INVESTIGATING THE STRESSES AND DETAILING ON LOAD BEARING TRANSFER WALL WITH OPENING BASED ON FINITE ELEMENT METHOD

AARON OON CHERN EN

UNIVERSITI TUNKU ABDUL RAHMAN

INVESTIGATING THE STRESSES AND DETAILING ON LOAD BEARING TRANSFER WALL WITH OPENING BASED ON FINITE ELEMENT METHOD

AARON OON CHERN EN

A project report submitted in partial fulfilment of the requirements for the award of Bachelor of Civil Engineering with Honours

Lee Kong Chian Faculty of Engineering and Science
Universiti Tunku Abdul Rahman

May 2023

DECLARATION

I hereby declare that this project report is based on my original work except for citations and quotations which have been duly acknowledged. I also declare that it has not been previously and concurrently submitted for any other degree or award at UTAR or other institutions.

Signature :

Name : Aaron Oon Chern En

ID No. : 1802661

Date : 24th April 2023

APPROVAL FOR SUBMISSION

I certify that this project report entitled **“INVESTIGATING THE STRESSES AND DETAILING ON LOAD BEARING TRANSFER WALL WITH OPENING BASED ON FINITE ELEMENT METHOD”** was prepared by **AARON OON CHERN EN** has met the required standard for submission in partial fulfilment of the requirements for the award of Bachelor of Engineering (Honours) Civil Engineering at Universiti Tunku Abdul Rahman.

Approved by,

Signature : ____________________

Supervisor : Ir. Ts. Dr Lau See Hung

Date : 28/04/2023

Signature : ____________________

Co-Supervisor : ____________________

Date : ____________________

ACKNOWLEDGEMENTS

I would like to thank everyone who had contributed to this final year research. I would like to express my gratitude to my supervisor, Dr. Lau See Hung, for her invaluable advice, guidance, and enormous patience throughout the research development.

In addition, I would also like to express my gratitude to my loving parents and friends who had helped and encouraged me throughout my undertaking of this research project.

ABSTRACT

Reinforced concrete (RC) transfer walls are widely used in high-rise structures nowadays to transfer a massive amount of load to the supporting structure below. The presence of openings on large structures such as transfer walls would often be designed by engineers to meet the functional, architectural or mechanical requirements of the buildings. However, these openings are the source of weakness that affects the load transfer mechanism and the bearing capacity of the transfer wall. This study aimed to investigate the stress of the transfer wall with different opening configurations by finite element analysis and design the most efficient reinforcement detailing by adopting Eurocode 2. This research study utilised SCIA Engineer and ANSYS software to simulate the numerical analysis of the model where 2D and 3D analysis was carried out respectively by both software to evaluate the stress distribution pattern of the transfer wall. Findings revealed that the stress path was smoother for transfer wall with staggered openings compared to vertical aligned openings due to less stress flow disturbance. High stress was concentrated at the opening corners and supporting region of the transfer wall. Findings show that the design of steel reinforcement was provided according to the critical locations where were highly stressed. For instance, high density of steel reinforcement was provided especially at the supporting region, whereas diagonal reinforcement was provided at the opening's corner to prevent cracking. The findings also show that the load path analysis of transfer walls can be well-presented in stress contour form by using numerical modelling through 2D and 3D finite element methods. However, the difference in stress value was huge due to the difference in analysis approach in terms of assumption and simplification, mesh density, the element type as well as the boundary conditions. The findings proved that 3D analysis provides more accurate results when dealing with geometrically complex structures such as the transfer wall with openings. Lastly, it was recommended that further study to be done for the finite element analysis method in terms of boundary conditions and mesh settings to improve the reliability and accuracy of the analysis results.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOL / ABBREVIATION

α	principal angle, °
σ	normal stress at x-axis, Mpa
σ_{max}	maximum Stress, Mpa
b	width, mm
d	thickness, mm
ε	strain, mm/mm
f_{yk}	yield strength of reinforcement steel bar, Mpa
fc'	concrete compressive strength, MPa
fu	ultimate stress, MPa
$A_{s,1}$	required reinforcement in longitudinal direction, mm^2
$A_{s,2}$	required reinforcement in transverse direction, mm^2
C_{dev}	concrete cover allowable deviation
E	Young's Modulus
v	Poisson ratio

DOF	degree of freedom
FEA	Finite Element Analysis
FEM	finite element method
G35/45	concrete grade (cylinder strength: 35 MPa, cube strength: 45 MPa)
G50/60	concrete grade (cylinder strength: 50 MPa, cube strength: 60 MPa)
NLFEM	nonlinear finite element method
RC	reinforced concrete
SLS	serviceability limit state
STM	strut-and-tie model
ULS	Ultimate limit state
1D	one dimensional
2D	two dimensional
3D	three dimensional

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 General Introduction

Load bearing transfer wall is defined as a vertical active structural element of a building that could resist the weight of the elements above it and transfer the weight to a foundation or a supporting member. It was commonly used in high-rise buildings and often located at the transition point between different sections of the building such as the podium. The transfer wall has been widely developed for decades as an effective structural solution for the purpose of efficient load distribution and space-saving.

Nowadays, the presence of openings on load bearing transfer walls can be seen frequently to meet the functional, architecture or mechanical requirement of the building without altering the design (Jaseela and Pillai, 2017). However, these openings are the source of weakness that affects the load carrying capacity of the load bearing member (Popescu, et al., 2015; Husain et al., 2019).

In view of this, the stress path of the load bearing transfer wall should be investigated to identify the stress flow and highly stressed regions. There are several stress evaluation methods which are widely used. For example, analysis by strut-and-tie (STM), matrix structural analysis, moment distribution or slope deflection methods could be adopted to study the stress behaviour of a structure. However, a complex stress state was developed due to the presence of openings which resulted in an unknown pattern in terms of load path and stress distribution for its force transition mechanism (Behfarnia and Shirneshan, 2017). Therefore, these conventional structural analysis approaches were not suitable to be adopted due to high complexity in stress transfer patterns. For this reason, finite element method is used in this study to simulate the entire transfer wall model by dividing the structure into small element where the behaviour of each elements could be observed (Dere, 2017).

In this study, two transfer walls with different openings configuration arranged in vertical align pattern and staggered pattern were simulated by using finite element modelling. SCIA Engineer software and ANSYS software were used for modelling to identify the stress pattern and the most efficient detailing method for the load bearing transfer wall with openings. Eurocode design standard (EC2) was adopted for the design of steel reinforcement in the structural wall. Lastly, finite element analysis of the reinforced transfer wall was carried out to study the stress behavior of the reinforcement and the concrete structure.

1.2 Importance of the Study

The study on the stress behaviour of load bearing transfer walls has a significant impact on the design of high-rise buildings. The performance and functionality of the transfer wall is characterised by the stress-strain relationship of concrete. This is important to study the failure pattern in terms of tensile cracking and compressive crushing when the load bearing capacity of the concrete has exceeded. Thus, structural analysis using the finite element method was used in this situation to simulate the nonlinear behaviour of the reinforced transfer wall with openings.

The transfer wall with different openings arrangement (vertical aligned and staggered) were performed by using finite element analysis to study the stress pattern of different opening arrangements in the transfer wall. The stress distribution and load path were altered when the arrangement of the opening changes. The failure pattern was examined to identify the highly stressed regions of the structural member based on the stress distribution and the load path. Therefore, a more efficient transfer wall with openings with better structural integrity can be modelled.

1.3 Problem Statement

The optimization of building space has required the engineers to design complex structural members such as transfer wall with openings and it has posed challenges to structural engineers due to the behaviour of stress distribution being different compared to transfer wall without openings. There were limited studies conducted on the transfer wall with openings. This was due to the presence of openings that have complicated the failure mechanism, and the failure behaviour of the structural member can be said to be complex (Sakurai, et al., 2017). With the help of finite element modelling, more experimental data is required to better understand the stress behaviour of structural load-bearing transfer walls.

There are few analysis approaches for complex load bearing structural members such as deep beam and transfer wall which includes the conventional Strut-and-Tie (STM) method and finite element method. However, the Strut-and-Tie (STM) method is not suitable to analyse the stress behaviour of transfer wall with openings as the nonlinearity of the structure in terms of redistribution of forces are not taken into account. The discontinuity in terms of stress path could not be captured accurately due to the presence of openings. Furthermore, the assumption of ductile material behaviour by STM is not suitable for analysis as the structure with openings could behave in a brittle manner (Metwally, 2017).

In current practice, there are several design standards such as ACI318 based on empirical models that can be adopted to design the RC wall, but there is no specific design standard for structural transfer wall with openings. Besides, these design codes were found to be over conservative, and it was proven that the design is not economical (Popescu, et al., 2015). As a result of the stress distribution and load path determined by the findings of the finite element analysis, effective detailing is needed to address this issue and improve the structural integrity of the transfer wall. Although there was no direct method provided in Eurocode 2 (EC2), the design concept of ultimate limit state (ULS) and serviceability limit state (SLS) was adopted by considering safety factors under extreme loads and normal service loads.

1.4 Aim and Objectives

The aim of the study is to investigate the stress and detailing of load bearing transfer wall with openings by using finite element analysis approach. The objectives of this study are listed as below:

i. to model the load bearing transfer wall with openings using finite element method;
ii. to evaluate the stress of load bearing transfer wall with openings.
iii. to propose detailing for load bearing transfer wall based on the finite element analysis.

1.5 Scope and Limitation of the Study

The stress-strain relationship of concrete under compression and tension was studied to evaluate the stress state of the structure when loaded. The nonlinear stress-strain behaviour of reinforced concrete in elastic and plastic stages were studied to identify the stress transfer interaction between the concrete and steel reinforcement.

Next, a two-dimensional (2D) load bearing transfer wall with openings is modelled using SCIA Engineer. 3D finite elements for both concrete model and reinforced concrete mode are modelled by ANSYS where the results were compared and studied. There are two different arrangements of openings (vertical aligned and staggered) on the transfer wall to be modelled. The transfer wall models are subjected to axial load from top and restrained on bottom which is supported by beam and column members. Efficient and economical detailing for the transfer wall with openings was proposed based on the result from finite element analysis by using Eurocode Standard (EC2).

The behaviour of reinforced transfer wall was analysed by using nonlinear FEM to study the stress transfer from the concrete to the steel reinforcement by their interaction. The improvement in terms of stress capacity of the concrete was evaluated at the same time.

There are several limitations to be considered in the study. The transfer wall with openings was only loaded axially. There was no transverse load and lateral load in terms of wind load to be exerted on the structure as these loadings were assumed to be taken by the shear wall of the building. The analysis of the support members below the transfer wall were not focussed in this study. The proposed detailing of the transfer wall with openings should be justified by real-life experimental data. Therefore, a scale-down laboratory model is recommended to be done to evaluate the actual stress behaviour of the model and the efficiency of the detailing.

1.6 Contribution of the Study

This study shows the stress distribution pattern and the critical location of transfer wall with vertical aligned and staggered openings. The proposed detailing method can become a reference for structural engineers when dealing with the design reinforced concrete transfer wall. With the outcome of this study, structural engineers could have a better understanding on the stress pattern of transfer wall with openings in order to design an economical structure.

Besides, finite element analysis was performed by SCIA Engineer and ANSYS software to study the complex stress state of the transfer wall. This highly reliable approach has greatly enhanced the ability of engineers to analyse complex structures which led to the development of more effective and efficient engineering structural solutions.

1.7 Outline of the Report

There are five chapters that make up this study. The first chapter began with a general introduction of transfer walls, followed by the importance of this subject and its difficulties. The objectives, scope, and limitations are presented in this chapter to outline the study's main focuses. Finally, the contribution of the study was discussed in the sub-chapter that followed.

Chapter 2 contains literature reviews related to the study topic. This chapter mainly focuses on the theory review on the behavior of the reinforced concrete, past experimental study related to shear wall with openings, and the introduction of finite element modelling and analytical model in studying the stress distribution pattern.

Chapter 3 outlines the methodology work plan for the numerical study. The overall study workflow is presented in this chapter. There are two main parts for the numerical study which include the numerical modelling, reinforcement design.

Chapter 4 discusses the numerical results obtained from the simulation process for both software. The stress pattern is studied and compared for the purpose of efficient reinforcement design. The result of the nonlinear finite element of the reinforced concrete model is studied as well.

Chapter 5 wraps up the entire analysis, conclusions, and discussion from Chapter 4. Based on the goals and objectives stated in Chapter 1, the conclusion was drawn. For future research purposes, recommendations and suggestions have also been made.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

The development of high-rise buildings has driven the design of load bearing structural members such as deep beams and transfer wall. When high load is applied on these structures, their durability and loading capability poses concern to the engineers. By comprehending the stress-strain relationship of the concrete material, the behaviour of these structures is examined. The concrete's stress-strain characteristic demonstrates the interaction between the amount of stress applied and the extent of deformation. The mixture of raw materials in concrete act as non-homogenous material in terms of non-uniform microstructure fracture mechanism. Due to concrete’s non-homogenous material behaviour, it exhibits non-linearity behaviour, that means the alteration of the concrete’s stiffness changes when excessive loading is applied. Besides, the non-linearity behaviour of reinforced concrete affects the stress concentration as well as the load path of a structural member. The stress applied has caused the concrete to undergo compression and tension where the load path and behaviour of the concrete can be observed by using finite element analysis.

Numerical approach is used in finite element modelling to solve complex engineering problems where the result is approximated by adopting suitable modelling approaches. In general, the modelling process of finite element is quite straightforward where it involves meshing or discretization, and then the discretized elements’ result is combined and assembled to show an overall approximate result. Finite element analysis uses numerical approaches with differential equations, the boundary conditions of the model and elements should be well defined where these factors govern the accuracy of the result. When the finite element model is correctly defined with suitable parameters, high accuracy results are expected to be obtained.

2.2 Concrete stress-strain behaviour

Concrete is made up of coarse aggregate, fine aggregate, cement and water with specified mix proportion. It is considered to be non-homogenous in micro-level. It exhibits a large number of microcracks primarily found at the interfacial zone or the transition zone of the coarse aggregate and mortar. These cracks failure has great effect on the mechanical behaviour of the concrete and eventually contributes to its nonlinear behaviour at low stress level. The response of the structure under loading is highly dependent on the stress-strain relation of the material and the magnitude of stress. The stress-strain relation of concrete in compression is of primary interest since concrete is strong in compression (Filippou, 2015).

The behaviour of the concrete is examined by load-deformation relation, cracking and crushing occur by failing under excessive tensile and compressive stresses. The behaviour of the concrete depends on several parameters including its composition of concrete material, the type of loading which is either compression or tension and the loading fashion which can be uniaxial, bi-axial or multiaxial (Dere, 2017). The concrete's stress-strain relationship is utilised to evaluate these parameters. The behaviour of concrete influences the design to detect the weak location, thus the stress-strain relationship of concrete under tensile and compressive stress is crucial. Therefore, concrete should be viewed in this situation as a quasi-brittle material, necessitating nonlinear fracture-mechanics study (Earij et al., 2017). Figure 2.1 shows the stress-strain diagram of concrete under compression.

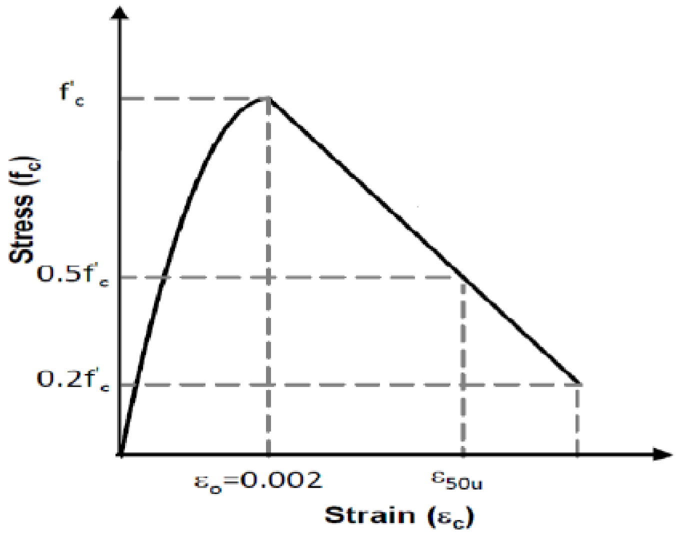


Figure 2.1: Stress-Strain Relationship of Concrete under Compression (Behfarnia and Shirneshan, 2017).

During the loading process at the initial stage, the stress-strain curve first behaves elastically up to the end of the linear segment which is around 40% to 50% of the ultimate compressive strength, that means the concrete deforms elastically where its stiffness remains unchanged. When the concrete reaches the value of initial yield, the curve starts to deviate from linear-behaviour to nonlinear behaviour up until the peak stress. The stress applied has caused inelastic straining which is non-recoverable (Chaudhari and Chakrabarti, 2012). This stage is where the concrete behaves plastically which is characterised by some stress hardening (Behfarnia and Shirneshan, 2017). The presence of these cracks in the interfacial transition zone is causing a weak link between the aggregates and the cement paste (Hanif, et al., 2018). The yield point has indicated that voids and cracks in the concrete body are increasing which result in the degradation of its stiffness (Wang et al., 2020). At this plastic stage, the strain increases gradually as the loading rate remains unchanged, the deformation rate is increased. Once the peak stress is achieved, the stress-strain will descend and eventually fail in crushing, it is due to the degradation of concrete stiffness which is known as softening behaviour.

Concrete is good in compression but very weak in tension, therefore the behavioural study of concrete tensile strength must be taken into consideration. The stress-strain relationship for concrete under tension is shown in Figure 2.2.

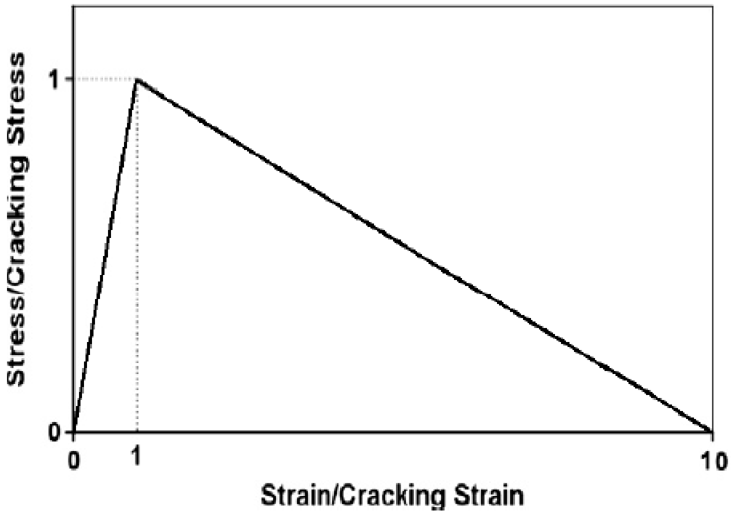


Figure 2.2: Stress-Strain Relationship for Concrete under Tension (Behfarnia and Shirneshan, 2017).

The stress-strain relationship for concrete under tension is said to be elastic all the way up until it reaches its maximum tensile strength. However, the concrete's highest tensile strength is just around 10% of its ultimate compressive strength. Once the peak tensile strength is reached, tensile cracks are observed on the concrete surface of the tensile region. The occurrence of these cracks is formed rapidly which is caused by the degradation of elastic stiffness (Chaudhari and Chakrabarti, 2012).

As tensile stress continues to be applied, the concrete is said to be undergoing tension stiffening. The impact of concrete operating in tension between cracks on the stress of steel reinforcement is known as this stage. The reinforcement totally resists the internal tensile force at the crack site, but between fissures, some of the tensile force is transferred by bond to the nearby concrete, reducing the stress-strain on the reinforcement (Allam, et al., 2013). Tension stiffening refers to the phenomena where concrete may hold tension even after fracture where tensile strength generally declines with increased tensile strain. The elastic stress-strain relationship implies that at a total strain of around ten times the strain during tensile cracking, the strain softening stage (tension stiffening) reduces linearly to zero (Shamass, et al., 2015).

2.3 Load bearing transfer wall with openings

Transfer wall is a structural wall that resists axial load from top of the building and transfers the load to bottom part of the building structure such as deep beam, transfer beam and transfer column. It is considerably different from shear walls, which are designed to resist horizontal loads that are generated in the wall's plane as a consequence of wind, seismic, and other events (Gandhi, 2016). Axially loaded transfer wall can be defined as a one-way action wall panel and two-way action wall panel. While a two-way transfer wall is supported on all four edges, a one-way transfer wall is typically assumed to be hinged at the top and bottom (Jaseela and Pillai, 2017).

The design of openings in a transfer wall is common nowadays as the structures are required to meet functional modification in order to comply with the current living standard (Popescu, et al., 2015). These openings have eventually affected the ultimate strength of the structural wall, especially the large openings tend to cause disturbance in the stress path when there is considerable amount of concrete and steel reinforcement being removed to make an opening (Mohammed, et al., 2013). Besides, the openings introduced into the structural wall reduces the overall performance in terms of flexural, axial strength, stiffness ,decreasing its ductility level and causing stress which cannot be classified in known patterns (Behfarnia and Shirneshan, 2017). It is found that large openings usually alter the ultimate load capacity of the entire structural system where it requires improvement in terms of strengthening the wall structural capability by steel reinforcement. Until today, the study of strengthening structural walls with openings is still in its early stages, thus further research and experimental testing are needed. Although there are design codes such as Eurocode is provided, however, the result is said to be over conservative when dealing with recent design models (Popescu, et al., 2015).

2.3.1 Load path on transfer wall with openings

The openings in the structural member are the source of weak points and cause deterioration of inside structure’s stiffness as well as load bearing capacity. The opening shape has a significant impact on the axial stress capacity at failure stage (Morsy and Ibrahim, 2019). The vertical rectangular direction opening is said to have the highest carrying capacity compared to square and circular shape, this is due to the loaded cross section area being larger where it can resist higher axial load. By comparing the structural integrity of circular opening and square opening, it is found that circular opening has higher load capacity than square opening. The stress transfer along the top edge portion of the circular opening has smoother change in load path and stress transfer (Senthil, et al., 2018; Chaudhary and Parekar, 2019). When it comes to stress concentration, the structural wall panel's openings affect the load flow by creating high concentrations close to its location.

Researches have shown that several factors of the openings will affect the structural integrity of the transfer wall. These parameters include the aspect ratio of the opening size, arrangement of the openings, locations of the openings, as well as thickness ratio of the structural wall. These are the main factors that will disrupt the load path and the stress distribution of a structural wall. Even a small alteration of these parameters bring significant impact to the load carrying capacity (Popescu, et al., 2015). Stress flow disturbance is experienced when the size of the opening is greater. This has caused initiation of concrete's nonlinear behaviour at low stress level, resulting in yielding and plastic straining (Musmar, 2013). According to Kim, Huang and Jin (2019), it is recommended that the ratio of opening area to be less than 20% to limit the degradation in stiffness and strength of the structure.

The stress distribution of the transfer wall with vertical aligned openings is shown in Figure 2.3 and the transfer wall with staggered openings is shown in Figure 2.4. This is apparent that staggered openings have less concentrated stress in surrounding openings. That means the transfer wall with staggered opening is having higher stiffness and more durable compared to transfer wall with vertical aligned openings. The reduction of stiffness has been proven to have lower load carrying capacity (Sharaf, et al., 2018). In terms of similarity, the weakest part of the transfer wall is located at the interval spacing region between the openings.

The location of the opening greatly influences the concrete's strength, especially when it is nearer to the wall's edge. It is recommended to keep the location of the openings as closer to the neutral axis of the structural wall (Borbory, 2020).

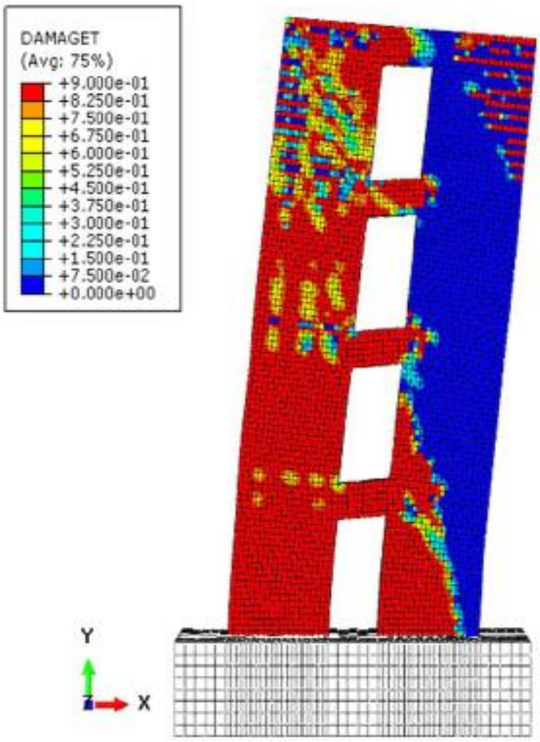


Figure 2.3: Stress on Transfer Wall with Vertical Aligned Openings (Husain, et al., 2019).

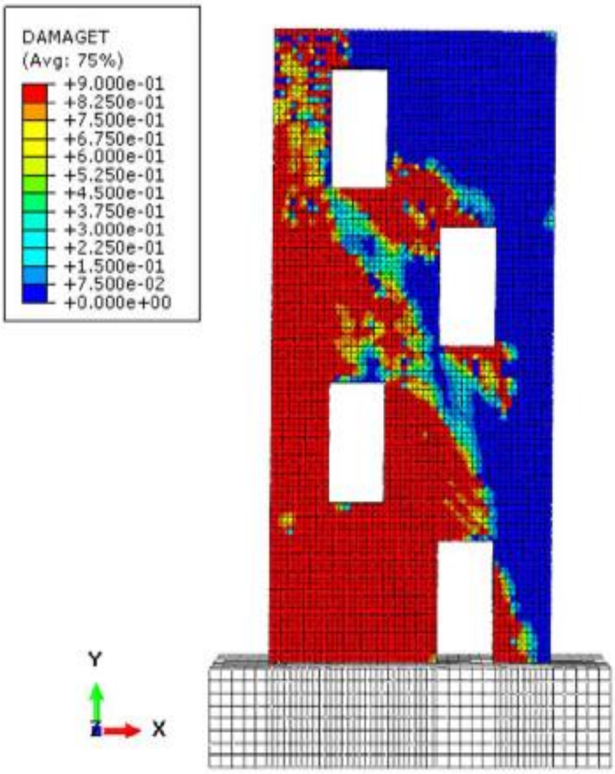


Figure 2.4: Stress on Transfer Wall with Staggered Openings (Husain, et al., 2019).

In response to the mean direction of the compressive stress, the load path mechanism of the load bearing transfer wall is developed. This disrupted load path is initially designed from the concept of strut and tie method (STM) (Wu and Li, 2003). As indicated in Figure 2.5, the structural wall can be split into four distinct zones: panel zones, nodal zones, column zones, and beam zones, depending on the nature of the stress distribution. By solving the strut and tie model, a truss analysis approach can be used to solve manually.

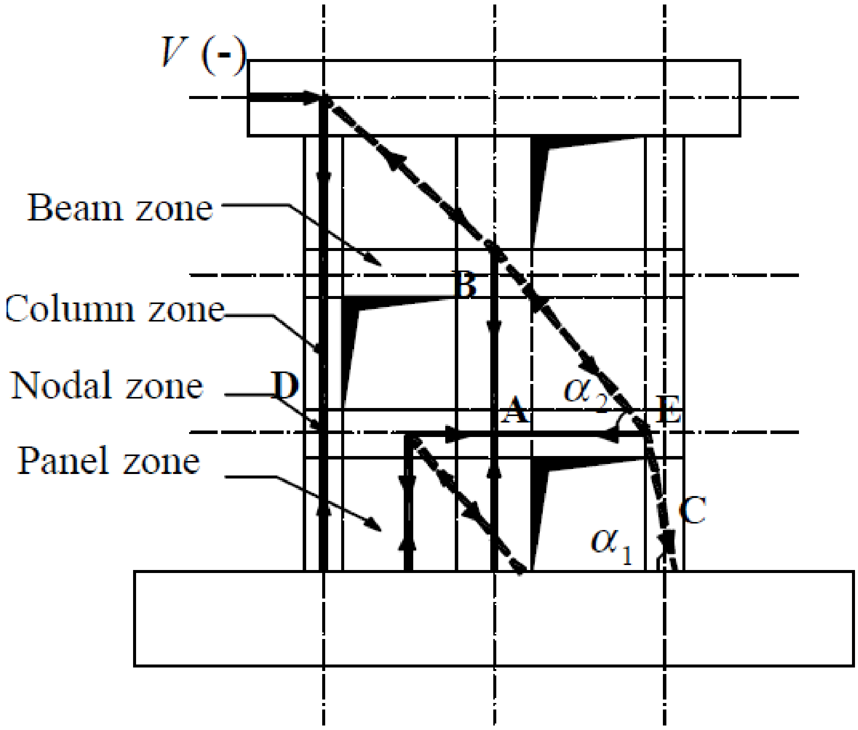


Figure 2.5: Strut and Tie model stress distribution (Wu and Li, 2003).

However, the application of finite element approach is proven to solve complex nonlinear structural problems by approximating the result compared to the strut and tie model. Therefore, a strain hardening plasticity model is employed for concrete in compression, and a nonlinear finite element approach is used to analyse the brittle fracture behaviour of concrete in tension (Morsy and Ibrahim, 2019).

2.4 Finite Element Modelling

Finite Element Modelling is an established numerical technique used to solve complex engineering problems which involves complex differential equations arising in engineering and mathematical modelling. It is ranged to simulate the deformation and stress analysis of rigid bodies to field analysis of heat and fluid flow etc. It is now well accepted as the most effective technique for numerical solution; therefore, stress analysis of structural transfer wall can be simulated by different modelling approaches.

Linear finite element analysis is widely used as a design tool. Similar goes to the application of nonlinear finite element analysis. The two major factors that to be considered while conducting nonlinear finite element analysis are the computational effort and the complexity of the nonlinear analysis problems. The development of improved and more efficient nonlinear algorithms has commenced to higher accuracy and reliable results (Filippou, 2015).

2.4.1 Finite Element Analysis (FEA)

Finite element analysis is used to examine how stress in a structure behaves and how load transfer systems work (Romans, 2010). This mathematical method for engineering analysis is typically characterised by a relatively complex set of algebraic, differential, or integral equations which are posed on geometrically complex domains. Numerical simulation, such as the finite element method (FEM), is the method of assessing a mathematical model of a process utilising numerical methods and computers (Reddy, 2019).

This type of numerical method is capable of finding approximate solutions even if the analytical model involves complicated domains such as geometry and material constitution, loads, boundary conditions, and interaction between various aspects of the system response. Additionally, the numerical method can be utilised to study and analyse the impact of different parameters. Because of the power of numerical methods in electronic computation, the mathematical model will include the most relevant features (Reddy, 2019).

The overall schematics of a model-based simulation of finite element method is shown in Figure 2.6 below.

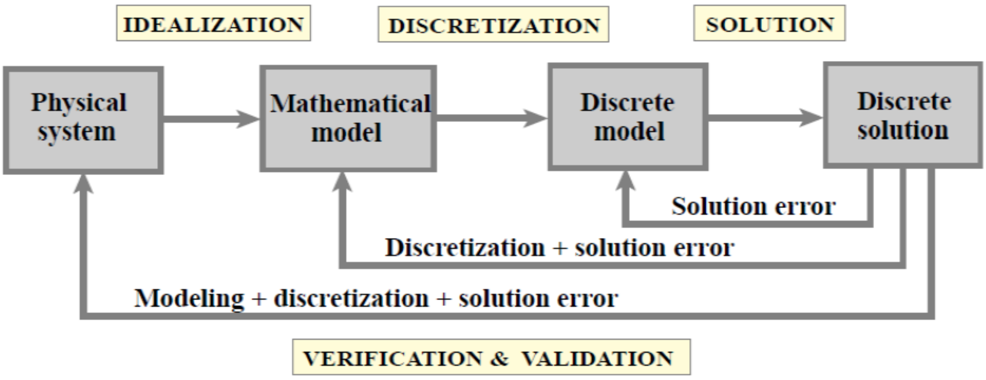


Figure 2.6: Flowchart of FEM (Jagota, et al., 2013).

Finite element method is using the concept discretization and solution technique. It reduces the problem by dividing a body of matter into finite elements. The values of the field variables at particular locations known as nodal points serve as the basis for the definition of the approximation functions. These nodes typically sit on the edges of elements when connections are made between them. The nodal values of the field variable become the unknowns in the finite element model of a problem. The interpolation functions define the field variable throughout the element assembly once these unknowns have been identified. In addition to the size and number of elements, the interpolation function chosen also affects the type of solution and degree of approximation. The finite element method's capability to generate solutions for each element separately before integrating them all together to represent the whole problem is one of its most significant features. For a stress engineering analysis problem, it is necessary to first identify the force-displacement or material stiffness properties of each individual element before assembling it to evaluate the stiffness of the entire system. A big problem will, in essence, be broken down into a number of much simpler problems (Jagota, et al., 2013).

2.4.2 Linear Finite Element Method

By directly deriving the necessary amount of reinforcement from the membrane force, followed by a linear elastic approach, reinforced concrete walls can be designed using the linear elastic finite element method. However, since it does not satisfy the necessity associated with crack control in a serviceability limit condition, the assumed linear elastic isotropic material of concrete for reinforcement design does not accurately reflect the behaviour of concrete (Romans, 2010).

Concrete is linear-elastic in response to compression up to around 40% of the ultimate peak stress. Beyond the linear elastic point, the concrete exhibits nonlinear behaviour up to the peak stress with a subsequent reduction in stiffness driven on by microcracking. The linear elastic finite element method cannot be used to determine the stage of micro cracking caused by stress. However, more advanced methods, such as nonlinear finite elements, can approximate the behaviour of concrete (Romans, 2010).

2.4.3 Non-linear Finite Element Method (NLFEM)

The nonlinear load-displacement and cracking behaviour of concrete structures is commonly evaluated and determined by using nonlinear finite element method (Dere, 2017). This sort of numerical analysis is utilised to evaluate failure behaviour and determine the maximum load carrying capacity. It includes the actual interaction of every structural member, including the effects of concrete crushing, tension stiffening, reinforcement bond slip, and yielding (Romans, 2010). The fracture plastic model of nonlinear concrete material response incorporates constitutive models for tensile and compressive behaviour (Sabau, et al., 2019). This model's incremental-iterative formulation implies that the external loading is applied to the concrete structure in a succession of increments. Due to the more stable result provided by lower increment sizes, this may result in higher computing costs. The less the degree of nonlinear reaction per increment, the higher the accuracy of the result (Romans, 2010).

The nonlinear response of concrete involves changing of element stiffness. The degradation of concrete due to high stress changes the stiffness matrices and causes deterioration of structural integrity of material. It is assumed that the concrete has been crushed when it fails at an integration point in uniaxial compression. Concrete failure is determined using the Von Mises failure criterion (Morsy and Ibrahim, 2019).

2.4.4 Boundary Condition in Finite Element Analysis

Finite element analysis uses a numerical approach with differential equations, that is why the boundary condition must be included (Brischetto et al., 2017). Meshing generation in finite element analysis is not a convenient choice when the condition of the boundary is not known. Therefore, essential boundary conditions should be prescribed on nodes of elements where placed along the immersed boundary, where the degree of freedom (DOF) of the elements are defined (Van Den Boom, et al., 2019). The degree of freedom can take many forms depending on the analysis to be performed, for instance, the degrees of freedom in structural analysis are displacement and rotation at a specific boundary or location.

Solid elements are typically employed in concrete modelling since they are more effective for solving complex geometrical and stress displacement problems. In typical 2D problems, two translational degrees of freedom (x-direction and y-direction) will be adopted for every node in the element so they are having consistent deformations in the location of these nodes. Concerning the general constraints of the model, the boundary condition is relying on the constraint operating on the model, such as the support condition at the bottom or side edges as well as the loading condition acting on the model. The imposed displacement is specified using a roller type boundary condition in order to maximise the model's deformation capacity (Rahnavard, et al., 2016).

2.4.5 Type of Finite Element Analysis

In a finite element model, elements are often divided into 1D, 2D or 3D elements. In general, all line elements, whether they are straight or curved, are classified as 1D elements since they have displacement functions for translation and rotation. In terms of numerical simulation, 1D element analysis can be more easily interpreted because they are well-organised. Because the assumptions employed in 1D elements are the same as those used in theory, it provides exactly the same stresses and displacements values as those estimated by theory (Wai, et al., 2013).

Next, 2D surface elements most often have the fundamental shape of a triangle or a quadrilateral. Three-node triangle elements, six-node triangular elements, and many more are examples of 2D elements. Plane elements or shell elements make up 2D elements. Therefore, rotational displacements and translational displacements are always approximated according to the x and y-axis. They are frequently employed to resolve 2D elasticity problems because they take into account plane stress and plain strain. The overall computational period is short compared to 3D analysis, however, the result does not accurately interpret the actual structural behaviour of the model (Wai, et al., 2013).

Besides, volumetric analysis is often meshed using 3D pieces. Translational displacements in x, y and z direction are taken into consideration by 3D solid elements. Normally, a 3D model is used for complex geometry as it is most accurately simulated according to the actual structure and it produces the clearest deformation that the model may suffer as a result of the applied load (Wai et al., 2013).

2.4.6 Meshing in Finite Element Analysis

The studies on numerical computation by finite element analysis have been used to compute the fracture phenomenon of rigid bodies. The defect of crack was found to be affecting the structural integrity as well as the durability of the material and structure. Engineers will be equipped with the knowledge necessary to minimise the negative impacts brought on by cracks by applying the meshing refinement illustrated in Figure 2.7 to effectively compute the propagation of a material's crack. If the geometry of a crack is small, it must be represented by a fine mesh discretized for the specific cracked area (Wang et al., 2017). However, the computational time for particular fine mesh is long (Areias et al., 2018), and in some cases, it is quite troublesome for the computational task to be successful.

An extremely fine grid's numerical accuracy determines how exact the output will be; as a result, the higher the mesh density, the more accurate the outcome (Vitali, et al., 2018). In terms of mesh sizing, there are no fixed mesh sizes for finite element analysis problems, but it has an upper limit depending on the size of the large coarse aggregate. Any mesh size 200 mm, 100 mm, 50 mm or 25 mm can be adopted in the modelling process. According to the numerical analysis done by Senthil, Gupta and Singh (2018), the size of element is carefully selected based on the mesh convergence study which involves trial and error of element sizing. It is found that the translation or deflection increases when the element size reduces (Senthil, et al., 2018). Eventually, the smaller size of mesh generates high accuracy results as the number of elements present in the model increases. It is impractical to use over fine mesh (10mm, 5mm or 1mm) as it requires long computational duration and the discrepancy in terms of result is not much different (Jain, et al., 2017).

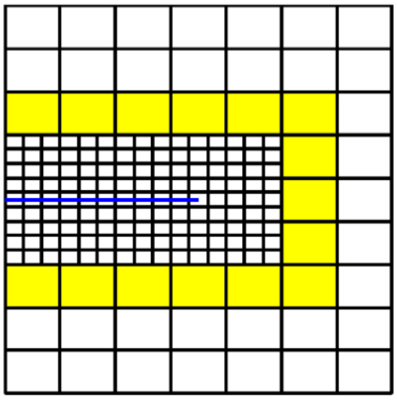


Figure 2.7: Mesh Refinement of a Rigid Body (Wang, et al., 2017).

2.5 Summary

In summary, researchers such as Behfarnia and Shirneshan (2017) have shown that the wall with openings has caused stress path disruption and affected bearing capacity. Besides, staggered opening arrangement has better stress transfer compared to vertical aligned opening arrangement as the stress flow disturbance is lesser. Moreover, the stress path obtained from finite element analysis is capable to graphically deriving the strut and tie analysis method (STM). Linear finite element analysis (FEM) numerically simulates the stress pattern of the entire structure and derives the required reinforcement to be designed for the concrete structure. Whereas nonlinear finite element analysis simulates the actual deformation and stress of the reinforced structure.

CHAPTER 3

METHODOLOGY AND WORK PLAN

3.1 Introduction

This chapter presents a comprehensive methodology and work plan to meet the aim and objectives of this study. The analysis of stress behaviour for transfer wall with openings was conducted by using finite element method. Two-dimensional (2D) analysis by SCIA Engineer and three-dimension (3D) analysis by ANSYS were carried out to represent the structure stress behaviour. Two transfer walls with different openings arrangement (vertical align and staggered openings) were modelled. On the basis of the finite element analysis results, detailing and design were proposed. For the reinforcing design of the structural wall, the Eurocode (EC2) design standard was adopted. Once the reinforcement is obtained, nonlinear analysis of the reinforced transfer wall was performed by ANSYS to study the stress behaviour and the interaction of the reinforcement and the concrete. Figure 3.1 shows the overall methodology of the study.

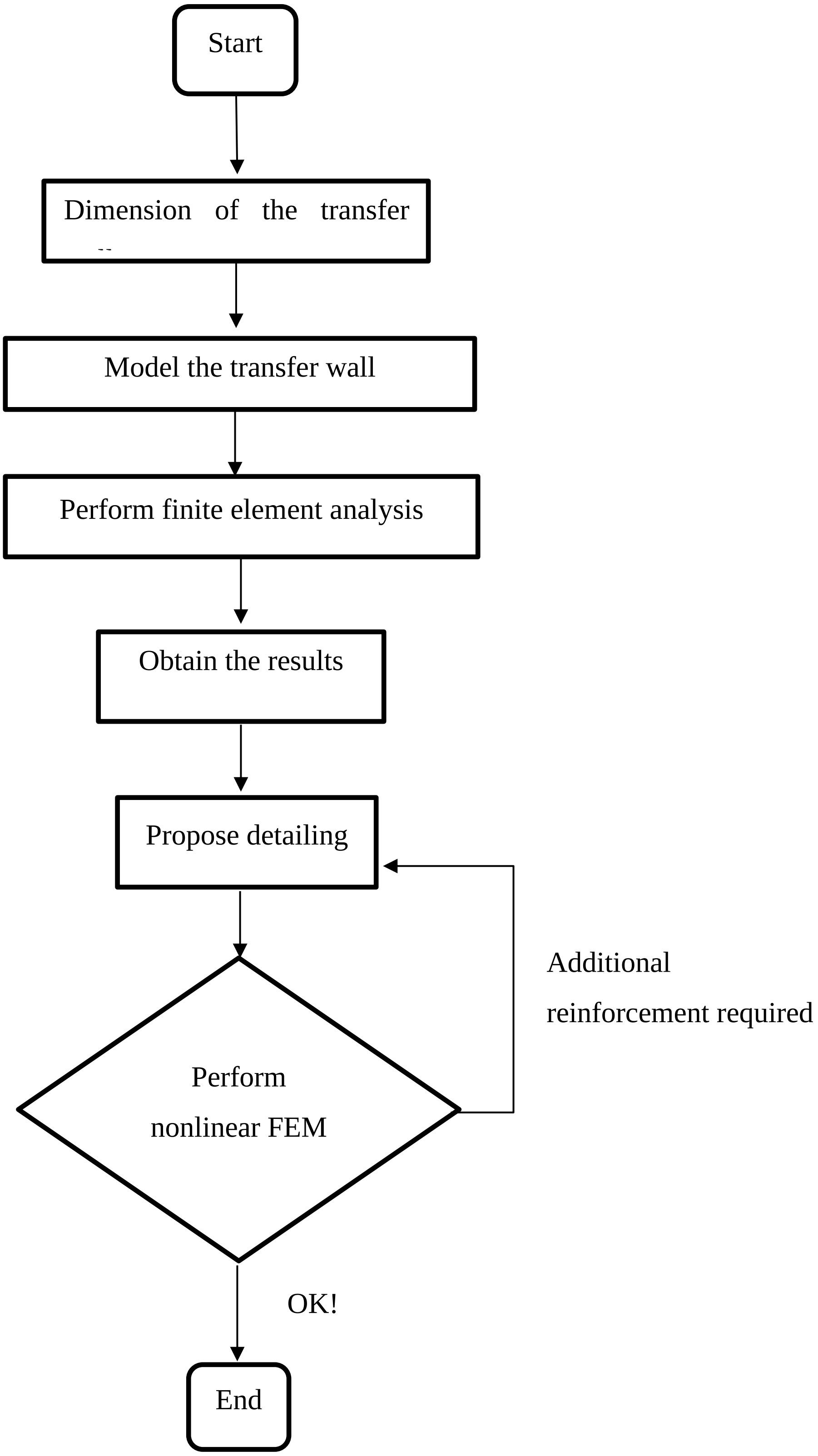


Figure 3.1: Overall Methodology.

3.2 Dimension of Transfer Wall with Openings

In this study, the transfer wall is 4-storey high with 3200 mm per storey, has a total height of 12800 mm and has a thickness of 275 mm. The structure is supported by a transfer beam with dimensions of (800 mm x 600 mm). The beam is supported by two columns with dimensions of (1200 mm x 800 mm). The concrete grade used for both beam and columns are C35/45.

The load exerted axially on top of the wall under ultimate limit state (ULS) are 5156 kN/m. The combination of dead load and variable load were obtained from the structure above the transfer wall by loading area analysis. Besides, the concrete grade used for the transfer wall is G50/60.

There are two openings' configurations to be modelled in this study which are vertical align openings and staggered openings. These openings have a fixed height of 2000 mm and width of 1000 mm and located at every storey of the transfer wall. Figures 3.2 and 3.3 show the transfer wall with vertical align openings and transfer wall with staggered openings respectively. The vertical align openings were located along the centre of the transfer wall whereas the staggered openings were located 1200 mm away from the edge of the transfer wall.

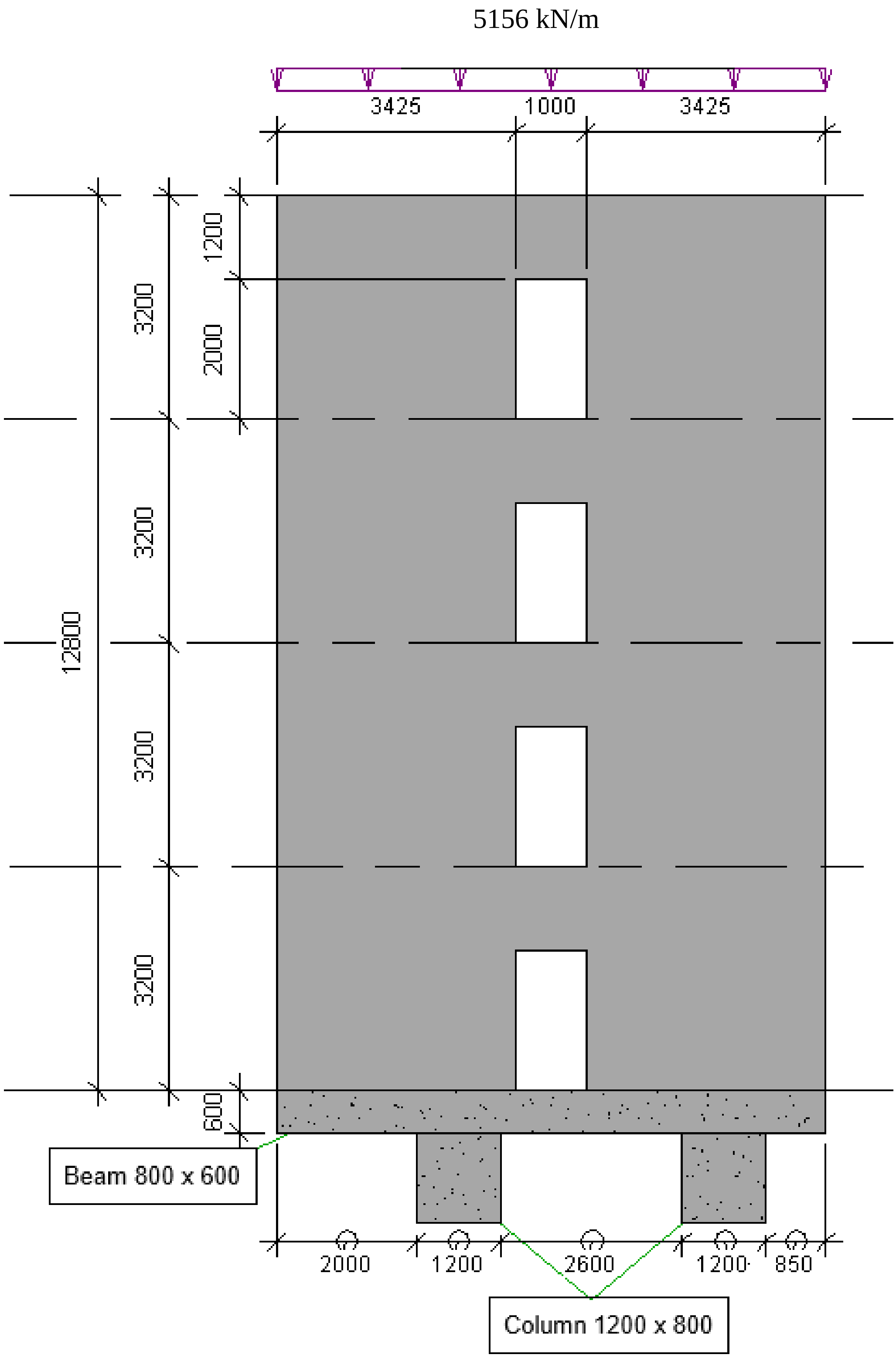


Figure 3.2: Transfer Wall with Vertical Openings.

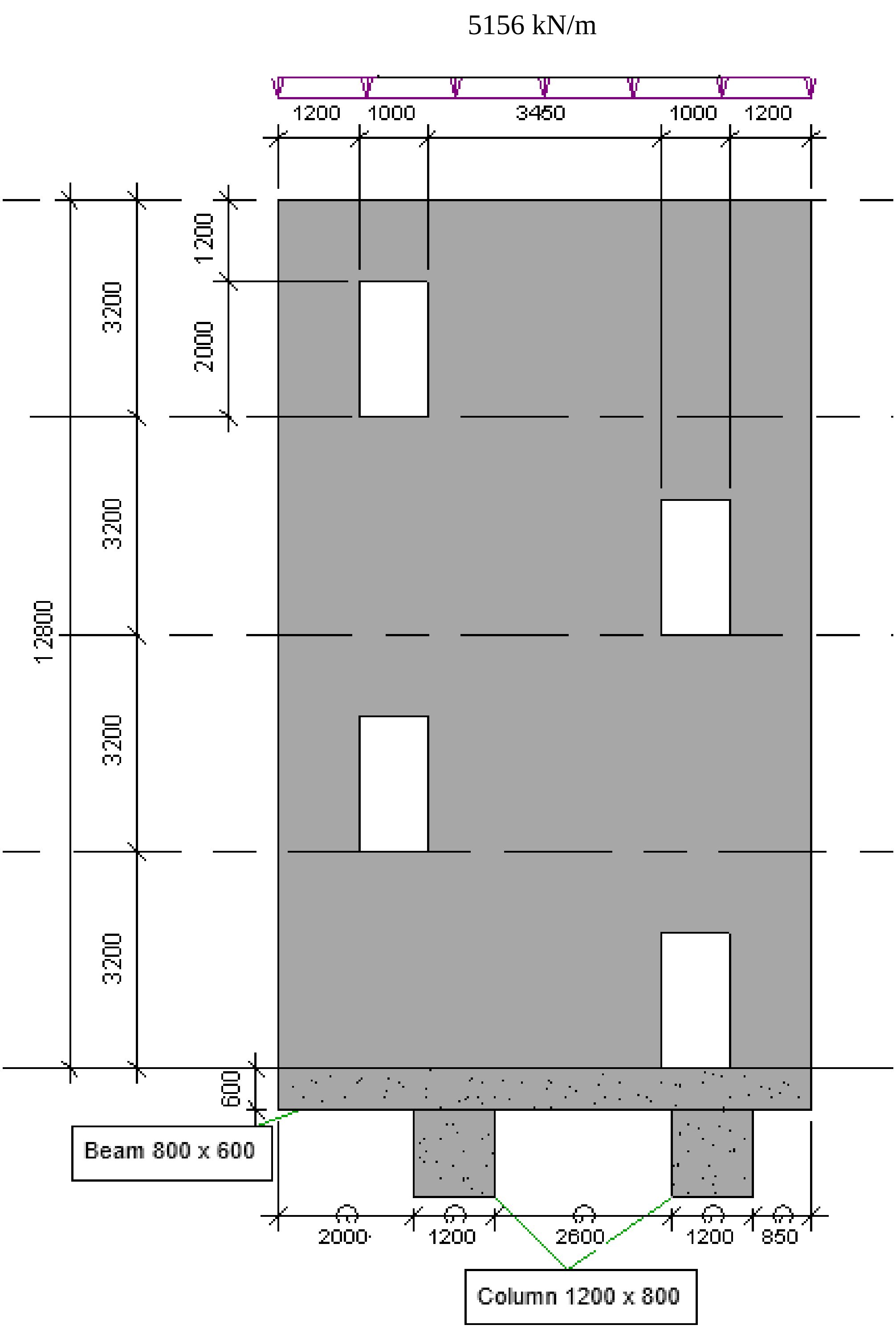


Figure 3.3: Transfer Wall with Staggered Openings.

3.3 Finite Element Analysis Modelling Procedure

Finite element Modelling generally involves three major processes which are pre-processing, finite element analysis and results processing.

Pre-processing is the initial procedure for data input. The information of the model such as structure geometry and material properties were defined. The model is developed according to its specific dimension. The attributes such as loading, boundary conditions, supports, type of element, mesh shape and mesh sizing has to be defined and assigned to the model. The type of analysis approach has to be defined as well, as the outcome of the analysis will be affected by using different approaches.

Once the model has been created, finite element analysis was performed. This process formulated the solutions for each individual element by numerical approach which involves the computation of stiffness matrix. Then, the solutions from each element were assembled and the entire solution were represented. The results can be processed by the assembled numerical solutions. The results were shown graphically where it represented the stress distribution and the load path of the model. Figure 3.4 shows the summarised finite element modelling procedure.

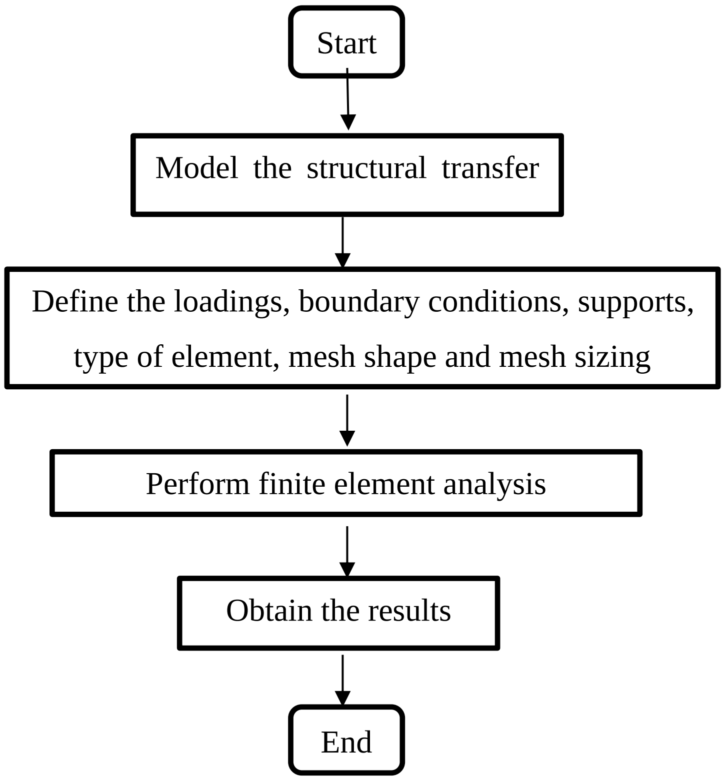


Figure 3.4: Summarized Finite Element Modelling Procedure.

3.3.1 Transfer Wall Finite Element Model by SCIA Engineer

The material properties of concrete were defined in the pre-interface. The Eurocode was utilised as reference codes for all material properties, loading assignments and design. Two-dimensional (2D) model space model space was selected before any data input and modelling process. Two-dimensional (2D) model space was selected. In this analysis, the transfer wall model will only act on the plane surface.

The models were drawn according to dimension as shown in Figures 3.2 and 3.3. As for the openings, a cut extrusion function was used to make a hole on the transfer wall. The transfer wall and the columns were modelled as 2D panels whereas the beam member was defined in 1D member to ensure the beam does not fail in bending and shear when stress is applied from the transfer wall. Concrete grade C50/60 was used for the transfer wall whereas C35/45 was used for the beam member and columns.

Next, the attributes such as loading condition, boundary condition, support condition type of element and meshing pattern were defined onto the model. The loading characterises the external influence which the model was subjected to. Uniformly distributed load of 5156 kN/m with the combination of dead load and variable load under ULS was defined on the top edge of the 2D transfer wall. Safety factors of 1.35 for dead load and 1.5 for variable load are taken into consideration to simulate the critical loading case for the structure.

The boundary condition was described by the way where the restraint is located. The transfer wall was set to be fully restrained at the top edge; the bottom edge was supported by a beam and two columns. As for the side edge of the transfer wall, the rotation and displacement in X and Y direction were restrained. As for the Z-axis in the global coordinate system of the software, the displacement and the rotation were not restrained as transverse movement such as wind load was allowed to the face of the transfer wall. Besides, the columns were fixed at the bottom where it was a continuous structure. For clearer visualisation purposes, Figure 3.5 below shows SCIA Engineer modelling (vertical aligned openings) with respective description as well as the global coordinate system for 2D model space.

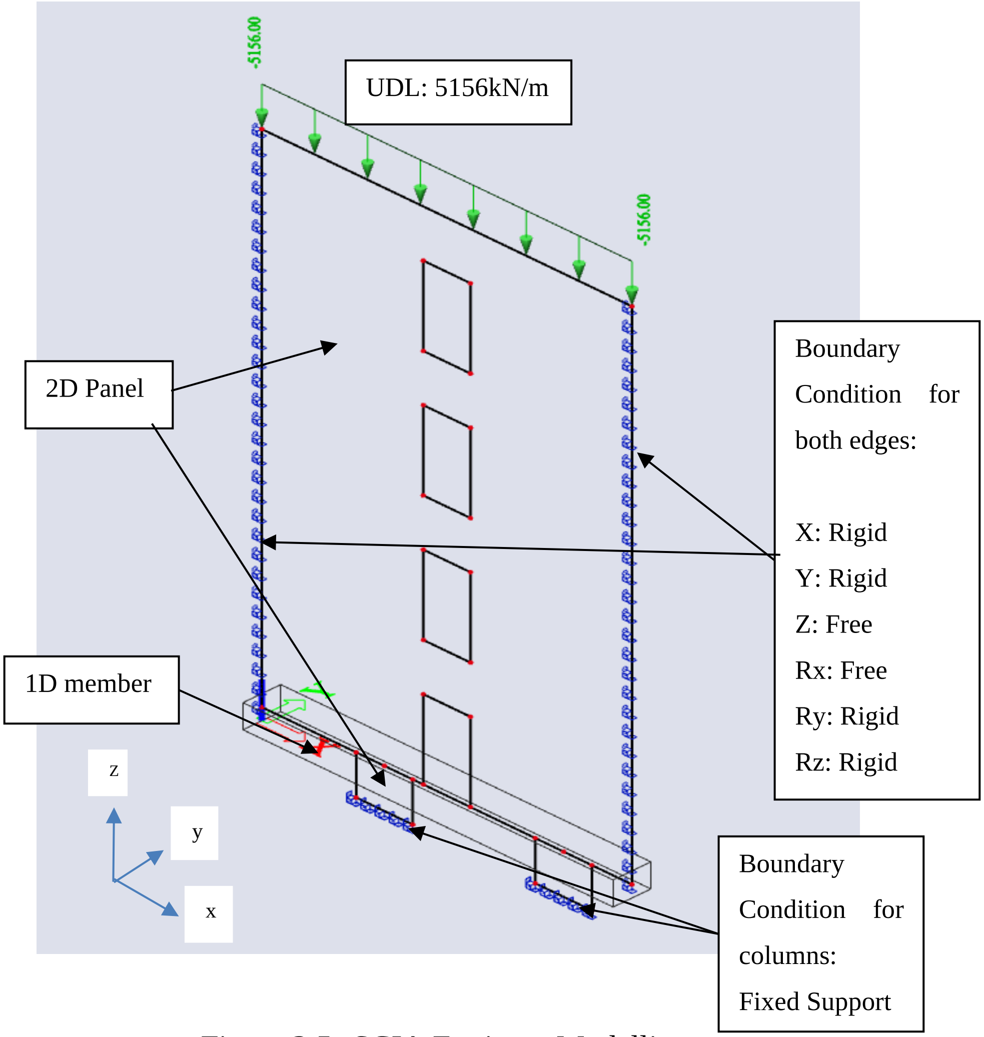


Figure 3.5: SCIA Engineer Modelling.

The finite element model used meshing to discretize the model into finite elements for computation. The number of mesh and meshing density will affect the accuracy of the analysis. The preliminary mesh size of 50 mm was used which is twice the size of the aggregate. 2D plate element type was used to represent the finite element structure. The shape of the transfer wall and the openings are rectangular, which is not complex, a four-nodes rectangular element type was used for the analysis.

Once the finite element model has been defined, finite element analysis was carried out. The results were processed at the end of the program, which was known as post-processing. The result shows the outcome and visualisation of the numerical modelling which includes the principal stress of the transfer wall, 1D stress of the beam and total deformation of the entire structure. Besides, the required reinforcement in transversal and longitudinal direction per finite element was determined to proceed with design and detailing of the transfer wall.

3.3.2 Transfer Wall Finite Element Model by ANSYS

The material properties were defined in the engineering data as the material behaviour was not yet defined in the project by default. Individual material properties such as compressive yield, compressive ultimate strength, Young Modulus, stress-strain behaviour and Poisson as shown in Table 3.1 were input into the software according to the Eurocode 2 standard. The analysis type static structural in 3D was selected before proceeding to modelling.

Table 3.1: Concrete Design Properties (EN 1992-1-1:2004).

Properties	Concrete C50/60	Concrete C35/45
Density (kg/m^3)	2500	2500
Young's Modulus, E (MPa)	37278	34077
Poisson's Ratio	0.2	0.2
Compressive yield Strength (MPa)	20	14
Tensile Ultimate Strength (MPa)	4.07	3.21
Compressive Ultimate Strength (MPa)	50	35

The transfer wall was sketched according to its dimension. The sketched model was then extruded to its thickness. Then, the beam and the columns were modelled as 3D members with extruded thickness of 800 mm. The transfer wall and the support structure were defined as separated bodies. Once the sketching was done, a connection was automatically generated by the software as a contact body to join the separated structures (transfer wall, beam and column). The contact body was set as bonded as these structures are separated supporting members.

The attributes such as boundary conditions and loading were defined accordingly. The structure was assigned with its respective material behaviour. As for the loading, a force was defined on the top surface of the transfer wall. The load was defined by the global coordinate system in Y direction with a value of -40762.5 kN after safety factor consideration. The negative sign represents the downward direction along the Y-axis.

As for the boundary condition, remote displacement with free X-axis rotation was allowed for transverse movement. The displacement of the Y-axis was defined as free at both side surfaces of the transfer wall, as the side surface of the transfer wall was not restrained to support the axial load. Besides, fixed support was defined at the bottom of the column edge as continuous structure. As for the mesh setting, the maximum element sizing used was 50 mm with linear mechanical physical preference. Figure 3.6 below shows the finite element model (vertical aligned openings) in 3D and the global coordinate system for ANSYS interface.

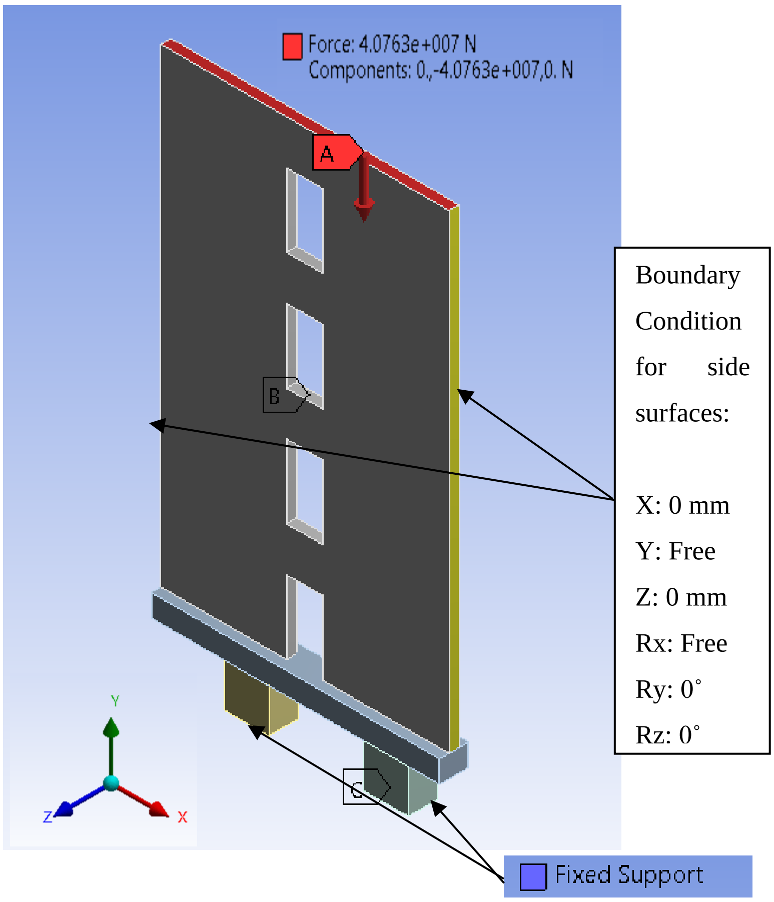


Figure 3.6: ANSYS FEM Modelling.

The finite element analysis was performed. The result showed the outcome and visualisation of the numerical modelling which includes the principal stress, von-mises stress, normal stress and deformation of the entire structure. Based on the result obtained, study was made by comparing the result obtained from SCIA Engineer software. Therefore, the design reinforcement of the transfer wall was optimised.

3.3.3 Finite Element Meshing Refinement

The finite element meshing is an important attribute in finite element modelling as it determines the overall accuracy of the analysis result. The shape and the size of the elements tend to change when the mesh is refined. In SCIA Engineer, mesh refinement was generated depending on the locations where high stress gradients were present. For the transfer wall modelling, high density meshes were generated especially at the edge and corner of the opening as these locations were highly stressed. Besides, the accuracy of translational displacement of the finite elements was determined by mesh refinement. The denser the mesh refined, the higher the accuracy of the deformation of meshes.

3.4 Finite Element Stress Analysis

In the 2D analysis of SCIA Engineer, the principal stress value was adopted to indicate the extreme value of the internal force which derived from the basic stress by transformation into the direction of the principal axes where the shear stress is equal to zero at a specific orientation.

Normal stresses were not used in this analysis as it just represented the stress components acting perpendicular to a particular plane or surfaces. The stress state at a point in a solid body is complex and cannot be fully characterised by a single normal stress. The angle of the principal magnitudes were derived from the local axis in x-direction and visualised as trajectories (SCIA Engineer, 2015). The formulation of maximum and minimum principal stress (σ_1 and σ_2) used by SCIA Engineer was shown in Equations 3.1 and 3.2 respectively. The equation for angle of the principal stress, α- was shown in Equation 3.3 as it was used to calculate the required reinforcement areas to the direction of the principal stress. In general, the principal magnitudes were defined with the help of the stress block and Mohr's Circle shown in Figure 3.7.

$$\sigma_1 = \frac{1}{2}(\sigma_x + \sigma_y + \sqrt{(\sigma_x - \sigma_y)^2 + 4\sigma_{xy}{}^2} \quad (3.1)$$

$$\sigma_2 = \frac{1}{2}(\sigma_x + \sigma_y - \sqrt{(\sigma_x - \sigma_y)^2 + 4\sigma_{xy}{}^2} \quad (3.2)$$

$$\alpha\text{-} = \frac{1}{2}atan^2(\sigma_{xy,}\sigma_x - \sigma_y) \quad (3.3)$$

where;

σ_x = normal stress at x-axis, MPa

σ_y = normal stress at y-axis, MPa

σ_{xy} = shear stress, MPa

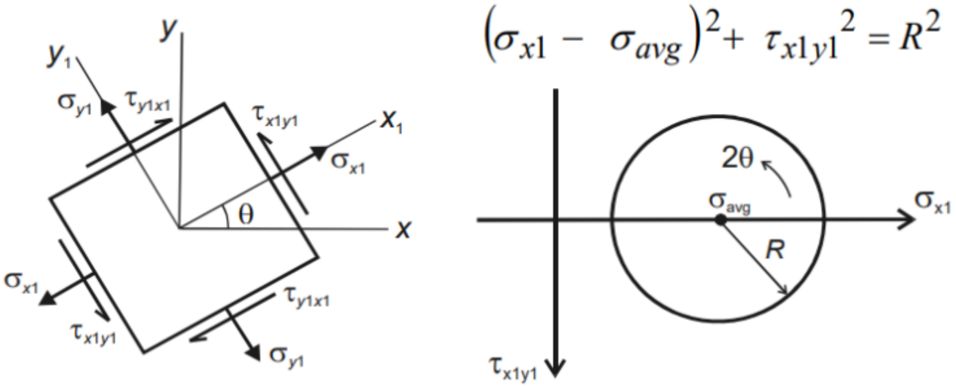


Figure 3.7: 2D Stress Block and Mohr's Circle.

As for the 3D analysis in ANSYS, the concept of stress tensor was adopted in the finite element model that represented the magnitude and direction of the stresses in 3D space. It was then solved by the eigenvalue-eigenvector method to determine the principal stress.

The principal stress analysis was computed from the normal stress components, whereas the derivation of normal stresses was dependant on the type of 3D element used. In this study, SOLID 45 elements were adopted for the analysis of transfer wall with three degrees of freedom (DOF) at each node by nodal translation in x, y and z directions (ANSYS inc., 2010). The computation of 3D principal stress, $\sigma_{1,2,3}$ was shown Equation 3.4. The concept of 3D stress block and Mohr's Circle was shown in Figure 3.8 for better visualisation.

$$\sigma_{1,2,3} = \frac{\sigma_x+\sigma_y+\sigma_z}{3} \pm [(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6({\tau_{xy}}^2 + {\tau_{xz}}^2 + {\tau_{yz}}^2)]/2 \tag{3.4}$$

where;

σ_x = normal stress at x-axis, MPa

σ_y = normal stress at y-axis, MPa

σ_z = normal stress at z-axis, MPa

τ_{xy} = shear stress at xy direction, MPa

τ_{xz} = shear stress at xz direction, MPa

τ_{yz} = shear stress at yz direction, MPa

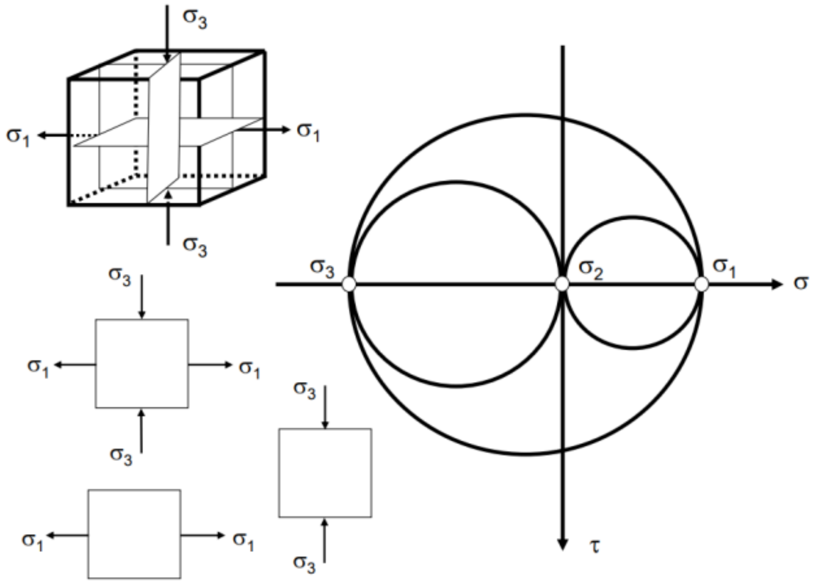


Figure 3.8: 3D Stress Block and Mohr's Circle.

In this study of finite element analysis, principal stresses were used in numerical analysis of the transfer wall, as it provided a more complete and accurate representation of the stress state at a point, compared to normal stresses.

3.5 Detailing of Transfer Wall with Openings in SCIA Engineer

The detailing or the required reinforcement was generated from the result of linear finite element from SCIA Engineer in terms of mm^2/m. Reinforcement template for wall structure was used by defining bar diameter and bar spacing manually as provided reinforcement. Eurocode 2 (EN1992-1-1) was adopted for the design reinforcement of the transfer wall.

The required reinforcement was calculated based on the principal stress value. Required reinforcement formula shown in Equation 3.5 was used to calculate the required reinforcement from design load of a structure in both directions with safety factor of 1.15. However, recalculation was done by SCIA Engineer based on the principal direction shown in Equation 3.6 iteratively until the optimal reinforcement area was achieved. Besides, the shear resistance, shear capacity and crack control were computed by SCIA Engineer to achieve its serviceability requirement (SLS). The design check with the provision of Eurocode 2 was also done by the software itself.

$$A_s = (\sigma_{max} * b * d)/0.87 * f_{yk} \tag{3.5}$$

where;

σ_{max} = Maximum Stress, Mpa

b = width, mm

d = thickness, mm

f_{yk} = yield strength of reinforcement steel bar, MPa

$$A_{s,ult} = A_{s,1}.cos\ (\alpha)^2 + A_{s,2}.cos\ (\alpha)^2 \tag{3.6}$$

where;

$A_{s,1}$ = required reinforcement in longitudinal direction, mm^2

$A_{s,2}$ = required reinforcement in transverse direction, mm^2

α = principal angle, $^\circ$

According to Eurocode 2 standard, the dimension of the load bearing structural wall with openings should not be less than 175 mm, the ratio of wall height to the wall thickness is limited to 40. The concrete grade used shall be higher than Grade 35. The nominal cover of 35mm plus allowable deviation (C_{dev}) should be achieved to reduce the risk of carbonation. The minimum transverse reinforcement required is 0.2% of the total area of concrete where the minimum longitudinal reinforcement required is selected based on the higher value of 0.1% of the area of concrete or 25% of the vertical reinforcement. Whereas, the maximum area of reinforcement is 4% of the total concrete area. Besides, at least 75 mm of bar spacing should be provided where 100 mm of bar spacing for rebar size greater than 40 mm. However, the maximum bad spacing should be limited to the lesser of 3 times the wall thickness or 400 mm. As for the shear link, the minimum vertical reinforcement of 2% of total concrete area should be provided. The vertical spacing of the links should be used to the lesser of 16 times of vertical bar diameter or 2 times the wall thickness, whereas the maximum spacing of horizontal link spacing should be lesser than twice the wall thickness

Besides, diagonal reinforcement should be provided at the corner of the openings to provide better crack control. The trimming bar used should be one size bigger than the diameter of rebar surrounding the walls, U-bars with the same size of horizontal bars should be placed around the openings for enclosing of trimming bars. In summary, Figure 3.9 summarises the design flow of the structural wall detailing according to EN1992-1-1.

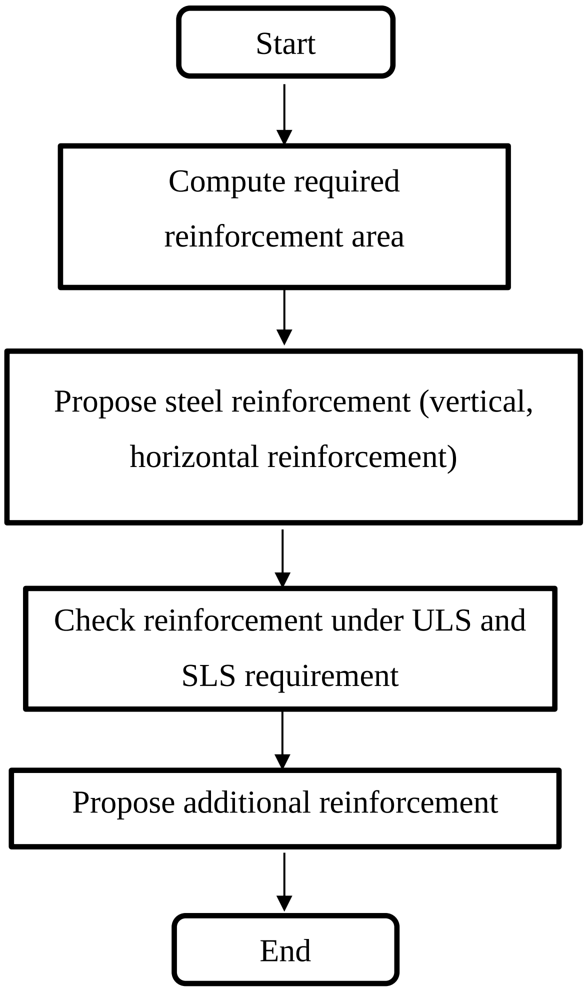


Figure 3.9: Transfer Wall Design and Detailing.

3.6 Nonlinear Analysis of Reinforced Transfer Wall by ANSYS

The 3D concrete model used for the previous analysis is continued by adding the reinforcement. Steel reinforcement and concrete behaviour were defined in the engineering data by adopting nonlinear behaviour of the materials. The main parameters needed to be defined for the rebar were Young's Modulus (*E*), Poisson ratio (*v*), and the Yield strength (f_{yk}). Table 3.2 below shows the properties of steel rebar.

Table 3.2: Rebar Properties.

Properties	Rebar
Density (kg/m^3)	7850
Young's Modulus, E (GPa)	200
Poisson's Ratio	0.3
Yield Strength (MPa)	500

The rebar was sketched as a 1D line body embedded in the concrete structure by the assistance of a reference plane. The offset defined while creating the reference plane was based on the concrete cover and the arrangement of the rebar (transverse and longitudinal). The cross-section of the rebar was defined as circular form with its respective diameter. For every line sketched in the concrete model, 1D line body and its respective cross-section were to be defined for these lines. Then, a repetitive pattern was created to thoroughly complete the drawing of line members. Figure 3.10 shows the nonlinear modelling of 1D line reinforcement.

The boundary conditions of the model remained whereas the interaction between the concrete and the steel reinforcement was defined. The model type of the line members was defined as reinforcement, therefore, APDL commands were not required anymore as the reinforcement-concrete interaction had been defined by ANSYS. It was required to enable beam section results as it included the stress result for the reinforcement.

Once the analysis definition was done, the finite element model was run to study its stress pattern and the behaviour of the reinforced structure. The

Von Mises stress contour of the reinforcement was obtained to study the overall structural behaviour of the model. Figure 3.11 shows the summarised nonlinear finite element modelling procedure of the reinforced transfer wall.

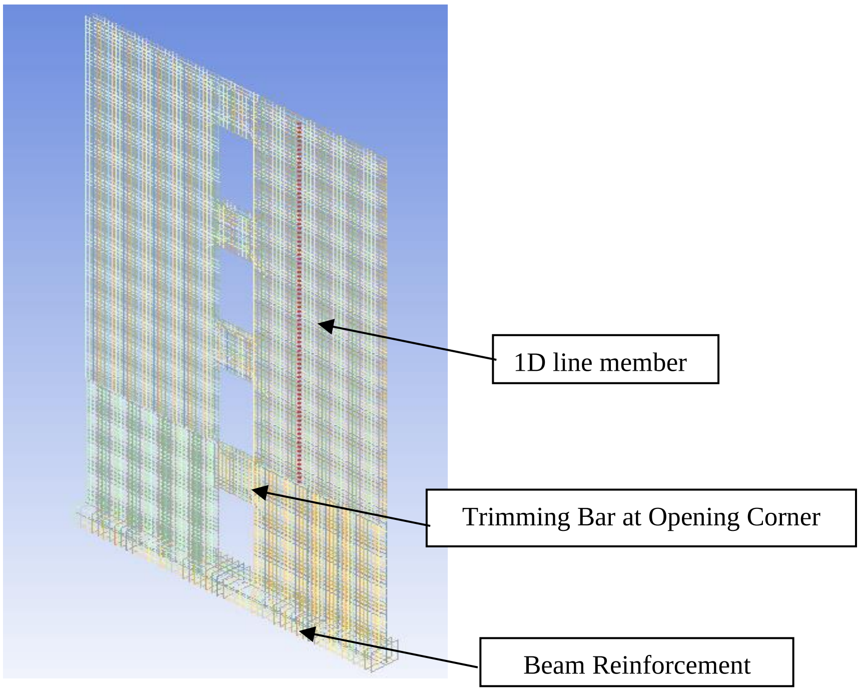


Figure 3.10: Nonlinear Modelling of 1D Line Reinforcement.

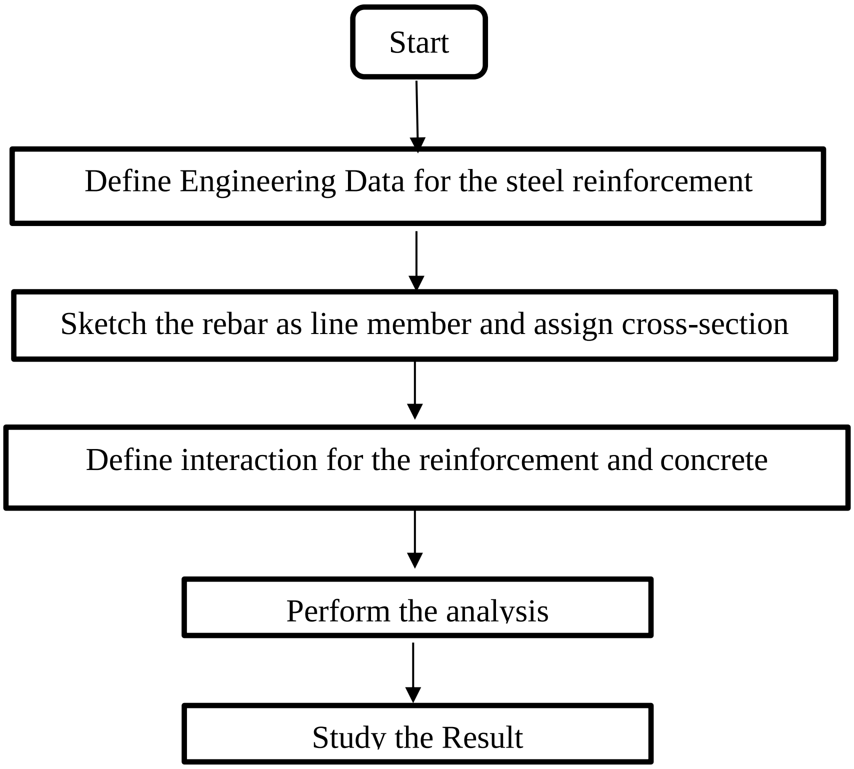


Figure 3.11: Nonlinear Finite Element Modelling.

3.7 Summary

In summary, the methodology workflow is divided into three major processes which are numerical modelling, analysis and modelling verification. Numerical modelling described the input data such as material properties, boundary conditions or meshing. Next, the numerical analysis was carried out to obtain the result such as maximum or minimum principal stress, deformation and Von Mises stress contour. Then, it was followed by result verification by comparing 2D analysis and 3D analysis to check the accuracy and the reliability of the modelling approach. The design of reinforcement was carried out by deriving the linear analysis result obtained from 2D analysis. In order to check the structural behaviour of the reinforced concrete model, nonlinear analysis was carried out to study the efficiency and stress behaviour of the reinforcement and the concrete structure.

In this study, 6 finite element models were required to be analysed which specified with annotation shown in Table 3.3.

Table 3.3: Specification of Numerical Model.

Model	Specification
Model A-1	Transfer wall model with vertical aligned opening (SCIA Engineer)
Model B-1	Transfer wall model with staggered opening (SCIA Engineer)
Model A-2	Transfer wall model with vertical aligned opening (Ansys)
Model B-2	Transfer wall model with staggered opening (Ansys)
Model A-RC	Reinforced transfer wall model with vertical aligned opening (Ansys)
Model B-RC	Reinforced Transfer wall model with staggered opening (Ansys)

CHAPTER 4

RESULTS AND DISCUSSION

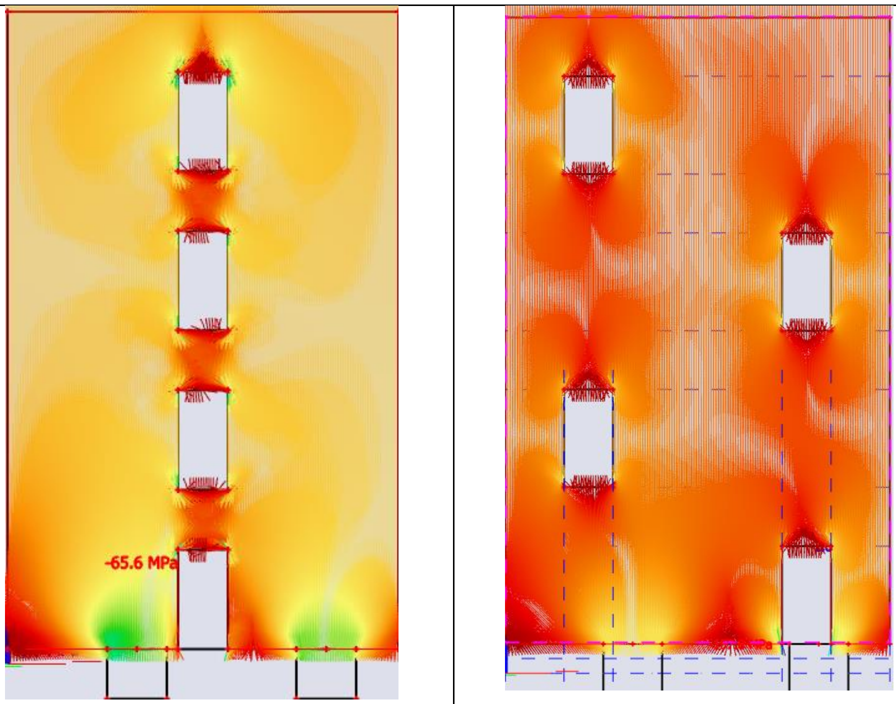
4.1 Introduction

This chapter presents the 2D stress analysis results obtained from finite element analysis of the transfer wall model from SCIA Engineer. The results were validated by the 3D simulation of the finite element model from ANSYS. Then, the required reinforcement design was proposed which derived directly from the stress of the finite element mesh. Lastly, this chapter ended with the analysis of the finite element model of reinforced transfer wall.

4.2 2D Stress Analysis from SCIA Engineer

This section discussed the stress analysis of finite element models from SCIA Engineer. Based on the 2D analysis results, it was found that high concentration of stress appears at the support region and opening corners of the transfer wall. The convergence of the stress path to the supporting beam and column structure is required to transfer the loading from top region to bottom structure. Besides, the punching force induced by the supporting column and beam has caused high stress concentration at the supporting region of the transfer wall. Therefore, it was considered as the most critical location that requires proper reinforcement design.

The stress trajectories shown in Figure 4.1 for both models have visualised the distribution of stress under loading. It helped to identify potential failure points in the structure where stress concentration was presented. The colour-coded contour shown in the figure represented the magnitude and the direction of the stress at different locations within the structure. The relationship between the stress and strain of the structure was described in Hooke's law in which the stress is directly proportional to the strain of the material.



(a) Model A-1	(b) Model B-1

Figure 4.1: Stress Trajectories.

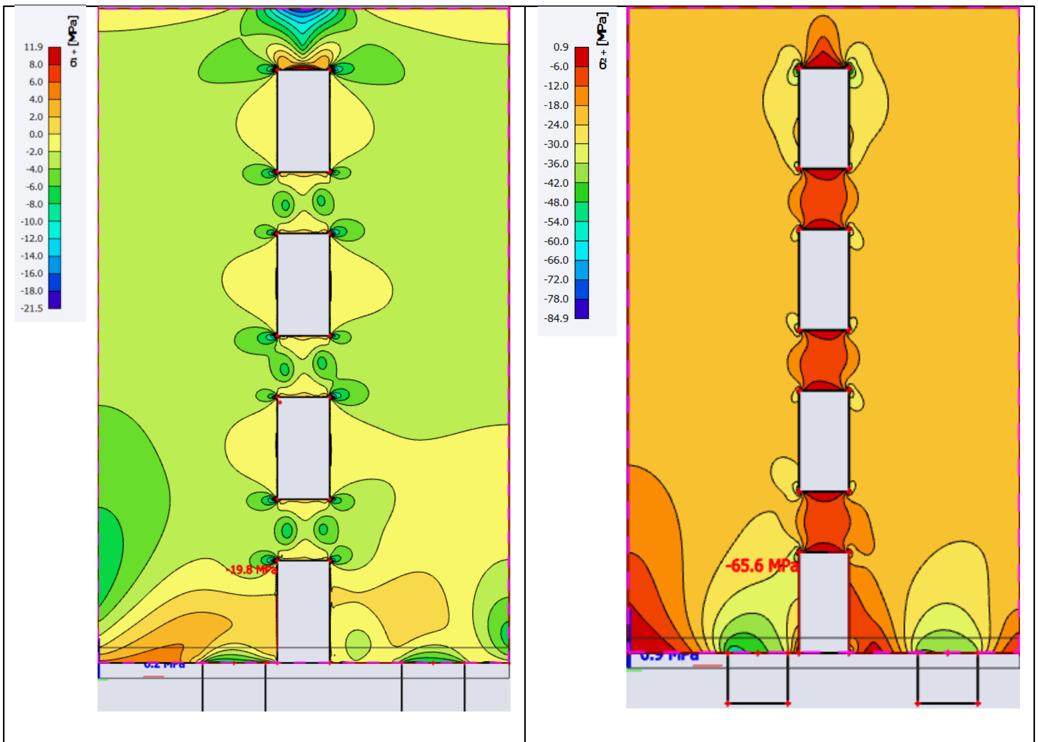
Stress trajectories for Model A-1 were shown in Figure 4.1(a), it was observed that the transfer wall structure functioned as a compressive arch where the load was transferred down from top through spaces without the openings. Besides, high tensile stress was found to be located above the top opening which has a stress pattern of tensile tie. It was due to the presence of the opening that disrupted the load path. The opening's corner was considered as one of the highly stressed spots in the structure and this disruption of stress path transition has led to the development of stress concentration in tension.

The stress trajectories for Model B-1 shown in Figure 4.1(b) react far differently compared to Model A-1. It was observed that the load path transferred along the section without opening (middle section) was having less disruption. However, the stress concentration at the opening's corner was similarly high compared to Model A-1. As for the support region, higher stress was concentrated at the right support compared to left support. This was due to the location of the bottom-most opening on top of the beam and column structure that converged the stress and induced high concentration of stress.

Therefore, the resultant compressive stress at the right support section was higher compared to the left section.

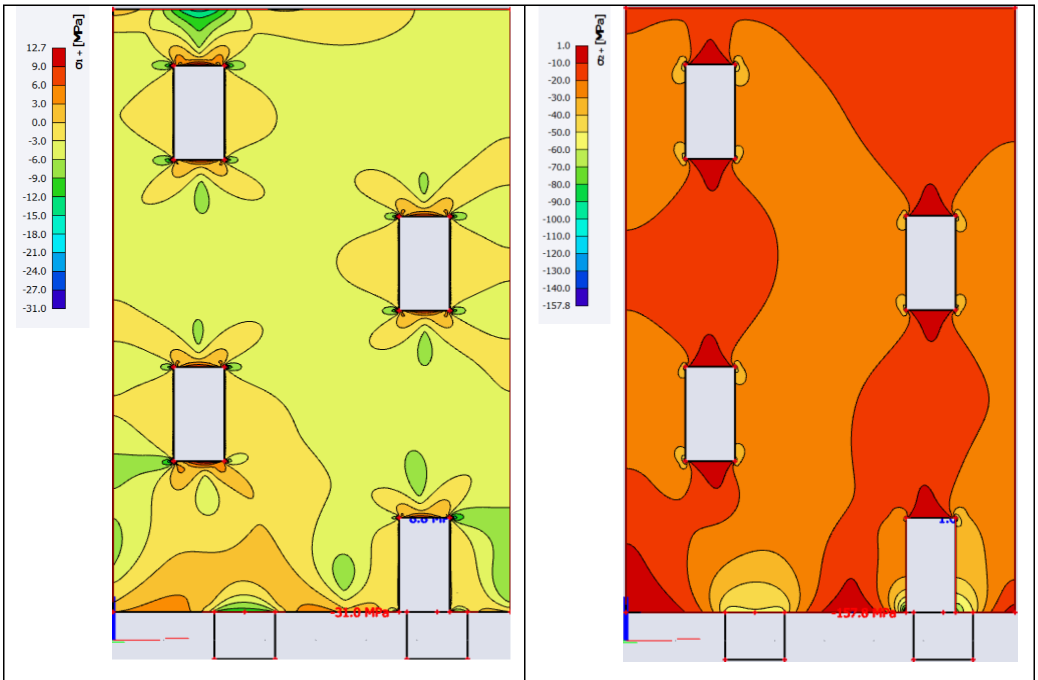
Principal stress was referring to the maximum and minimum values of stress at a specific location within the structure. The principal stress value was used to determine the orientation and the magnitude of the stresses that were most critical to the strength and behaviour of the transfer wall. The potential failure points in the structure were easily identified when the strength of the material was exceeded by principal stress. The optimization of the structural design was based on the principal stress value which was obtained from SCIA Engineer.

Figures 4.2 and 4.3 show the maximum principal stress and minimum principal stress of Model A-1 and Model B-1 respectively. Maximum principal stress results were used to determine the highest tensile area whereas the minimum principal stress results tracked the highest compression region. Based on Figure 4.2(a) and Figure 4.3(a), it was observed that both models had similar tensile stress patterns at the loading area above top opening and opening corners. These regions were considered critical as concrete is weak in tension.



(a) Maximum Principal Stress	(b) Minimum Principal Stress

Figure 4.2: Principal Stress Contour of Model A-1.



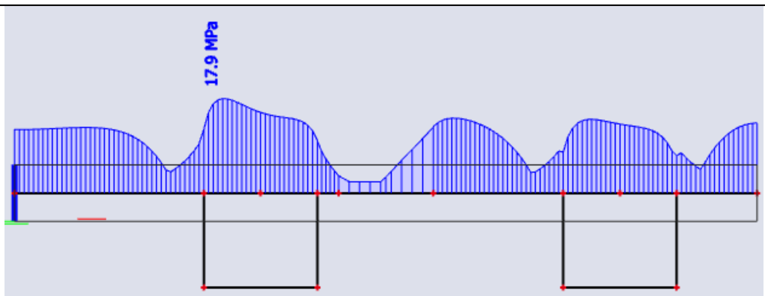
(a) Maximum Principal Stress

(b) Minimum Principal Stress

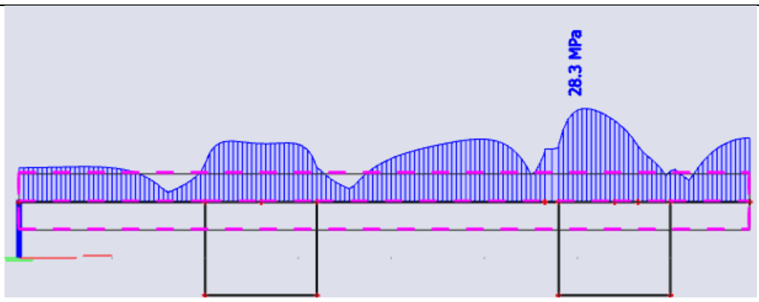
Figure 4.3: Principal Stress Contour of Model B-1.

Based on Figure 4.2(b) and Figure 4.3(b), highest compressive stress was found to be located at the support region right above the column supports. Due to unsymmetrical support regions, the support regions had different compressive reaction force induced from column structure. Besides, it was found that the compressive stress tended to be concentrated at the support region located nearer to the bottom opening due to stress convergence. It was validated that the compressive stress value was higher at the left support of Model A-1 and higher at the right support of Model B-1.

The overall compressive stress experienced by Model B-1 was higher than Model A-1, the maximum compressive stress at the support region of Model A-1 ranged between 36 MPa to 48 MPa. Whereas for Model B-1, the maximum compressive stress at the support region ranged between 50 MPa to 60 MPa which had exceeded the uniaxial compressive strength of the concrete (C50/60). Besides, the 1D stress of beam support shown in Figure 4.4(a) and Figure 4.4(b) has proved that the location with highest concentration of stress was located above the column structure. The stresses from the transfer wall were transferred to the nearest support columns through the beam. It was clearly seen that the critical location of the beam happened to be located at the point of the face of the columns. The maximum 1D stress carried by the beam Model B-1 was 28.3 MPa which was higher than the 1D stress of 17.9 MPa carried by beam Model A-1.



(a) Model A-1



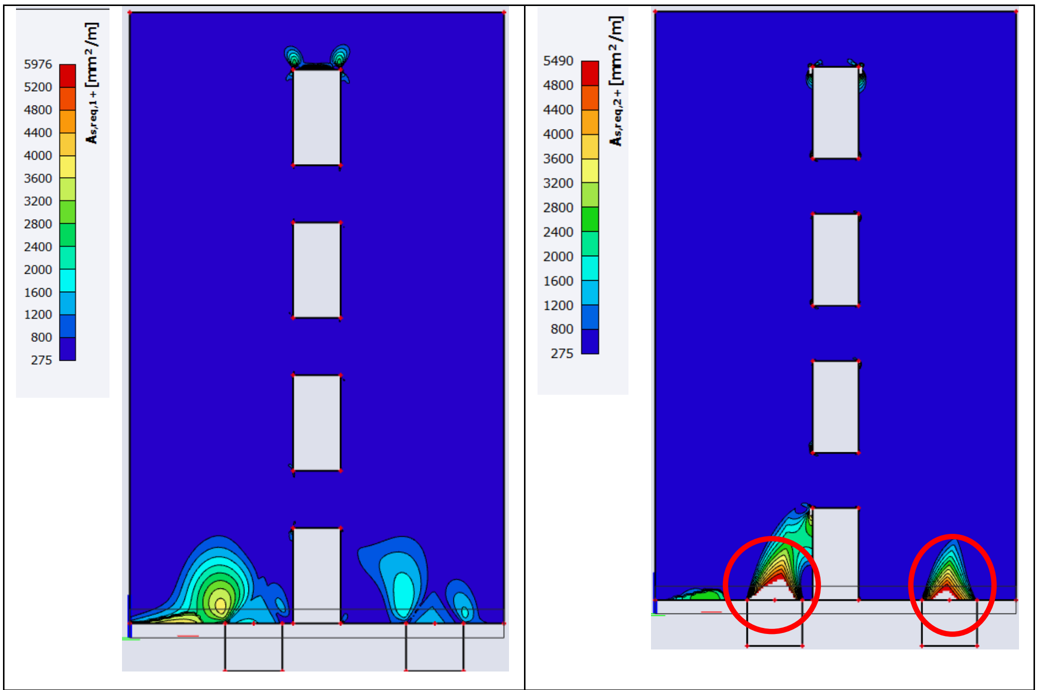
(b) Model B-1

Figure 4.4: Maximum 1D Beam Stress.

It was concluded that the Model A-1 (vertical aligned opening arrangement) had more influence in stress distribution compared to Model B-1 (staggered opening arrangement). Besides, the presence of openings has induced weak tensile regions around the transfer wall. Moreover, the critical compressive region was governed by the location of the opening which was nearest to the support region.

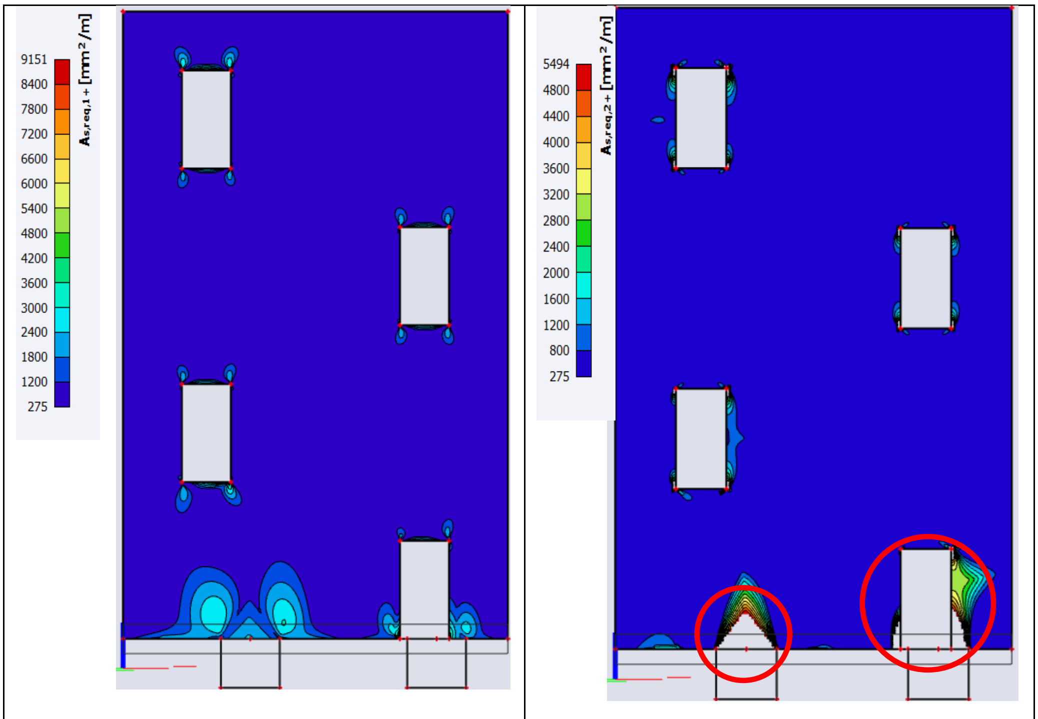
4.3 Design of Reinforcement

Based on the 2D analysis, the reinforcement design of the model was according to the stress result at a specific location. Transverse (y-direction) and longitudinal reinforcement (x-direction) shown in Figure 4.5 (Model A-1) and Figure 4.6 (Model B-1) were the main elements required to enhance the structural performance. The required reinforcement shown in the figures was the front face of the model. The back face of the models adopted the same reinforcement with the front face as both faces were symmetrically reinforced.



(a) Longitudinal Reinforcement | (b) Transverse Reinforcement

Figure 4.5: Required Reinforcement of Model A-1.



(a) Longitudinal Reinforcement | (b) Transverse Reinforcement

Figure 4.6: Required Reinforcement of Model B-1.

High tensile stress was located at the loading area above the opening and support region. Therefore, a large amount of longitudinal reinforcement was required at these critical locations. As for the required transverse reinforcement or vertical reinforcement, it was mainly concentrated at the support region to sustain the high compressive stress. The presence of the white region highlighted in the figures represented the undesignable region which indicated that the required reinforcement has exceeded the maximum reinforcement allowed in Eurocode 2. In this case, additional transverse rebars were required to be added and extended from the beam structure as starter bar to meet the required reinforcement especially at the region where columns were located. As seen in Figure 4.6(b), the additional transverse reinforcement at the right support region was to be placed at both sides of the opening in order to resist the concentration stress. Trimming bars were provided as diagonal reinforcement at each opening corner to prevent cracking induced by high stress concentration.

The provided reinforcement should not be over-reinforced so that the durability of the entire reinforced structure could be preserved. Sudden failure by compression concrete crushing might happen if the concrete compression strain has been exceeded before the yielding of the reinforcement. Therefore, the reinforcement should achieve the required reinforcement area to cater the compressive stress and tensile stress especially at the critical region. For visualisation purposes, the detailed drawing for both models are shown in Appendix A.

SCIA Engineer derived the required reinforcement based on the design load (n_{ed}) of the membrane stress from the stress analysis. Then, the recalculation of design load was done based on the direction of the principal stress to ensure the tensile stress can be withstand by sufficient reinforcement. Table 4.1 (Model A) and Table 4.2 (Model B) shows the summarised calculation of reinforcement design at maximum condition.

Table 4.1: Required Reinforcement Calculation (Model A).

Model A			
Values		Design load (n_{ed}, kN)	As, required(mm^2)
X-direction (Longitudinal)		625.92	720
Y-direction (Transverse)		-10,717.70	5,490
After Recalculation			
Values	α- (˚)	Design load (n_{ed}, kN)	As, required (mm^2)
X-direction (Longitudinal)	-10.8	-1,192.39	1,373
Y-direction (Transverse)	79.2	-13,393.97	5,490

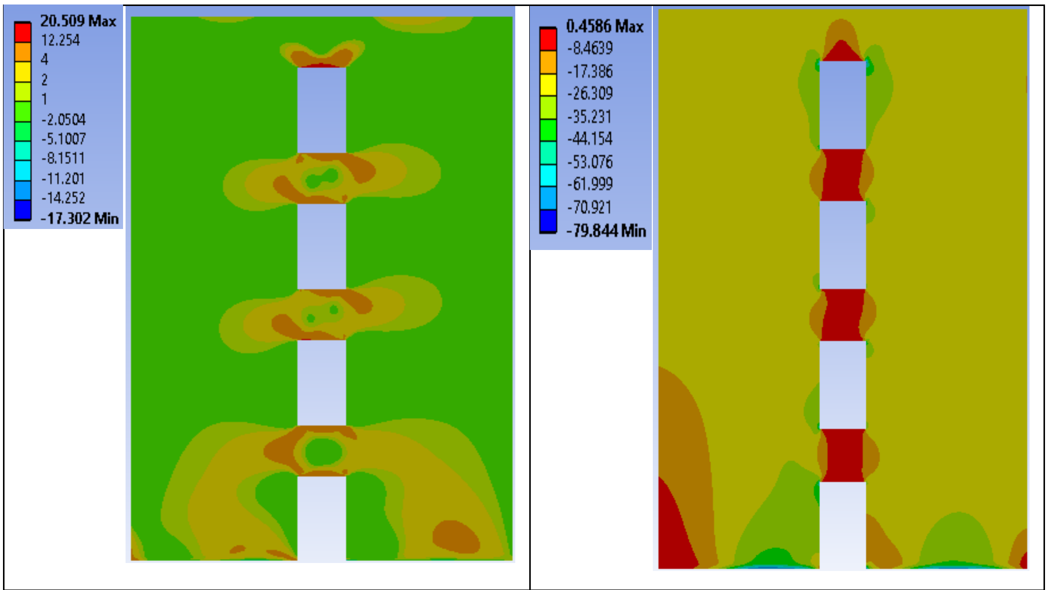
Table 4.2: Required Reinforcement Calculation (Model B).

Model B			
Values		Design load (n_{ed}, kN)	As, required (mm^2)
X-direction (Longitudinal)		576.33	663
Y-direction (Transverse)		-10,720.95	5,494
After Recalculation			
Values	α- (˚)	Design load (n_{ed}, kN)	As, required (mm^2)
X-direction (Longitudinal)	4.5	-256.25	1,400
Y-direction (Transverse)	94.5	-11,697.47	5,494

In summary, the design of reinforcement in each direction was required to meet the amount of calculated required reinforcement. Although there were undesignable locations which exceeded the allowable reinforcement, additional reinforcement was required to be added which extended from the beam structure to meet the actual required amount of reinforcement.

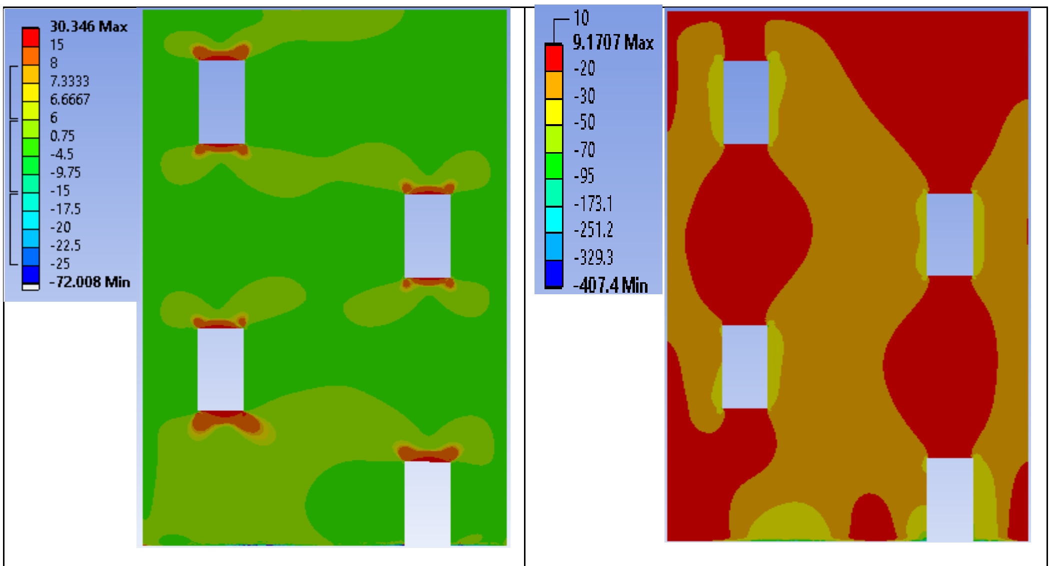
4.4 Stress Evaluation by ANSYS

In this study, 3D analysis performed by ANSYS has shown clearer structural behaviour visually compared to 2D analysis. Figure 4.7 and Figure 4.8 show the maximum and minimum principal stress of Model A-2 and Model B-2 respectively. The colour contours shown in the figure were referred to the entire model including the support members. Therefore, the stresses at the critical locations of the transfer wall were probed to easily identify the stress value.



(a) Maximum Principal Stress	(b) Minimum Principal Stress

Figure 4.7: Model A-2.



(a) Maximum Principal Stress	(b) Minimum Principal Stress

Figure 4.8: Model B-2.

It was found that stress pattern results obtained from ANSYS were quite similar compared to respective Model A-1 and Model B-1 results as shown at section 4.2. However, the stress values obtained in ANSYS were different from SCIA

Engineer due to the difference approach in analysis type. Table 4.2 shows the percentage difference in stress value obtained from both analysis approaches at the critical locations.

Table 4.3: Difference in Stress Value of Model A and Model B.

Stress	**Model A-1 (MPa)**	**Model A-2 (MPa)**	**Percentage Difference (%)**
Maximum Principal Stress (Highest)	11.9	20.5	72.27
Maximum Principal Stress (Lowest)	-21.5	-17.30	19.53
Minimum Principal Stress (Highest)	0.90	0.45	100
Minimum Principal Stress (Lowest)	-84.9	-79.84	5.95
Stress	**Model B-1 (MPa)**	**Model B-2 (MPa)**	**Percentage Difference (%)**
Maximum Principal Stress (Highest)	12.7	30.35	138.9
Maximum Principal Stress (Lowest)	-31.5	-72.01	128.6
Minimum Principal Stress (Highest)	1.00	9.17	817
Minimum Principal Stress (Lowest)	-157.8	-167	5.8

It was observed that the results from each analysis had a high percentage difference in stress value. It was mainly attributed to differences in assumption and simplification, mesh density, the element type as well as the boundary conditions. In general, the 2D analysis assumed that the structure was infinitely thin or the loads were only applied in one plane whereas 3D analysis considered full geometry of the structure and captured the load in all directions. This factor has resulted in significant differences in stress value. In terms of mesh density, 2D analysis had coarser mesh compared to 3D analysis which led to inaccurate stress computation especially at location of high stress gradient. Besides, the boundary conditions had a significant impact on the stress value such that the boundary condition only applied in single plane for 2D analysis whereas the boundary condition for 3D was more realistic as it was applied on the entire structure geometry. Based on the comparison, the 3D analysis result was preferred as the suggested length-to-thickness ratio for 2D analysis ranged from 10-20 (Zienkiewicz et al., 2005). The thickness of the transfer wall was considered too thick to be adopting 2D analysis where the length-to-thickness (w/d) ratio was approximately 29.1.

By comparing 2D and 3D results of Model A, it was verified that the highest tensile stress was found to be located above the top opening, whereas the highest compressive stress was located near to the support region. Besides, the compression strut region and tensile tie region can be clearly seen from the stress trajectories pattern shown in Figure 4.8. The stress concentration at the opening's corner was clearly presented where cracking is most likely to happen.

As for Model B, the stress distribution pattern shown in Figure 4.9 was quite similar compared to the 2D analysis result. It was observed that high concentration of compressive stress was located at the right support region as it was due to the presence of bottom opening that converged the stress path. Besides, it was found that the tensile stress experienced on top and bottom edge of each opening shown in Figure 4.9 were quite high compared to Figure 4.2 shown at above subsection 4.2. The tensile stress had affected the durability of the structure especially at the opening's corner as the concrete behaves poorly when subjected to tension. Prior to the stress pattern, the reinforcement proposed should have the capability to enhance the structural integrity of the transfer wall especially at the critical locations.

Based on the comparison done by both types of analysis, it was concluded that the results of stress and deformation patterns were quite similar. Model B was relatively geometrical complex compared to Model A; therefore, 3D analysis will be preferred to be used for higher accuracy results. The result obtained from 3D was more accurate theoretically compared to 2D, but the derivation of reinforcement design was done by 2D analysis from SCIA Engineer. Therefore, nonlinear analysis of reinforced concrete in 3D analysis was required to be conducted to study the behaviour of the transfer wall and the sufficiency of the proposed reinforcement.

4.5 Nonlinear Analysis of Reinforced Transfer Wall

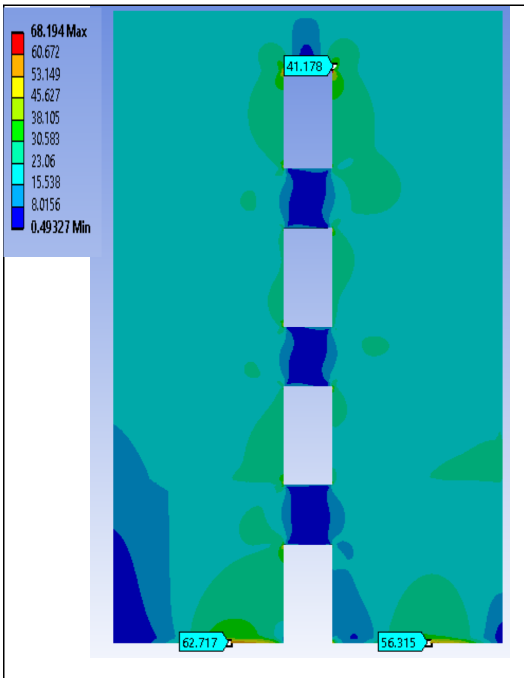
The nonlinear analysis was done to evaluate whether the applied reinforcement was sufficient to achieve the bearing capacity. It has shown improvement of stress capacity and stress behaviour when the reinforcement was included. In short, the overall behaviour of the reinforced concrete transfer wall model was re-evaluated.

Based on the nonlinear analysis, the stress trajectories or distribution pattern remained for both models. It was observed that the overall stress has reduced significantly as some stress has been transferred to the reinforcement by their interaction. Therefore, it was concluded that the reinforcement has improved the stress bearing capacity of the structure. The stress distribution of this nonlinear analysis was represented by Von Mises stress contour of concrete and the reinforcement.

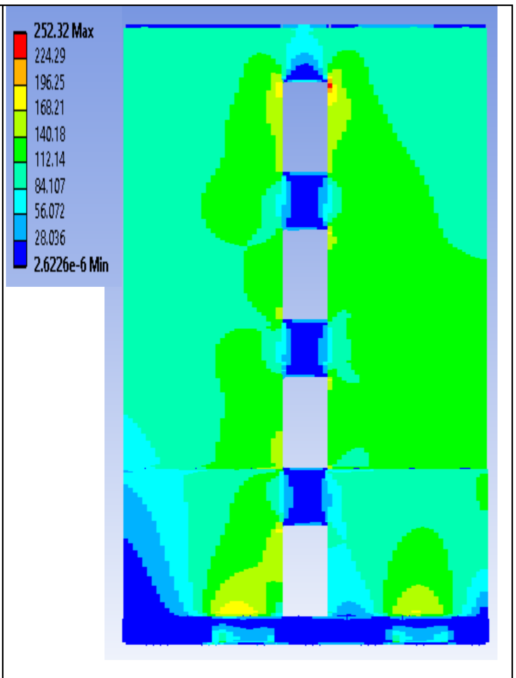
4.5.1 Overall Behaviour of Model A-RC

Figure 4.9(a) shows the Von Mises stress of concrete of Model A-RC, it was observed that the tensile stress located at the opening's corner was highly concentrated. The formation of tensile cracking may appear at this region as the tensile stress of 41 Mpa has exceeded the concrete tensile capacity. In such a situation, it was proven that the diagonal reinforcement provided has taken the tensile stress when the yielding of the concrete started. Besides, high compressive stress was also concentrated at the support region near to the columns which was caused by the convergence of compressive stress from top structure. Therefore, brittle failure of concrete in crushing may occur as the uniaxial compressive stress of concrete has been exceeded. It has shown that the highest Von Mises stress experienced at the support region was 68.19Mpa which has still exceeded the concrete strength (G50/60) as some of the stresses have been distributed to the reinforcement. Although the compressive stress experienced by the concrete has reduced by 14.6% from 79.844 Mpa to 68.19 Mpa as shown in Figure 4.7(b), the capacity of the reinforcement to resist compression stress was still insufficient.

Figure 4.9 (b) shows the Von Mises stress contour of the reinforcement. It was observed that the highest stress occurred at the corner of the top opening which resonates with the tensile stress experienced by the concrete. Besides, it was clearly noted that the stress path of the reinforcement and the concrete behaved similarly as the reinforcement was responsible to counteract the stress induced by the compressive strut and tensile tie. By considering the safety factor of the steel rebar of 1.15 based on Eurocode, the occurrence of rebar yielding would not happen as the stress was still within the stress capacity. The highest von mises stress experienced by the reinforcement was 252.32 Mpa which is lesser than the yield strength of 500 Mpa. Whereas the highest Von Mises stress of the reinforcement at the support region was within 168.21 Mpa to 196.25 Mpa due to the limitation of reinforcement in resisting compressive stress.



(a) Von Mises Stress Contour of Concrete



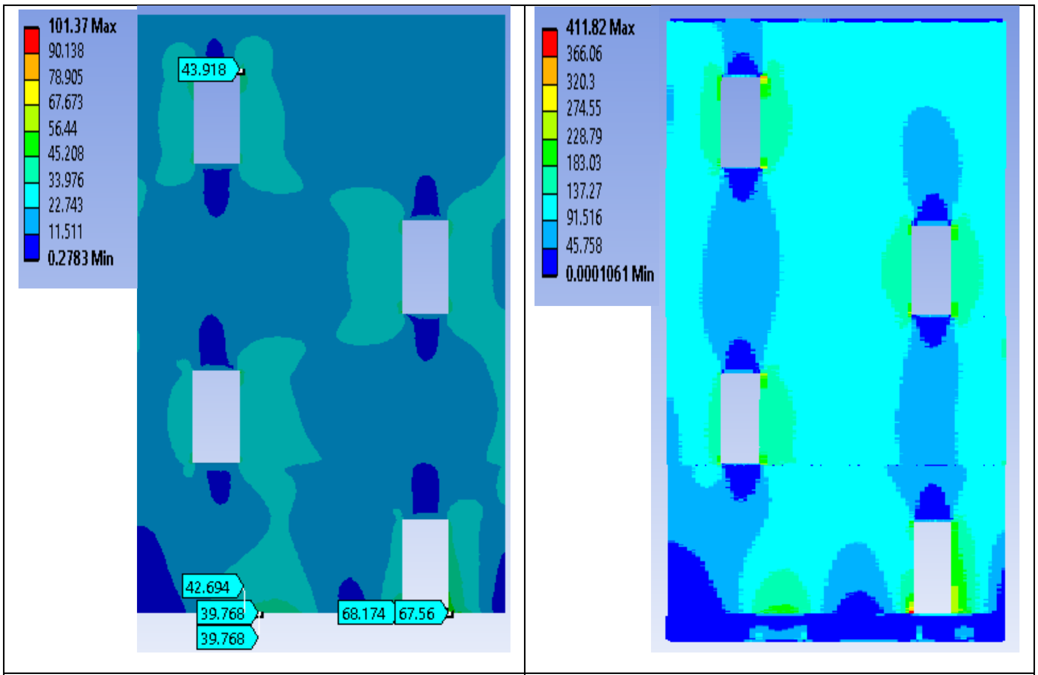
(b) Von Mises Stress Contour of Reinforcement

Figure 4.9: Model A-RC.

4.5.2 Overall Behaviour of Model B-RC

Figure 4.10 (a) shows the Von Mises stress of concrete of Model B-RC. It was found that the tensile stress was highly concentrated at opening's corners where the same phenomenon was also observed in Model A-RC. Moreover, high compressive stress up to 68.174 Mpa was concentrated at the right support region which may lead to compressive crushing failure as the strength of the concrete has been exceeded. The compressive stress at left support was within the allowable compressive stress limit of the concrete.

Figure 4.10(b) shows the Von Mises stress contour of the reinforcement. It was observed that the highest stress experienced by the rebar was 411.82 Mpa which was higher compared to Model A-RC. The overall tensile stress experienced by the reinforcement was higher compared to Model A-RC. The development of the tensile ties from high tensile stress has caused a width increment of the compressive strut. Therefore, it led to smoother overall stress distribution throughout the transfer wall section (Popescu et al., 2015).



(a) Von Mises Stress Contour of Concrete	(b) Von Mises Stress Contour of Reinforcement

Figure 4.10: Model B-RC.

4.6 Summary

In summary, the modelling of the transfer wall by using 2D or 3D finite element method was capable of presenting the stress analysis. However, 3D analysis would be preferred to be adopted for geometrically complex structural models as it produces more accurate and realistic results. The study of stress behaviour of the transfer wall has been presented in the form of stress contour for easier understanding and visualisation. The stress trajectories or deformation pattern could identify the location with high stress concentration. Besides, the design reinforcement of the transfer wall model was simplified with the assistance of contour as it resembled the stress pattern of the structure. In terms of stress distribution, Model B (staggered openings) has overall less stress path disruption compared to Model A (vertical aligned openings) due to smoother stress transfer. In terms of critical location, high tensile and compressive stress were experienced at the corners or surrounding the opening and the support region. Although the reinforcement has reduced the bearing stress of the concrete, it should be sufficient to prevent the failure of the structure in tensile cracking or compression crushing.

CHAPTER 5

RESULTS AND DISCUSSION

5.1 Conclusion

In conclusion, finite element analysis was completed using SCIA Engineer and ANSYS with the objective to study the stress behaviour of transfer wall with openings with vertical aligned and staggered openings. Besides, effective reinforcement detailing was designed to reinforce the transfer wall model which was derived by linear finite element analysis. Lastly, the reinforced transfer wall model was analysed by nonlinear finite element method by ANSYS to identify the stress at the critical region of the transfer wall. All the objectives of study were accomplished throughout the numerical analysis modelling approach. The key findings of this study corresponding to the objectives are stated as below:

i. the transfer wall with openings were modelled by SCIA Engineer and ANSYS software. The types of analysis used were 2D analysis by SCIA Engineer and 3D analysis by ANSYS respectively. These numerical models were modelled according to the modelling methodology presented in Chapter 3 which covered the assignment of material properties, dimension of the models, boundary conditions, loading definition and meshing assignment. The numerical models were analysed and compared. Besides, the modelling of reinforcement as 1D line body embedded in the 3D model was conducted by Ansys to further analyse the stress behaviour and critical region of the entire model.

ii. The transfer wall experienced the highest concentration of stress at its support region and opening corners. To transfer loading from top to bottom structure, stress paths must converge to the supporting beam and column structure. The compressive punching from the supporting column and beam has caused stress concentration at the supporting region. The stress trajectories of Model A show that the transfer wall functions as a compressive arch where the load was transferred down through spaces without openings. The presence of opening has led to stress concentration at the corner in tension due to the disruption of stress path transition. The load path of Model B was transferred along the section without an opening with less disruption, but the opening's corner experiences similarly high stress concentration. The right support section had higher stress concentration due to the location of the bottom-most opening on top of the beam and column structure that converged the stress.

iii. The reinforcement design of transfer wall models was based on the stress experienced at specific locations. Longitudinal reinforcement was required at critical locations with high tensile stress, while transverse reinforcement is concentrated at the support region to sustain high compressive stress. Diagonal reinforcement was also provided at each opening corner to prevent cracking from high stress concentration. The reinforcement should not be over-reinforced to preserve the durability of the entire reinforced structure. The stress paths of the reinforcement and concrete behaved similarly, with the reinforcement counteracting compressive strut and tensile tie stress. Model B-RC had higher overall tensile stress and experienced higher stress in the reinforcement. The development of tensile ties caused an increment in compressive strut width, leading to a smoother stress distribution throughout the transfer wall section.

5.2 Recommendations

The research on stress investigation in reinforced concrete transfer walls is relatively rare at this moment. It has a large room of improvement and modifications to be done to further enhance the structural analysis by finite element analysis. The following are some recommendations for deeper study:

i. Scale down laboratory structural models can be conducted to verify the result of the study. As the transfer wall specimen model is considered very large and costly to be conducted. Most of the researchers has conducted shear wall finite element analysis but not transfer wall. Therefore, experiments could be conducted in a smaller scale to validate the reliability of this structural model as it is vastly used in structural engineers nowadays.

ii. The boundary conditions of the finite element model can be further analysed. The type of support condition such as partially restrained or flexible support could be studied. Besides, lateral load or transverse load such as wind load could be included to perform a more realistic and practical model. However, there is a significant lack of published research on the type of boundary conditions especially for transfer walls. Therefore, conducting a study on this topic would be very valuable.

REFERENCE

Allam, S.M., Shoukry, M.S., Rashad, G.E. and Hassan, A.S., 2013. Evaluation of tension stiffening effect on the crack width calculation of flexural RC members. *Alexandria Engineering Journal*, 52(2), pp.163–173. Available at: http://dx.doi.org/10.1016/j.aej.2012.12.005.

Ansys inc., 2010. Element Reference. , 15317(November), p.9. Available at: http://inside.mines.edu/~apetrell/ENME442/Documents/SOLID187.pdf.

Areias, P. et al., 2018. Effective 2D and 3D crack propagation with local mesh refinement and the screened Poisson equation. *Engineering Fracture Mechanics*, 189, pp.339–360. Available at: https://doi.org/10.1016/j.engfracmech.2017.11.017.

Behfarnia, K. and Shirneshan, A., 2017. A numerical study on behavior of CFRP strengthened shear wall with opening. *Computers and Concrete*, 19(2), pp.179–189.

Van Den Boom, S.J., Zhang, J., van Keulen, F. and Aragón, A.M., 2019. A stable interface-enriched formulation for immersed domains with strong enforcement of essential boundary conditions. *International Journal for Numerical Methods in Engineering*, 120(10), pp.1163–1183.

Borbory, E., 2020. Openings effects in reinforced concrete shear walls; a literature review on experimental and finite element studies. , (2), pp.1–6.

Brischetto, S., Tornabene, F., Fantuzzi, N. and Bacciocchi, M., 2017. Interpretation of boundary conditions in the analytical and numerical shell solutions for mode analysis of multilayered structures. *International Journal of Mechanical Sciences*, 122(December 2016), pp.18–28. Available at: http://dx.doi.org/10.1016/j.ijmecsci.2016.12.017.

Chaudhari, S.. and Chakrabarti, M.., 2012. Modeling of Concrete for Nonlinear Analysis using Finite Element Code ABAQUS. *International Journal of Computer Applications*, 44(7), pp.14–18.

Chaudhary, S.S. and Parekar, S.R., 2019. Stress Distribution of Different Shapes of Opening in Shear Wall. *SSRN Electronic Journal*, 5(August), pp.8–11.

Dere, Y., 2017. Nonlinear FE Modeling of Reinforced Concrete. *International Journal of Structural and Civil Engineering Research*, pp.71–74.

Earij, A., Alfano, G., Cashell, K. and Zhou, X., 2017. Nonlinear three–dimensional finite–element modelling of reinforced–concrete beams: Computational challenges and experimental validation. *Engineering Failure Analysis*, 82(August), pp.92–115. Available at: http://dx.doi.org/10.1016/j.engfailanal.2017.08.025.

Filippou, F.C., 2015. Mechanics and Materials Finite Element Analysis of Reinforced Concrete Structures. , (January 1990).

Gandhi, B.H., 2016. Effect of Opening in Shear Wall. *International Journal For Technological Research In Engineering*, 6890(5), pp.572–577.

Hanif, M.U. et al., 2018. Damage assessment of reinforced concrete structures using a model-based nonlinear approach – A comprehensive review. *Construction and Building Materials*, 192, pp.846–865. Available at: https://doi.org/10.1016/j.conbuildmat.2018.10.115.

Husain, M., Eisa, A.S. and Hegazy, M.M., 2019. Strengthening of reinforced concrete shear walls with openings using carbon fiber-reinforced polymers. *International Journal of Advanced Structural Engineering*, 11(2), pp.129–150. Available at: https://doi.org/10.1007/s40091-019-0216-6.

Jagota, V., Sethi, A.P.S. and Kumar, K., 2013. Finite element method: An overview. *Walailak Journal of Science and Technology*, 10(1), pp.1–8.

Jain, R., Pal, S.K. and Singh, S.B., 2017. Numerical modeling methodologies for friction stir welding process, *Computational Methods and Production Engineering: Research and Development*, pp.125-169.

Jaseela, C.A. and Pillai, P.R.S., 2017. Optimum Shape of Cut-Out Opening on Concrete Structural Wall. *International Research Journal of Engineering and Technology(IRJET)*, 4(7), pp.881–887. Available at: https://irjet.net/archives/V4/i7/IRJET-V4I7210.pdf.

Metwally, I.M., 2017. Three-dimensional nonlinear finite element analysis of concrete deep beam reinforced with GFRP bars. *HBRC Journal*, 13(1), pp.25–38. Available at: http://dx.doi.org/10.1016/j.hbrcj.2015.02.006.

Mohammed, B.S., Ean, L.W. and Malek, M.A., 2013. One way RC wall panels with openings strengthened with CFRP. *Construction and Building Materials*, 40, pp.575–583. Available at: http://dx.doi.org/10.1016/j.conbuildmat.2012.11.080.

Morsy, A. and Ibrahim, Y., 2019. Parametric Study for Performance of R.C. Wall with Opening using Analytical F.E. Model. *Athens Journal of Technology & Engineering*, 6(1), pp.31–62.

Musmar, M.A., 2013. Analysis of shear wall with openings using solid65 element. *Jordan Journal of Civil Engineering*, 7(2), pp.164–173.

Popescu, C., Sas, G., Blanksvärd, T. and Täljsten, B., 2015. Concrete walls weakened by openings as compression members: A review. *Engineering Structures*, 89, pp.172–190. Available at: http://dx.doi.org/10.1016/j.engstruct.2015.02.006.

Rahnavard, R., Hassanipour, A. and Mounesi, A., 2016. Numerical study on important parameters of composite steel-concrete shear walls. *Journal of Constructional Steel Research*, 121, pp.441–456. Available at: http://dx.doi.org/10.1016/j.jcsr.2016.03.017.

Reddy, J.., 2019. Introduction to the finite element method, *McGraw-Hill Education*.

Romans, M., 2010. Design of walls with linear elastic finite element methods. *Technische Universiteit Delft*

Sabau, C. et al., 2019. Local and global behavior of walls with cut-out openings in multi-story reinforced concrete buildings. *Engineering Structures*, 187(February), pp.57–72. Available at: https://doi.org/10.1016/j.engstruct.2019.02.046.

Sakurai, M., Nishida, T. and Kobayashi, J., 2017. Effect of Opening Size on Compressive Strut of Rc Shear Walls With Openings Based on Fem Analysis.

SCIA Engineer., 2015. Topic Training – Finite Element Method. , p.74.

Senthil, K., Gupta, A. and Singh, S.P., 2018. Computation of stress-deformation of deep beam with openings using finite element method. *Advances in Concrete Construction*, 6(3), pp.245–268.

Shamass, R., Zhou, X. and Alfano, G., 2015. Finite-Element Analysis of Shear-Off Failure of Keyed Dry Joints in Precast Concrete Segmental Bridges. *Journal of Bridge Engineering*, 20(6), p.04014084.

Sharaf, T., El, M., Badr, G. &s and Lotfy, E., 2018. Soil-Structure Interaction of Tall Buildings Comprising Shear Walls with Openings Under Seismic Loads. *Electronic Journal of Structural Engineering*, 18(2), p.2018.

Vitali, O.P.M., Celestino, T.B. and Bobet, A., 2018. 3D finite element modelling optimization for deep tunnels with material nonlinearity. *Underground Space (China)*, 3(2), pp.125–139. Available at: https://doi.org/10.1016/j.undsp.2017.11.002.

Wai, C.M., Rivai, A. and Bapokutty, O., 2013. Modelling optimization involving different types of elements in finite element analysis. *IOP Conference Series: Materials Science and Engineering*, 50(1).

Wang, G. et al., 2020. A stress-path-independent damage variable for concrete under multiaxial stress conditions. *International Journal of Solids and Structures*, 206, pp.59–74. Available at: https://doi.org/10.1016/j.ijsolstr.2020.09.012.

Wang, Z. et al., 2017. 3-D local mesh refinement XFEM with variable-node hexahedron elements for extraction of stress intensity factors of straight and curved planar cracks. *Computer Methods in Applied Mechanics and Engineering*, 313, pp.375–405. Available at: http://dx.doi.org/10.1016/j.cma.2016.10.011.

Wu, H. and Li, B., 2003. Investigating the load paths of RC shear wall with openings under reversed cyclic loadings.

APPENDICES

Appendix A: Detailed Drawings

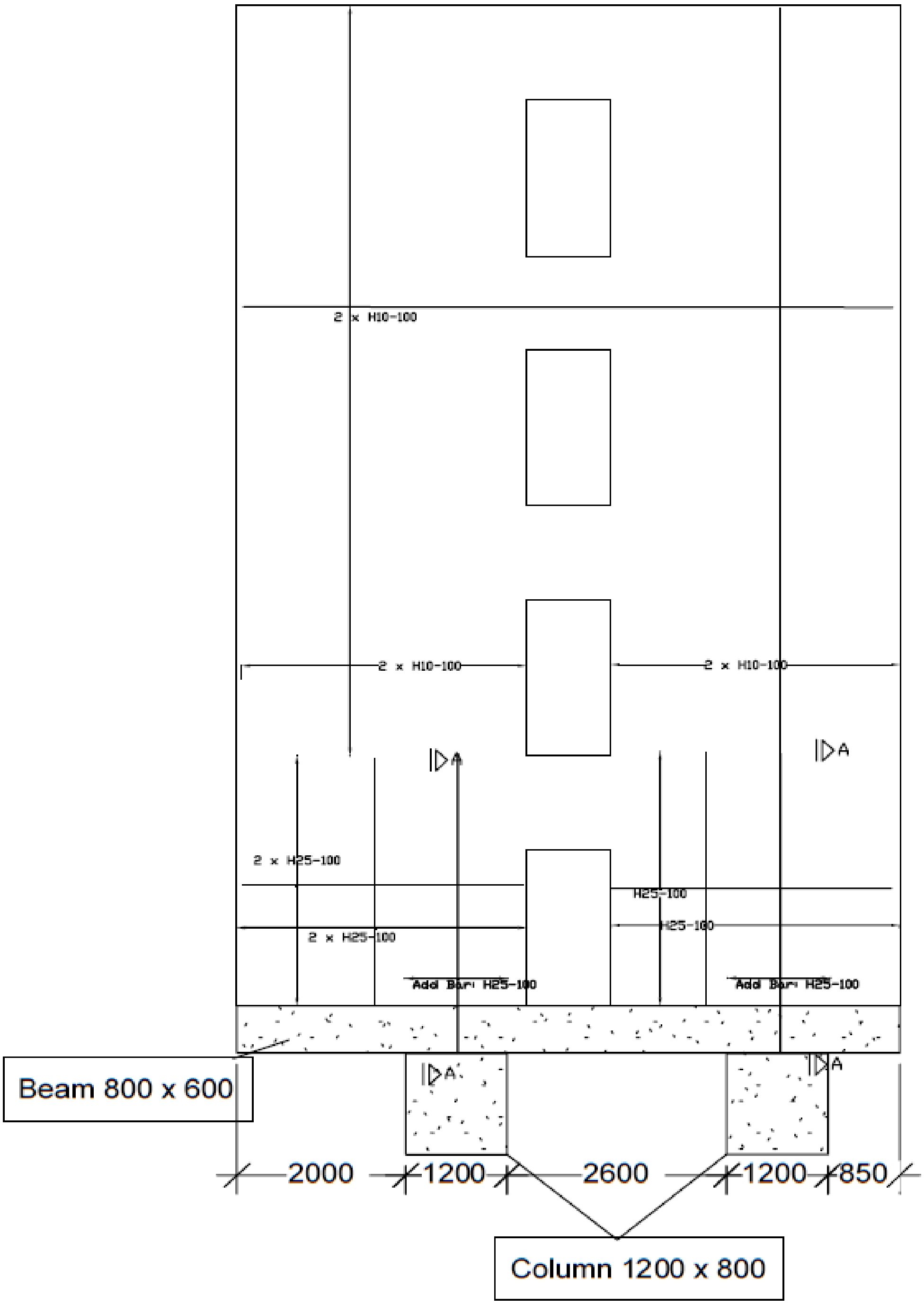


Figure A-1: Detailed Reinforcement of Model A

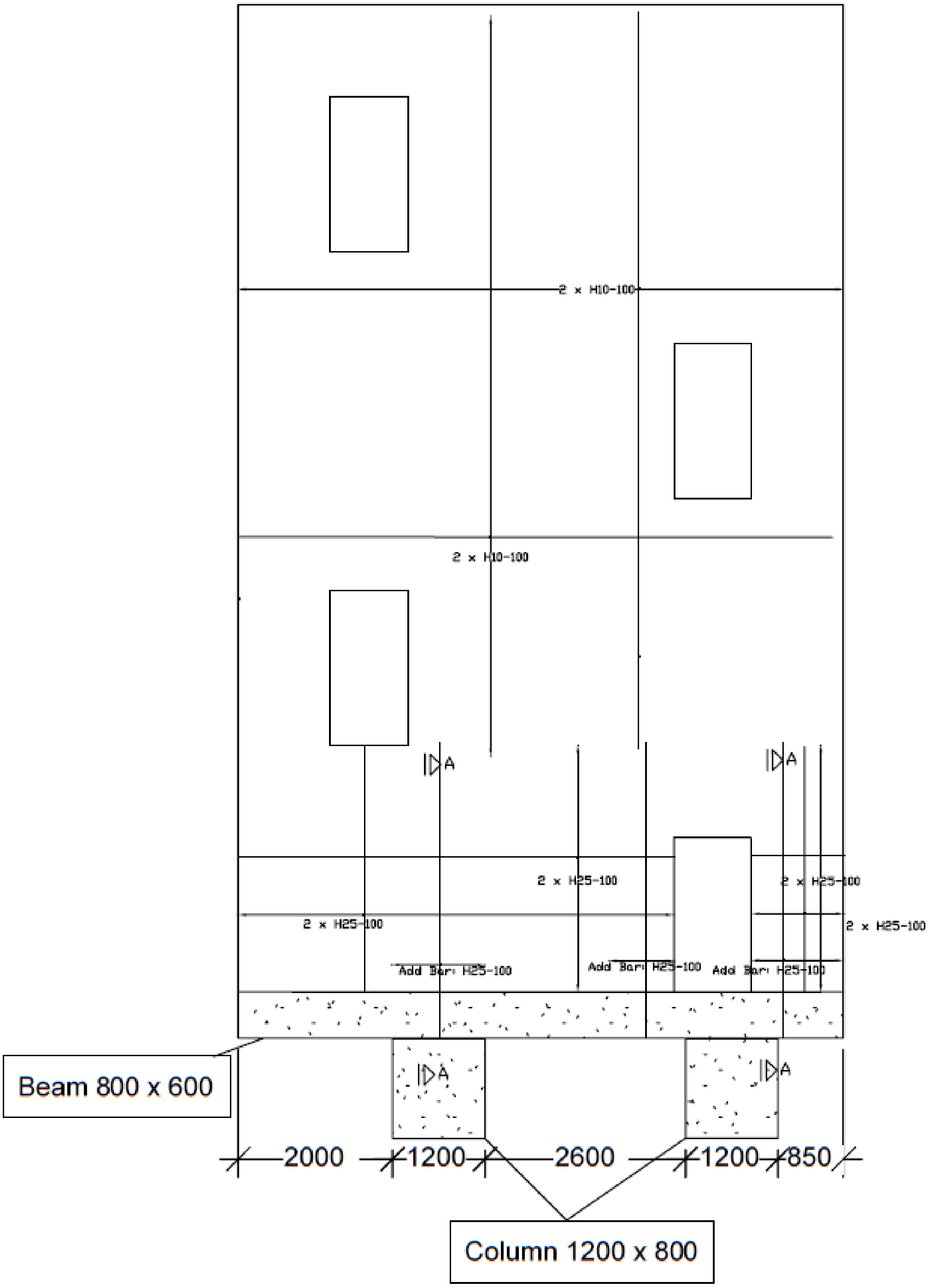


Figure A-2: Detailed Reinforcement of Model B

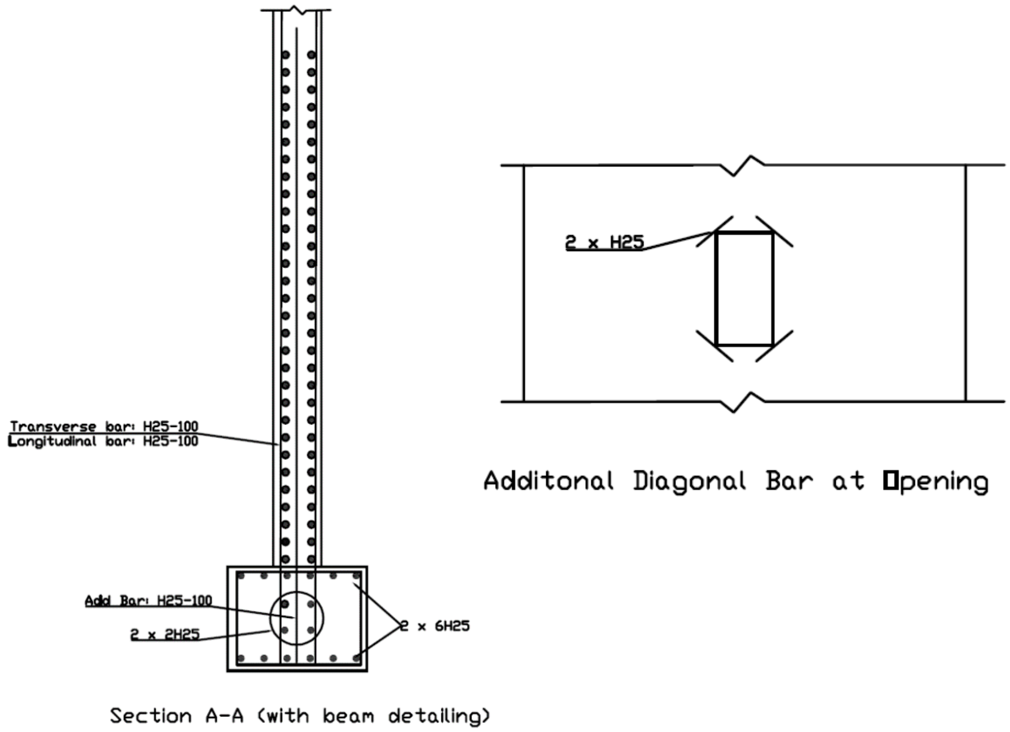


Figure A-3: Typical Reinforcement for Section View and Opening