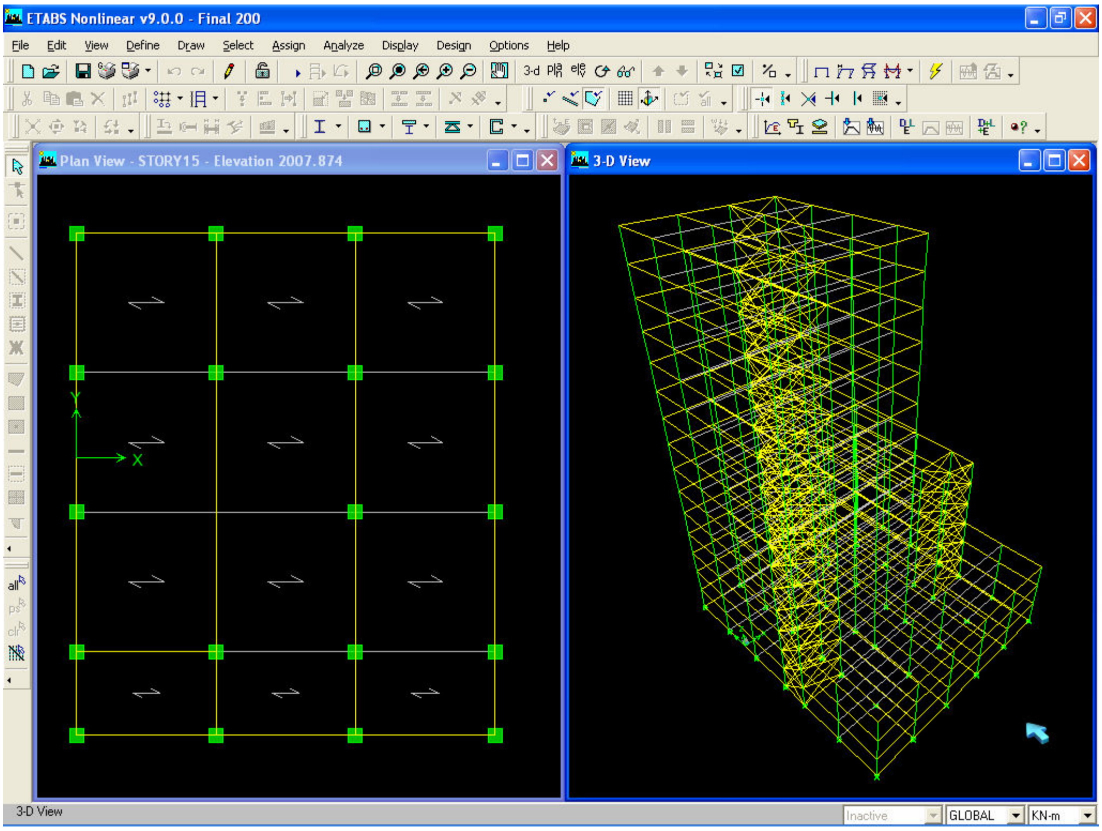


Figure 4.11: ETABS interface

As in ETABS, upon receiving the exported model from Autodesk Revit Structure 2, the sections (columns and beams) from Autodesk Revit Structure 2 cannot be recognized by the software as the precast concrete section library is not provided with the software.

However, this problem can be overcome by redefining the frame sections in the ETABS interface. The frame sections are created by defining the dimension of the cross sections. After the suitable frame properties have been assigned, the beams are partial released to model the semi-rigid connection at both ends (Appendix B).

As to create the wall element, the area in between frame elements where the wall would be placed will be selected. The respective area element will be assigned.

Conversely, when wall elements are included in the analysis, the results was not as expected. When the wall elements are meshed for analysis, the load transfer from wall to beam is incorrect. As a wall is placed on a beam, the self weight of the wall should act downwards (-Z of the global axis) onto the beam, however, in the software, the some of the resulting forces act upwards.

Therefore, all walls are removed from the model and the self weight of wall that is acting on the beam is calculated separately and the load is applied as line load acting on the respective beams. In addition, an equivalent diagonal strut (Appendix C) will be placed on the critical points, such as at the lift shaft and staircase (Figure 4.12). The dimension of the strut will be calculated based on Equation (2.2) and (2.3). Meanwhile, the mechanical properties of the strut will be of precast concrete. However, the self weight of the material will be assumed not to contribute to any dead load as the load has been considered as a line load acting on the beam. To ensure the analysis can proceed, the self weight for the strut is input as 0.01 kN/m^3 instead of 0. In addition, constraints at both ends of the strut are released to allow pinned connection.

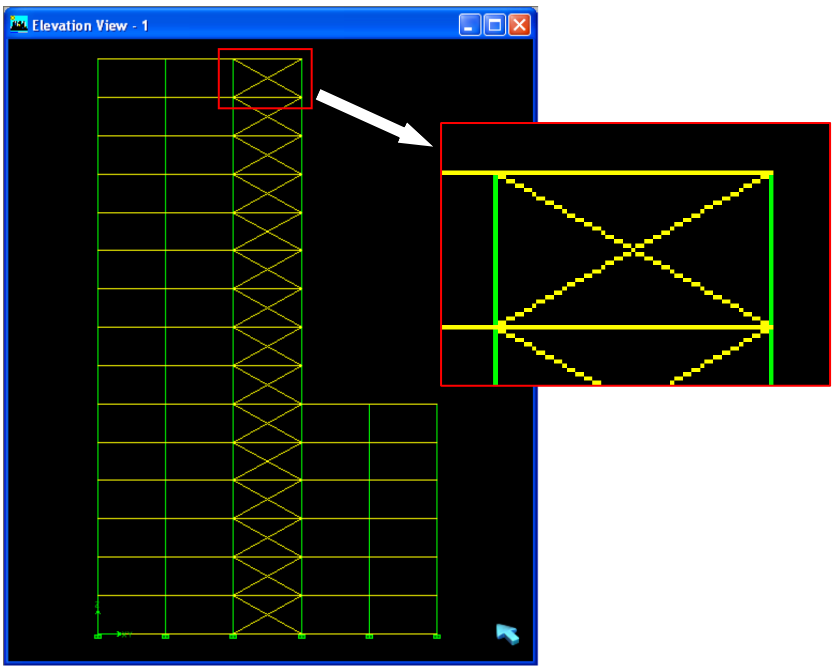


Figure 4.12: Equivalent diagonal strut modelling

Due to the absence of wall elements in the model, consequently, the wind loads are calculated manually and applied as point load onto the structure (Appendix D). The calculation is carried out according to BS 6399-2:1997.

When all the elements are well assigned, the Check Model option is selected to check for errors in the model. Some available options are overlapping lines elements, overlapping area elements, gaps between points, gaps between points and line elements, and gaps between points and area elements.

Analysis options and output setup can be set at the Set Analysis Option. Users can define the types of analysis to be carried out, such as 3D analysis or frame analysis. As for this study, the 3D analysis is carried out. Besides, there are options for dynamic analysis setup – number of mode shapes required.

The ETABS analysis software is capable in generating many results. Just to name a few, the deformed shape, lateral drift, support reaction, shear force diagram, bending moment diagram, stress distribution and mode shapes.

After all the analysis procedures have been carried out, the designed model can be exported and updated to Autodesk Revit Structure 2 model. This is where the change in member section and properties will be updated. However, due to the absence of the precast concrete section library in ETABS, therefore, this option is not performed in this study.

4.5 Analysis Results

From the analysis, a column which has the largest horizontal displacement is selected as a reference column. In addition, the natural period for the first mode of shape is obtained. The maximum deflection for each category of beam element is checked to ensure that the deflection is less than the maximum allowable value.

From the modelling, the required dimensions based on the basic span/depth calculation for the 6 m span beam are 300 mm width × 400 mm depth. The initial step of the analysis is to analyze the structure with various walls of different thickness to determine the most suitable wall thickness. From the analysis, a graph of horizontal displacement parallel to the wind load is plotted (Figure 4.13). The comparison is made on a same reference column for all the cases.

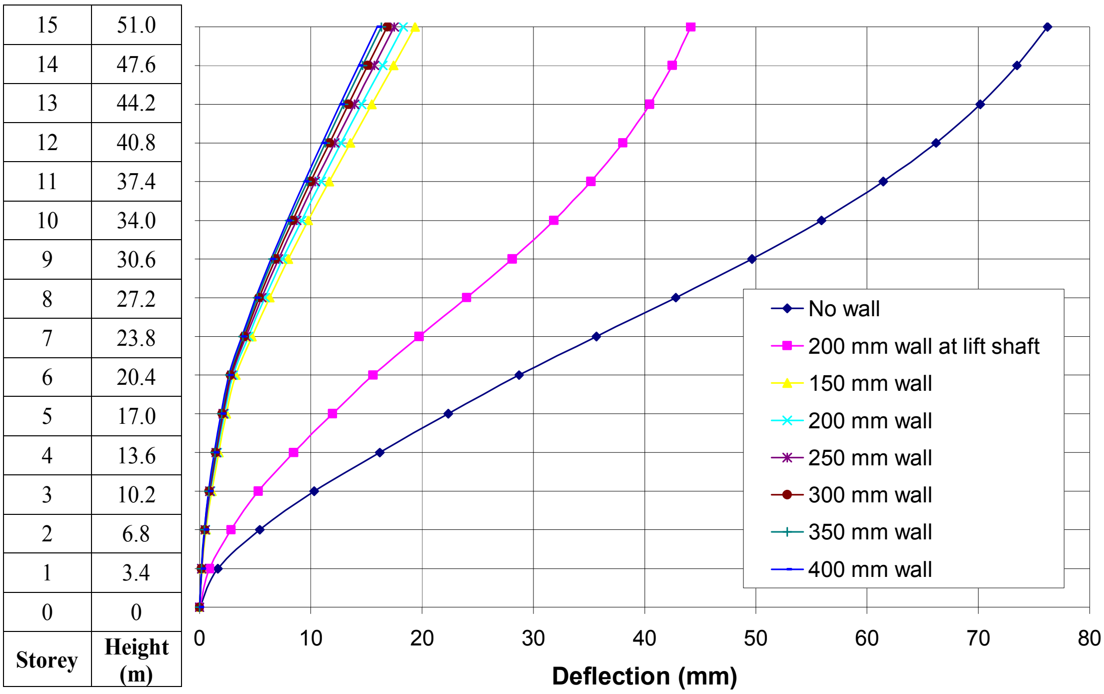


Figure 4.13: Graph of storey height versus horizontal deflection for various wall thicknesses

The graph shows that when the structure is not provided with any wall elements, the deflection is large (76.5 mm). When the wall of 200 mm thick around the lift shaft is added, the deflection significantly reduced by about 30 mm to 45.5 mm. With the presence of 200 mm wall around the lift shaft and staircase, the deflection reduced to less than 20 mm.

The effect of wall thickness is studied by varying the wall thickness, and the deflection is observed. From the graph, the displacement reduces as the thickness of wall increase. Hence, the stiffness of the overall structure will increase when the thickness of wall increases. However, for a wall to resist lateral load, the wall should be place parallel to the load direction.

Therefore, it is also noted that the reduction of deflection is very small with every increment of 50 mm wall thickness. In another word, the increasing wall thickness does not play a significant role in reducing the deflection nevertheless the wall is needed to provide isolation for noise and heat.

From the analysis, the presence of wall around the lift shaft and staircase is sufficient to keep the building deflection less than the maximum allowable deflection as stated in the code of practise; hence, the surrounding wall panel for the structure can be replaced with other façade panel or glass panel, besides precast concrete wall panel.

Therefore, with the model of 200 mm thick wall placed at the lift shaft and the staircase, the analysis for various connections rigidity is carried out. According to McGuire (1995), the joint stiffness determines the types of connection in a structure. For a connection with joint stiffness, α, less than 1, the connection behaves as a pinned connection; whilst for a connection with joint stiffness greater than 100, the connection will behave as a rigid connection.

The deflection of the building is observed and the results are as shown in Figure 4.14.

Storey	Height (m)
15	51.0
14	47.6
13	44.2
12	40.8
11	37.4
10	34.0
9	30.6
8	27.2
7	23.8
6	20.4
5	17.0
4	13.6
3	10.2
2	6.8
1	3.4
0	0

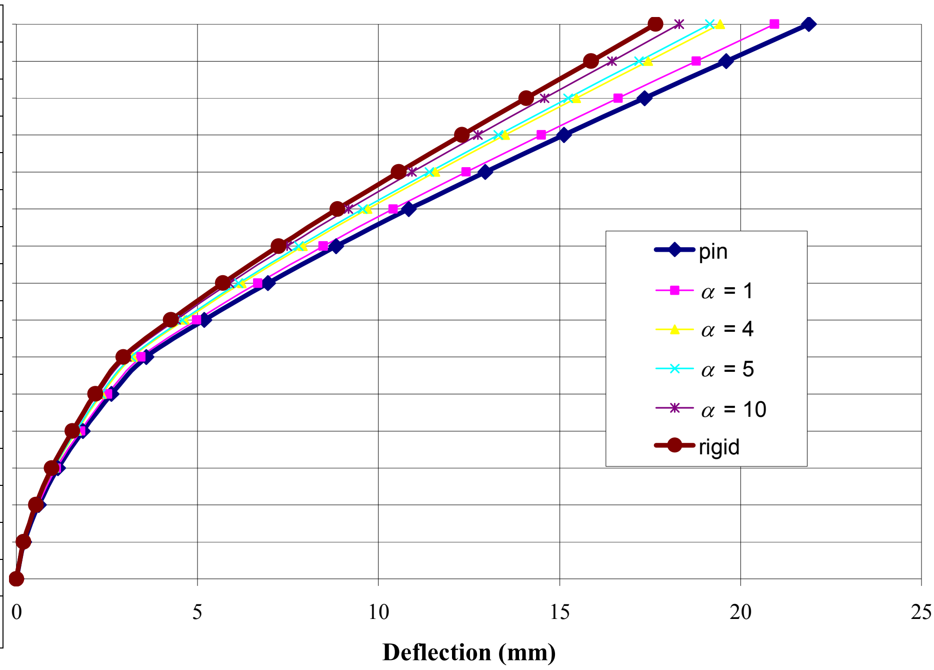


Figure 4.14: Graph of storey height versus horizontal deflection for various connections rigidity

Based on Figure 4.13, the deflection of the building does not vary much with the variation of connection rigidity. However, the types of connection highly influence the member section selection. As for a pinned connection, the mid-span moment for the beam will be relatively large as compared to the rigid connection. Therefore, a larger section will be required to resist the moment. On the other hand, if the connection is rigid, the moment resisting capacity of the beam will relatively smaller. However, in a precast concrete construction, the rigid connection could hardly be achieved. Therefore, the semi-rigid connection will be used in the model for analysis in order to obtain more accurate results.

There is a change of gradient in the graph shown in Figure 4.14. This could probably due to the reduction of effective floor area after level 6 and the stiffness of the structure is less as compared to the lower part of the structure. However, for all the connection rigidity cases, the deflection of the structure is in the form of cantilever structure.

Based on the graph developed by McGuire (1995), the mid value for the semi-rigid region is 10 as the graph is plotted on semi-log axis; therefore, the semi-rigid connection is assumed to have a joint stiffness of 10. Further analysis will proceed with a joint stiffness of 10.

Besides displacement, the natural period of the first mode shape can be used to show the relative stiffness of the structure (Figure 4.15).

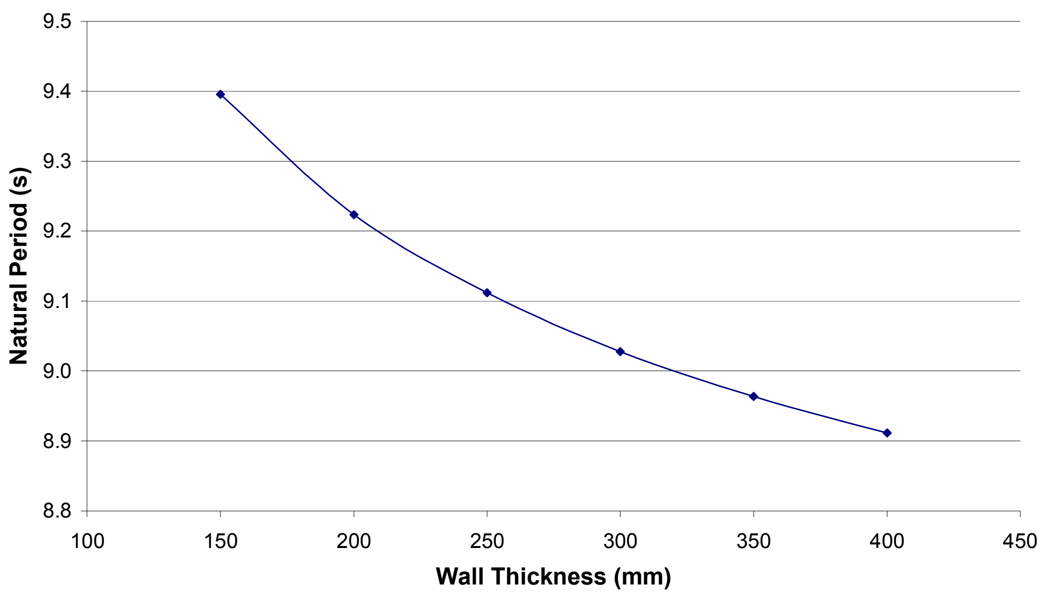


Figure 4.15: Graph of natural period of the first mode shape versus wall thickness

From the graph, it can be observed that the increase of wall thickness will reduce the natural period of the first mode shape of the structure. Hence, indicating that the stiffness of the structure has been relatively increased. Besides the mode shape of the structure, the natural period of the structure should be observed tightly.

This aspect is especially important as the potential of resonance could occur, resulting in severe damage to the building. The natural period of the structure should be compared with the natural period of wind which ranges from 3 seconds to 60 seconds. Modification to the design of the structure should be carried out if resonance occurs as resonance could cause the structure to oscillate and finally fail.

The increment of wall thickness will also increase the weight of the structure. Besides, the self weight of the wall will increase the burden on the supporting beam, resulting in additional deflection.

From the analysis, a load combination for serviceability limit state [(1.0 × dead load) + (0.8 × imposed load) + (0.8 × wind load)] ($1.0G_k + 0.8Q_k + 0.8W_k$) is created. The deflection for the beam with the highest load is observed. The deflection check is made on each category of beams classified based on the length. The maximum deflection of the beams should meet the requirement as stated in BS 8110-2:1985, Clause 3.2.1.1 (Appendix E). The results are tabulated in Table 4.3 and summarized in Figure 4.16.

Table 4.3: Maximum deflection for each beam span

Wall thickness	**Beam span**	**Maximum deflection (mm)**	**Note**
150 mm	3.6 m	0.257	< 7.2 mm OK
	6 m	8.396	< 12 mm OK
	12 m	17.104	< 20 mm OK
200 mm	3.6 m	0.326	< 7.2 mm OK
	6 m	8.857	< 12 mm OK
	12 m	17.101	< 20 mm OK
250 mm	3.6 m	0.395	< 7.2 mm OK
	6 m	9.317	< 12 mm OK
	12 m	17.096	< 20 mm OK

Table 4.3: Maximum deflection for each beam span (con't)

Wall thickness	Beam span	Maximum deflection (mm)	Note
300 mm	3.6 m	0.464	< 7.2 mm OK
	6 m	9.777	< 12 mm OK
	12 m	17.091	< 20 mm OK
350 mm	3.6 m	0.534	< 7.2 mm OK
	6 m	10.239	< 12 mm OK
	12 m	17.085	< 20 mm OK
400 mm	3.6 m	0.603	< 7.2 mm OK
	6 m	10.698	< 12 mm OK
	12 m	17.079	< 20 mm OK

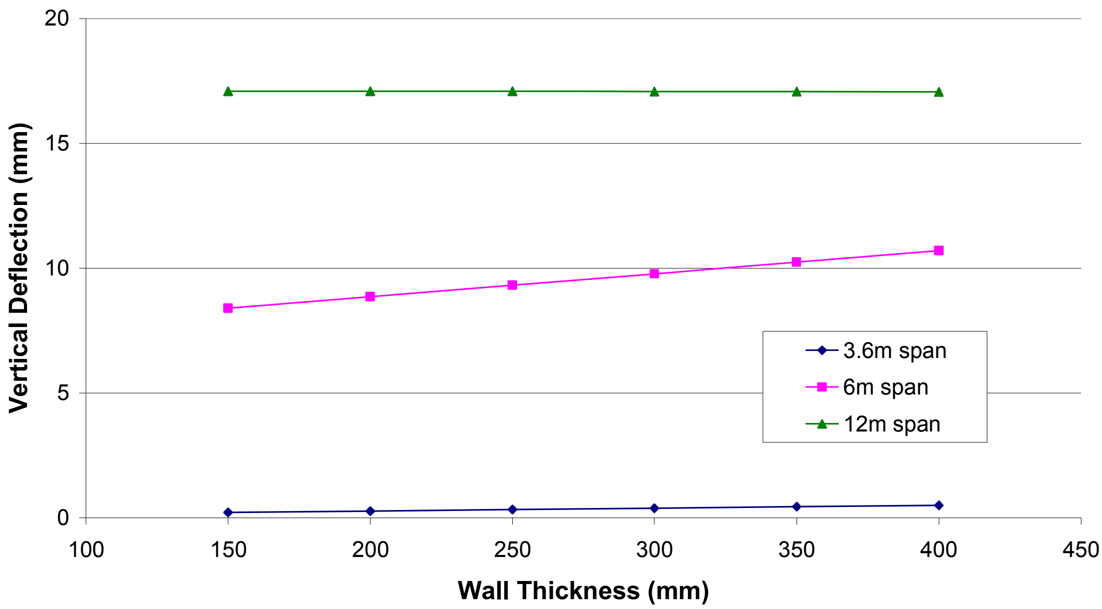


Figure 4.16: Graph of vertical deflection versus wall thickness

From the results obtained above, the limitation for use of wall with various thicknesses is from the deflection of the 6 m span beam. Nevertheless, walls are usually provided to transfer load, provide sound and heat insulation. Excessive thickness of wall will only create more problems during the erection and transportation process besides increasing the overall weight of the structure. As a result, the wall with 200 mm thickness will be used for this analysis, as recommended by the International Precast Concrete Federation.

After all the sectional properties for every member is checked, the analysis is carried out.

The appropriate load combination is chosen to view the analysis results. In the software, the commonly used load combination will be automatically generated. In this analysis, the load combination of [(1.2 × dead load) + (1.2 × imposed load) + (1.2 × wind load)] ($1.2G_k + 1.2Q_k + 1.2W_k$) for the ultimate limit state is used. Shear force diagram and bending moment diagram can be viewed for the overall structure, the chosen elevation view or for the particular member of interest. Figure 4.17 shows the shear force diagram for the frame perpendicular to wind direction. The maximum shear force for 6 m span beam at the support are at the ground floor beam, 247.15 kN and -247.15 kN. As for the column, the maximum shear force is 305.41 kN at level 2 and minimum shear force is -264.42 kN at level 2.

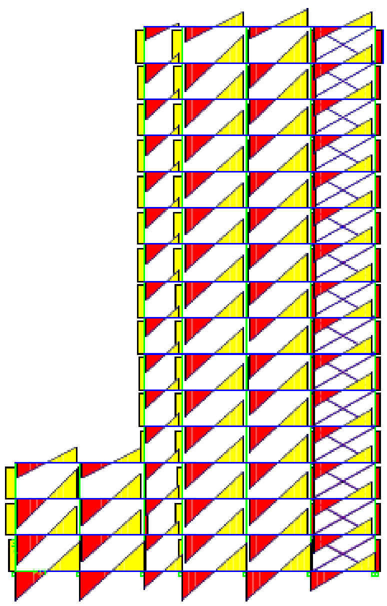


Figure 4.17: Shear force diagram for frame perpendicular to wind direction

Figure 4.18 shows the bending moment diagram for frame perpendicular to the wind direction. These moment induced in the member are mostly due to member self weight and imposed load from the floor element. The effect of wind is not significant on the frame placed perpendicular to wind direction. In addition, the moment in the short span beams are relatively small as compared to the longer span. Besides, the moment in the column is not significant. The maximum bending moment for 6 m span beam are 164.765 kNm and -231.989 kNm, located at ground level and level 2 respectively. Whilst, the maximum bending moment for column is 536.97 kNm and the minimum bending moment is -466.48 kNm, both located at level 2.

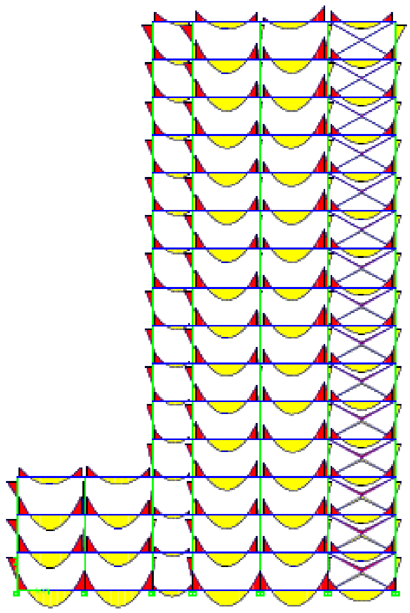


Figure 4.18: Bending moment diagram for frame perpendicular to wind direction

Figure 4.19 shows the shear force diagram for the frame parallel to wind direction. Due to the one-way slab action, the dead and imposed load from the slab is not transferred to the supporting beam. Instead, the beam only has the self weight. The maximum shear force in the 6 m span beams are 67.98 kN and -67.98 kN, both located at the base level. As for the column, the maximum shear force found within the frame are 305.41 kN and -264.42 kN both located at level 2.

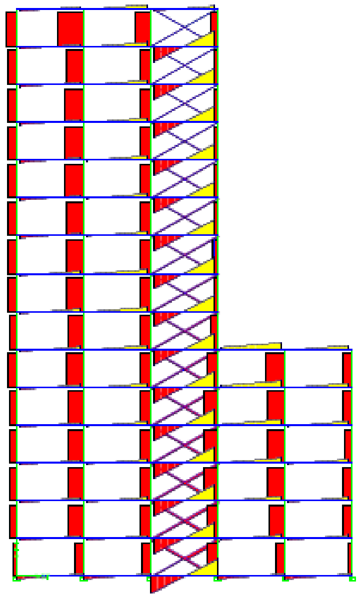


Figure 4.19: Shear force diagram for frame parallel to wind direction

Figure 4.20 shows the bending moment diagram for the frame parallel to wind direction. As observed, the maximum bending moment for 6 m span beam are 45.317 kNm and -56.646 kNm located at the base level. As for the columns, the maximum bending moment are 536.97 kNm and -466.48 kNm located at level 2.

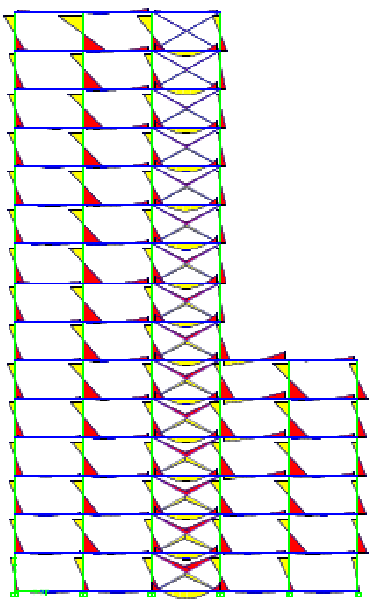


Figure 4.20: Bending moment diagram for frame parallel to wind direction

Meanwhile, as for the 3.6 m span beam, the maximum shear force are 138.57 kN and -165.27 kN at ground floor beam. For moment, the maximum bending moment are 55.427 kNm and -116.206 kNm at the ground floor beam.

For the 12 m span beam, the maximum shear force are 547.28 kN at storey 2 and -532.28 kN in beam at base level. As for the bending moment, the maximum value is 715.998 kNm in level 13 beam and -887.092 kNm in base level beam.

Figure 4.21 shows the axial force in the column for frame perpendicular to the wind direction and Figure 4.20 shows the axial force in columns for frame parallel to wind direction. The maximum axial force exists at the lowest level as the axial force is accumulated from the top level to the bottom level in the column. The maximum axial force in the column is 8633.48 kN. Axial force in column is independent to wind direction as it is the load that being transferred from the superstructure to the foundation of the structure through the column.

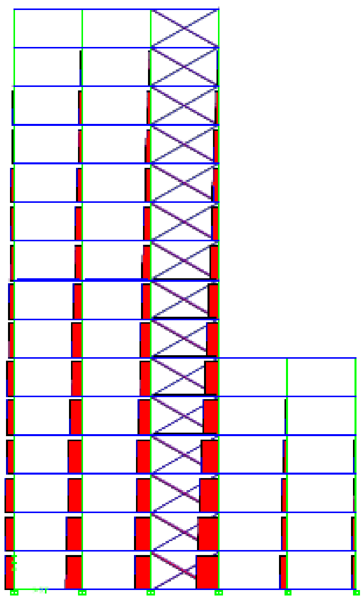


Figure 4.21: Axial force in column for frame perpendicular to wind direction

Table 4.4 shows the summary of the shear force and bending moment for both frames. From the table, it can be summarized that the beams, regardless to be perpendicular or parallel to wind direction, are mainly needed to support the floor slab; whilst for columns, they are required to resist the wind load. Hence, the stiffness of column is important in designing the wind resisting structure.

Table 4.4: Maximum shear force and bending moment in beam and column

Frame	**Element**	**Span**	**Shear Force (kN)**		**Bending Moment (kNm)**	
			Support 1	**Support 2**	**Mid-span**	**Support**
Perpendicular to wind direction	Beam (300 mm × 400 mm)	3.6 m	138.57	-165.27	55.427	-116.206
		6 m	247.15	-247.15	164.765	-231.989
	Beam (400 mm × 700 mm)	12 m	547.28	-532.28	715.998	-887.092
	Column (600 mm × 600 mm)	-	305.41	-264.42	536.97	-466.48
Parallel to wind direction	Beam (300 mm × 400 mm)	3.6 m	-	-	-	-
		6 m	67.98	-67.98	45.317	-56.646
	Beam (400 mm × 700 mm)	12 m	-	-	-	-
	Column (600 mm × 600 mm)	-	305.41	-264.42	536.97	-466.48

With the information from the above table, envelop for shear force and bending moment can be developed. The design for each category of element can be carried out.

Besides showing the overall diagram of a structure, clicking on the element of interest can view the details of the particular element. A dialogue box providing all the information regarding to the element will be shown (Figure 4.22).

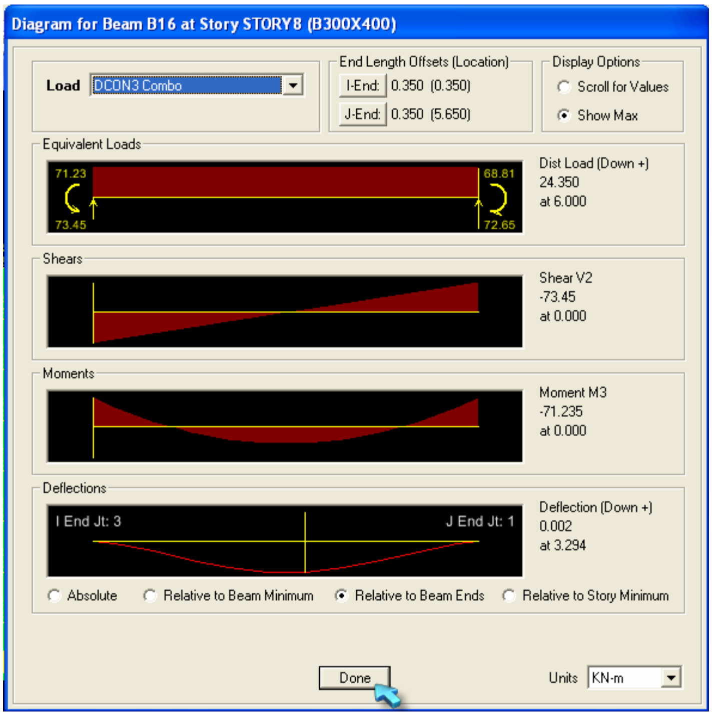


Figure 4.22: Detailed load, force and deflection diagram for a particular element

In precast concrete structure, the variations of structural elements are kept to minimum. Often, the elements used to resist the largest load and moment will be selected to be mass produced for the entire structure. Elements in a precast structure often categorized according to the length. In another word, elements with different length are usually designed separately and categorized as different group of elements.

Therefore, after the most critical element has been identified, the suitability of the section will be checked with the catalogue provided by the precast concrete company to ensure the particular section is sufficient to sustain the maximum forces. If the section is not sufficient, the new section will be reassign and the analysis will be carried out again.

As for this precast concrete structure, the elements used are standardized. Therefore, the element which has the capacity of sustaining the highest load within the structure will be used as the standard element for all the other similar elements. The choice of frame sections is tabulated in Table 4.4.

Table 4.5: Frame section

Member	**Section information**
3.6 m and 6 m span beam	300 mm × 400 mm
12 m span beam	400 mm × 700 mm
Column	600 mm × 600 mm
Slab	200 mm thick
Wall	200 mm thick

Ideally, after all the analysis and designing procedure has been carried out. The latest results will be updated to Autodesk Revit Structure 2. In Autodesk Revit Structure 2, all the detailing will be displayed in the interface, where this will eventually helps the surveyor and contractor in carrying out their works. The quantity surveyor can produce the Bill of Quantity easily with the aid of 3D model for the superstructure. Likewise, the contractor will be able to save more time in preparing the tender for bidding.

If any alteration where to be done to the structure, the same procedure will be carried out and the analysis results can be updated to the 3D model in Autodesk Revit Structure 2.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

With the advancement of IT, problems can be solved easily provided the available technology are been fully make use of. With the BIM application available, information transfer and sharing can be fully utilized. All the parties involved in the construction can share the information related to the project within an interface. Besides, any modification done can be updated automatically where all the parties will receive the updated information at the same time.

With the BIM software, sketching of initial idea can be visualized easily. Once the idea is confirm, the model can be used as the design basis. The BIM approach keeps all the documents and information in the digital format. In addition, sharing within the same interface reduces the amount of documents and information storage to be handled.

Upon confirm on the model, the model can be sent to bridged analysis software to perform analysis. ETABS is an example of software that is bridged with Autodesk Revit Structure 2 as third party structural analysis software.

Although there are many problems that occur during the transferring process, these software still help in reducing working time. Creating a 3D model in ETABS is not as easy as thought, especially if the structure is a complicated and irregular ones. Therefore, by using modelling software like Autodesk Revit Structure 2, the modelling work for analysis software can be made much easier. Besides modelling, applying loading on the respective element base on the space usage will be easier with the aid of Autodesk Revit Structure 2 software.

In Autodesk Revit Structure 2, the architectural plan is imported and placed as the background of the modelling process. Hence, the space usage can be known easily as compared to ETABS which does not provide any information on the space use for reference.

Analysis in ETABS provides a wide range of results variation, from the deflection to the detailing of reinforcement. Ideally, when the analysis and design work is done in ETABS, the model is again updated to the Autodesk Revit Structure 2. In case of any amendment to the model, the same process to transfer the model to ETABS for re-analyze will be repeated.

This process helps reduce a lot of working time for an engineer. Information loss between parties is no more an issue of not getting work done correctly at the right time.

5.2 Recommendation

There are many aspects in this study that should be improved. In future studies, a section library especially for precast elements can be developed. With the development of the precast concrete section library, the transfer of information can be done easier. In addition, add-ons can be developed in order to allow wall elements transfer between software.

The analysis should be carried out in a full 3D model, where the beams, columns, slabs and walls are assigned as it will be constructed. This will increase the accuracy of the analysis. In addition, the stress distribution on the slab and wall elements can be evaluated, and the highly stress area can be redesign to reduce the stress concentration at the particular point.

As for the modelling, future studies may consider the effect of openings on the structure. Openings highly influence the behaviour of structure under loading, especially lateral load and the dynamic effect of wind. Studies can also take into consideration of the effect of earthquake.

REFERENCES

Al-Chaar, Ghassan K., Lamb, Geogory E. (2002). *Design of Fiber-Reinforced Polymer Materials for Seismic Rehabilitation of Infilled Concrete Structures*. U.S. Army Corps of Engineer (USACE): Construction Engineering Research Laboratory.

Allen, Edward (1999). *Fundamental of Building Construction: Materials and Methods*.

Amos, F. J. C. (1974). High Hopes and Low Life. *Tall Buildings and People?* Oxford: Residential Conference St. Catherine's Collage.

Beall, Christine, Jaffe, Rochalle (2003). *Concrete & Masonry Databook 2003*. New York: McGraw Hill. 12.23.

Bell, G. R. (1989). Civil Engineering Investigation. In: Carper, K. L. *Forensic Engineering*. New York: Elsevier Science Publisher Co., Inc. 190-232.

Bor, Walter (1974) High Buildings: A Blessing or A Curse? *Tall Buildings and People?* Oxford: Residential Conference St. Catherine's Collage.

British Standard Institution (1996). *Loading for Buildings – Part 1: Code of Practice for Dead and Imposed Loads.* London, BS 6399.

British Standard Institution (1996). *Loading for Buildings – Part 2: Code of Practice for Wind Loads*. London, BS 6399.

British Standard Institution (1997). *Structural Use of Concrete – Part 1: Code of Practice for Design and Construction*. London, BS 8110.

British Standard Institution (2004). *Common Rules for Precast Concrete Products*. London, BS EN 13369.

Chew, Y. T., Cheong, H. F., Lee, S. L. (1984). Some Studies of Wind Effects on Tall Buildings. *Proceedings of International Conference on Tall Buildings*. October 22-26. Singapore. 457-464.

Dutt, A. J. (1984). Assessment of Wind Loading in the Design of Structural Systems for Tall Buildings. *Proceedings of International Conference on Tall Buildings*. October 22-26. Singapore. 465-

Elliott, K. S. (1996). *Multi-storey Precast Concrete Framed Structures*. London: Blackwell Science.

Farshad, M. (1994). *Stability of Structures*. Netherlands: Elsevier Science Amsterdam. 10-11.

Fédération Internationale de la Précontrainte (1982). *Design Philosophy for Precast Buildings of Two or More Storeys*. Slough.

Gauld, B. J. B. (1995). *Structures for Architects*. England: Longman. 6-7.

Haas, A. M. (1983). *Precast Concrete: Design & Applications*. England: Applied Science Publishers Ltd.

Hanaor, A. (1998). *Principles of Structures*. United Kingdom: Blackwell Science. 87-90.

Hibbeler, R. C. (2003). *Mechanics of Materials*. 5th ed. New Jersey: Prentice Education Inc.

Hibbeler, R. C. (2005). *Structural Analysis*. Singapore: Prentice Hall.

Holmes, M., Martin, L. H. (1983). *Analysis and Design of Structural Connections Reinforced Concrete and Steel*. England: Ellis Horwood Limited. 86-89.

Macdonald, A. J. (1975). *Wind Loading on Buildings*. London: Applied Science Publishers Ltd.

McGuire, John, PhD. (1995). *Notes on Semi-Rigid Connections*. FEMCI Online. (http://femci.gsfc.nasa.gov/semirigid/)

Murashev, V. I., et al. (1971). *Design of Reinforced Concrete Structures*. Moscow: Mir Publishers.

Nilson, A. H., Darwin, D. (1997). *Design of Concrete Structures*. 12th ed. New York: McGraw Hill.

Schultz, D. M., Burnett, Eric. F. P., Fintel, Mark (1977). *Design & Construction of Large-Panel Concrete Structures: A Design Approach to General Structural Integrity*. United States of America: Office of Policy Development and Research.

Smolira, M. (1975). *Analysis of Tall Building by the Force-Displacement Method*. England: McGraw Hill. 1.

APPENDIX A

Initial member size estimation for beam

For 6 m beams:

$$\frac{L}{d} < 26$$
$$\frac{6000}{d} < 26$$
$$d > \frac{6000}{26}$$
$$d > 230$$

Assume reinforcement modification factor = 1.3,

$d = 230 \times 1.3$

$d = 300$ mm (try $h = 400$ mm)

For 12 m beams:

$$\frac{L}{d} < 26$$
$$\frac{12000}{d} < 26$$
$$d > \frac{12000}{26}$$
$$d > 460$$

Assume reinforcement modification factor = 1.3

$d = 460 \times 1.3$

$d = 600$ mm (try $h = 700$ mm)

APPENDIX B

Connection Stiffness Calculation

Referring to Figure 2.2, for semi-rigid connection stiffness, $\alpha \approx 10$ (dimensionless).

First Trial for beams 400 mm × 700 mm (12 m beams)

E_{conc} = 29910129.1 kN/m^2

$$I = \frac{1}{12}bh^3 = \frac{1}{12} \times 0.4 \times 0.7^3 = 0.01143 \text{ m}^4$$

$$k = \alpha\frac{EI}{L} = 10 \times \frac{29910129.1 \times 0.01143}{12} = 284893.98 \text{ kNm}$$

First Trial for beams 300 mm × 400 mm (6 m beams)

E_{conc} = 29910129.1 kN/m^2

$$I = \frac{1}{12}bh^3 = \frac{1}{12} \times 0.3 \times 0.4^3 = 0.0016 \text{ m}^4$$

$$k = \alpha\frac{EI}{L} = 10 \times \frac{29910129.1 \times 0.0016}{6} = 79760.34 \text{ kNm}$$

First Trial for beams 300 mm × 400 mm (3.6 m beams)

E_{conc} = 29910129.1 kN/m^2

$$I = \frac{1}{12}bh^3 = \frac{1}{12} \times 0.3 \times 0.4^3 = 0.0016 \text{ m}^4$$

$$k = \alpha\frac{EI}{L} = 10 \times \frac{29910129.1 \times 0.0016}{3.6} = 132933.91 \text{ kNm}$$

APPENDIX C

<u>Load Calculation</u>

Load from wall acting on beam

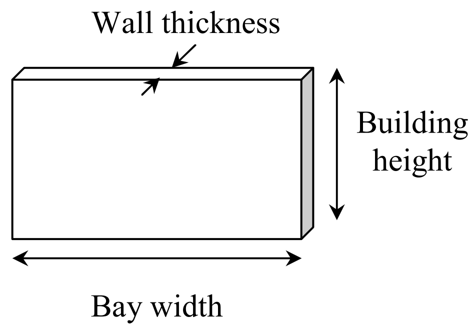
Storey height = 3.4 m

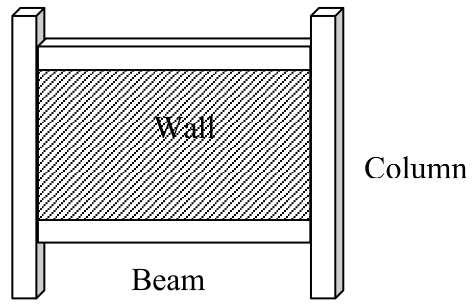
Wall thickness = 0.2 m

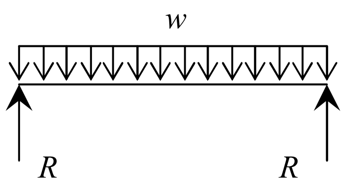
w = Density of Concrete ×
Building height ×
Wall thickness

$w = 23.6 \times 3.4 \times 0.2$

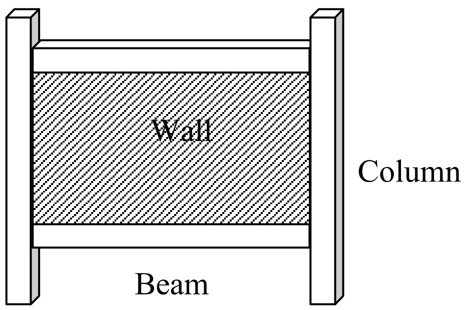
w = 16.05 kN/m length

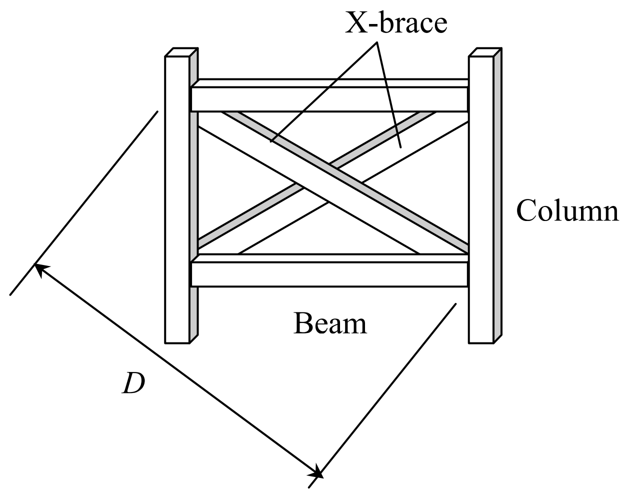






The wall is modelled as a simple X-brace within the frame structure.





Equivalent Strut Width Calculation

Storey height, $H = 3.4$ m

Clear storey height, $h_m = 3.4 - 0.4 = 3.0$ m (beam supported)

Young's Modulus of wall panel, $E_m = 29910129.1$ kN/m^2

Young's Modulus of frame, $E_c = 29910129.1$ kN/m^2

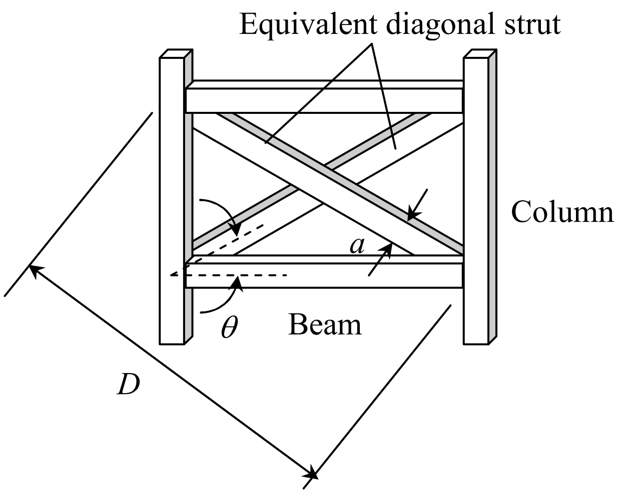
Wall thickness, $t = 0.2$ m

For 6 m span beam, $\theta = \tan^{-1}\dfrac{3.4}{6} = 29.54^\circ$

For 3.6 m span beam, $\theta = \tan^{-1}\dfrac{3.4}{3.6} = 43.36^\circ$

600 mm × 600 mm $I_{col} = \dfrac{1}{12}bh^3 = \dfrac{1}{12}\times 0.6\times 0.6^3$

$I_{col} = 0.0108$ m^4



Relative infill-to-frame stiffness, $\lambda_I H = H\left[\frac{E_m t \sin 2\theta}{4E_c I_{col} h_m}\right]^{\frac{1}{4}}$

For 6 m span beam

$$\lambda_I H = 3.4\times\left[\frac{29910129.1\times 0.2\times \sin\left(2\times 29.54^{\circ}\right)}{4\times 29910129.1\times 0.0108\times 3.0}\right]^{\frac{1}{4}}$$

$\lambda_I H = 3.6471$

For 3.6 m span beam

$$\lambda_I H = 3.4\times\left[\frac{29910129.1\times 0.2\times \sin\left(2\times 43.36^{\circ}\right)}{4\times 29910129.1\times 0.0108\times 3.0}\right]^{\frac{1}{4}}$$

$\lambda_I H = 3.7880$

Equivalent strut width, $a = 0.175D(\lambda_I H)^{-0.4}$

$a = 0.175 \times D\ (3.6471)^{-0.4}$

$a = 0.1043D$ m

At 6 m bay,

$a = 0.1043D$

$a = 0.1043\times\left(\sqrt{6^2+3.4^2}\right)$

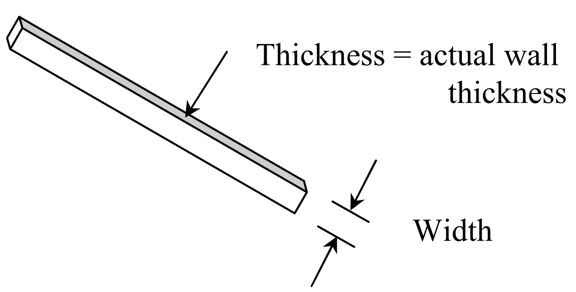
$a = 0.719$ m

At 3.6 m bay,

$a = 0.1043D$

$a = 0.1043\times\left(\sqrt{3.6^2+3.4^2}\right)$

$a = 0.516$ m



APPENDIX D

Wind Load Calculation

Basic data:

Basic wind speed V = 33.5 m/s (50-year return period)

Topography factor, S_1 = 1.0 (flat area)

Ground roughness, building size and height above ground, factor, S_2 = 0.98

Factor S_3 = 1.0

C_p factor

$$\frac{h}{w} = \frac{51}{30} = 1.7$$

$$\frac{l}{w} = \frac{33.6}{30} = 1.12$$

Windward C_p = +0.8

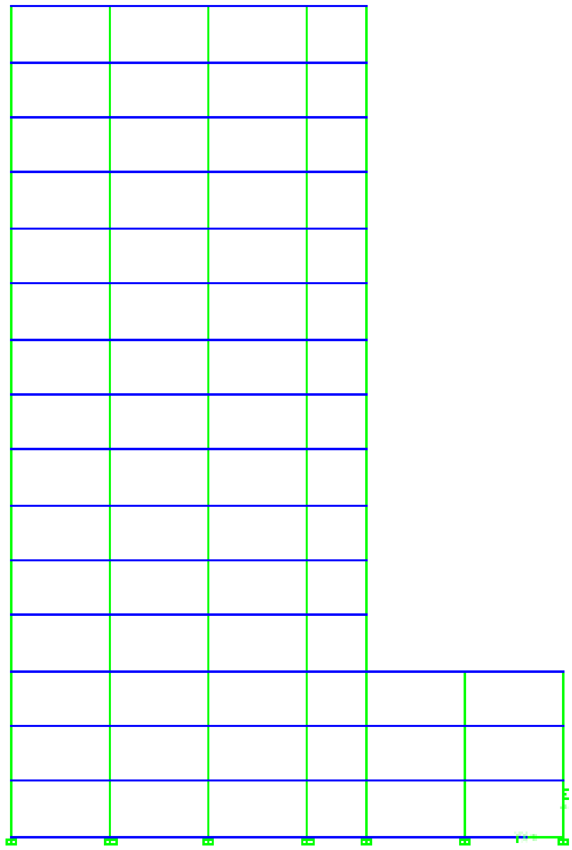
Leeward C_p = -0.25

$$
\begin{aligned}
V_s &= V \times S_1 \times S_2 \times S_3 \\
&= 33.5 \times 1.0 \times 0.98 \times 1.0 \\
&= 32.83 \text{ m/s}
\end{aligned}
$$

$$
\begin{aligned}
q &= k\, V_s^2 \\
&= 0.613 \times (32.83)2 \\
&= 0.8 \text{ kN/m}^2
\end{aligned}
$$

Case 1 (Wind from 0°, acting on Elevation A surface)

Elevation A (Windward - Pressure)



a *b* *c* *d* *e* *f* *g*

Level 1 – 2 & 3 (Node *a, b, c, d*)

Node *a, g*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 3 \times 3.4$

$= +6.5$ kN

Node *b, c, f*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 6 \times 3.4$

$= +13.1$ kN

Node *d, e*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 4.8 \times 3.4$

$= +10.4$ kN

Level 3 (node *e, f, g*)

Node *e*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times [(4.8 \times 1.7) + (1.8 \times 1.7)]$

$= +7.2$ kN

Node *f*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 6 \times 1.7$

$= +6.5$ kN

Node *g*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 3 \times 1.7$

$= +3.3$ kN

Level 3 – 14 (node *a, b, c, d, e*)

Node *a*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 3 \times 3.4$

$= +6.5$ kN

Node *b, c*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 6 \times 3.4$

$= +13.1$ kN

Node *d*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 4.8 \times 3.4$

$= +10.4$ kN

Node *e*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 1.8 \times 3.4$

$= +3.9$ kN

Level 15

Node *a*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 3 \times 1.7$

$= +3.3$ kN

Node *b, c*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 6 \times 1.7$

$= +6.5$ kN

Node *d*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 4.8 \times 1.7$

$= +5.2$ kN

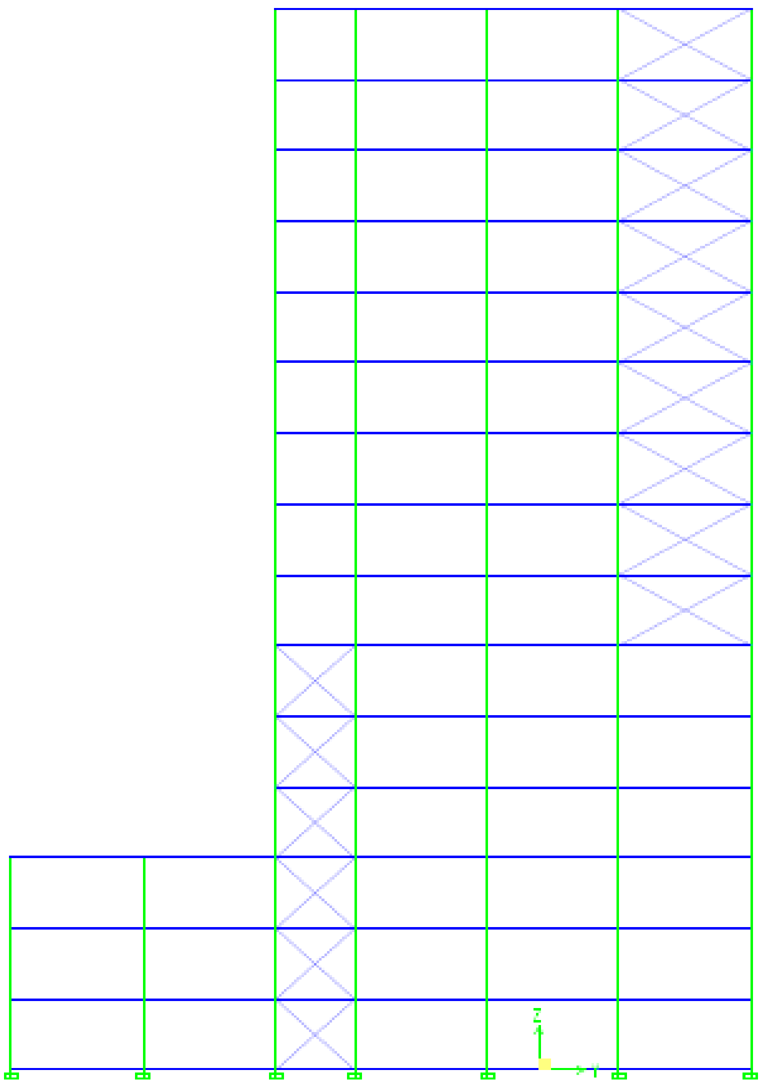
Node *e*

$F = C_p\, q\, A$

$= +0.8 \times 0.8 \times 1.8 \times 1.7$

$= +2.0$ kN

Elevation F (Leeward - Suction)



a *b* *c* *d* *e* *f* *g*

Level 1 – 2 & 3 (Node *d, e, f, g*)

Node *a, g*

$F = C_p\ q\ A$

$= -0.25 \times 0.8 \times 3 \times 3.4$

$= -2.0$ kN

Node *b, e, f*

$F = C_p\ q\ A$

$= -0.25 \times 0.8 \times 6 \times 3.4$

$= -4.1$ kN

Node *c, d*

$F = C_p\ q\ A$

$= -0.25 \times 0.8 \times 4.8 \times 3.4$

$= -3.3$ kN

Level 3 (node *a, b, c*)

Node *a*

$F = C_p\, q\, A$

$= -0.8 \times 0.8 \times 3 \times 1.7$

$= -3.3$ kN

Node *b*

$F = C_p\, q\, A$

$= -0.25 \times 0.8 \times 6 \times 1.7$

$= -2.0$ kN

Node *c*

$F = C_p\, q\, A$

$= -0.25 \times 0.8 \times [(4.8 \times 1.7) +$

$(1.8 \times 1.7)]$

$= -2.2$ kN

Level 3 – 14 (node *c, d, e, f, g*)

Node *c*

$F = C_p\, q\, A$

$= -0.25 \times 0.8 \times 1.8 \times 3.4$

$= -1.2$ kN

Node *d*

$F = C_p\, q\, A$

$= -0.25 \times 0.8 \times 4.8 \times 3.4$

$= -3.3$ kN

Node *e, f*

$F = C_p\, q\, A$

$= -0.25 \times 0.8 \times 6 \times 3.4$

$= -4.1$ kN

Node *g*

$F = C_p\, q\, A$

$= -0.25 \times 0.8 \times 3 \times 3.4$

$= -2.0$ kN

Level 15

Node *c*

$F = C_p\, q\, A$

$= -0.25 \times 0.8 \times 1.8 \times 1.7$

$= -0.6$ kN

Node *d*

$F = C_p\, q\, A$

$= -0.25 \times 0.8 \times 4.8 \times 1.7$

$= -1.6$ kN

Node *e, f*

$F = C_p\, q\, A$

$= -0.25 \times 0.8 \times 6 \times 1.7$

$= -2.0$ kN

Node *g*

$F = C_p\, q\, A$

$= -0.25 \times 0.8 \times 3 \times 1.7$

$= -1.0$ kN

APPENDIX E

Maximum allowable deflection

According BS 8110-2:1985 Clause 3.2.1.2, to avoid damage to non-structural elements, the deflection should not exceed $\frac{\text{span}}{500}$ or 20 mm whichever is lesser.

For 3.6 m span,

Maximum deflection = $\frac{\text{span}}{500} = \frac{3600}{500} = 7.2 \text{ mm}$.

For 6 m span,

Maximum deflection = $\frac{\text{span}}{500} = \frac{6000}{500} = 12 \text{ mm}$.

For 12 m span,

Maximum deflection = $\frac{\text{span}}{500} = \frac{12000}{500} = 24 \text{ mm}$ >20 mm;

Use maximum deflection=20 mm