Project report in TMR 4320

Simulation-Based Design

Yaolin GE

November 20, 2018

DEPARTMENT OF MARINE TECHNOLOGY

NORWEGIAN UNIVERSITY OF SCIENCE AND TCHNOLOGY

Norwegian University of Science and Technology

Preface

This report is for the project in the course TMR 4320 Simulation-Based Design in the Fall of 2018 at the Norwegian University of Science and Technology (NTNU). The stated goals of this project are to learn to identify, delimit, formulate and solve simulation-related design problems relevant for the marine industry. Additionally, I aim to synthesize the theories and methods learned in the course to understand and reflect on them through application to this specific project for designing a point-loaded balcony which represents very classical structural units of a ship. Through the entire project, a couple of computer-aided software packages were employed to design, analyze, optimize and visualize the object.

I would like to thank my Professor, Ekaterina Kim and Student Assistants, Emil Severin Moen Ramsvik, Andreas Breivik Ormevik, Erlend Røilid Vollan, and Bendik Folden Nyttingnes for their assistance and support throughout the project. Some questions were also consulted and clarified by Prof. Kiendl and Prof. Erikstad. Also a great thanks to them. Eventually, I would like to thank my girlfriend who provides me tons of useful practical information on shipbuilding since she works at a modern shipyard.

By my signature below, I hereby certify that all work was conducted independently in terms of writing the report, coding the necessary computation packages and carrying out essential software verifications.

Signature: Yaolin Ge

Place: Trondheim, Norway

Date: 20-NOV-2018

Summary

This project deals with the process of designing a point-loaded balcony made from plate reinforced by stiffeners and girders. The balcony is connected to the cabins with two transverse beams while its plate is stiffened longitudinally.

Basic assumptions were made in order to simplify the mathematical model and support its corresponding finite element analysis. They are listed as follows:

1) The side close to the cabin is clamped on all degrees of freedom, while the remaining edges are free
2) The vertical displacement is relatively small compared to the other two dimensions in terms of length and width, then the deflection would be linearly proportional to the applied load and linear theory can be used to analyze the linear behavior of the structures, which is much easier than nonlinearity
3) This balcony design is uncoupled with the global design to the entire ship
4) The water pressure is assumed to be an average height of distributed pressure load, and the point load was assumed to be located at the most dangerous location
5) All materials are assumed to be the isotropic ASTM-A36 pure iron steel
6) Weather conditions, especially in terms of temperature will not affect the material strength too much.
7) Ice may not cause damage effect on this balcony
8) Some specific profile stiffeners or girders may be manufactured by the shipyard itself

Hand calculations give the maximum deflection and stress of the plate based on the elementary beam theory, which shows that the maximum displacement of the girder is 30*mm,* while it is 7*mm* in the stiffener. And the corresponding maximum stress is 230*MPa* and 130*MPa* respectively, which is slightly exceeding the requirement, then the optimization is necessary.

Finite element model demonstrates the validity of the simplified mathematical model, also proves a little bit on the model that was used in the hand calculations. FEM results present the maximum stress in the plate is 238*Mpa* and maximum displacement is 41*mm,* which shows a good correlation with the beam model. Through the mesh convergence study, the mesh size of 0.15625 was selected because of its decent accuracy and cheap computational efforts.

Particle swarm optimization gives the local optimal design (probably global optimal) with the length of 10*m*, width 3.285*m*, which are fixed anyway, then 9 150×25 stiffeners, and two 250×250×10 T-girders, which are located at the symmetric location 2.15*m* far away from the free edge. The maximum deflection then becomes 44*mm* while maximum stress is reduced to 196*MPa.*

Table of Contents

1 Introduction

The background information for this project is based on the course TMR4320 Simulation-Based design. This report is aimed to solve a structural design problem based on a simulation model and conduct a finite element analysis to determine the behavior of the structure. The goal of this project is to create a complete procedure to solve real-life structural problems on the basis of theory enhanced by the development of computer software packages. For this project, I aim to propose specific methods, theories and logistics for an optimal stiffened-plate design to have the capability of performing tasks well with minimum total weight corresponding to minimum total cost.

1.1 Task description

The major procedures of the project will be conducted as three main steps:

Step 1: Develop an initial design based on DNV and CCS rules and regulations

Step 2: Conduct a FEA study to evaluate the strength

Step 3: Optimize the structure for minimal weight

1.2 Planned approach

In order to capture the main purpose of the project and also to sharpen the understanding of principles and physics of the supporting theories, researches were done on the relevant theories, which will be discussed in detail in chapter 2. To build a geometric model and a technical drawing, Inventor was applied to do this task. To step further into the analysis part, essential software skills were learnt as prerequisites which have relevance in employing analysis software packages such as Mechanical APDL and Workbench. To optimize, codes were developed for parametric model as well as optimization model in MATLAB based on the knowledge from this course and some previous experience also.

2 Develop the initial design

2.1 Select parent vessel and desired materials

In order to apply rules and regulations to this specific balcony design, the expected parent ship needs to be selected so as to maintain consistence. Here, MS Symphony of the seas which is the world's largest cruise vessel at present has been chosen, technical details are listed as below.

Table 1 Technniccal configuratin of the vessel[1]

Ship	Length	Beam(max)	Beam(sealine)	Height	Draught	Speed
MS Symphony of the Seas	361 m	66 m	47.45 m	72.5 m	9.32 m	22 kn

For balcony design, the main material is steel, and in order to simplify the case, ASTM-A36 steel was used, more specification information of this type of steel is shown in the table below.

Table 2 Properties of ASTM-A36 steel[2]

Steel number	Density	Young's modulus	Possion's ratio	Yield strength	Ultimate tensile strength
ASTM-A36	7850 kg/m^3	207 Gpa	0.3	250 Mpa	400~550 Mpa

2.2 Determine the initial dimensions

Regarding to the DNV rules, some requirements can be found to determine the expected parameters and properties for this specific case. The focus of this project is to determine the optimal combination of different elements, layouts and so on. The following section will establish the minimum requirements for each component.

2.2.1 Determine initial minimum thickness

Referring to DNV structural ship design rules, the net thickness of the plate should satisfy the minimum thickness threshold, expressed as below,

$$t = a + bL_2\sqrt{k}$$

Where, a and b are balance coefficients

L_2 is the length of the vessel

k is the material factor

Therefore, for each of those components, they can be computed as in the table below:

Table 3 Minimum thickness based on DNV rules[3]

Component	a	b	Minimum thickness[m]
Plate	0.0045	0.01	0.00811
Girder	0.0045	0.012	0.00883
Stiffener	0.0045	0.009	0.00775

2.2.2 Determine the section properties

The standard formula for section properties can be expressed as:

$$Z = \frac{f_u \left| P \right| s l_{bdg}^2}{f_{bdg} C_S R_{eH}}$$

Where, f_u is the factor for unsymmetrical profiles, here it is equal to 1.0 since I use flat bar which has a symmetrical profile along its middle axis, P is the design pressure needs to be considered in compatible in this specific case, which is the water pressure. s is the gap between each two stiffener or each two girder respectively, and l_{bdg} is the effective bending spam while f_{bdg} is the bending moment distribution factor. C_s is the permissible stress distribution, 0.75 in this case. Finally, R_{eH} is the yield strength of the material. Therefore, two sections can be computed separately by using their own properties, which is shown as below:

$$Z_{Girder} = \frac{1 \times \left| 10.055 \right| \times 5 \times 10^2}{0.5 \times 0.75 \times 250} = 53.6cm^3$$

$$Z_{Stiffener} = \frac{1.1 \times \left| 10.055 \right| \times 0.657 \times 3.285^2}{0.5 \times 0.75 \times 250} = 0.84cm^3$$

2.2.3 Dimension selection

The length and width of the plate is fixed for this case, which can be derived from the formula in the lectures:

$$L = month$$

$$W = \frac{10}{ln(day + 1)}$$

Where month and day come from my birthday date, which is 10 and 20 respectively. Therefore, length and width were set to 10m and 3.285m respectively. Then, from the above requirements, the initial design dimension was chosen basen on DNV rules, the main dimensions are shown as below:

Table 4 Parameter value for initial design

Component	Parameter	Dimension, [m]
	Length	10
Plate	Width	3.285
	Thickness	0.015
	Height	0.45
	Width	0.35
T profile girder	Thickness of web	0.015
	Thickness of flange	0.015
	Gap	5
	Number	6
Stiffener	Height	0.2
	Thickness	0.01

Gap	0.547.5

2.3 Model formulation

2.3.1 Basic assumptions

To idealise the situation, some assumptions should be established first in order to support the idealization:

1) The side close to the cabin is clamped on all degrees of freedom, while the remaining edges are free.
2) The vertical displacement is relatively small compared to the other two dimensions in terms of length and width, then the deflection would be linearly proportional to the applied load and linear theory can be used to analyse the linear bahaviour of the structures, which is much easier than nonlinearity
3) This balcony design is uncoupled with the global design to the entire ship
4) The water pressure is assumed to be a average height of ditributed pressure load, and the point load was assumed to be located at the most dangerous location
5) All materials are assumed to be isotropic ASTM-A36 pure iron steel, which behaves linear-elasticity well in this case.
6) Weather conditions especially in terms of temperature will not affect the material strength too much.
7) Ice may not cause damage effect on this balcony
8) Some specific profile stiffeners or girders may be manufactured by the shipyard itself

2.3.2 Visualize the geometric CAD model

From the above derived dimensions and assumptions, the computer-aided design model can be developed as shown in the picture below, it is worth noting that it is a bottom view so as to illustrate the locations and properties of stiffeners and girders. Of course, in reality, people cannot stand on this side. The more detailed technical drawing view is discussed in the following section.

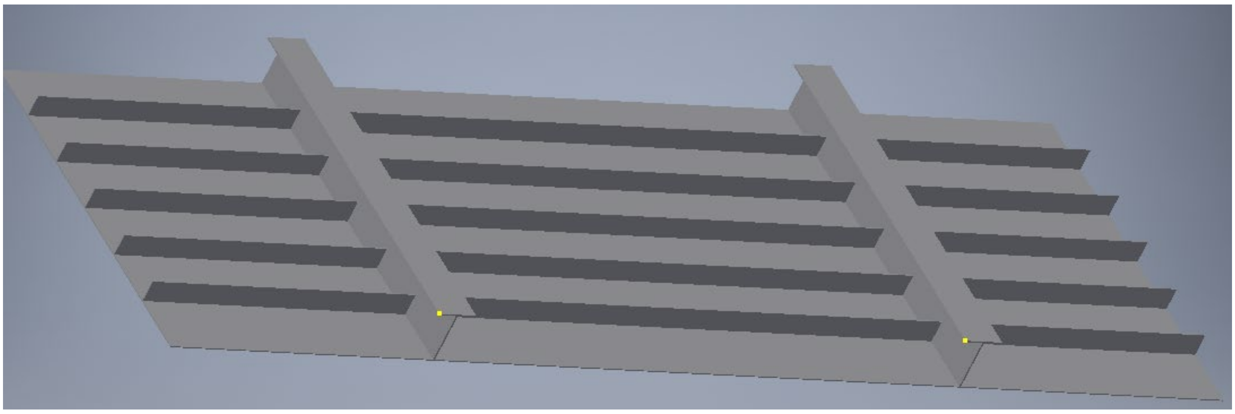


Figure 1 CAD model for representation

2.4 Sketch the technical drawing

Technical drawing is quite an important prerequisite file when it comes to manufacturing. It can interpret the willing of the designers to the manufacturers' product, and it has the capability to interact with different sectors without any misunderstandings. As for this specific case, the technical drawing for initial design was developed and detailed view for the type of stiffeners and girders were also sketched as seen in the graph below. It is sketched on the basis of GB/T14689-93[4] and GB/T 18229-2000[5].

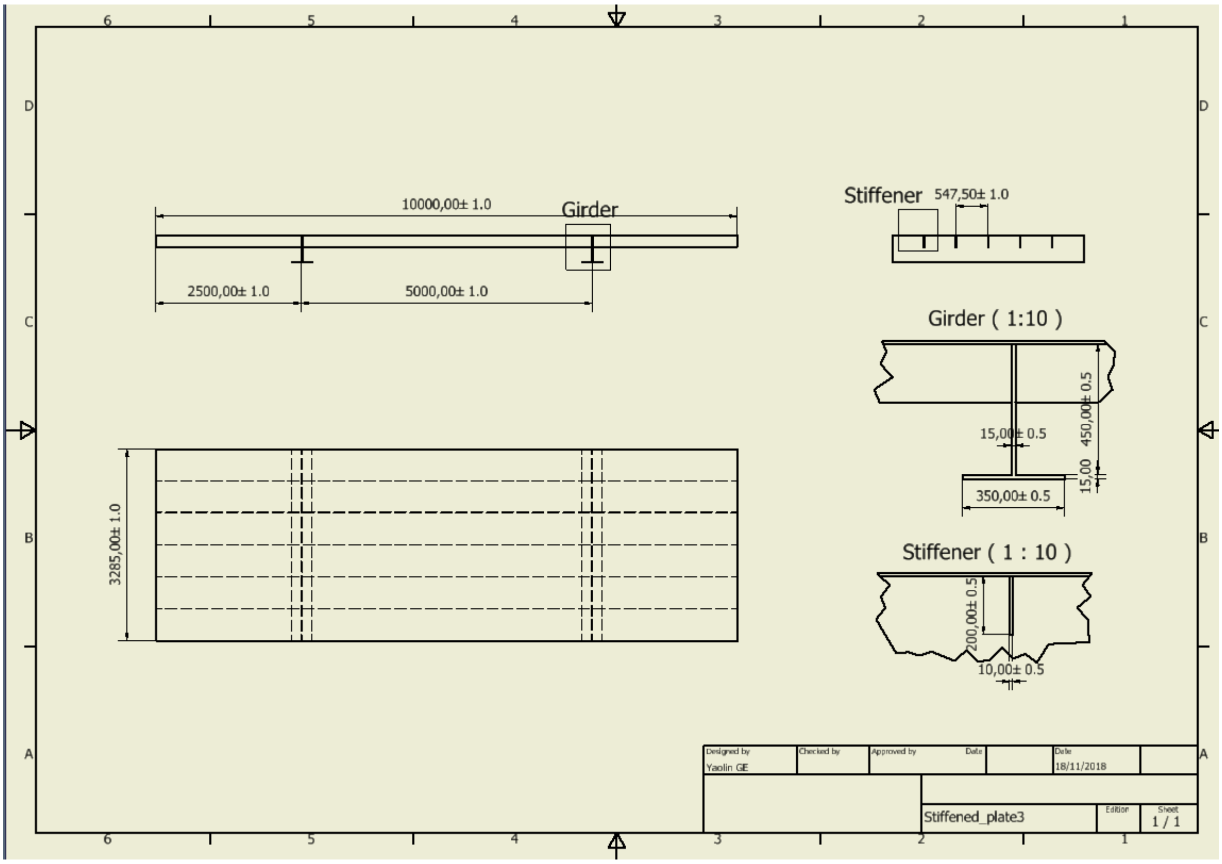


Figure 2 Technical drawing of initial design

2.5 Hand calculation and verification

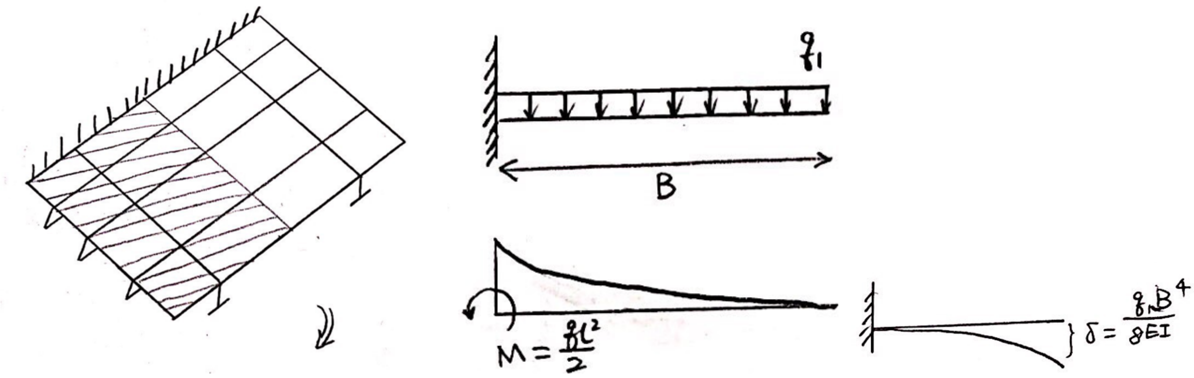
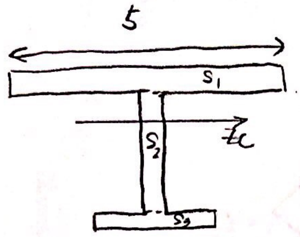


Figure 3 strip theory in the transversal direction

The above figure shows how the stiffened plate can be simplified as a beam in the transversal direction and how pressure loading can be specified onto the idealized beam. As this stage, the pointed load was not applied to any of the above position, since the dangerous location was still hidden in this unsolved problem. By only considering the pressure load, the hand calculation results can be computed respectively. The water pressure load is calculated from the formula below:

$$p = \rho g h$$

Where, ρ is the sea water density, known as 1025 kg/m^3, g is the gravity acceleration, also set as 9.81m/s^2, and h is the water height, which is required to be set as 1 meter, then the pressure becomes 100550*N/m²*, or 100550 *Pa*. Then for the simplified beam, the distribution loading should be the pressure times the effective length that the pressure loading acts on, it should be *100550*L/2,* which is 502750 *Pa.*



$$S_1 = 5 \times 0.015 = 0.075\ m^2$$

$$S_2 = 0.45 \times 0.015 = 0.00675\ m^2$$

$$S_3 = 0.35 \times 0.015 = 0.00525\ m^2$$

$$Z_c = \frac{S_1 \cdot (0.015 + 0.45 + \frac{0.015}{2}) + S_2 \cdot (0.015 + \frac{0.45}{2}) + S_3 \cdot \frac{0.015}{2}}{S_1 + S_2 + S_3}$$

$$= 0.426$$

$$I_{total} = \frac{0.35 \times 0.015^3}{12} + S_3 \cdot (0.426 - 0.015/2)^2 + \frac{0.015 \times 0.45^3}{12} + S_2 \cdot (0.426 - 0.015 + \frac{0.45}{2})^2$$

$$+ \frac{5 \times 0.015^3}{12} + S_1 \cdot (0.015 + 0.45 + 0.015/2 - 0.426)^2$$

$$= 0.0124\ m^4$$

Figure 4 moment of inertia of the crossectional area of the simplified beam

The moment of inertia was calculated in the above draft, and then the rest of those unknowns can also be computed:

$$\delta = \frac{502750 \times 3.285^4}{207 \times 10^9 \times 0.0124} = 30mm$$

$$M = \frac{502750 \times 3.285}{2} = 165152Nm$$

$$\sigma_{max} = \frac{MI_z}{z_{max}} = \frac{825760 \times 0.0124}{0.426} = 240MPa$$

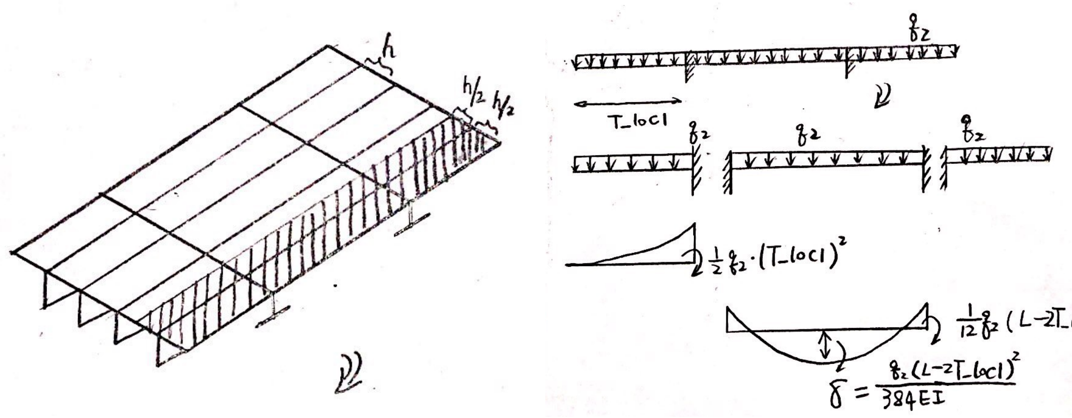


Figure 5 Stripe theory inf the longitudinal direction

In the similar way, the structural cam also be simplified as a longitudinal beam whose detail subdivision is shown in the above figure. The moment of inertia and the rest of those unknown can be computed for the longitudinal beam as well. Also some previous assumptions were implied implicitly. However, for this case, ths only additional assumption is that the stiffness of the girder is relatively huge compared to the stiffeners, hence, it can be idealized to the fixed support for the longitudinal beam which is illustrated before. The distributed load for this case is equal to 55050 *Pa*.

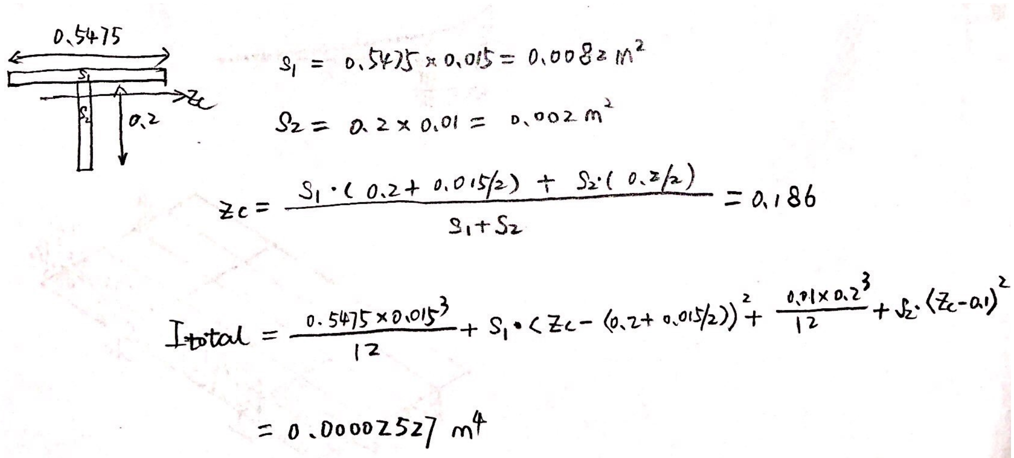


Figure 6 moment of inertia of the crossectional area of the simplified beam

After the moment of inertia has been derived, the rest of those unknowns can be calculated as follows:

$$\delta = \frac{55050\times(10-2\times2.5)^2}{384\times207\times10^9\times0.00002527} = 7mm$$

$$M = \frac{55050\times(10-2\times2.5)}{12} = 22937.5Nm$$

$$\sigma_{max} = \frac{MI_z}{z_{max}} = \frac{22937.5\times0.02527}{0.186} = 130MPa$$

2.6 Result discussion

Since the entire stiffened plate was extremely simplified, and only a few limited points were chosen as the unknown probers. Therefore, the process of hand calculations only gives the general feeling about how to solve the real-world problem in an analytical way. The real-world problems are really complicated and most of them may not be solved exactly using the analytical methods. Therefore, those genius engineers came up with some good ideas which can be utilized to solve almost every structural problem in a numerical way, but of course at the cost of accuracy, but at least it really gave an engineering-friendly results, the following section will discuss more details.

3 Finite element model analysis

Regarding to numerical methods, there are thousands of papers that are devoting to this field, here finite element method (FEM) was selected. Finite element method is a numerical method for solving problems of engineering and mathematical physics. Typical problem areas of interest include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. Normally the solution from analytical approach requires partial differential equations which need to be established with considering boundary conditions, which is also known as strong form. The finite element method formulation of the problem results in a system of algebraic equations, and by using principle of virtual work and principle of virtual displacement can alter the direction to an easier way of solving complex structural problems. The method yields approximated values of the unknowns in terms of displacement at discrete number of nodes over the domain. To solve the problem, it has three major procedures:

Step 1: Preprocessing phase: create and discretize the problem into finite elements, which involves:

- choosing an appropriate element type
- creating a FE mesh with proper mesh density
- discretize the representation of loads and boundary conditions

Step 2: Solution phase: solve a set of linear algebraic equations to obtain nodal results including displacements, temperatures and so on.

Step 3: Postprocessing phase

- Compute strains and stresses from nodal results
- Assess the accuracy of the FE solution

Here, ANSYS workbench was used to solve this problem for the initial design, basic knowledge was disseminated in the lecture notes, only critical procedures are illustrated as follows.

3.1 Conduct a FEA study

Preprocessing is the first step that needs to be done, it contains three typical missions: discretization, meshing and applying loads and boundary conditions.

Discretization is typically regarded as the process to divide the continuous domain into a finite number of subdomains, which might lose the exact smoothly continuous property, but it is still worth doing so since it makes the unsolvable problems into relatively easier problem to solve.

For this project, discretization means to divide the entire stiffened plate into pieces of elements and assemble them together on the basis of finite element principle such as virtual displacement and virtual work. Consequently, in this phase, selection of element types is very crucial to the accuracy of the final result. Since the plate thickness is comparably small to the width and length, shell elements can be potential candidates. By browsing ANSYS content packages, it shows that there are around 11 different shell elements for solving plate problems. only two very common shell elements are discussed into detailed so as compare their advantages and disadvantages. They are shell 181 and shell 281, the following descriptions give the general comparison between those two elements.

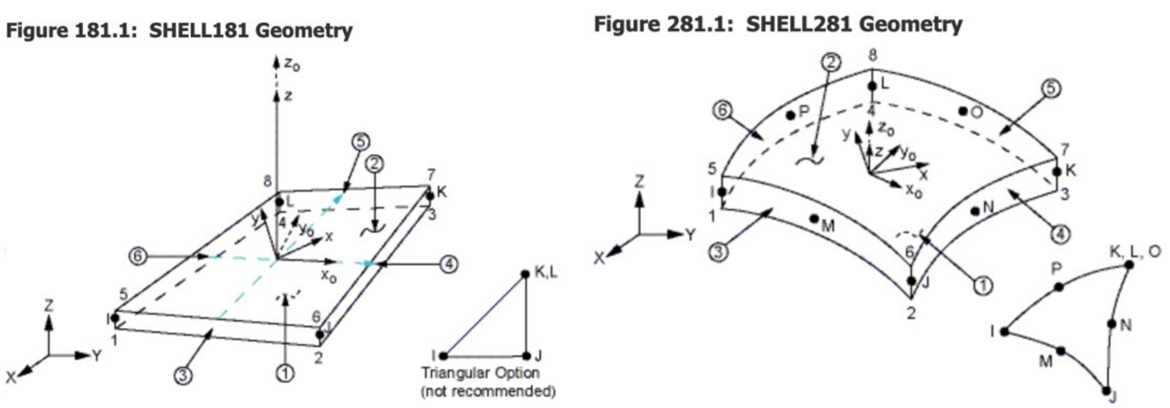


Figure 7 Element type explantation[6]

- They are both suitable for analyzing thin to moderately-thick shell structures.
- Shell 181 is a four-node element while shell 281 is a eight-node element, and all nodes of both elements have six degrees of freedom: translations in the x, y, and z directions, and rotations about the x, y, and z-axes.
- The accuracy of both elements in modeling composite shells is governed by the first-order shear-deformation theory (usually referred to as Mindlin-Reissner shell theory).
- Both element formulation is based on logarithmic strain and true stress measures. The element kinematics allow for finite membrane strains (stretching). However, the curvature changes within a time increment are assumed to be small.

It seems the only difference is the node number, but that is just an illusion. shell 281 was selected as the specific element type for this project, the reason why it was standing out will be discussed after the description of singularity in the right next following paragraph.

A stress singularity is a point of the mesh where the stress does not converge towards a specific value. As the mesh is refined, the stress at this point keeps increasing and increasing. Theoretically, the stress at the singularity is infinite. Typical situations where stress singularities occur are the appliance of a point load, sharp re-entrant corners, corners of bodies in contact and point restraints. Therefore, stress singularity is a very typical phenomenon in finite element analysis.

Again, come back to the reason why eight-node shell element was selected, it is because higher-order elements can capture a relatively detailed and complex data representation with relatively fewer required number of elements, which can improve the accuracy of the results. Since it can reduce the number of the points located at the corners, by recalling the memory on singularity, it often happens in the sharp corners and so on, accordingly, using fewer elements with fewer nodes located at corners will to some extent reduce the effect of stress singularity. However, nothing is perfect, it can bring the drawbacks in terms of larger memory for store matrices and long-time analysis onto the table. Regarding to this project, it is not a huge problem which might need millions of nodes to capture the required unknowns. Therefore, using eight-node shell 281 element can be reasonable and acceptable. The mesh size for this stage is roughly 0.1 m, which is not the mesh size that will be applied in the optimization part. The optimal mesh size will be determined during the mesh convergence test. CAD model built in Inventor was imported into ANSYS workbench and the built-in mesh generator was employed to generate automatic decent mesh size, as shown below. Afterwards, boundary conditions and pressure loads were applied to the corresponding location as normal routines which can be found in the lecture notes, no details are to be discussed here.

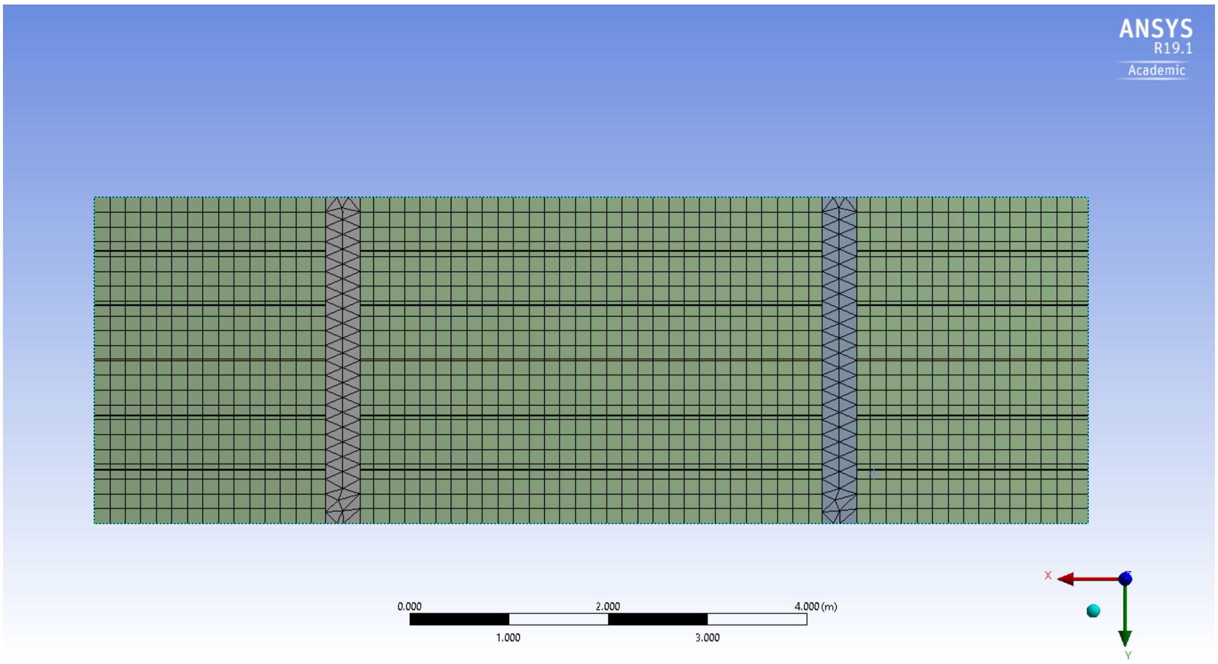


Figure 8 global mesh generation

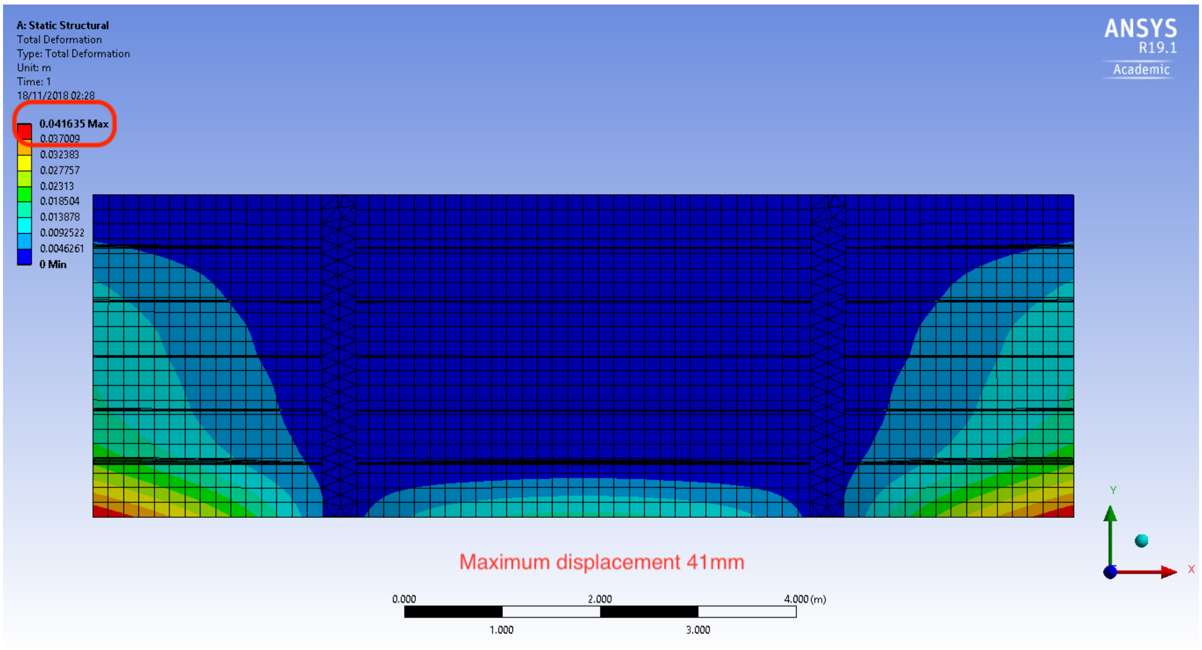


Figure 9 Displacement contour in workbench

These above figure gives the evaluation of maximum displacement in the postprocessing phase. It can be seen that the maximum displacement is around 41mm, which occurs at two corners on the opposite side to the fixed support of the plate. In practical, since the length of the plate is 10 meters long, the ratio of the vertical displacement over the length is less than 1%, say that it is still acceptable. Subsequently, to improve this initial design, the point load will be applied to one of those two corners since it is relatively more dangerous than other points.

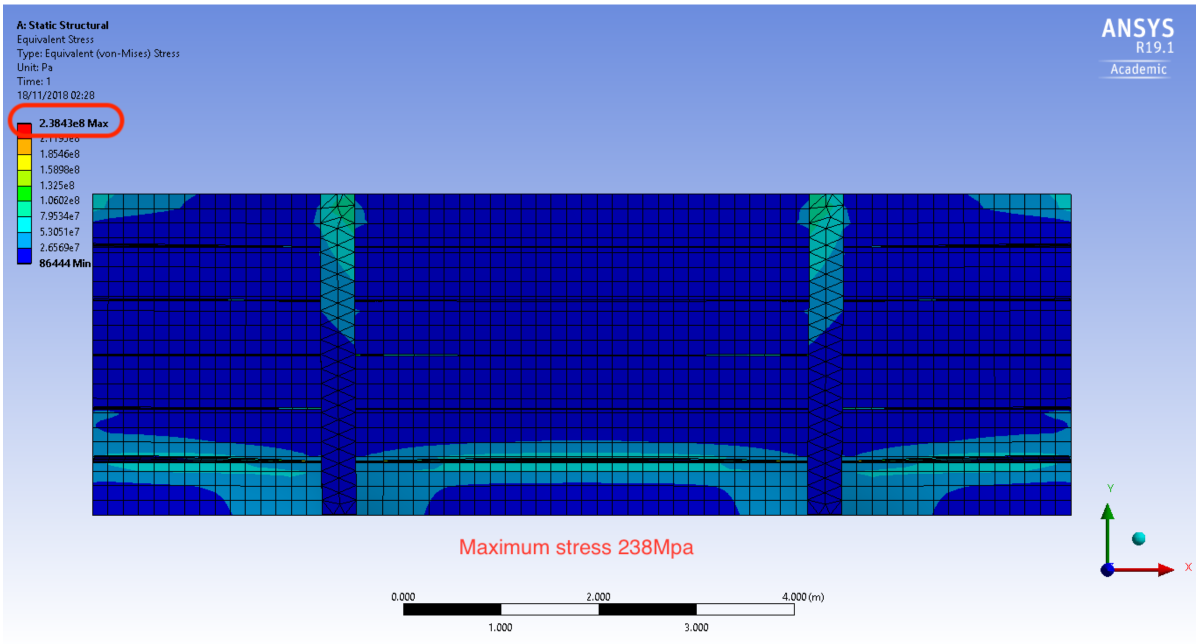


Figure 10 von Mises stress in workbench

The above figure gives the information about the maximum stress distribution Since the yield strength is 250 *Mpa*, and the safety factor is 1.25, then the permissible maximum stress is 200 *Mpa*. But the maximum stress from the initial design is 238 *Mpa*, still in the lower range than the yield strength, but over exceeds the permissible maximum strength. The detailed view on where the maximum stress happens is shown below. As discussed before, this sharp corner connecting to other areas might cause stress singularity, which needs to be considered when it comes to the optimization part. Here the material strength under the temperature changes was investigated a bit. The tensile strength of ice varies from 0.7–3.1 *MPa* and the compressive strength varies from 5–25 *MPa* over the temperature range $-10^{\circ}C$ to $-20^{\circ}C$. The ice compressive strength increases with decreasing temperature and increasing strain rate, but ice tensile strength is relatively insensitive to these variables. Assumingly, ice will not affect the result too much, the material strength stays steady, which is also stated in the last assumption in previous stage. After the mesh size is determined from the mesh convergence study, the optimization needs to be conducted so as to improve the initial design.

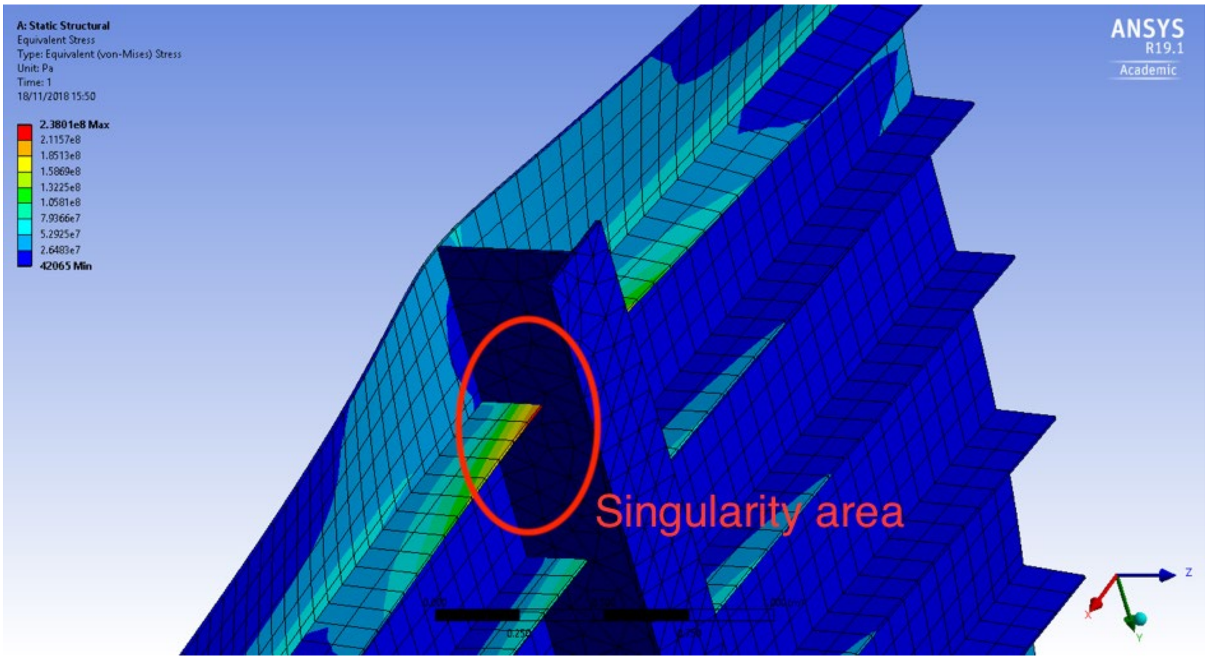


Figure 11 location where stress singularity occurs

3.2 Mesh convergence study

After the model has been formulated, an important topic occurs, which is using mesh convergence test to validate the finite element model and also to find the optimal mesh size which plays a crucial role in the mesh refinement study and the accuracy of the result. In the next optimization part, it requires a huge amount of computational resources to be able to capture the possibly global optimal solution but not certainly. Therefore, the test for mesh convergence in terms of displacement, stress and consuming time was carried out. The main reason to conduct a mesh convergence test is that by plotting those graphs which can indicate the tendency how the displacement, stress and consuming time curves behave during a sequence of reduction on element size, which in a different way it can be plotted as against the element number as well. Here, element number was chosen as the self-variable and located in the x-direction while consuming time, displacement, and stress were labelled in y-directions in the following three different graphs.

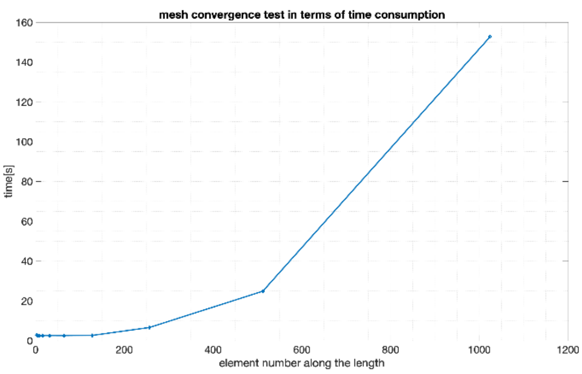


Figure 12 Convergence consuming time

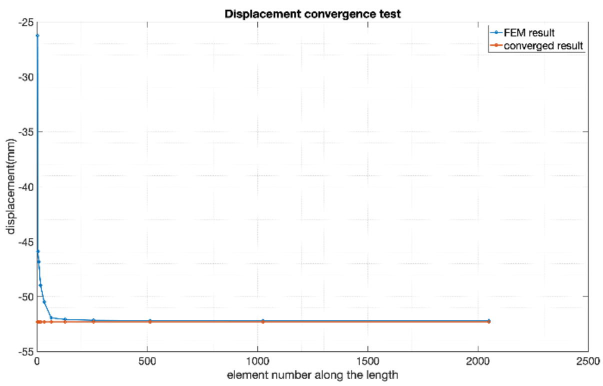


Figure 13 Displacement convergence test

From the above graph, it can be seen clearly that as the number of elements increases, the consuming time increases rapidly, and also nonlinearly. Each time the mesh size is halved (or the elements number is doubled), the consuming time increases more than doubled. Hence, it is sensible to choose a sufficient decent mesh size which can not only result in an accurate result but also lead to a relatively acceptable time consumption. In order to find the optimal mesh size, the displacement convergence test and stress convergence test were conducted to provide a criterion for the optimal mesh size.

The above figure gives the tendency how the maximum displacement goes when the mesh is refined. It shows that when the element number was increased to 64 along the length of the plate, the result is likely to converge to a certain value, in this initial design case, it is -53mm, and the corresponding consuming time is around 10 seconds, which is relatively good compared to 160 seconds or even more when the mesh size is finer and finer. the more detailed data can be found in appendix. From the above discussion, it basically means that the result obtained from using 64 elements is capable of satisfying the expected solution, and its mesh size is 0.15625 *m* at this step.

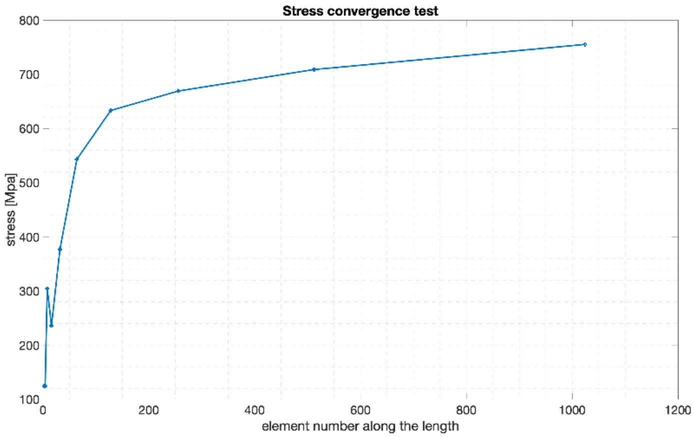


Figure 14 Stress convergence test

The above figure shows how the maximum stress is going to change when the mesh size is refined. It shows that the maximum stress in the balcony always increases with element number. Seemingly, it provides an explicit proof of the effect of stress singularity and it also makes sense when singularity occurs. When the mesh is refined, the number of elements increases as well, which leads to an increase in the number of nodes locating at the sharp corners, hence, the possibility of having stress singularity is increasing also. Theoretically, the stress will becomes infinite when the mesh size is keeping decreasing, but due to limited computational resources, it was restricted to a certain level of mesh size which can be handled by my computer, when the element number keeps increasing, then memory will overflow and gives no sense results at all, then it is sensible again to limit the element number.

3.3 Discussion

This chapter gives the information on how to make use of rules and regulations to develop a feasible initial design and what assumptions should be established first to support the process of developing the model either for hand calculation or finite element model. ANSYS workbench was employed to conduct the finite element analysis and MATLAB codes were developed to conduct the mesh convergence study. Up till now, the basic steps for initial design have been completed and an optimal design needs to be searched to satisfy the needs of stakeholders and beneficiaries.

4 Optimization

4.1 Introduction

Thanks to the technological innovation and high development in computer-aided packages, some sophisticated design problems can be tackled using some high-level design methods. The followings are various of advanced design methods.

4.1.1 Tabu search

Tabu search is a global optimization algorithm and a Metaheuristic or Meta-strategy for controlling an embedded heuristic technique. Tabu Search is a parent for a large family of derivative approaches that introduce memory structures in Metaheuristics, such as Reactive Tabu Search and Parallel Tabu Search.[7]

4.1.2 Simulated annealing

Simulated annealing is a method for solving unconstrained and bound-constrained optimization problems. The method models the physical process of heating a material and then slowly lowering the temperature to decrease defects, thus minimizing the system energy.[8]

4.1.3 Particle swarm optimization

In computational science, particle swarm optimization (PSO) is a computational method that optimizes a problem by iteratively trying to improve a candidate solution with regard to a given measure of quality. It solves a problem by having a population of candidate solutions, here dubbed particles, and moving these particles around in the search-space according to simple mathematical formulae over the particle's position and velocity. Each particle's movement is influenced by its local best known position, but is also guided toward the best known positions in the search-space, which are updated as better positions are found by other particles. This is expected to move the swarm toward the best solutions.[9]

4.1.4 Genetic algorithms

In computer science and operations research, a genetic algorithm (GA) is a metaheuristic inspired by the process of natural selection that belongs to the larger class of evolutionary algorithms (EA). Genetic algorithms are commonly used to generate high-quality solutions to optimization and search problems by relying on bio-inspired operators such as mutation, crossover and selection.[10]

In this project work, particle swarm optimization was used to find the optimal solution of a series of variables such as plate thickness, stiffener number and sections of girders. particle swarm optimization (PSO) was applied to search for a decent optimal solution. PSO is a stochastic optimization technique on the basis of large population, which has some similarities with Genetic Algorithm (GA) that I learnt from another course TMR4115. They both produce a large random population first to initialize the system and search for the optimal solution in those generated population. However, GA requires more flexible but costly technique such as crossover and mutation, PSO only scans the entire feasible region and navigate the next generation to fly close to the optimal direction. The reason why PSO was selected is that PSO is a more mature and faster way to find a decent optimal solution with costing fewer computational resources compared to other optimization methods. This chapter will deal with the development of parametric model and the interaction between PSO codes and Mechanical APDL.

4.2 Parametric modelling

Parametric is a term used to describe a dimension’s ability to change the shape of model geometry as soon as the dimension value is modified. Developing a parametric model is friendly to the interaction with PSO codes and also easy to update the parameters in search of optimal solution. The following description gives the benefits of using parametric model to optimize the design.[11]

- Capability to produce flexible designs

- Better integration with downstream applications and reduce engineering cycle time
- Existing design data can be reused to create new designs
- Quick design turnaround, increasing efficiency

4.3 PSO formulation

Since the length and width of the stiffened plate has been set as a fixed value, the rest of those parameters are listed as below. An optimal solution must be the optimal combination of those parameters which can have the best performance in their feasible region.

Table 5 Parameter representation

Variables	Description
Plate_thickness	Thickness of plate
N_stiffener	Number of stiffeners
H_stiffener	Height of stiffeners
Stiffener_thickness	Thickness of stiffeners
T_flange	Width of girders
T_web	Height of girders
T_web thickness	Thickness of the web of T-girders
T_flange thickness	Thickness of the flange of T-flange
T_loc1	Location of T-girders

Then the problem becomes to set an objective function and by changing those variables to find the optimal values for each parameter so as to find the optimal value for the objective function. The objective for this balcony design of course is to minimize the total cost, which can be interpreted to minimize the total weight in order to reduce the cost on the material. The objective function can be expressed as below:

$$Min\ \ Cost = Min\ \ Volume$$

Where, volume is the summation of the volume of each single plate (here, T-girders and stiffeners are assumed to be regarded as different profile steels welded by different plates), it can be obtained from ANSYS commands *VOLU*, which automatically compute the volume for each plate according to their given dimensions. Since volume has been extracted, the total weight is proportional to the total volume, hence, no need to calculate the weight any more.

In order to define a feasible region for the final solution computed from those parameters, some constraints need to be established first. Two constraints were applied for this case:

$$MaximumStress \leq SafetyFactor \times YieldStress$$

$$MaximumDisplacement \leq 1\% \times L\_plate$$

Where, safety factor was chosen as 1.25, and yield strength was 250 *Mpa*, then the maximum stress should be less than 200 *Mpa*. It is worth noting that von Mises stress was extracted from ANSYS solution file. von Mises stress is a value used to determine if a given material will yield or fracture. It is mostly used for ductile materials, such as metals. The above criterion states that if the von Mises stress of a material under load is equal or greater than the yield limit of the same material under simple tension. In order to avoid the stress singularity when extracting the maximum stress in solution file, some sharp corners' nodes were deselected. The picture below shows nodes contained in which area should be deselected. These areas are somewhere close to the corners such as the edge where the fixed support boundary conditions were applied and the connecting areas where the stiffeners and girders intersect with each other. It is also worth noting that this figure below only shows one case when the girder at the one quarter position along the length, since the location of the girder is also a variable, it may depend on the situation to deselect those nodes that are close to the girder position. Also, the maximum displacement should be less than 100 *mm*. Both criterions should satisfy the requirements simultaneously.

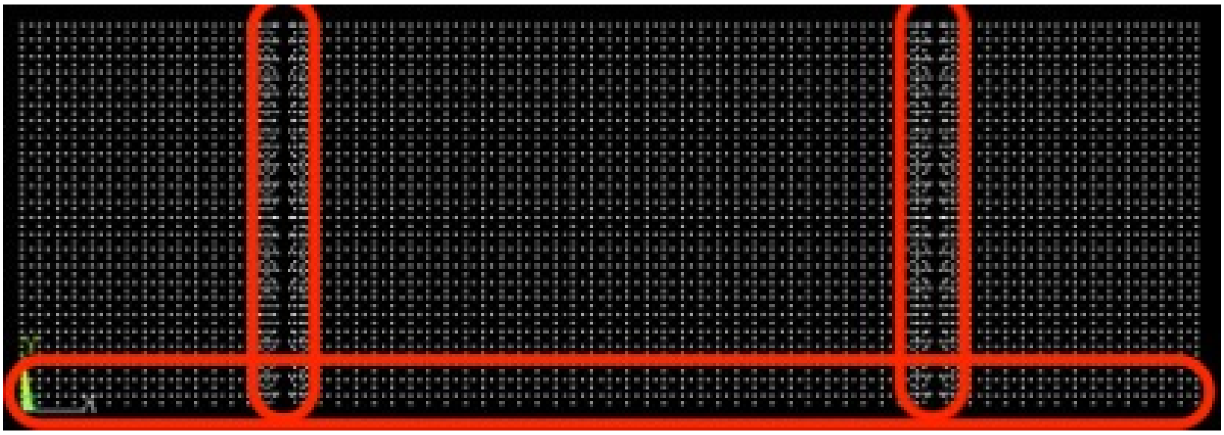


Figure 15 nodes contained in which area should be deselected.

4.4 PSO operation

The main codes related to PSO optimization were disseminated in the courses, only a few limited changes on some specific sets. The main principles of PSO are to generate a set of random particles with initial velocity, then the large population of particles will fly through the problem space in search of the best solution including local optimal solution and global optimal solution. The main parameters need to be modified are listed as follows:

Table 6 Paramenter range

Variables	Feasible region
Plate_thickness	0.003:0.001:0.05
N_stiffener	2:1:9
H_stiffener	0.01:0.001:0.5
Stiffener_thickness	0.003:0.001:0.05
T_flange	0.1:0.001:0.5
T_web	0.1:0.001:1.2
T_web thickness	0.003:0.001:0.05
T_flange thickness	0.003:0.001:0.05
T_loc1	0.5:0.01:4.5

Table 7 Algorithm preset

Swarm size	Feasible initial population	Generations	Inertia	Beta	Beta_k	Penalty factor
40	-1	150	1.4	0.8	3	2

In order to capture the global optimal solution, the relatively large swarm size and generations were set to be able to fly through as many objective points as possible. However, it takes almost four hours per click to obtain the converged solution. Theoreticality, the possibility of finding the global optimal solution increases as the generation rounds increase, and if it is set to infinite, the global optimal solution must be scanned. As a matter of this project, it cannot be set to infinite due to limited time restriction, the above preset is able to give a decent local optimal (probably global optimal) solution.

4.5 Results and discussion

During the entire project, I conducted a couple of tests for different cases, at least half of them failed, and a quarter of the rest gave some stochastic or unstable results, and eventually 10 to 20 groups of seemingly successful results might bring about some new troubles, say, the thickness of girder flange may not be possible to manufacture, or the number of stiffeners are so high which is unreasonable when considering the interaction of each stiffener and welding restrictions. Subsequently, in order to cope with such chaos properly, some practical problems were consulted, and some advices were given from the company GUANGZHOU WENCHONG DOCKYARD CO., LTD., and it is worth noting that this company produces tropical ships, some extremely icy weather condition may be neglected, and for this project, as described in the assumptions, ice would not influence the structure in a damage way. Consequently, some considerations on real-life manufacturing were taken into account. For instance, the thickness of plate cannot be larger than 60 *mm.* In addition, it is widely known that most of those T-profile girders will be manufactured by the shipyard itself to satisfy their specific needs. Equally important, the space between each stiffener should be larger than a expected value, otherwise, it is not reasonable due to welding condition as it has been mentioned before. To compare with different optimal solutions, two of those optimal solutions are given in the table below.

Table 8 Final optimal solution

Variables	Optimal value(G1)	Optimal value(G2)
Plate_thickness *[m]*	0.005	0.007
N_stiffener *[#]*	9	5
H_stiffener *[m]*	0.5	0.176
Stiffener_thickness *[m]*	0.003	0.013
T_flange *[m]*	0.137	0.291
T_web *[m]*	0.89	0.235
T_web thickness *[m]*	0.003	0.016
T_flange thickness *[m]*	0.05	0.005
T_loc1 *[m]*	2.31	2.13
Max_Stress *[Mpa]*	196	199
Max_displacement *[mm]*	43	98
Volume *[m^3]*	0.322	0.405

The above optimal combination (G1) gives the minimum volume of 0.322 m^3, which has the minimum weight of 2.528 tons. The corresponding maximum stress in this optimal solution is 196 *Mpa*, which fairly satisfies the requirement. While (G2) gives the minimum value of 0.405 with weight of 0.318 tons, and larger delfection which is 98*mm* and higher maximum stress which is 199*Mpa.*

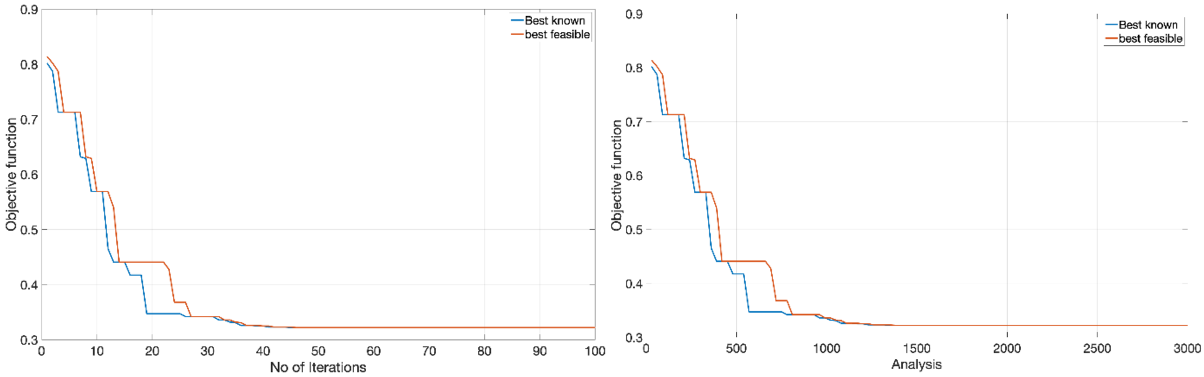


Figure 16 development of objective values (G1)

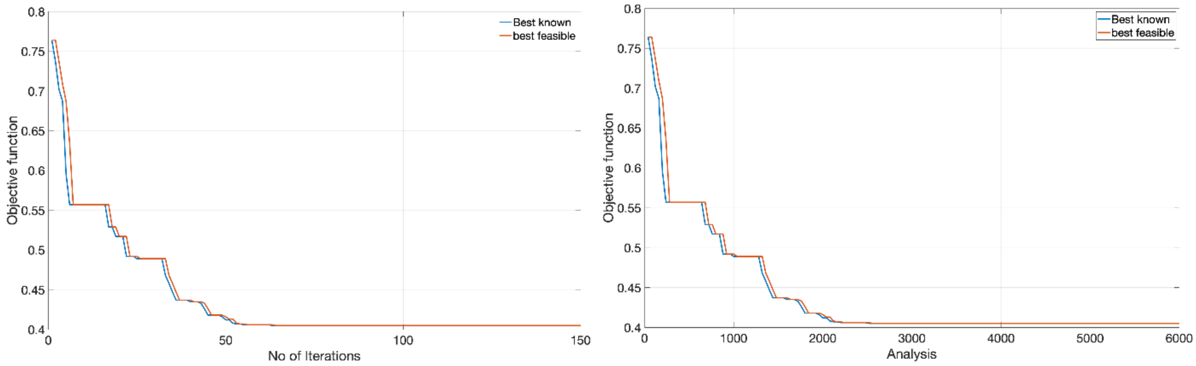


Figure 17 development of objective values (G2)

The figures below give information on how constraint was developing for both two cases during the entire optimization process, it can be seen that in the first 50 cycles of iteration, the constraint is developing stochastically, and as the time went on, it gradually converges to a stable solution which is stated as above. Is it a global optimal solution? Not so sure, since every time when I rerun the codes try to double check the solution, it always gives a different answer but keeps the volume below 0.45 m^3. Probably, yes, there must be a global optimal solution, due to the limited experience and computation resources however, therefore, it is reasonable only to find the local optimal solution.

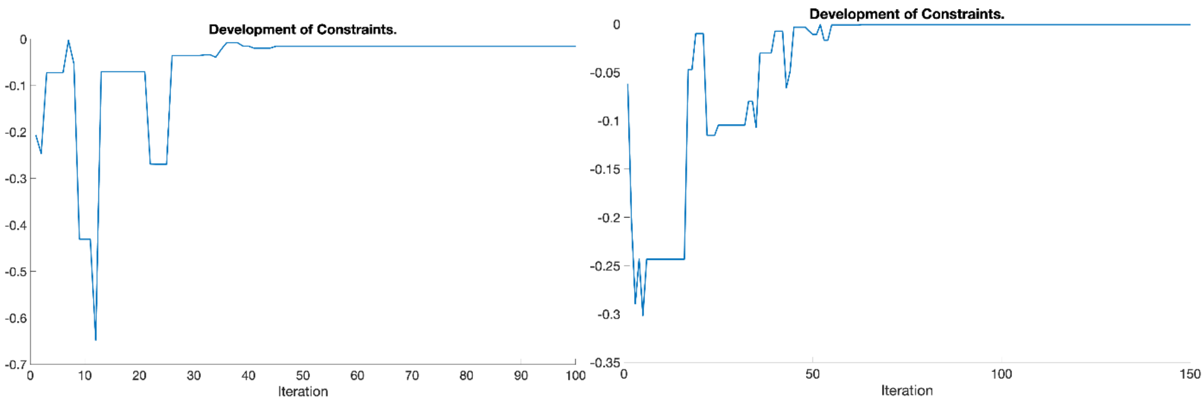


Figure 18 development of constraint (G1 left and G2 right)

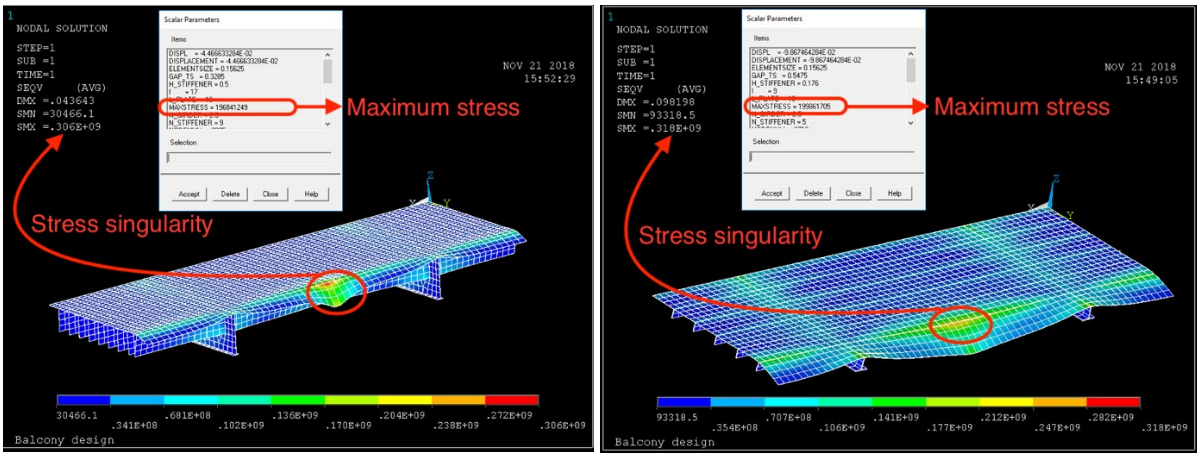


Figure 19 stress and comparison between G1(left) and G2(right)

It can be seen clearly that G1 has a larger T-girder with 0.9m web and 0.1m flange, and 9 stiffeners, which is not realistic, it will cause some ill conditions during welding. G2 gives a more reasonable combination, with only 5 stiffeners and feasible T profile girders with 0.291m width and 0.235m height.

5 Conclusion

The structural design is a comprehensive topic which requires not only theories but also skilled experience. There is no absolute perfect design, as long as it satisfies the stakeholders' needs which would have a market to manufacture and then it can be a feasible optimal deisgn. In this project, the objective function is to minimize the weight but keep in the safe area, some important conclusions are drawn as follows:

Materials can also influence on the optimal design in terms of minimum weight, for instance, if high yield strength steel is applied, intuitively, the thicknesses for those plates, stiffeners and girders can be reduced and total weight can be reduced as well. However, the unit cost for that specific type of steel is more likely to be higher than this type of the steel that was used in this project, consequently, the total cost may be hard to say that it will be reduced definitely.

Every design work depends on the criterion that the designers set. Of course, it is a multivariable function and only by changing one or few variables individually would not tell the influence from other variables. In this case, if the designers give priority to safety as well as small deflection, then optimal value (G1) might stand out from the crowed, but it may not be manufacturing-friendly. Nevertheless, if the shipyard asks the designers to design something that is easy to manufacture but still the optimal one, it is sensible that optimal value (G2) would come out, though it may have a large deflection in the future.

It is hard to say that which one is definitely better than the other one, as a designer, to have an instant communication with different sectors is also important, which could provide a supportive practical advice and local routines when designing some specific area with no standard rules and regulations. As for different customers, different designs should be submitted, then the customers might tell which one they prefer best, this might be also another criterion to judge whether it is a good design or not.

Above all, this project provides tons of useful knowledge on the entire procedure of designing based on simulation model, it is worth mentioning that this report is 30 pages limited and of course it cannot cover all of those details that the project met in the process, even though it still brings the critical information.

Reference

[1] MS Symphony of the seas, world's largest cruise vessel, available : MS Symphony of the seas: https://en.wikipedia.org/wiki/List_of_largest_cruise_ships

[2] Standard Specification for Carbon Structural Steel. Accessed:https://www.astm.org/Standards/A36

[3] Chapter 3 Structural design principles, DNV rules. Accessed: http://rules.dnvgl.com/docs/pdf/DNVGL/RU-SHIP/2018-07/DNVGL-RU-SHIP-Pt3Ch3.pdf

[4] GBT14689-1993, technical drawing rules. Accessed: https://www.chinesestandard.net /PDF/English.aspx /GBT14689-1993

[5] GBT14689-1993, technical drawing rules. Accessed: https://www.chinesestandard.net/PDF/English.aspx/ G BT18229-2000

[6] Shell 281 and shell 181, ANSYS document files. Accessed: https://www.sharcnet.ca/Software /Ansys/15.0.7/enus/help/ans_elem/Hlp_E_SHELL281.html;https://www.sharcnet.ca/Software/Ansys/15.0.7/enus/help/ans_elem/Hlp_E_SHELL181.html

[7] Tabu search, optimization. Accessed: http://www.cleveralgorithms.com/nature-inspired/stochastic/tabu_search.html

[8] Simulated annealling optimization. Accessed: https://www.mathworks.com/help/gads/what-is-simulated-annealing.html

[9] PSO optimization. Accessed: http://www.swarmintelligence.org

[10] Genetic algorithm, Optimization. Accessed: https://en.wikipedia.org/wiki/Genetic_algorithm

[11] Parametric modelling. Accessed: https://www.designtechsys.com/articles/parametric-modelling

[12] Finite element methods. Accessed: https://www.stratovan.com/blog/higher-order-elements

[13] Singularity and stress concentration. Accessed: http://www.acin.net/2015/06/02/stress-singularities-stress-concentrations-and-mesh-convergence/

[14] Stress concentraction. Accessed: https://coefs.uncc.edu/mwhelan3/files/2010/10/ICD_Saint_Venant1.pdf

[15] Von mises stress. Accessed: R. von Mises, On Saint-Venant's Principle. Bull. AMS, 51, 555–562, 1945.

http://ethesis.nitrkl.ac.in/4689/1/211ME1180.pdf

[16] Euler–Bernoulli beam theory. Accessed: https://en.wikipedia.org/wiki/Euler–Bernoulli_beam_theory

[17] Analysis of Stiffened Rectangular Plate. Accessed: https://engineering.purdue.edu/~aprakas/CE474/CE474-Ch3-ForceMethod.pdf

[18] Steel crossection. Accessed: https://www.rolledalloys.com/processing/astm-a480-plate-thickness-flatness-tolerances.html

[19] TMR4320 lecture notes and class work

Appendix 1 APDL main codes

```
FINISH
/CLEAR
/units,si  ! set the international standard units
! Import dimensions
/input,Main_Dimension,txt

! Define the system
/PREP7
/title, Balcony design

! Define geometry
*set,L_plate,10              ! set plate length
*set,W_plate,3.285           ! set plate width
*set,N_girder,2              ! number of girders
*set,T_loc2,L_plate-T_loc1
*set,Gap_TS,W_plate/(N_stiffener+1)
*set,ElementSize,0.15625
*set,Pressure,1025*9.81
*set,Point_load,75*3*9.81
ACEL,,,9.81                  ! Apply gravity force

*if,T_web,LE,H_stiffener,then
T_web=H_stiffener+0.001
*endif

! Create the keypoints for all stiffeners
*do,i,0,N_stiffener+1
        k,,0,Gap_TS*i,0
        K,,L_plate,Gap_TS*i,0
*enddo
*do,i,1,N_stiffener
        K,,0,Gap_TS*i,-H_stiffener
        K,,L_plate,Gap_TS*i,-H_stiffener
*enddo

! Create the keypoints for all girders
*do,i,1,1
        K,,T_loc1*i,0,0
        K,,T_loc1*i,0,-H_stiffener
        K,,T_loc1*i,0,-T_web
        K,,T_loc1*i-T_flange/2,0,-T_web
        K,,T_loc1*i+T_flange/2,0,-T_web
        K,,T_loc1*i,W_plate,0
        K,,T_loc1*i,W_plate,-H_stiffener
        K,,T_loc1*i,W_plate,-T_web
        K,,T_loc1*i-T_flange/2,W_plate,-T_web
        K,,T_loc1*i+T_flange/2,W_plate,-T_web

        K,,T_loc2*i,0,0
        K,,T_loc2*i,0,-H_stiffener
        K,,T_loc2*i,0,-T_web
        K,,T_loc2*i-T_flange/2,0,-T_web
        K,,T_loc2*i+T_flange/2,0,-T_web
        K,,T_loc2*i,W_plate,0
        K,,T_loc2*i,W_plate,-H_stiffener
        K,,T_loc2*i,W_plate,-T_web
        K,,T_loc2*i-T_flange/2,W_plate,-T_web
        K,,T_loc2*i+T_flange/2,W_plate,-T_web
*enddo
```

```
! Create the areas for the plate
*do,i,1,N_stiffener+1
        a,2*i-1,2*i,2*i+2,2*i+1
*enddo

! Create the areas for all girder_webs
*do,i,N_stiffener*4+5,N_stiffener*4+16,10
        a,i,i+5,i+6,i+1
        a,i+1,i+1+5,i+1+5+1,i+1+1
*enddo

! Create the areas for all girder_flanges
*do,i,N_stiffener*4+5,N_stiffener*4+5+11,10
        a,i+1+1+1,i+1+1+1+5,i+1+1+1+5+1,i+1+
1+2
*enddo

! Create the areas for all stiffeners
*do,i,1,N_stiffener*2,2
        a,i+2,N_stiffener*2+4+i,N_stiffener*2+5+
i,i+3
*enddo

! Glue all the areas together
aovlap,all
aglue,all

! Assign areas to plates
ASEL,S,LOC,Z,-0.0001,0.0001
cm,Plate,area
allsel,all

! Assign areas to girder_web1
ASEL,S,LOC,X,T_loc1-0.0001,T_loc1+0.0001
cm,Girder_web1,area
allsel,all

! Assign areas to girder_web2
ASEL,S,LOC,X,T_loc2-0.0001,T_loc2+0.00001
cm,Girder_web2,area
allsel,all

! Assign areas to girder_flange
ASEL,S,LOC,Z,-T_web-0.0001,-T_web+0.0001
cm,Girder_flange,area
allsel,all

! Assign areas to stiffener_gap1
ASEL,S,LOC,Z,-H_stiffener-0.0001,-0.0001
ASEL,R,LOC,X,0.001,T_loc1-0.001
cm,Stiffener_gap1,area
allsel,all

! Assign areas to stiffener_gap2
ASEL,S,LOC,Z,-H_stiffener-0.0001,-0.0001
ASEL,R,LOC,X,T_loc1+0.001,T_loc2-0.001
cm,Stiffener_gap2,area
allsel,all

! Assign areas to stiffener_gap3
ASEL,S,LOC,Z,-H_stiffener-0.0001,-0.0001
ASEL,R,LOC,X,T_loc2+0.001,L_plate-0.001
cm,Stiffener_gap3,area
allsel,all
! Create material properties
mptemp,1,0
mpdata,ex,1,,207e9
mpdata,pr xy,1,,0.3,
mpdata,dens,1,,7850

! Define element type
et,,SHELL281
! Define element size and mapped mesh
esize,ElementSize
mshape,0,2d

! Assign plate section properties
sect,1,shell,,Plate,
secdata,Plate_Thickness,1
secoffset,bot

! Assign stiffener section properties
sect,2,shell,,Stiffeners,
secdata,Stiffener_thickness,1
secoffset,mid

! Assign girder_web section properties
sect,3,shell,,Girder_webs,
secdata,T_web_thickness,1
secoffset,mid

! Assign girder_flange section properties
sect,4,shell,,Girder_flanges,
secdata,T_flange_thickness,1,,
secoffset,bot

! Create structural mapped mesh
!AESIZE,ALL,ElementSize
CMSEL,ALL

! Mesh those components one by one
SECNUM,1
CMSEL,S,Plate,Area
AMESH,Plate
CMSEL,ALL
SECNUM,2
CMSEL,S,stiffener_gap1,area
CMSEL,A,stiffener_gap2,area
CMSEL,A,stiffener_gap3,area
AMESH,stiffener_gap1
AMESH,stiffener_gap2
AMESH,stiffener_gap3
CMSEL,ALL
SECNUM,3
CMSEL,S,girder_web1,area
CMSEL,A,girder_web2,area
AMESH,girder_web1
AMESH,girder_web2
CMSEL,ALL
SECNUM,4
```

```
CMSEL,S,Girder_flange,area
AMESH,Girder_flange
CMSEL,ALL

! The model is entirely built and ready to solve
ANTYPE,0         ! Static analysis

! Apply boundary conditions
NSEL,S,LOC,Y,-0.0001,0.0001
D,ALL, ,0, , , ,ALL, , , , ,
ALLSEL,ALL
! Apply pressure load
NSEL,S,LOC,Z,-0.0001,0.0001
SF,ALL,PRES,-Pressure
ALLSEL,ALL
!Apply a pointed load at the dangerous location
*if,T_loc1,LE,2.5,then
NSEL,S,LOC,X,L_plate/2-
0.0001,L_plate/2+0.0001
NSEL,R,LOC,Y,W_plate-0.0001,W_plate+0.0001
F,ALL,FZ,-Point_load
ALLSEL,ALL
*else
NSEL,S,LOC,X,-0.0001,0.0001
NSEL,R,LOC,Y,W_plate-0.0001,W_plate+0.0001
F,ALL,FZ,-Point_load
ALLSEL,ALL
*endif

! Solve the system
FINISH
/SOL
SOLVE

! Extract the results
/POST1

! Extract the volume
ETABLE,EVolume,VOLU,
           ! Volume of single element
SSUM   ! Sum all volumes
*GET,Volume,SSUM,,ITEM,EVOLUME   !
Create parameter 'Volume' for volume of beam
! Extract the maximum stress
allsel
nsel,s,loc,x,-0.0001,T_loc1-0.0001
nsel,a,loc,x,T_loc1+0.0001,T_loc2-0.0001
nsel,a,loc,x,T_loc2+0.0001,L_plate+0.0001
nsel,r,loc,y,0.0001,5
nsort,s,eqv,0,0,all
*get,Stress,sort,0,imax
*get,Maxstress,node,Stress,s,eqv
allsel

*SET,SMAX,Maxstress   !   Create   parameter
'SMax' as max stress

! Export the results
*CFOPEN,OUTVOL.txt
*VWRITE,Volume
(F20.3)
*CFCLOS
*CFOPEN,OUTSTRESS.txt
*VWRITE,SMAX
(F20.3)
*CFCLO
```

Appendix 2 Matlab codes-PSO main codes

```
%=========================================================================
% Particle swarm optimization main codes
%=========================================================================
function StructOpt
%-------------------------set value source---------------------------------
source = 1;                       % source: 1 = random or 2 = read from file
%-----------------------set PSO parameters---------------------------------
Algorithm = [40 -1 150 1.4 0.8 3 2 1];
% PSO-parameter
% swarm_size = 40;
% feasibles_initial_population = -1; % Feasible designs in intial
population (-1 dont want to use).
% generations = 150;               % Calculation rounds.
% inertia = 1.4;                   % Intertia at start.
% beta = 0.8;                      % Factor for dynamic inertia reduction
% beta_k = 3;                      % Number of rounds when it should
improve, otherwise make inertia smaller
% penalty_factor = 2;              % Penalty factor for violated constraints
% print_results = 1;               % 0 not printed, 1 is printed (results)
%---------------------Set discrete variables-------------------------------
Feasible_Set=[];
  nvariables=9;
    Feasible_Set=inf(nvariables,1101);
    Feasible_Set(1,1:48)=[0.003:0.001:0.05];      % Plate thickness
```

```
    Feasible_Set(2,1:8)=[2:9];                    % Number of stiffeners
    Feasible_Set(3,1:491)=[0.01:0.001:0.5];       % Height of stiffeners
    Feasible_Set(4,1:48)=[0.003:0.001:0.05];      % Thickness of stiffeners
    Feasible_Set(5,1:401)=[0.1:0.001:0.5];        % Width of T_flange
    Feasible_Set(6,1:1101)=[0.1:0.001:1.2];       % Height of T_web
    Feasible_Set(7,1:48)=[0.003:0.001:0.05];      % Thickness of T_web
    Feasible_Set(8,1:48)=[0.003:0.001:0.05];      % Thickness of T_flange
    Feasible_Set(9,1:401)=[0.5:0.01:4.5];         % Girder location
%------------------Carry out the optimization------------------------------
if source == 1
    [best_f,best_x,best_g,history_f,history_x,history_g,iterations,...
        particle,particle_history,t_history] = PSO('particle_fun',...
        continous_variables,Feasible_Set,LowerB,UpperB,Algorithm,[]);
elseif source == 2
    [best_f,best_x,best_g,history_f,history_x,history_g,iterations,...
        particle,particle_history,t_history] = PSO('particle_fun',...
        continous_variables,Feasible_Set,LowerB,UpperB,Algorithm,...
        'initial_feasible_population.txt');
end
%-----------------Display best feasible particletion-----------------------
format compact;
disp(' ')
disp(' ')
disp('Results:')
best_f,best_x,best_g
disp(' ')
disp(' ')
%===============================END=========================================
```

Appendix 3 Matlab codes-Particle function

```
%==========================================================================
% generate particle function
%==========================================================================
function [f,g] = particle_fun(Xxx)
%----------------------set particle parameters------------------------------
Plate_Thickness = Xxx(1);                % plate thickness
N_stiffener = Xxx(2);                    % number of stiffeners
H_stiffener = Xxx(3);                    % height of stiffeners
Stiffener_thickness = Xxx(4);            % stiffener thickness
T_flange = Xxx(5);                       % width of T-girder flange
T_web = Xxx(6);                          % height of T-girder web
T_web_thickness = Xxx(7);                % thickness of T-girder web
T_flange_thickness = Xxx(8);             % thickness of T-girder flange
T_loc1 = Xxx(9);                         % location of left T-girder
%------------------read and rewrite those parameters-----------------------
fid = fopen('Main_Dimension.txt','w');
fprintf(fid,'Plate_Thickness = %0.6f\n',Plate_Thickness);
fprintf(fid,'N_stiffener = %0.6f\n',N_stiffener);
fprintf(fid,'H_stiffener = %0.6f\n',H_stiffener);
fprintf(fid,'Stiffener_thickness = %0.6f\n',Stiffener_thickness);
fprintf(fid,'T_flange = %0.6f\n',T_flange);
fprintf(fid,'T_web = %0.6f\n',T_web);
fprintf(fid,'T_web_thickness = %0.6f\n',T_web_thickness);
```

```
fprintf(fid,'T_flange_thickness = %0.6f\n',T_flange_thickness);
fprintf(fid,'T_loc1 = %0.6f\n',T_loc1);
fclose(fid);
%-----------------------run ANSYS in batch mode-----------------------------
tic
!modeling.bat;
toc
%-------------------import the results obtained in ANSYS-------------------
Volume = load('OUTVOL.txt');              % import the autocomputed volume
from result file
MAXStress = load('OUTSTRESS.txt');      % import the maximum stress
MAXDispl = load('OUTDISPL.txt');        % import the maximum displacement
%-------------------set objective function and constraint------------------
objective=Volume;                        % set objective function as volume
constraint_1=MAXStress/200e6-1;          % set stress constraint
constraint_2=abs(MAXDispl)/0.1-1;        % set displacement constraint
%---------------return objective and constraint values to PSO--------------
f=objective;                             %Objective
if constraint_1>constraint_2             %Constraint feasible -1 to 0 an
infeasible is from >0 to 1
    g = constraint_1;
else
    g = constraint_2;
end
%==================================END=====================================
```

Appendix 4 Matlab codes-Convergence test

```
%==========================================================================
% Mesh Convergence test
%==========================================================================
%---------------------------initialization---------------------------------
ElementSize = [];                      % initialize the mesh size vector
z = [];                                % initialize the displacement vector
stress = [];                           % initialize the stress vector
time = [];                             % initialize the time vector
ElementSize(1) = 10.0;                 % set the prelimiary mesh size as 10m
fid = fopen('ElementSize.txt','w');    % import the element size file
fprintf(fid,'ElementSize=%d',ElementSize(1));% rewrite the element size
fclose(fid);                           % close the file
%----------------------------refinement------------------------------------
time(i) = 0;
for i = 1:10                           % halve the element size ten times
    tic
    !modeling.bat;
    time(i+1) = toc
    ElementSize(i+1) = ElementSize(i)/2;% reduce the element size s
    fid = fopen('ElementSize.txt','w');
```

```
    fprintf(fid,'ElementSize=%d',ElementSize(i)/2); %halve the element size
    fclose(fid);
    z(i+1) = load('OUTZ.txt');           % import the displacement
    stress(i+1) = load('OUTSTRESS.txt');% import the stress
end
%---------------------------postprocessing----------------------------------
ElementSize(1) = [];                    % remove the initial value
z(1) = [];                              % remove the initial value
time(1) = [];                           % remove the initial value
stress(1) = [];                         % remove the initial value
Y = -19.3*ones(size(ElementSize));      % set the converged value
%------------------------plot those figures--------------------------------
figure;
plot(10./ElementSize,z,'-o',10./ElementSize,Y,'-o','linewidth',1.5);
grid minor;
box on;
xlabel('element number along the length')
ylabel('displacement')
title('Displacement convergence test');
figure;
plot(10./ElementSize,time,'-o','linewidth',1.5);
grid minor;
box on;
xlabel('element number along the length')
ylabel('time[s]')
title('Time');
figure;
plot(10./ElementSize,stress,'-o');
grid minor;
box on;
xlabel('element number along the length')
ylabel('displacement')
title('stress convergence test');
%===============================END=====================================
```