Effect of Plastic Deformation on Residual Strength of Ship Structures after Grounding

Master Thesis

You Zhou
Effect of Plastic Deformation on Residual Strength of Ship Structures after Grounding

Author: You Zhou
Student ID: 4255399
Email: zhouyou1225@gmail.com

Ship and Offshore Structures,
Faculty 3mE
Delft University of Technology
Delft, the Netherlands
www.3me.tudelft.nl

SARC BV
Brinklaan 109 A 11
1404 GA Bussum
The Netherlands
sarc@sarc.nl
www.sarc.nl

Bureau Veritas Marine
Netherlands BV
De Witte Keizer
Vissersdijk 223 - 241
ROTTERDAM NL-3011 GW
www.bureauveritas.com

Damen Schelde Naval Shipbuilding
Glacisstraat 165
4381 SE Vlissingen
P.O. Box 555
www.damennaval.com
Effect of Plastic Deformation on Residual Strength of Ship Structures after Grounding

ABSTRACT

Ship grounding decreases the strength of ship hull structures once plastic deformation occurs during the event. In practice of ship societies, current method of determining residual strength of ship hull structures after grounding is to totally remove the plastically damaged parts. As plastically deformed part of ship hull structures might still contribute considerable amount of strength, the method of total removal might be overly conservative. The article is oriented to quantify the residual strength of ship hull structures after grounding according to varied levels of plastic deformation. Literature study indicates that most of studies are focused on longitudinal residual strength of damaged ship structures, and transverse residual strength is rarely researched. Hence, it is target to study transverse residual strength after grounding damage, with implementation of a joint study of test and finite element simulation. The experiment gives validation to finite element models, and then an empirical formula can be derived by a series of FE simulations with the verified finite element model.

Keywords: FEM, Experiment, Residual Strength, Grounding, Buckling, Ultimate Strength, Elastic Strength, Digital Image Correlation.

Thesis Committee

Chair Prof. Dr. Ir. M.L. Kaminski, TU Delft
University Supervisor Dr. Ir. X. Jiang, TU Delft
Company Supervisor Dr. Ir.Herbert Koelman, SARC B.V.
Company Supervisor Martijn Nieuwenhuijs, Bureau Veritas
Company Supervisor Ir. Gerhard Vreugdenhil, Bureau Veritas
Committee Member Ir. Joep Broekhuijsen DAMEN B.V.
Committee Member Dr. Ir. Arie Romeijn, TU Delft
Preface

This document is the result of research on effect of plastic deformation on residual strength of ship structures after grounding, which concludes my master program in Offshore & Dredging Engineering at the Delft University of Technology.

The research has been initiated and sponsored by SARC BV and Bureau Veritas, Rotterdam office. Also, acknowledgement is given to DAMEN BV for producing testing specimen.

This thesis could not have been completed without the support of many people. Prof. Dr. Ir. M.L. Kaminski has been an inspiring teacher, and his enthusiasm and scientific supervision is highly appreciated. I also express my sincere thanks to Dr. Ir. Herbert Koelman from SARC and Martijn Nieuwenhuijs from Bureau Veritas, for their guidance and patience during the various phases of my thesis. Their broad knowledge and experience in the field of offshore structures have been of great value. I would like to thank my university supervisors, Dr. Ir. Xiaoli Jiang sharing her knowledge and ideas, and giving inspiration and encouragement. Also, I would like to thank Ir. Gerhard Vreugdenhil from Bureau Veritas to arrange my work in BV and his close involvement in this project.

You Zhou
Delft, the Netherlands
May 11, 2015
## Content

Content.................................................................................................................................................... 1

1 Introduction........................................................................................................................................... 4

1.1 General............................................................................................................................................ 4

1.2 Literature Review .......................................................................................................................... 4

1.3 Scope............................................................................................................................................. 6

1.4 Thesis Plan ................................................................................................................................... 6

2 Numerical Simulation ..................................................................................................................... 8

2.1 General.......................................................................................................................................... 8

2.2 Material Property .......................................................................................................................... 8

2.3 Treatment of Non-linear Simulation ........................................................................................... 12

2.4 Initial Imperfection ..................................................................................................................... 14

2.5 Springback .................................................................................................................................. 15

3 Specimen Design .......................................................................................................................... 17

3.1 General........................................................................................................................................ 17

3.2 Bending Moment Capacity ......................................................................................................... 18

3.2.1 Introduction.......................................................................................................................... 18

3.2.2 Bureau Veritas ..................................................................................................................... 18

3.2.3 Det Norske Veritas ............................................................................................................... 19

3.2.4 Eurocode3 ............................................................................................................................ 20

3.2.5 Results Comparison ............................................................................................................. 21

3.3 Shear Buckling design ................................................................................................................ 23

3.3.1 Introduction .......................................................................................................................... 23

3.3.2 Shear Buckling Capacity ...................................................................................................... 24

3.3.3 Design of Transverse Stiffener ............................................................................................ 25

3.3.4 Stiffening Plan for Shear Buckling ...................................................................................... 26

3.3.5 Conclusion ........................................................................................................................... 28

3.4 Lateral Torsional Buckling Design ............................................................................................. 29

3.4.1 Introduction.......................................................................................................................... 29

3.4.2 Design Approach ................................................................................................................. 30

3.4.3 Finite Element Design Approach ......................................................................................... 31

3.4.4 Discussion ............................................................................................................................ 33

3.5 Concluded Specimen Specification ............................................................................................ 33

4 Test................................................................................................................................................ 35

4.1 General........................................................................................................................................ 35
A.3 Design of Transverse Stiffener

Appendix B
  B.1 Specimen Fabrication

Appendix C
  C.1 Testing Accessories Fabrication

Appendix D
  D.1 Material Certificate

Appendix E
  E.1 Ansys APDL Sample
Chapter 1

Introduction

1.1 General

Waterway transportation has an indispensable role in the worldwide transport chain. Because of increasing activity on the waterways, the probability of grounding or collision increases by the minute (Fig1-1). For salvage companies, shipping companies and ship owners, correct decision making on ship recovery relies on an adequately accurate residual strength calculation of damaged ship structures.

![Ship grounding.](image)

Fig1-1: Ship grounding.

Ship structures are generally designed to service in the elastic range. When the hull of the ship is subjected to a large impact load due to grounding, plastic deformation occurs in the damaged parts. In practice of ship society, the damaged zones are completely taken off for residual strength calculation. But plastically deformed structures still has residual strength. Determining residual strength of damaged structures according to severity of damage can provide ship society with extra accuracy in calculation. It is expected that the strength of ship structures decreases as the magnitude of the plastic deformation increases. The plastic deformation affects the geometry and stress distribution in material negatively. The objective of this research is to quantify the effect of the plastic deformation on the residual strength of the damaged ship structure through numerical and experimental studies.

1.2 Literature Review

A large number of studies on grounding accidents have been carried out in terms of damage predictions (Simonsen and Hansen, 2000 [1]), crushing resistance (Lin and Jørgen, 2008; Paik and Seo, 2007 [2]), full-scale grounding test (Ohtsubo, 1994 [3]), risk assessment (Soares and Teixeira, 2001 [4]).

With respect to residual strength, American Bureau of Shipping (1995) [5], published the guide of hull-girder residual strength assessment; Ge Wang et al. (2002) [6] quantified longitudinal residual strength according to loss of section modulus due to grounding; Paik et al. (1998) [7] developed a fast method for assessing the collapse of the hull girder in the damaged condition using the formulation of the

Refer to the attached diagram for an illustration of the total removal of damaged elements (grey part) as per Wang et al. (2002) [6].

The methodology of assessing residual strength in these literature is generally adopted as by defining damaged zones after grounding, structural elements in these zones are removed (Fig1-2). This approach of design is currently adopted by ship societies (Bureau Veritas, ABS, etc.). In most of grounding cases, the total removal of damaged structural elements is an overestimation as usually plastic deformation is produced on structure instead of the loss of material. Due to this fact, the current design method yields an overly conservative results in determining residual strength of damaged ship structures. However, to what extent the structural strength remains a question, by determining which a more accurate calibration of residual strength can be achieved.

Refer to the attached diagram for an illustration of a common grounding situation with varied degrees of damage.

In the other perspective, the target of these literature is to calculate longitudinal strength of damaged ship structures. The transverse residual strength is rarely documented. Hence, the orientation of this thesis is inclined to study residual strength in transverse direction.
1.3 Scope

As noted above, thesis is conducted in two respective:
(1) Calibration of residual strength of plastically deformed ship structural element due to grounding and collision.
(2) Study ship residual strength in transverse direction.

Hence, primary transverse supporting member, a transverse frame, is selected as the study object. Considering a double-bottom ship structure (Fig1-4). A transverse frame is connected to the upper and lower bottom, the structural combination could be resembled as I-shaped girder with intermediate transverse stiffeners.

![Fig1-4: Model selection from transverse ship cross-section.](image)

In transverse direction, the general loading can be interpreted as bending moment created by incoming (wave pressure) or still external loading. Therefore, the study is orientated to a transverse frame simulated as I-shaped beam under bending moment.

1.4 Thesis Plan

Two major study tools are adopted: numerical simulation and test. Numerical simulation is a reproductive method to quantify residual strength of damaged ship structures. Debatably, the simulation results could deviate from the fact due to misinterpretation of boundary conditions, material properties and improper solving. To compensate and calibrate those situations, an identical test is designed to validate numerical simulation. A validated numerical model by testing results could be utilized to obtain
residual strength of damaged structures according to varied degrees and formation of plastic damages due to grounding at ignorable cost. Finally, formulation of residual strength according to levels of plastic damage could be developed by numerical study in this plan (Fig1-6).

The numerical simulation is a design tool for test as well. The specimen in test could fail due to multiple modes: lateral torsional buckling, shear buckling and pre-buckling at unwanted positions. Analytical calculation is limited for the design purpose of test. But combination of analytical calculation and numerical simulation could guarantee a proper design of test.

Test mainly consists of two procedures. A four point bending test is designed to acquire the bending limit at the center web of specimen; indentation process deforms the web to varied extent to create different levels of plastic deformation which is comparable to grounding damage.

Fig1-7: Four point bending (L). Indentation process (R).
Chapter 2

Numerical Simulation

2.1 General

Finite element package Ansys is adopted as tool of simulation. Selection is made due to its reproductive nature of using APDL language. As non-linear simulation is involved, the proper set-up of material non-linearity is first step to gain an accurate result. Boundary conditions, mesh quality, element type, load application method and choice and configuration of solver are also critical factors.

The set-up of finite element model is identical as it is in test. In indentation process, a semi-cylinder is modelled as rigid body to create plastic deformation on specimen, and at the bottom of specimen, a rigid plate to simulate ground is used to support the loading from the indenter. In four point bending procedure, four rigid semi-cylinders are used to create boundary condition and loading. Both models are laterally supported as it is indicated in the red circle (Fig2-1). Non-linear contact analysis is built between semi-cylinders and specimen to simulate contact behaviour of these bodies. When analysing the contact of these bodies, fiction needs to be included as in practical test case it is inevitable to have fiction at interfaces. As geometric difference and residual stress produced by indentation process should be inherited to four point bending procedure, indentation process and four point bending procedure are simulated in one inheriting process which contains a spontaneous transfer of geometric deformation and stress variation.

![Indentation Model (L). Four point bending model (R).](image)

Fig2-1: Indentation Model (L). Four point bending model (R).

2.2 Material Property

Steel S235JR is used to fabricate specimen, as it is commonly seen in ship structures. The material features minimum yielding stress of 235N/mm², Young’s module 210000 and poison ratio of 0.3. Two models are adopted for different purposes in compliance with DNV-RP-C208 [13]; the characteristics of material is isotropic hardening rather than kinematic hardening.
**Model1**: A simple bilinear model has been established for the design of test, with yielding stress of 235N/mm², Young’s module 210000 and work hardening ratio of 2100.

**Model2**: A multi-linear model has been established for data comparison between test and numerical simulation. In this way, more accuracy of simulation is gained.

![Stress-strain curve, bilinear hardening model (L); Multi-linear hardening model (R).](image)

For test design, a simple bilinear model would sufficient, as the demand of accuracy is not high. Testing design is safe due to conservative critical values are taken. For data comparison between test and numerical simulation, demand of accuracy is high. The multi-linear hardening material model is based on the material features given by producer of specimen (Appendix D), with yielding stress 293N/mm² and ultimate tensile strength 437N/mm². Combining rules of DNV-RP-C208, corresponding shape is set for the multi-linear stress-strain curve (Fig2-2). This two mentioned values are based on tensile strength test by material producer, it provides engineering stress-strain relationship. Since the engineering stress–strain curve does not give a true indication of the deformation characteristics of steel, it is necessary to use the true stress–strain curve that represents the basic plastic-flow characteristics of the material (Dieter 1986, [14]). Both true stress-strain curve and engineering stress-strain curve are provided in the plot (Fig2-2). The true stress–strain relationship was divided in two parts, the true stress $\sigma_t$ and the true strain $\varepsilon_t$ were expressed in terms of engineering stress $\sigma_e$ and engineering strain $\varepsilon_e$ by Dieter:

$$\sigma_t = \sigma_e (\varepsilon_e + 1)$$
$$\varepsilon_t = \ln(\varepsilon_e + 1)$$

(2-1)

In multi-linear stress-strain relationship, the lines after ULS point are depicted in a zero slope. In fact, steel material often shows a gradual drop from ULS point until fracture point (Fig2-3). This behaviour is not needed in model as the simulation is mainly conducted in compression cases so that necking strain is not necessary. Setting of zero slope after ULS point is due to Ansys does not allow a negative slope in stress-strain relationship taking convergence problems into account.
As two simulation, indentation process and four point bending successively included, the loading should be examined in terms of loading direction. This is important to determine which plastic material model should be used, namely isotropic hardening model or kinematic hardening model. Under reversed loading, kinematic model shows strength degradation due to Bauschinger effect once work hardening occurs to material due to plastic deformation. On the contrary, isotropic material model adopts equal strength when reverse loading is applied (Fig2-4), by which isotropic hardening states that the yielding surface expands uniformly during plastic flow.

Ansys is used to perform this check by giving the x-stress distribution under the indentation loading and four point bending loading respectively. In the examining procedure, the indentation is first applied to 25mm at the centre of web, and then a four point bending procedure is performed until a post-buckling stage of specimen. From Fig2-5, change of loading direction on the central web of specimen can be noted as:

<table>
<thead>
<tr>
<th>Area of Interest/ Status</th>
<th>Indentation 25mm</th>
<th>Springback</th>
<th>Four point bending, pre-buckling</th>
<th>Four point bending, post-buckling</th>
</tr>
</thead>
<tbody>
<tr>
<td>Red Zone</td>
<td>Tension</td>
<td>Compression</td>
<td>Compression</td>
<td>Tension</td>
</tr>
<tr>
<td>Yellow Zone</td>
<td>Compression</td>
<td>Compression</td>
<td>Compression</td>
<td>Compression</td>
</tr>
<tr>
<td>Blue Zone</td>
<td>Tension</td>
<td>Tension</td>
<td>Tension</td>
<td>Compression</td>
</tr>
<tr>
<td>Green Zone</td>
<td>Compression</td>
<td>Compression</td>
<td>Compression</td>
<td>Compression</td>
</tr>
</tbody>
</table>

Table2-1: Stress state at different stages of four point bending.
As noted in red zone, indentation process makes the elements in this zone extend in x-direction, so it is indicated that the elements are under tension stress. When at offloading of indentation, the total elements in web are relaxed from the compression. Springback happens when the elastic strain in material releases, causing an overall movement of elements moving upwards. This opposite movement makes the elements in red zone compressing in x-direction. As this zone has already buckling, further bending from four point bending procedure tends to move the element moving out-of-plane, resulting increasing tension stress in this zone. Therefore, in the status of post-buckling, the elements in red zone shows a high tension stress. At loading status of indentation process and springback, elements in yellow zone show compression stress due to the containment of transverse stiffener at both sides of web, and according to simple beam theory, further bending will cause the upper zone of web in compression, which arise the compression stress in yellow zone from a lower value to a final higher value of 388N/mm². Same to green zone, the containment of side transverse stiffeners compresses the elements in this zone at indentation and springback stages. In bending stage, the elements in this zone shows a decline in compression stress due to tension created by bending in lower part of beam.

![Simulation results of x-stress plot at central web of specimen, at different loading stages.](image)

Fig2-5: Simulation results of x-stress plot at central web of specimen, at different loading stages.
The change of loading direction is barely seen in this particular simulation, except in a minor area at red zone and a few element in blue zone. As the elements in blue zone is mainly in tension state before the post-buckling stage, the material hardening model will impose no influence on strength behaviour of specimen. Besides, red zone is so confine so that this area does not affect buckling behaviour of specimen at a large extent. Thus, Bauschinger effect can be ignored in the simulation. In terms of Ansys simulation, non-convergence usually occurs in the use of kinematic material hardening model when large strain analysis is involved and no experimental evidence shows a significant Bauschinger effect [15] in a large strain analysis case. On the contrary, isotropic hardening is often used for large strain or proportional loading simulations [16]. Consequently, it can be concluded that isotropic hardening model is accurate for this simulation.

2.3 Treatment of Non-linear Simulation

The causes of non-linearity in numerical simulation are material non-linearity, geometric non-linearity and changing status, etc. The direct consequence is resulting in a non-linear load-displacement curve. Material non-linearity might change the slope of this curve gradually, whereas geometric non-linearity can cause a radical slope variation. The four cases can be referred to bifurcation, buckling, snap-through and snap-back (Fig2-6) [17]. This degree of change in slope may cause serious convergence problems in simulation, resulting in false critical values or even non-convergence. To avoid this problem, choice of load application is key to convergent solution. After following discussion, method of displacement control in terms of applying load is accepted for simulation (Fig2-10). However, a proved convergence setting is not necessarily the best solution, the time consumption of simulation also needs to be taken into account to make simulation easily re-generable.

Fig2-6: Load-displacement curve of four non-linear cases.

Generally, Ansys adopts Newton- Raphson iterative method to solve non-linear problems. It works by guessing a trial solution, and then successively improve the previous guess by using the slope of load-displacement curve [17]. In Newton- Raphson iterative method, the solution is driven by load control, which means a positive load increment in every iterative step (Fig2-7). This does not work when the slope of load-displacement curve becomes zero or changes its sign. The load must subsequently remain constant or decrease in order to follow the load-displacement curve. The alternative is using arc-length method. The arc-length method uses the explicit spherical iterations to maintain the orthogonality between the arc-length radius and orthogonal directions [17]. General applications of the arc-length method involves the tracing of a complex path in the load-displacement response into the buckling/post buckling regimes. However, unsmooth or discontinuous load-displacement response in the cases often seen in cycling loading-offloading procedures cannot be traced effectively by the arc-length solution
method [18]. As indentation process and four point bending simulation are both involved, it gives enormous trouble to exchange residual stress and geometric deformation if two processes are simulated separately, such that a single inheriting simulation is preferred. However, discontinuity of load is inevitable in this particular simulation. When switching from indentation process to four point bending process, offloading is involved, which makes arc-length method abandoned as well.

A third option is using displacement control method. It follows the load path correctly in such simulations as bifurcation, buckling and snap-through (Fig2-8). Although the displacement control cannot cope with snap-back as the slope of load-displacement curve becomes vertical, as far as this thesis is concerned, bifurcation and buckling is the only two non-linear simulations involved. Indentation process is a bifurcation simulation and bending crush is a buckling simulation (Fig2-9). The displacement control would suffice the purpose of simulation. In an indentation process, the tangent of load-displacement curve changes its sign after passing the ULS point, so load incremental method would fail to solve the problem, and on the contrary displacement control would fully meet the demand of simulation.

Fig2-7: Newton-Raphson method (L) vs spherical arc-length method (R) [17].

Fig2-8: Displacement control method [17].
However, setting a proper load application method cannot fully solve convergence problems. Some other examples may be improper boundary conditions causing rigid body motion, large load increments causing non-convergence, material instabilities, or large deformations causing mesh distortion that result in element shape misalignments. As the most common ones, boundary conditions and mesh quality should be first checked when the solution is not convergent. On the other hand, a more precise result is at the cost of computational power. Ansys allows the setting of tolerance for convergence, when it is too small a refined mesh should be used and usually demands for a large period of time for every iteration step. Additionally, a smaller time step of solving means a smaller displacement/load increment at each individual substep of calculation, which results in a more precise solution but longer computational time. A balance solution is preferred in terms of accuracy of results and time spent.

Welding and other fabrication process will cause geometric instraightness and residual stress at structures, which has considerable influence on buckling resistance of the structure. Generally, these two factors decrease the buckling resistance of structures which can be summed up as initial imperfection. As it is observed in the test of this article, fabrication induced geometric instraightness on specimen is largely random in occurrence of its shape and magnitude. While shape of residual stress distribution induced by welding could be simplified as it is indicated by Paik, 2003 [24], the magnitude
of stress could be varied according to different welding procedures, and therefore its influence is largely unpredictable.

In order to account for influence of initial impaction on buckling resistance of structure, equivalent imperfections and/or residual stresses by calibrating the magnitude of the imperfections (and, or the residual stresses) to the resistance of a known case that with regard to the stability resistance resembles the buckling problem at hand [13]. Generally, considering simplification of FEM analysis, initial imperfection is usually presented and inputted into non-linear FEM Analysis as a certain amount of geometric imperfection upon structure and its shape is defined by the eigenvalue mode with the least buckling resistance (Fig3-9). In accordance to DNV-OS-C401 [31], a bow shaped deflection over the length of structure is proposed in magnitude of 0.0015 length (2-3) for lateral torsional buckling simulation (Fig2-11, L), and local web deflection is defined in magnitude of the minimum number of 0.005 web height and 0.005 web span (2-4) for local buckling simulation (Fig2-11, R). Such imperfection is also required in fabrication tolerance of specimen in Appendix B.

\[
e_0 = 0.0015L
\]  
\[
e_{ow} = Min(0.005a, 0.005h_w)\]

\(e_0\) – Total strain.  
\(e_{ow}\) – Elastic strain.  
\(e_p\) – Total strain.

2.5 Springback

Indentation process is a large deformation non-linear analysis, in which the indenter deforms the specimen beyond its elastic range. The deformation of material can be interpreted as a function of strain, which consists two parts:

\[
\varepsilon_t = \varepsilon_e + \varepsilon_p
\]

In which:
\(\varepsilon_t\) – Total strain.  
\(\varepsilon_e\) – Elastic strain.  
\(\varepsilon_p\) – Total strain.
After offloading of indenter, plastic strain will remain in the material which is presented in form of visible plastic deformation on specimen. But elastic strain is tend to release itself, and therefore this part of deformation on specimen tend to restore itself back to its initial state, which is called springback. In tensile test which defines yielding stress of material, 0.2% plastic strain is defined as engineering proof of yielding as elastic strain of steel is so tiny that it can be hardly measured or seen because of its magnitude and springback nature. In this article, springback is incorporated in the simulation and thus its tiny influence on geometry of specimen is accounted for. Compared to influence on geometry, however, springback has a significant influence on stress distribution in material after offloading (Fig2-5). This is considered and transferred into next stage of simulation, namely, four point bending procedure, as well.
3.1 General

The test specimen should be able to simulate a real ship structure. Since the transverse strength is the object of study, a transverse girder (a primary transverse supporting member) is selected as the configuration of specimen. It has an I-shaped cross section, with top flange and bottom flange attached to the web, and sectioned by four transverse stiffeners (Fig3-1, L).

In a four bending test, only the middle cross-section is subjected to pure bending moment (Fig3-1). As only bending moment is interested in, it is granted to take the middle cross-section as the focus of study. To control the deformation of specimen becomes the key of specimen design, the deformation is expected and accepted at the middle cross-section. Any other deformation modes should be excluded, such as shear buckling at side cross-sections, pre-buckling at transverse stiffeners and lateral torsional buckling of whole specimen. In a further step, the capacity of specimen should be calculated. It serves an important instruction to choose loading machine and application of load.

The design generally consists of five aspects:

1. Determine bending capacity of the middle cross-section.
2. Shear buckling design of side webs.
3. No pre-buckling at transverse stiffeners.
4. Lateral torsional buckling design.
5. Considering the slenderness of cross-section in real ship structure, make the design comparable to practice.

The approach to designing is combination of analytical solution and numerical simulation. The two tools guarantees the results of each other. On the other hand, in some situations, the existing analytical solution does not fully tackle designing problems. Numerical simulation can provide sufficient aids in these design situations.
3.2 Bending Moment Capacity

3.2.1 Introduction

In a four point bending test, the cross-section between two loading points is subjected to uniform pure bending moment (Fig3-1). Therefore, it can resemble the case that ship structures are subjected to bending moment transversely. As applied loading increasing, the I-shaped cross-section will collapse at the compression part due to buckling or yielding of material.

Extensive analytical solutions have been developed on bending strength of steel plates. Design rules from two ship societies, Bureau Veritas [19] and Det Norske Veritas [20] are reviewed for bending moment capacity. Also, Eurocode3 [21] is examined and compared with above two.

All three rules are based on Euler buckling theory of elastic plate (3-1), but approaches to bending moment capacity are varied. In two design rules from ship societies, the acquirement of critical buckling stress is the key to bending moment capacity. Bureau Veritas applies simple correction of Johnson-Ostenfeld formula (3-2) to the value of elastic buckling stress, in order to obtain critical buckling stress. On the other hand, DNV uses method of effective width of plate to obtain critical buckling stress (3-4). Once this value is known, the bending capacity can be simply calculated by beam theory. Differently, Eurocode3 applies different section modules to calculate bending capacity, the way not concerning with critical buckling stress. The methodologies of DNV rules and Eurocode3 are both developed upon the effective width theory, but Eurocode3 gives a better approximation up to practical solution as the stress redistribution of a buckling plate is incorporated in the calculation. To choose a suitable analytical solution for test design, results from three rules are calculated and compared, subsequently compared with numerical results as a reference.

3.2.2 Bureau Veritas

The buckling check from the rules of Bureau Veritas is based on the Euler elastic buckling of plate (3-1), and the critical buckling stress does not exceed the yielding stress of material. To achieve this, a simple correction by Johnson-Ostenfeld formula (3-2) is used. The procedure is ruled as follow:

1) Determine the elastic buckling stress of plate:

\[ \sigma_E = k \frac{\pi^2 E}{b^2 t} = k \frac{\pi^2 E}{12(1-\nu^2)} \left( \frac{t_w}{h_w} \right)^2 \]  

(3-1)

\( k \) is buckling coefficient. Where the boundary condition is taken as simply supported, thus \( k = 23.9 \) for bending moment.

2) Then, the critical buckling stress is obtained as:

\[ \sigma_{cr} = \begin{cases} \sigma_E & f o r \ \sigma_E \leq 0.5\sigma_F \\ \sigma_F \left[ 1 - \sigma_E/(4\sigma_F) \right] & f o r \ \sigma_E \geq 0.5\sigma_F \end{cases} \]  

(3-2)

Where \( \sigma_F \) equals the yielding stress of material.
3) The check criteria is:

\[ \frac{\sigma_{cr}}{\gamma_m \gamma_R} \geq |\sigma_b| \]  

(3-3)

Where \( \gamma_m \) is material partial safety factor and \( \gamma_R \) resistance safety factor, which are taken to 1.02 and 1.10 respectively. \( \sigma_b \) is the compression stress resulted from loading.

### 3.2.3 Det Norske Veritas

For DNV, buckling checks of plates is made according to the effective width method. The reduction in plate resistance for in-plane compressive forces is expressed by a reduced (effective) width (Fig3-2) of the plate which is multiplied by the design yield strength to obtain the design resistance.

![Fig3-2: Effective width method [20].](image)

For bending moment, the determination of buckling stress is as following:

\[ \sigma_{cr} = C_x \frac{\sigma_y}{\gamma_m} \]  

(3-4)

\( \gamma_m \) is the material partial safety factor.

Where \( C_x \) is the reduction factor, given by:

\[ C_x = 1 \quad \text{when } \bar{\lambda}_p \leq 0.673 \]  

(3-5)

\[ C_x = \frac{\bar{\lambda}_p - 0.055(3 + \Psi)}{\bar{\lambda}_p^2} \quad \text{when } \bar{\lambda}_p > 0.673 \]  

(3-6)

Where \( \bar{\lambda}_p \) is the plate slenderness given by:

\[ \bar{\lambda}_p = \frac{\sqrt{\frac{\sigma_y}{\sigma_{cr}}} \cdot \frac{1}{28.4 \varepsilon \sqrt{k_\sigma}}}{s} \]  

(3-7)

In which:
s - plate width; Ψ, stress ratio. For pure bending plate, taken to -1; t, plate thickness; σ_{cr}, plate critical buckling stress.

\[ \varepsilon = \frac{235}{\sigma_y} \]

\[ k_\sigma = 23.9, \text{ for pure bending.} \]

### 3.2.4 Eurocode3

The Eurocode3 is used to determine the capacity of bending moment resistance here. By NEN-EN 1995-1-5 [21], cross-sections for local buckling check are divided into four categories: class1 and class2 cross-section with plastic resistance, class3 cross-section with elastic resistance and class4 cross-section with resistance calculated by effective cross-section modulus.

The definition of classification in Eurocode3 is annotated as:

**Class 1**: cross-sections are those which can form a plastic hinge with the rotation capacity required from plastic analysis without reduction of the resistance.

**Class 2**: cross-sections are those which can develop their plastic moment resistance, but have limited rotation capacity because of local buckling.

**Class 3**: cross-sections are those in which the stress in the extreme compression fiber of the steel member assuming an elastic distribution of stresses can reach the yielding strength, but local buckling is liable to prevent development of the plastic moment resistance.

**Class 4**: cross-sections are those in which local buckling will occur before the attainment of yielding stress in one or more parts of the cross-section.

The classification of cross-section is realized by the section slenderness, which depends on length (or height)/ thickness ratio of single component of the cross-section and material yielding stress. Taking design of I-shaped cross-section as an example, the slenderness of web is defined as web height/ thickness \( (h_w/t_w) \) and the slenderness of flange is likewise defined as flange half-width/ thickness \( (c/t_f) \) provided flanges attach to the web along their centerline. For which the formulation of slenderness is:

\[ \lambda_w = h_w/t_w \]

\[ \lambda_f = c/t_f \]

Boundaries of different classification are defined as:

<table>
<thead>
<tr>
<th>Class</th>
<th>Flange slenderness, ( c/t_f )</th>
<th>Web slenderness, ( h_w/t_w )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Class1</td>
<td>9ε</td>
<td>72ε</td>
</tr>
<tr>
<td>Class2</td>
<td>10ε</td>
<td>83ε</td>
</tr>
<tr>
<td>Class3</td>
<td>14ε</td>
<td>124ε</td>
</tr>
<tr>
<td>Class4</td>
<td>If the slenderness ratio exceeds class3 boundary, it is class4.</td>
<td></td>
</tr>
</tbody>
</table>

\[ \varepsilon = \sqrt{\frac{235}{f_y}}, \text{ where } f_y \text{ is the yielding stress of material.} \]
The classification for an entire cross-section is selected by choosing the least favorable classification of an individual component of the cross-section. For instance, an I-shaped cross-section with class1 flanges and class4 web is labeled as class4 cross-section.

Once the classification of cross-section is determined, the ultimate bending resistance can be determined subsequently by referring to Eurocode3 (article 6.5.2, NEN-EN 1993-1-1 [22]), which is formulated as:

$$M_{c,rd} = \frac{W_y f_y}{\gamma M_0}$$  

(3-10)

Where $M_{c,rd}$ is the bending resistance of cross-section, and $f_y$ is the yielding stress of material. $\gamma M_0$ is partial safety factor which is taken to 1.0 for bending moment. Especially noted that $W_y$ is the cross-section modulus which depends on the cross-section classification:

(1) For class 1&2 cross-section, take section modulus as plastic section modulus:

$$W_y = W_{pl} = A_c y_c + A_t y_t$$  

(3-11)

(2) For class 3 cross-section, take section modulus as elastic section modulus:

$$W_y = W_{el} = \frac{I_{el}}{\Delta y}$$  

(3-12)

(3) For class 4 cross-section, take section modulus as effective section modulus:

$$W_y = W_{eff} = \frac{I_{eff}}{\Delta y}$$  

(3-13)

In which $\Delta y$ is the neutral axis height. One can refer to Appendix A for example calculation and the definition of nomination.

3.2.5 Results Comparison

The collapsing modes of cross-section are basically depend on its slenderness. As it is can be seen from DNV rules and Eurocode3, a slenderness cross-section collapse under bending moment due to the buckling of plate, but a compact cross-section deforms due to the yielding stress of material is reached. To give a review of the accuracy of three rules, a wide range of cross-section slenderness is selected, and calculations of bending capacity are made according to those different cross-section. Besides, the bending capacity obtained from numerical simulation are also illustrated, to make comparison with a more realistic situation (Fig3-4).

The middle cross-section of test specimen is taken as study object. The flange is designed to 8mm thickness to prevent it fails before the collapse of web. The range of web slenderness is from 60 to 400, so it will encapsulate all three types of cross-section, namely, a slender one, an intermediate one and a compact one. To achieve this slenderness range, the thickness of web changes from 1mm to 6.5mm, while keeping web height constant at 400mm. The specification of the middle cross-section of specimen is illustrated (Fig3-3).
Results comparison set-up.

Analytical solution of BV is based on critical buckling stress obtained from Johnson-Ostenfeld transformation, and therefore the maximum allowable stress is the yielding stress of material. Likewise, with buckling stress calculated by effective method, the critical buckling stress of DNV does not exceed this value.

Analytical results from BV and DNV takes a conservative approach, as a conservative design needs to be guaranteed. It is especially obvious at high slenderness regime (slender cross-section) that the values from these two codes are much lower than numerical results, as two codes neglect the development of plastic stress in material. In lower slenderness regime (compact cross-section), two results are gradually approaching each other, as for both method, the critical buckling stresses are approaching the material yielding stress. But the plastic strength of material is hidden, and consequently, two results are lower than numerical value.

Fig3-3: Results comparison set-up.

Fig3-4: Bending resistance comparison between different design codes and numerical results.
It is shown that for cross-section in high slenderness regime, Eurocode3 has a closest match with numerical results, because it takes a different approach to calculate bending resistance: it uses effective section modulus to multiply yielding stress of material to calculate bending resistance. The better agreement with numerical results is because this approach reflects the underlying principle of buckling of a slender plate: the stress will redistribute away from the buckled zone, and causing buckled zone to be ineffective, and neutral axis of bending is therefore lowered so that cross-section regain more resistance [23]. At low slenderness regime, Eurocode3 has the best match with numerical method as well, since it takes plastic section modulus to calculate bending resistance, and by doing this, the plastic strength of material is taken into account. Thus, it is recommended to use analytical results from Eurocode3, as it reflects the behavior of material in a better way. Also, more accuracy is gained.

3.3 Shear Buckling design

3.3.1 Introduction

A four point test is planned for the verification of numerical results. As the two side webs are subjected to shear force during test, it is necessary to study shear buckling resistance of two side webs. For the given loading, two side webs are subjected to shear force of magnitude, $P$ (Fig3-1). The intended specimen failure for the test is only the middle web collapsing due to bending moment resistance exceeded, and shear buckling of two webs needs to be excluded.

The Task of shear buckling design consists of several aspects:

1) Predict shear buckling at unstiffened two side webs using tension filed method.
2) Design of transverse stiffeners to withstand tension field action in web.
3) Discuss possible stiffening plans, and make the optimal choice.

![Flow chart shear buckling design.](image-url)

Fig3-5: Flow chart shear buckling design.
A flow chart of the design procedure is as followed (Fig3-5). The calculation of ultimate bending moment at middle web is elaborated in previous section, it can also be referred to sample calculation in Appendix A. Design of enforcing stiffeners for shear buckling and transverse stiffeners design can be found in this section.

3.3.2 Shear Buckling Capacity

When a panel is subjected to shear force, the shear deformation tends to elongate the panel at one diagonal and at meanwhile the other diagonal is shorten due to parallelogram transformation (Fig3-6). As a result, one diagonal is under tension stress and the other is subjected to compression force. The prediction of shear capacity is made by tension field method (Basler, 1961 [24]), in accordance to above two loading paths. The capacity of the panel to carry load in two alternative load-paths is how the action in I-girders are assumed to be in the tension field method, with respect to shear loading. The maximum capacity in the ultimate loading state consists of three parts – first a part, \( V_{cr} \), is governed by the shear buckling strength, and then a second (post-buckling) part, \( V_{tf} \), is represented by the diagonal in tension [24]. For the third part, which is the contribution from flanges, is often neglected for practical reasons, which means \( V_f = 0 \). As above, the ultimate strength of shear panel is given as:

\[
V_u = V_{cr} + V_{tf} + V_f
\]  

(3-14)

The prediction of shear capacity in this section is favorably approached by the calculation from tension field method. Design codes are also reviewed for shear capacity calculation. However, in design code, taking Eurocode3 for instance, the second part of resistance, a post-buckling contribution from tension field action, is taken off from the calculation for a simplified design procedure, as it is also on conservative side. But it is leading to an inaccurate prediction for specimen design as transverse stiffeners at sides provides considerable post-buckling strength. A special notice is that tension field theory is only applicable for I-girders where the panel aspect ratios of the girder lie within limits of \( 1 \leq a/b \leq 3 \), and for panel that is outside these limits the tension field theory is not valid.

According to (Paik,2013 [24]), the contribution from first part, the shear buckling strength of web along compression diagonal, is the major part of shear resistance of a plate girder if the web is not too thin, which is formulated as:

\[
V_{cr} = h_w t_w \tau_{cr}
\]  

(3-15)

Where \( V_b \) is shear resistance due to shear buckling strength, \( h_w \) and \( t_w \) are web height and web thickness respectively. \( \tau_{cr} \) is critical shear buckling stress, which is determined as following two steps:

1) Determine elastic shear buckling stress by using formula of Euler elastic buckling of plate:

\[
\tau_E = k_\tau \frac{\pi^2 E}{b^2 t} = k_\tau \frac{\pi^2 E}{12(1 - \nu^2)} \left( \frac{t_w}{h_w} \right)^2
\]

(3-16)

Where shear buckling coefficient is determined according to the aspect ratio of web panel:

\[
k_\tau = 5.34 + \frac{4}{\left( \frac{a}{h_w} \right)^2} \left( \frac{a}{h_w} \geq 1 \right)
\]

(3-17)
2) Determine critical buckling stress by substituting $\tau_E$ into the Johnson-Ostenfeld formula:

$$
\tau_{cr} = \begin{cases} 
\tau_E & \text{for } \tau_E \leq 0.5\tau_f \\
\tau_f \left[1 - \frac{\tau_f}{(4\tau_E)}\right] & \text{for } \tau_E \geq 0.5\tau_f 
\end{cases}
$$

(3-19)

Where $\tau_f = \sigma_f / \sqrt{3}$. $\sigma_f$ is the fielding stress of web.

3.3.3 Design of Transverse Stiffener

The transverse stiffeners must be strong enough to withstand the transmit force from tension field action. According to the Basler model, the following criteria has to be satisfied [24]:

$$
A_s \geq \frac{P_s}{\sigma_{yw}}
$$

(3-22)
\[ I_s \geq \frac{P_s h^2}{\pi^2 E} \]  \hspace{1cm} (3-23)

Where \( A_s \), \( I_s \) are cross-sectional area and or moment of inertia of the transverse stiffeners respectively, \( P_s \) is the stiffener bearing force due to tension field action. According to (Trahair & Bradford 1988), it is:

\[ P_s = \frac{\sigma_{yw} h t_w}{2} \left( 1 - \frac{\tau_{cr}}{\tau_y} \right) \left[ \frac{a}{h} - \frac{(a/h)^2}{\sqrt{1 + (a/h)^2}} \right] \]  \hspace{1cm} (3-24)

### 3.3.4 Stiffening Plan for Shear Buckling

A thick web has a larger shear resistance than a thin panel, meanwhile the bending moment resistance also increase due to a greater moment of inertia of a larger cross-section. For a uniform web thickness design in side webs and the middle web, due to increases both in shear resistance and bending resistance with an increasing web thickness, which one happens first, shear buckling at side webs or local buckling at the middle web, is unknown. The comparison is therefore made for every specified specimen to check the possibility of shear buckling at side webs that happens before the collapse of middle web.

An example calculation according to tension field method is attached in Appendix A. For the chosen specification of panel (see, specimen cross-section number 3):

The shear capacity is:

\[ P_{t,ult} = V_u = V_{cr} + V_{tf} + V_f = 72.472 \text{ KN} \]

While the ultimate strength of middle web due to bending is:

\[ P_{b,ult} = 164.36 \text{ KN} \]

With:

\[ P_{b,ult} = 164.36 \text{ KN} \geq P_{t,ult} = 72.472 \text{ KN} \]

So, the shear buckling of side webs occurs prior to the local buckling of middle web.

Uniform web thickness is adopted for side webs and middle web taking fabrication convenience into account. At a certain thickness of web, the shear strength of side webs equals the bending strength of middle web. Designing specimen’s web to this thickness are always compromised by the magnitude of this value: web thickness needs to be very large to reach this threshold. This will results in a cross-section slenderness exceeds the practical range in real ship structures. Thus, instead of increase the web thickness, a stiffening plan is preferred to prevent shear buckling. A list of stiffening plans at side webs expected is:

1) **Longitudinal stiffeners.**
2) **Stiffeners along tension diagonal of web.**
3) **Stiffeners along compression diagonal of web.**
Simulation of different stiffening plans. As buckling is prone to occur when steel is compressed, the favored assumption is that stiffening side webs in compression diagonal is the optimal choice. Non-linear ANSYS simulation is used to find out which one is the best, taking the same specification of specimen as above (Fig3-7). Check is first performed on an unstiffened specimen, which indicates shear buckling at two side webs happening before the local buckling of middle web as predicted by tension field method. For longitudinal stiffened panel, the shear buckling capacity is enforced, but still not stiff enough to satisfy demand. For a tension diagonal stiffened panel, the shear buckling would occur aside the stiffeners, which is also unintended. However, stiffening side webs at compression diagonal would meet such demand of eliminating shear buckling at side webs (Fig3-7).

Fig3-7: Simulation of different stiffening plans.

As buckling is prone to occur when steel is compressed, the favored assumption is that stiffening side webs in compression diagonal is the optimal choice. Non-linear ANSYS simulation is used to find out which one is the best, taking the same specification of specimen as above (Fig3-7). Check is first performed on an unstiffened specimen, which indicates shear buckling at two side webs happening before the local buckling of middle web as predicted by tension field method. For longitudinal stiffened panel, the shear buckling capacity is enforced, but still not stiff enough to satisfy demand. For a tension diagonal stiffened panel, the shear buckling would occur aside the stiffeners, which is also unintended. However, stiffening side webs at compression diagonal would meet such demand of eliminating shear buckling at side webs (Fig3-7).
3.3.5 Conclusion

The shear buckling at two side webs are possible to happen if the side webs are not thick enough. Thus, the prediction of shear buckling should be made, and if it happens, a stiffening plan should be carried out.

To compute shear buckling capacity, different methods are compared. Eurocode3 is a simplified method and it is on the conservative side, but the mechanism of shear buckling is hidden so that it is insufficiently accurate. By referring to (Paik, 2013), analytical method of a full tension field action, is adopted to determine the shear capacity of two side webs by using tension field method which is based on Basler model and subsequently transverse stiffeners should be designed to withstand the stiffener force created by tension field action in side webs. In some designs, shear buckling strength of the web is the major contribution to shear panel resistance and tension field action contributes at second place. But for a thin web design, the strength contribution from tension field action weights over that of the shear buckling strength of web. In such cases, which is mostly common in offshore and ship structures, the design with a full tension field method is indispensable.

If the shear buckling of side webs happens prior to the local buckling of middle web due to bending, for test damage control, a stiffening plan needs to be carried out to prevent shear buckling. Expectation is postulated that to stiffen compression diagonal of a shear panel is the optimal choice as buckling is prone to occur when material is compressed. Non-linear ANSYS simulation is utilized to check optimal stiffener alignment, and it shows that a stiffener along compression diagonal of shear panel is the best choice for shear buckling enforcement. Thus, the choice is made to stiffen the compression diagonal of shear panel if the prediction of analytical method shows that shear buckling happens first.
3.4 Lateral Torsional Buckling Design

3.4.1 Introduction

Consider a beam in major axis bending subjected to increasing loading. If the beam is slender, it may buckle before the sectional capacity is reached. This kind of buckling involves both lateral deflection and twisting, and is called lateral torsional buckling (LTB) (Fig3-8).

Fig3-8: LTB of a cantilever beam (L). LTB of an I-girder due to central loading, test set-up (R).

Longer beams become instable at smaller loads compared to shorter beams. Also, for beams with the same length but different cross-sections, a slender cross-section buckle at smaller loads than beams with stout cross-sections. However, not only above two factors are taken into LTB design, a list parameters which give influences on the resistance of LTB of a beam are shown (Table3-2), where (+) means a positive contribution to LTB resistance when it increases, and on the other hand (-) means a negative contribution to LTB resistance when it increases.

<table>
<thead>
<tr>
<th>Factors that affect lateral torsional buckling</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Material properties</td>
<td>Shear modulus (G) (+)</td>
</tr>
<tr>
<td></td>
<td>Young’s modulus (E) (+)</td>
</tr>
<tr>
<td>Cross-section properties</td>
<td>Torsion constant ($I_t$) (+)</td>
</tr>
<tr>
<td></td>
<td>Warping constant ($I_w$) (+)</td>
</tr>
<tr>
<td></td>
<td>Second moment of inertia about weak axis ($I_x$) (+)</td>
</tr>
<tr>
<td>Geometry properties</td>
<td>Length of the beam (L) (-)</td>
</tr>
<tr>
<td>Boundary conditions</td>
<td>Bending about major axis</td>
</tr>
<tr>
<td></td>
<td>Bending about minor axis</td>
</tr>
<tr>
<td></td>
<td>Warping</td>
</tr>
<tr>
<td>Second moment of inertia about</td>
<td>Type of loading (distributed, concentrated etc.)</td>
</tr>
<tr>
<td></td>
<td>Height of load application (-)</td>
</tr>
</tbody>
</table>

Table3-2: Influential factors of LTB [25].
The modes of LTB of a single beam is related to its boundary condition. Modes at which LTB happens are listed in the table 3-3.

<table>
<thead>
<tr>
<th>No restraints</th>
<th>Compression flange restrained</th>
<th>Flange in tension restrained</th>
<th>No restraints</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Free to buckle due to lateral torsional buckling</td>
<td>- No lateral torsional buckling is possible</td>
<td>- Distortional buckling is possible</td>
<td>- Lateral distortional buckling is possible for beams with slender flexible webs and rigid flanges</td>
</tr>
</tbody>
</table>

Table 3-3: Modes and restraints of LTB [25].

The design is to avoid test specimen collapsing due to LTB in a four bending test. The possible modes at which LTB is more likely to be “Flange in tension restrained, Table 3-3”, also comparing with Fig 3-8(R), test set-up. The task is to choose a proper specification of specimen, including length, cross-section, material, to eliminate the possibility of LTB. The orientation to change this values could be referred to Table 3-2.

### 3.4.2 Design Approach

A list of codes are examined for LTB design. For instance, Eurocode3, NEN-EN 1993-1-1, provides an analytical solution for lateral torsional buckling under bending moment, by applying a reduction factor to the ultimate bending capacity of a specified cross-section. However, due to the specimen has an inconstant configuration of cross-section over the length (diagonally stiffened at side webs), the analytical solution for LTB is inaccurate for design purpose. Instead of an analytical approach, ANSYS will be used to determine the resistance of specimen to LTB. The determination process consists of three steps by referring to Young Bong Kwon, 2012 [26]:

30
1) ANSYS eigenvalue analysis to elastic buckling strength of two buckling forms: local buckling of the middle web and LTB of the whole specimen.
2) Compare two elastic strength to determine which form of buckling is dominant.
3) Non-linear ANSYS solution to inspect the LTB in a more practical and realistic situation.

3.4.3 Finite Element Design Approach

ANSYS eigenvalue analysis is to determine the elastic buckling resistance of two buckling modes: LTB and local web buckling for the specimen. The values acquired are elastic buckling strength. Theoretically, it is a mathematical prediction by assuming that plates is perfectly straight. It is a direct and clear approach to acquire which mode of buckling happens first.

Eigenvalues for two different buckling forms can be obtained for one single specimen. By comparing this two eigenvalues, it is at ease to determine which buckling forms is dominant, or under a special condition, two buckling forms will interact with each other.

Fig3.9: Two buckling forms of a single specimen, LTB (L) and local buckling (R).

Table3-4 gives numerical solutions to the elastic resistance of varied specified specimens of two buckling forms.

<table>
<thead>
<tr>
<th>NO.</th>
<th>tw(mm)</th>
<th>tf(mm)</th>
<th>Local buckling (KN)</th>
<th>Lateral torsional buckling (KN)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>a=400 (mm)</td>
</tr>
<tr>
<td>1</td>
<td>3</td>
<td>25</td>
<td>802,7</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>25</td>
<td>2443,7</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>25</td>
<td>9273,4</td>
<td>3850</td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>8</td>
<td>585,6</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>6</td>
<td>8</td>
<td>1253,0</td>
<td>1681,9</td>
</tr>
<tr>
<td>6</td>
<td>9</td>
<td>8</td>
<td>2607,1</td>
<td>1773,9</td>
</tr>
</tbody>
</table>

Table3-4: Comparison of local buckling and LTB.

**Note that crossed cells means that no LTB exists.**
Influence of lateral torsional buckling on different specimens.

Dominant web local buckling, specification NO.4 at a=500mm.

Interaction between web local buckling and lateral torsional buckling:

Specification NO.5 at a=500mm.

Dominant LTB, specification NO.6 at a=500mm.

Fig3-10: Influence of lateral torsional buckling on different specimens.
Conclusion can be subsequently drawn as:

(1) For specimen number 1&4, web local buckling is dominant. The structure fails only due to local buckling.
(2) For specimen number 2&5, lateral buckling will interact with local buckling of web.
(3) For specimen number 3&6, lateral buckling is dominant. Local buckling is too restrained to take place.

In practical situation, more factors should be taken into account, such as geometric non-linearity, material plasticity, imperfections, and interaction of two buckling forms. The eigenvalue analysis is not sufficient to involve all the factors in a single simulation. On the contrary, the non-linear analysis can be ideal option to illustrate real-time deformation pattern of specimen. But at mean time, it is an adequate tool to give a direct numbers of buckling resistance of different modes in addition to more computational cost and more time consumed. Therefore, it is only used as an inspecting tool to check the involvement of LTB for test specimen. The specimen numbers 4 to 6 at a=500 mm in Table 3-4 are chosen for non-linear ANSYS inspection, the deformation patterns are illustrated.

3.4.4 Discussion

The non-linear inspection shows that conclusions based on ANSYS eigenvalue analysis are essentially correct. It is noticeable that, for specimen NO.4 at a=500mm, although the local web buckling is dominant, the top flange of specimen still triples laterally at a subtle extent. This means LTB cannot, in essence, be eliminated entirely, although ANSYS eigenvalues analysis fails to predict the existence of LTB. Therefore, it is better to define this situation as dominant local buckling of middle web. On the other hand, for specimen NO.6 at a=500mm, although the LTB is dominant, local buckling of web still exists at a minor influence.

In practice, test specimen can be laterally buckled due to its own fabrication imperfection and instraightness at testing boundary conditions beyond the prediction of simulation. Those factors are not easily controlled and avoided, as fabrication imperfection is random in terms of its location, shape and magnitude, and it also depends on the mancraftship and the precision of fabricating equipment. Therefore, the optimal LTB design is to separate the eigenvalues of LTB and local buckling wide apart. This will leave some margin to avoid the occurrence of LTB and minimize its magnitude when it occurs.

3.5 Concluded Specimen Specification

As noted in the previous elaboration, the design methods of damage control with regard to bending capacity, shear buckling and lateral torsional buckling have been established for test specimen design. The finalized specimen specification should not only examined and designed by these three procedures, but also should be comparable to practice in ship building. As noted in previous section, 3.2 Bending Moment Capacity, rules about bending strength of an I-shaped from design codes of BV, DNV and Eurocode3 etc are based on Euler elastic buckling theory. Considering Euler buckling elastic theory of a plate under bending moment of a ship cross-section:

\[
\sigma_E = k_b \frac{\pi^2 E}{b^2 t} = k_b \frac{\pi^2 E}{12(1-v^2)} \left( \frac{t_w}{h_w} \right)^2
\]

(3-25)
In which $k_b$ is buckling coefficient, and $k_b = 23.9$ when the cross-section is longitudinal loaded (Paik, 2003 [24]). Other parameters, $\pi$, Younges’ module $E$ and Possion ratio, $v$ are also constant when the same steel is chosen. $t_w$ and $h_w$ are web thickness and web height respectively. Therefore, the value of $\frac{t_w}{h_w}$ has a deciding influence on buckling strength of steel-plated structures.

In compliance with Eurocode3, define non-dimensional slenderness ratio of web as:

$$\lambda_w = \frac{h_w}{t_w}$$  \hspace{1cm} (3-26)

Slenderness ratio is the critical parameter for the strength of ship cross-section as it is shown in Fig3-4. Another study has also been conducted in terms of slenderness ratio by G.Y. Grondin, 1999 [27] to study strength of ship structures, which indicates different buckling mechanism of steel plate according to varied slenderness ratio. Therefore, the range of slenderness ratio has been decided as a criterion to choose the configuration of I-shaped specimen which is comparable to real ship structure. Investigation shows that the range of slenderness ratio of transverse frames in ship structures is 75-150. With all these criteria of design, the specification of specimen is finalized as:

<table>
<thead>
<tr>
<th>Length (mm)</th>
<th>Height (mm)</th>
<th>Width (mm)</th>
<th>Web thickness (mm)</th>
<th>Flange thickness (mm)</th>
<th>Transverse stiffener thickness (mm)</th>
<th>Diagonal stiffener thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1700</td>
<td>500</td>
<td>200</td>
<td>5</td>
<td>8</td>
<td>10</td>
<td>10</td>
</tr>
</tbody>
</table>

Table3-5: Concluded specimen specification.

The slenderness ratio of cross-section is $500/5 = 100 \in [75, 150]$. The detailed drawing is enclosed in Appendix B with fabrication requirement (fabrication tolerance, welding and welding test).
Chapter 4

Test

4.1 General

The design of test consists of determining testing procedures, load application, testing fixture design and test measurement. Two individual processes, indentation process and four point bending, are involved, which respectively serves different purpose of tests. Indentation process is used to create plastic deformation on specimen to simulate grounding situations, whereas four point bending is to acquire bending limit of specimen. With regard to this, plastic deformation due to four point bending should be contained, which indicates the loading of four point bending test should be within elastic strength of specimen. Considering design of testing fixtures, loading components and end supports should be designed such that the specimen can bend freely in a simply supported boundary condition and indentation process can be carried out in a deformation controlled approach.

As for test measurement, Digital Image Correlation (DIC) is adopted to monitor the lateral displacement of central web of specimen during the testing. The acquired displacement at certain testing circumstance, for example, at certain load and testing stage. Etc., can be subsequently compared with numerical simulation, by which means verification of simulation can be accomplished. The other data can be procured is the load-displacement curve by sensor of loading machine, which is similarly provided by Ansys simulation as well.

4.2 Test Procedure

A four point bending test acquires the bending limit at the center web of specimen; indentation process deforms the web to varied extent to create different levels of plastic deformation which is comparable to grounding damage. The test is designed as a cycling process of five stages, and each stage a corresponding indentation process and four point bending test are carried out to achieve test goal (Fig4-2).

![Diagram](Fig4-1: Four point bending (L). Indentation process (R). Using displacement control method.)
To minimize test expense, only one specimen is tested. An indentation process is designed to simulate the grounding situation. The web of middle panel is indented to 5%, 10%, 15% and 20% of the web height in different testing stages (Fig4-2). Thus, the verification for numerical simulation of differently damaged ship panel can be achieved. For data collection, technique of Digital Image Correlation will be adopted to measure lateral displacement at deformed web of specimen.

![Flow chart test procedure (L). Indentation process to varied depth (R).](image)

**4.3 Test Load**

Either the load of indentation or four point bending can be created by a compression test machine. The selected test machine is MTS 3500KN hydraulic compressor (Fig4-3). Determining the loading application of indentation processes and four point bending procedures is critical for test operation. In test, to avoid overall collapse of specimen, the displacement at loading point is controlled (Fig4-1), for prevention from a total collapse of specimen. This method of applying load is adopted by finite element simulation as well. As the degree of indentation is measured by its depth, it is convenient to use displacement control to create indent to varied levels. The indentation process is performed to 25mm, 50mm, 75mm and 100mm depth during different testing stages. A numerical check is performed by finite element method to acquire the maximum possible indenting force (Fig4-4). The maximum force needed is 203KN, which is below the machine capacity.

Indentation process can create plastic deformation on specimen, and four point bending procedure can also plastically deform the specimen as well once the load is high enough. As the intended plastic deformation is only produced by indentation process, this should be prevented. With the displacement of loading headers increasing, the force exerted on specimen also rises. A stopping load should be set once it is reached the machine will stop loading.
Under bending moment, a perfect plate has a distinct critical buckling value, which can be referred as the initiation of plastic deformation. However, for a plate with initial imperfection, this critical value becomes ambiguous (Fig4-5). At beginning, structure will respond proportionally in terms of load and displacement. At certain loading, non-linear behaviour initiates, but buckling still not occurs. When load surpassing the critical value, buckling starts but instead of a sudden collapse, the structure stabilizes itself by its membrane stress until reaching the ultimate strength. The critical buckling value is uncertain but it is between the ultimate strength and proportional strength. For a conservative design of test, the proportional strength is taken as the test stop loading which guarantees that four point bending procedure will not create any plastic deformation on the specimen. A finite element analysis shows the same characteristics of load-displacement curve (Proportional-Critical-Ultimate), Fig4-6.
Fig4-5: Critical load, perfect plate vs imperfect plate (M.J.M. van der Burg, 2011 [23]).

Fig4-6: FEM results of load-displacement curve with varied degrees of indentation -with material model1.

For each of testing stage, a simulation is applied to determine the loading at this stage by finding proportional limit in Fig4-6. The results are summarized in the Table4-1.

<table>
<thead>
<tr>
<th>Test stage K</th>
<th>Indentation(mm)</th>
<th>Four point bending load (proportional limit), F(KN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>846</td>
</tr>
<tr>
<td>2</td>
<td>25</td>
<td>568</td>
</tr>
<tr>
<td>3</td>
<td>50</td>
<td>345</td>
</tr>
<tr>
<td>4</td>
<td>75</td>
<td>245</td>
</tr>
<tr>
<td>5</td>
<td>100</td>
<td>Crushing load</td>
</tr>
</tbody>
</table>

Table4-1: Test load at different stages.
4.4 Test Fixture

The boundary condition created by test fixture should be simply supported, and loading parts should be able to connect to testing machine and can perform two procedures, namely indentation process and four point bending. Surface finish is required at the interface of specimen and test fixture to reduce the fiction, making specimen bend freely in the test. The detailed design drawing of test fixtures is enclosed in Appendix C with fabrication requirement (fabrication tolerance, welding and surface finish).

The loading parts consist of a loading beam, two loading headers and an indenter. In indentation process, indenter is connected to loading beam to create plastic deformation on specimen (Fig4-7). Once this procedure is accomplished, the indenter is uninstalled from loading beam, and then four point bending can be applied by two loading headers (Fig4-8).

Fig4-7: Indentation process 25mm (L). Indentation Process 50mm (R).

Fig4-8: Four point bending, intact specimen (L). Four point bending, specimen with 50mm indent (R).

An end support consists of a bottom plate with a semi-cylinder and two side supports welded to it. Spherical surface with a small surface finish makes bending freely and two side supports provides the specimen with lateral restrain. The end supports is closely fixed at testing bed to prevent its own rigid body motion (Fig4-9).
4.5 Test Measurement

Digital Image Correlation (DIC) is adopted as test measurement tool. It is used to measure the out-of-plane displacement of web under deformation. This procedure allows the determination of the object deformation in a plane parallel to the image plane of the camera. For 3-dimensional measurement, two cameras are used. If the object is observed by two cameras from different directions, the position of each object point is focused on a specific pixel in the camera plane. If the positions of the two cameras relatively to each other, the magnifications of the lenses and all imaging parameters are known, the absolute 3-dimensional coordinates of any surface point in space can be calculated, (Fig4-10). If this calculation is done for every point of the object surface, the 3D surface contour of the object can be determined in all areas, which are observed by both cameras [28].

![Fig4-9: End supports on testing bed (L). Specimen on end supports (R).](image)

![Fig4-10: Principle of 3D image correlation with 2 cameras (L). Determination of the three-dimensional displacement vector (R) [28].](image)

Once the 3D contour has been determined, the second step in digital 3D correlation is the measurement and determination of the three-dimensional deformation of the object surface. This process is carried out by correlation of the images, taken by both cameras with their original reference images. In figS4-10 (R), the displacement vector of a surface element is shown. The center point P has been displaced
from the reference state \( u \) to the deformed state \( v \). Additionally, the surface element has been rotated, tilted and distorted. With the known displacement vectors of each surface point and the reference contour, the strains can be calculated. They can be derived either directly by the differentiation of the displacements of adjacent surface points or by the analysis of the distortion of each local facet, which has been used for correlation.

In test, the DIC system is produced by VIC-3D with two cameras and a lighting system (Fig4-10, L). The central web of specimen was filmed to acquire data. Painting was first applied to web, with white background and black dots to provide a meshable grid for position tracking (Fig4-10, R). During the testing, proper lighting was applied to web.

![DIC cameras and lighting (L). Dotted painting on specimen web (R).](image1)

The other type of measurement is machine load-displacement curve. Test load is driven by displacement control of machine loading component. Once displacement is at the contacting point of specimen and machine loading part, the machine sensor begins to feedback force (Fig4-12, L). As displacement increases, load rises. Then a load-displacement curve can be plotted (Fig4-12, R). This curve illustrates the structural response to load application, giving proportional point, critical buckling value and ultimate strength.

![Machine measurement (L). Typical four point bending load-disp. curve (R).](image2)
Chapter 5

Data Verification

5.1 General

Two sets of data from test will be compared with Finite Element solutions: DIC measurement and machine load-displacement curve. Test data provides validation to finite element solutions. If the data from experimental solution and numerical solution closely agree with each other, the numerical solution can be verified. If not, the problems in simulation will be diagnosed, the model will be scientifically tuned to test results. A list of experimental data will be compared to numerical solution is:

<table>
<thead>
<tr>
<th>Test stage</th>
<th>Indentation process, DIC</th>
<th>Four point Bending, DIC</th>
<th>Indentation process, load-displacement curve</th>
<th>Four point Bending, load-displacement curve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Intact</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>N</td>
</tr>
<tr>
<td>25mm dent</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td>N</td>
</tr>
<tr>
<td>50mm dent</td>
<td>N</td>
<td>Y</td>
<td>N</td>
<td>N</td>
</tr>
<tr>
<td>75mm dent</td>
<td>N</td>
<td>Y</td>
<td>N</td>
<td>N</td>
</tr>
<tr>
<td>100mm dent</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>Y</td>
</tr>
</tbody>
</table>

*Note: Y-Compared, N-Not Compared.*

Table 5-1: Comparison made between experimental and numerical results.

The numerical model can be referred to ‘Chapter2, 2.1 General’. Material model can be referred to ‘Chapter2, 2.2 Material Property’, Model2. The contact fiction between loading headers and specimen is important to the final results. Thus, fiction in terms of fiction coefficient ($f_c$) will be studied in this chapter.

In practice, the manufactured steel plate is usually thicker than it is specified. For an accurate FEM results, the plate thickness of tested specimen is measured and FEM model is changed to this set of plate thickness:

<table>
<thead>
<tr>
<th>Test stage</th>
<th>Web thickness (mm)</th>
<th>Flange thickness (mm)</th>
<th>Transverse stiffener thickness (mm)</th>
<th>Diagonal stiffener thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design</td>
<td>5</td>
<td>8</td>
<td>8</td>
<td>10</td>
</tr>
<tr>
<td>Test</td>
<td>5.5</td>
<td>8.1</td>
<td>8.1</td>
<td>10.3</td>
</tr>
</tbody>
</table>

Table 5-2: Measured plate thickness of specimen.

5.2 Test Misalignment

5.2.1 Problem Definition

Test misalignment are inevitable in testing data. Post-processing methodology should be discussed and built for eliminate those misalignment to make test data usable. Test misalignment generally consists of two components, which are:
Random misalignment might originate from measurement misalignment, and it may occur in the measuring instruments or in the environmental conditions. Those misalignment often appear as noises, featuring a value protruding out from a set of rational looking and consistent data, and this type of misalignment can often be filtered and processed, but they are inherently unpredictable. Systematic misalignment is misalignment that is not determined by chance but is introduced by an inaccuracy inherent in the system. Systematic misalignment often results from an improper set-up of test and operational impropriety, and they are often due to a problem which persists throughout the entire experiment. Consequently, systematic misalignment are reproducible inaccuracies that are consistently in the same direction. Hence, this type of misalignment is traceable to some resulting factors, and the factors can be defined and consequently solution of misalignment cancellation can be developed.

5.2.2 Misalignment Cancellation

One concept should be clear that DIC takes a reference picture to calculate lateral displacement: the reference picture is taken at zero loading of indentation process/four point bending. Although previous testing procedure makes web buckled and displace in lateral direction, the lateral displacement at entire web plane is assumed to be zero at the beginning of each successive testing procedure. The results of plot is not applicable in terms of data verification. The systematic misalignment has to be corrected. Thus, an assumption has been proposed as:

**Assumption:** The lateral displacement at web edges is only due to fabrication instraightness or inclination at specimen or test fixtures. At a certain loading, the specimen and test fixtures will closely attached to each other, and the deflection or distortion at web edges due to fabrication misalignment will be ceased after this point.
Lateral displacement is superposition-able, which can be expressed as:

\[ \Delta U_{z,t} = \Delta U_{z,F_1} + \Delta U_{z,(F_2-F_1)} + \cdots + \Delta U_{z,(F_n-F_{n-1})} \tag{5-2} \]

In which \( \Delta U_{z,t} \) is the total lateral displacement due to load \( F_n \) in four point bending procedure, while \( \Delta U_{z,(F_n-F_{n-1})} \) is the increase of lateral displacement due to increase in load by \( F_n - F_{n-1} \).

To eliminate systematic misalignment due to fabrication imperfection, for four point bending with 25mm indentation, it is assumed that at load of 200KN, the specimen and test fixtures will closely attached to each other, and this systematic misalignment has been fully developed. Thus, a plot is made by subtracting lateral displacement at 200KN (Fig5-3) from lateral displacement at 568KN (Fig5-1). The revised plot (Fig5-5) gives validation of the assumption previously proposed: the lateral displacement at edges of web is zero, which means increased load from 200KN to 568KN does not create deflection and distortion at web edges. The lateral displacement at web edges is fully developed at load of 200KN. Subsequently, the revised lateral displacement plot from test result (Fig5-5) can be compared to revised lateral displacement plot from FEM results (Fig5-6) as nearly identical deformation contours have been seen between two plots.

![Web Lateral Displacement](image)

**Fig5-2:** Experiment, four point bending with 25mm indentation, lateral disp. at 568KN loading.
Fig 5.3: FEM, four point bending with 25mm indentation, lateral disp. at 568KN loading.

Fig 5.4: Experiment, four point bending with 25mm indentation, lateral disp. at 200KN loading.
Fig 5.5: Experiment, revised lateral displacement plot.

Fig 5.6: Test, revised lateral displacement plot.
5.3 Data Comparison

5.3.1 Indentation Process Comparison

Load-disp. curves of indentation process 25mm are compared for test result and FEM results. Also, lateral displacement plots at the end of indentation process 25mm are compared with each other. As in indentation process, specimen is attached at one single line to the loading part on machine, the influence of fabrication imperfection is not significant: Fig5-7 and Fig5-8 indicate a quite close maximum lateral displacement at identical position on web. The other explain of this behaviour is that the lateral displacement on web in indentation process is mainly governed by the depth of indentation, other factors such as material input and initial imperfection are of minor influence.

Load-disp. Curves from test and FEM (Fig5-9) indicate that testing model and FEM model have a similar mechanical characteristics. The configuration of two curves is nearly identical, which means material input of numerical simulation is basically correct in terms of yielding stress, ultimate tensile stress and corresponding strain. The ultimate indentation resistance of FEM result is still higher:

\[
\frac{[234(FEM) - 211(Test)]}{211(Test)} = 10.9\%
\]

It is reasonable when considering that the fabrication of test specimen usually gives a larger initial imperfection that that inputted in FEM simulation. The tangents of two curves (Fig5-9, dashed lines) present the stiffness of two models. The tangent of FEM curve is 77.515KN/mm compared to test tangent of 68.153 KN/mm is stiffer:

\[
\frac{[77.515(FEM) - 68.153(Test)]}{68.153 (Test)} = 13.7\%
\]

Fig5-7: Experiment, indentation process 25mm.
While this could be explained by large fabrication imperfection in test model as well, the other reason is that the displacement measurement also includes the deformation of upper structures, namely indenter, loading beam (Fig4-7, R) in the test, which makes the test curves appear less stiff. Two curves after ultimate strength are almost paralleling to each other, which indicates the inputted work hardening rate is significantly close to test model.
Considering the fact that lateral displacement plots at centre web have an identical shape and maximum values, and characteristics of loading curves have a reasonable explanation, the simulation of indentation process could be therefore verified.

5.3.2 Four Point Bending Comparison

Fig5-5 and Fig5-6 illustrate that numerical solution has a deviation from experimental solution, where maximum experimental lateral displacement is -1.4mm and maximum numerical lateral displacement is -1.18mm. However, the location of maximum displacement and displacement contour are identical, which means two models can be compared in terms of maximum lateral displacement.

Multiple numerical simulations indicate that this deviation is heavily influenced by the fiction coefficient \( f_s \) between the contact bodies in simulation (Fig2-1). Data in forms of DIC data (maximum lateral displacement value at web) and machine load-disp. curve will be compared in order to optimize the numerical set-up and analyze deviation of two data sets. A higher fiction coefficient in simulation will result in a higher strength, which results in a higher ultimate strength of FEM model (Fig5-13). Machine load-displacement curve from both models at last testing stage is also compared to check the consistency with maximum lateral displacement comparison.

The other factor is imperfection on specimen. Higher imperfection will result in a lower lateral resistance both in simulation and test. Initial imperfection of simulation is expected much lower than that of test, as fabrication of specimen usually produces unexpected large initial imperfection on specimen. Indentation also produce imperfection on specimen, which is at much higher magnitude than initial imperfection if large indentation is applied. Consequently, the total imperfection consists of two components:

\[
\text{Total Imperfection} = \text{Initial Imperfection} + \text{Imperfection by Indentation}
\]  

When specimen is intact or subtly indented, it can be expected that initial imperfection takes dominant proportion in total imperfection, and the strength behavior of specimen is mainly governed by initial imperfection. When the indentation is increased, the initial imperfection becomes less influential as the imperfection by indentation is at much larger magnitude than that of initial imperfection. Such behavior can be seen from Fig5-10, Fig5-11 and Fig5-12:

1. For four point bending with 25mm indentation (Fig5-10), test data line is below the FEM lines with varied fiction coefficient: the indentation induced imperfection is still low such that initial imperfection is dominant at this stage. Therefore, test results with high initial imperfection shows a lower lateral resistance than that of simulation with low initial imperfection.

2. For four point bending with 50mm indentation (Fig5-11), test data line is below the FEM lines with fiction coefficient 0.1 and 0.15: the indentation induced imperfection is raised due to more indentation, which makes larger proportion of this part in total imperfection. Although the initial state still holds that test model has more initial imperfection than simulation model, the influence of initial imperfection becomes less dominant in terms of lateral resistance. The test results become more stiffness when comparing with numerical results as it rises above lines of simulation with fiction coefficient 0.1 and 0.15, but still below the lines of simulation of fiction coefficient 0.2, 0.25 and 0.3.
Fig 5-10: Revised maximum lateral displacement, four point bending with 25mm indentation.

Fig 5-11: Revised maximum lateral displacement, four point bending with 50mm indentation.
For four point bending with 75mm indentation (Fig5-12), test data line is close to the line of simulation with friction coefficient 0.25: the indentation induced imperfection is further raised due to more indentation, which makes more proportion of this part in total imperfection. The influence of indentation induced imperfection becomes dominant in terms of lateral resistance.

As a gradual rise of proportion of indentation induced imperfection due to the larger indentation from 25mm to 100mm can be seen, it is expected that during indentation process 75mm to 100mm, the imperfection induced by indentation will become dominant in total imperfection. Therefore, the difference in terms of specimen’s imperfection between simulation and test is at subtle influence on the strength of model. The test line will become close to the line of simulation with certain friction coefficient. This friction coefficient has been found between 0.25-0.3, which can be seen in both Fig5-12 and Fig5-13.

Table5-3 summarizes comparison of simulations of different friction set-up in terms of ultimate strength for four point bending with 100mm indent. In previous section, the indentation resistance comparison shows that FEM result is higher than testing result by 10.9% because of higher initial imperfection testing model has. The expected ultimate bending strength of FEM solution is higher than that of test, but it should be much less than 10.9% due to less proportion of initial imperfection in total imperfection as indentation is applied to 100mm. Therefore, a reasonable friction in testing situation is expected between 0.25 and 0.3. From above, it can be concluded that the numerical results of four point bending has a reasonable agreement with test results.
Friction Coefficient  |  ULS FEM (KN)  |  ULS Test (KN)  |  Difference  \\
--- | --- | --- | ---  \\
0.1  |  305.1  |  373.3  |  -18.3%  \\
0.15  |  323.6  |  ---------  |  -13.3%  \\
0.2  |  348.1  |  ---------  |  -6.7%  \\
0.25  |  368.5  |  ---------  |  -1.2%  \\
0.3  |  402.3  |  ---------  |  7.7%  \\

Table 5-3: ULS comparison between FEM solution with varied friction coefficient and test result.

5.4 Concluding Remarks

At first, indentation process 25mm is compared for FEM results and experimental results. The ultimate indentation resistance from Fem result is higher than that of test by 10.9%. It is rational when considering that test model has more initial imperfection than it is inputted into FEM model. The comparison of lateral displacement plots from both models shows an identical shape and maximum value, which shows that the FEM set-up is basically correct. It provides a solid base for next stage of simulation, namely four point bending process, and also it gives part of credits that the present simulation is trust-worthy.

For data comparison of four point bending, the test misalignment should be first diagnosed and corrected. The fabrication imperfection of testing fixtures causes unwanted distortion at specimen, which makes non-zero displacement at edges of central web of specimen. On the contrary, the FEM model is much more ideal, it shows no displacement at edges of central web as bending moment is applied. To compare two sets of data, distortion in test model needs to be eliminated at first. The assumption is proposed that the distortion is only cause by imperfect configuration of test fixtures and specimen, but it is not inherent in the nature of specimen’s mechanical characteristics. Therefore, the distortion of specimen would be ceased after the fixtures and specimen are fully contacted. A graphic
subtracting method is adopted to cancel distortion of specimen in test. Once it is eliminated, the lateral displacement due to bending moment can be compared with both models.

The test has been done in such way that the imperfection control of fabrication is not ideal. On the other hand, numerical model has been established in a way which contains less imperfection. Several graphs are depicted from the experimental data and FEM data to investigate the influence of initial imperfection on lateral resistance of web. At beginning stages of test stages, the initial imperfection is dominant and affect lateral strength of specimen negatively. Therefore, test model shows less strength than model. As the indentation is successively increased in last few stages of test, the imperfection by indentation created on specimen over-weights the initial imperfection enormously. As a results, due to the same amount of indentation exerted both on test model and FEM model, the strength of two models become convergent.

Friction coefficient at contacts has a significant influence on strength of FEM model as well. The higher friction coefficient will result in a higher stiffness in numerical model. In practice, steel-to-steel contact often features a friction coefficient of 0.15–0.3. This coefficient value has been found in this numerical case between 0.25 and 0.3. Little misalignment has been seen between two models, the FEM model can be served for formulation purpose.
Chapter 6

Analytical Solution

6.1 General

A series of parametric studies is conducted to investigate the residual bending strength of damaged ship structures based on verified finite element models. First, the friction coefficient at contacts should be set to zero to study pure bending case. The methodology of this study is to modified single factor at each of individual simulation to study the influence of modified factor. These factors could be grouped into three categories:

1. Indentation location: different indentation locations at longitudinal direction of specimen.
2. Material difference: different grades of steel will be inputted in model, S235, S355 and S420, which is based on DNV-RP-C208.
3. Specimen geometry: different aspect ratios of indented web, varied slenderness of flange and web.

Finally, an analytical expression will be derived based on these parametric studies in form of residual bending strength ratio (residual strength/ intact strength) versus indentation percentage (Indentation depth/ total web height). The first formulation is based ultimate strength of specimen, and the second formulation is based elastic strength of specimen as Bureau Veritas required.

6.2 Specimen Geometry

6.2.1 Aspect Ratio

Two models with different aspect ratio (web width/ web height) are utilized to show the influence of this factor (Fig6-1). As bending buckling coefficient is insensitive to aspect ratio, it is expected two models will yield the same result. Model1 is test verified model which serves as a reference to other models. The aspect ratio of model1 is 1, and in model2, aspect ratio is changed into 1.4. The indentation location is selected at the center of specimen for both models.

The load-disp. curves show little different between two models (Fig6-2), which proves the prediction that specimen strength is insensitive to aspect ratio. Subsequently, residual strength ratio is depicted according to the indentation percentage by normalizing ultimate strength of each curves and indentation depth which also indicates that aspect ratio has subtle influence on residual strength of indented specimen. By sampling ultimate strength of each curve, the curve of residual strength ratio versus indentation percentage is plotted, and it adopts rational fit method, the formulation of this type is indicated in Fig6-3.
Other perception of this comparison is that the size of indenter exerts no influence on residual strength, as it is can be notified that indenter is relatively larger in model1 than in model2. The degradation of residual strength is significant: 20% of indentation on total web height results a strength reduction of 80%. Especially, the drop of bending strength is significant at first few fraction of indentation, as the first 5% indentation reduces 40% bending strength of web. Afterwards, the rate of drop is slowed down, as it is can be seen that the last 5% indentation only reduces 5% bending strength of web.
6.2.2 Flange Slenderness Ratio

The other factor might be influential is the slenderness ratio of flange ($\lambda_f$) and web ($\lambda_w$). As it is indicated by Eurocode3 (Chapter3, 3.2.4 Eurocode3): A very slender cross-section (Class4) buckles before reaching the yielding stress of material; An intermediate cross-section (Class3) buckles at the reaching of yielding stress but buckling can prevent the cross-section from developing plastic strength; A compact cross-section (Class1&Class2) buckles at development of its plastic strength. The assumption is made that difference in buckling mechanism might result in the different residual strength ratio. Thus, both the slenderness of flange and web is investigated to reveal if slenderness ratio is relevant to residual strength ratio.

The first is to see the influence of flange slenderness ratio on residual strength ratio. Model3 is introduced with 300mm width compared to original 200mm. So the flange slenderness ratio, $\lambda_f = \frac{b_f}{t_f}$, will increase by 50%. The resulted residual ratio is plotted in Fig6-5. It indicates that the residual...
strength ratio of Model 3 is slightly higher than model 1 when indentation is lower than 9%, and lower than that of Model 1 when indentation is higher than 9%. The difference between two lines is subtle, which might result from the different scales used in finite element model. Hence, the slenderness ratio of flange has ignorable influence on residual strength ratio.

Fig6-4: Model 3, flange width increased to 300mm.

Fig6-5: Residual strength ratio vs indentation percentage for model 1 and model 3.
6.2.3 Web Slenderness Ratio

The next is to investigate the influence of web slenderness ratio \( \lambda_w = t_w / h_w \) on residual strength ratio. Webs of four varied slenderness ratio are investigated. They are categorized from Class 1 to Class 4 by Eurocode 3 (Table 6-1).

<table>
<thead>
<tr>
<th>Web thickness (mm)</th>
<th>Slenderness ratio, ( \lambda_w )</th>
<th>Classification</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>167</td>
<td>4</td>
<td>Slender</td>
</tr>
<tr>
<td>5.5 (ref)</td>
<td>91</td>
<td>3</td>
<td>Intermediate</td>
</tr>
<tr>
<td>7.5</td>
<td>67</td>
<td>2</td>
<td>Compact</td>
</tr>
<tr>
<td>9</td>
<td>59</td>
<td>1</td>
<td>Compact</td>
</tr>
</tbody>
</table>

Table 6-1: Webs of different slenderness ratios.

Fig 6-6 indicates that a compact web has more residual strength when compared to a slender cross-section. Therefore, the web slenderness ratio has considerable influence on residual strength ratio. It can be notified that when the slenderness of web is different, indentation percentage is influence as well. At low indentation percentage, webs of varied slenderness shows closer residual strength ratio and it is convergent at zero indentation as 1. On the contrary, when indentation percentage is high, the gaps between different lines also become larger, which demonstrates that more difference in residual strength has developed.

As it is observed that web slenderness has a significant influence on residual strength ratio, the parameter of web slenderness ratio, \( \lambda_w \) should be incorporated into formulation. It can be notified that residual strength ratio is monotonically decreasing with variable \( \lambda_w \) decreasing and with respective to the other variable \( d \), residual strength ratio is monotonically decreasing with \( d \) increasing. Which can be
also notified is that at indentation percentage zero, due to normalization is applied, residual strength ratio unconditionally equals 1. To sum up, the characteristics of mathematical model should satisfy:

\[
\frac{\partial R(d, \lambda_w)_{ULS}}{\partial \lambda_w} < 0, \lambda_w \in [59, 167] \quad (6-1)
\]

\[
\frac{\partial R(d, \lambda_w)_{ULS}}{\partial d} < 0, d \in [0, 20]
\]

\[
R(0, \lambda_w) = 1
\]

(6-3)

Different mathematical forms are postulated to fit the solution. The basic rule of postulation is to minimize the discrepancy between model proposed and the data by applying least square method. The above three conditions should also be satisfied. Polynomial fitting and rational fitting always finds a minimal discrepancy but they are not accurate in describing a monotonic relationship between variables and result. Relative research is accomplished by M.L. Kaminski, 1992 in which a mathematical model is postulated to predict strength of a uniaxially compressed imperfect plate due to the influence of magnitude of imperfect and plate reference slenderness:

\[
\Phi = \frac{1 + \mu + 2\lambda_r^2}{3\lambda_r^2} - \sqrt{(1 + \mu + 2\lambda_r^2)^2 - 4\lambda_r^2}
\]

(6-4)

In Which:

\( \Phi \) is the strength of uniaxially compressed plate

\( \mu \) is the knockdown factor on initial deflection of plate, \( \mu = 0 \) for perfect plate.

\( \lambda_r \) is the reference plate slenderness ratio.

The formulation postulated presents a monotonically decreasing of result on both variables \( \mu \) and \( \lambda_r \) (Fig6-7), but on condition \( \mu = 0 \), the strength is also decreasing with \( \lambda_r \) increasing. The physically meaning of this characteristic is that a slender plate \( (\lambda_r > 1) \) buckles before it reaches the yielding stress of material and initiation of plastic stress development. Obviously, this characteristic is governed by plate slenderness ratio. By studying of present calculation (Fig3-4) of intact strength from different organizations, it is learned that slenderness ratio has already been incorporated. Hence, to calibrate residual strength of damaged structures, residual strength ratio should be obtained by normalizing residual strength with intact strength. Consequently, at \( d = 0 \), residual strength ratio should equal 1. Given that indentation percentage \( d \) is comparable to knockdown factor \( \mu \) on initial imperfection and web slenderness ratio \( \lambda_w \) is comparable to reference plate slenderness ratio \( \lambda_r \), to perform this normalization, the model can be postulated as:

\[
R(d, \lambda_w)_{ULS} = \frac{\Phi(d, \lambda_w)}{\Phi(0, \lambda_w)} = \frac{p_1 + p_2 \left( \frac{d}{100} \right) + p_3 \left( \frac{\lambda_w}{100} \right)^2 - \sqrt{(p_4 + p_5 \left( \frac{d}{100} \right) + p_6 \left( \frac{\lambda_w}{100} \right)^2)^2 - p_7 \left( \frac{\lambda_w}{100} \right)^2}}{p_1 + p_3 \left( \frac{\lambda_w}{100} \right)^2 - \sqrt{(p_4 + p_6 \left( \frac{\lambda_w}{100} \right)^2)^2 - p_7 \left( \frac{\lambda_w}{100} \right)^2}}
\]

(6-5)
In which \( p_1 \sim p_7 \) are parameters of the formula. The normalizing value 10 and 100 under \( d, \lambda_w \) respectively are used to obtain uniform values of parameters \( p_1 \sim p_7 \) in terms of magnitude and avoid complex number in results. Besides, in the least square problem, a large parameter value may result in a dominant influence in result and hence it will influence distribution of resulting values significantly. Therefore, it is a balanced way to normalize variables by a factor to achievement a more even set of parameters in formula.

![Graph showing Strength, Imperfect Plate](image)

**Fig6-7**: Strength prediction of uniaxially compressed impact plate, M.L. Kaminski, 1992.

The non-linear curve fitting problem is solved by performing the Levenberg–Marquardt algorithm (LMA), also known as the damped least-squares (DLS) method, which is used to solve non-linear least squares problems. Consequently, parameters are determined and listed in Table6-2 and acquired mathematical model is depicted in Fig6-8 with scattered FEM data.

| TABLE 6-2: Values of parameters. |
|-------|-------|-------|-------|-------|-------|-------|
| VALUE | 2.32201 | 2.25975 | 0.65543 | 2.10536 | 2.29692 | 0.97501 | 7.84200 |
6.2.4 Discussion of Postulated Model

The mathematical model postulated is fitted into the data from experimentally verified FEM model. The data consists of different values varying with indentation percentage and web slenderness ratio. The approach to deciding parameters (6-5) in model is to apply Levenberg–Marquardt algorithm to solve the problem of least square minimization. The postulation of model is required to satisfy three basic characteristics of obtained FEM data, (equation 6-1~ equation 6-3): a monotonically decreasing result with two variables (Fig6-8) and normalization at zero indentation percentage. This is accomplished by comparing existing relative research and then converting the model into requirement of this research.

As it is illustrated by Fig6-8, a monotonically decreasing result with two variables is achievement by the postulated model and normalization is accomplished at zero indentation percentage as well. The surface of the model is plotted with FEM data: some data points are scattered above the surface of model and some data points is below the surface. The difference is at an ignorable extent: maximum 2.5% in absolute value in terms of residual strength ratio. Besides, some of the data points are right on the surface of model. It can be concluded that a reasonable fitting is achieved by using mathematically model presented.

Mathematically, the model presents a hyperbolic relationship of residual strength ratio and indentation percentage, and it is monotonically decreasing with indentation percentage increasing. At first few
indentation percentage, the decreasing of residual strength ratio is significantly fast: the first 5% indentation will result in about 40% of strength degradation. Afterwards, the rate of decreasing is slowed down: last 15%-20% of indentation will result in 5% of strength degradation. Physically, it presents the shifting of neutral axis of indented cross-section: the neutral axis is lowered due to large indentation, which gives a smaller offset from neutral axis itself to the most compressed part of the cross-section in bending. With respective to the other variable, web slenderness ratio, at a given indentation percentage, the model also presents a hyperbolic relationship of residual strength ratio and slenderness ratio. In a monotonically decreasing relationship, the rate of strength degradation is slowed down as the web slenderness ratio increasing. Physically, it presents a considerable influence of the capability of a cross-section at developing its plastic stress distribution on residual strength. A compact cross-section is able to develop plastic strength before buckling to gain more residual strength while a slender cross-section buckles before it reaches the yielding strength of material.

The model is only fitting into data with range: $0 \leq d \leq 20, 59 \leq \lambda_w \leq 167$. When the indentation of cross-section exceeds 20% strength degradation is too significant (70%-80%) so that it is not necessary to calibrate the residual strength of damage cross-section. The total cross-section between two transverse stiffeners should be removed in global strength calculation. The range of web slenderness should be able to satisfy most of situations in ship structures as it is previously investigated (Chapter 3). The slenderness beyond presented range should be considered a conservative calculation.

### 6.3 Indentation Location

Using Model2, different locations of indentation in longitudinal direction of specimen is applied Fig6-8). As the center cross-section has a constant bending moment (Fig3-1), the only factors can affect residual strength of specimen is the adjacency of indentation to transverse stiffeners and symmetry of indentation damage. However, this variation has been proved not concerning. The residual strength ratio plot is given in Fig6-9, which shows that indentation location is irrelevant to residual strength ratio.

![Different indentation location](image)

**Fig6-9:** Different indentation location.
Residual strength ratio vs indentation percentage for three varied indentation locations.

6.4 Material Difference

Different steels have varied yielding stress and ultimate tensile stress, with varied corresponding strain. This might exerts influence on the residual strength ratio of specimen. Hence, by using Model1, material characteristics of steel S235, S355 and S420 according to DNV-RP-C208 are inputted to investigate whether properties has influence on residual strength ratio. The true stress-strain curves of adopted material input is plotted in Fig6-10.
Three curves intercept each other at different points but difference between lines is ignorable for engineering purpose (Fig6-11). The minor difference between curves might result from different material having varied corresponding strain to featuring stress, namely yielding stress, ultimate tensile strength, etc., but the influence is still ignorable. It can be concluded that the same formula of residual strength ratio versus indentation percentage can be applied for different materials.

6.5 Elastic Strength

Bureau Veritas is interested in elastic strength of a damaged ship structure part. Hence this section gives an insight into the relation of ultimate strength and elastic strength provided that structure is plastically deformed.

By definition, the elastic strength of a structure is:

The maximum strength a structure can provide before initiation of plastic deformation due to increasing external loading.

To evaluate plastic deformation, equivalent plastic strain is procured from Ansys simulation with FEM Model1 (test verified model). The location of equivalent plastic strain measurement is at the node of maximum lateral displacement after the specimen is buckled (Fig6-12) for each of indentation stages, as plastic deformation is most likely to initiate at this node during increasing load application. Equivalent plastic strain is plotted versus machine load in Fig6-13 for varied levels of indentation. According to the definition of elastic strength, the initiation of equivalent plastic strain can be considered as elastic strength of structure (Fig6-13). By procuring the value of elastic strength, elastic residual strength ratio is plotted (Fig6-14) according to varied slenderness ratio of web. Fig6-6 and
Fig6-14 are based on the same simulations. Compared to Fig6-6, curves in Fig6-14 are irrational, which does not fit any mechanic explanation. This might result from two reasons:

1. Strain is of significantly small value, which means in order to acquire accurate initiation of plastic strain, a refined mesh and minimized time-step should be used in simulation. Due to confinement of computational power available, this is hard to achieve.

2. The inputted stress-strain curve is a simplified multi-linear model based on DNV-RP-C208, which does not fully describe a real stress-strain relationship.

Fig6-13: Lateral displacement plot, point of measurement.

Fig6-14: Machine Force vs Equivalent plastic strain.
As it is discussed above, numerical solution might not be accurate enough due to the confinement of computational value available and limited stress-strain information. Developing an analytical solution based on simulation can be biased and inaccurate. Instead, a conservative design is used to guarantee the elastic residual strength will meet the demand of reliability analysis. The analytical solution of ultimate residual strength ratio is unbiased and rational, to design analytical solution of elastic residual strength ratio based on formulation of $R(d)_{ULS}$ is a correct approach. Studying the relation of elastic residual strength ratio and ultimate residual strength ratio, a residual plot (Fig6-16) is made by subtracting elastic residual strength ratio in Fig6-14 from ultimate residual strength ratio in Fig6-6.
Bureau Veritas prefer residual strength ratio based on elastic strength of structures because a conservative calculation should be applied in reliability analysis of grounded vessels. Considering following facts, designing elastic residual strength of damaged structures by applying formulation (6-5) based on ultimate strength results is conservative:

1. Residual plot Fig6-16 shows that elastic residual strength ratio is possibly higher than ultimate residual strength at only maximum 8% based on available FEM data.

2. Calibration of residual strength is based on calculation of intact strength. The residual strength ratio is multiplied to intact strength to obtain residual strength. Fig3-4 shows intact bending strength calculation from different organizations, in which results from Bureau Veritas indicates at least 25% more conservative results from the fact.

3. In ship society, when applying reliability analysis, yielding stress of material is taken as it is specified but real yielding stress is commonly higher. In this testing case, S235JR steel actually features yielding stress of 293N/mm² rather than 235N/mm².

4. Plate thickness is usually thicker than it is specified. In this test case, ship building steel is used to build specimen. At web of specimen, thickness is actually 5.5mm rather than specified 5mm which is 10% higher.

Based on above facts, it can be concluded that applying residual strength ratio based on formulation 6-5 will fully meet the requirement of a conservative calculation as required by Bureau Veritas.

6.6 Conclusion and Recommendation

A wide series of parametric studies have been conducted to formulate analytical solution of residual strength ratio of an indented specimen, which simulate the transverse frame with attaching double bottom in ship hull structure. Two types of residual strength ratio are studied: one is based on ultimate strength of structure and the other is based on elastic strength of structure. FEM results of ultimate residual is rationally distributed but results based on elastic residual strength ratio is irrational due to limitation of computational power available. Hence, instead of directly developing elastic residual strength ratio from FEM results, reasons are given to indicate using residual strength ratio based ultimate strength is conservative to the design of elastic strength.

The conclusion has been drawn that the residual strength ratio (ultimate/elastic) is only relevant with indentation percentage and web slenderness ratio, which can be expressed in formula (6-5).

Generally, a compact cross-section has more residual strength when plastically deformed compared to a slender cross-section (Fig6-6). This difference becomes more significant when indentation becomes larger.

The same analytical solution above can be applied for:

1. Calculation of ultimate residual strength and elastic residual strength.
2. Cross-sections with different aspect ratios, when \( \text{web width/web height} \geq 1 \).
3. Cross-sections with different indentation width and size of indenter.
4. Cross-sections with different attached flanges, measuring in flange width and flange thickness.
(5) Cross-sections with different indentation location.
(6) Cross-sections with different building steel.

The utilization of the analytical solution is simple: after grounding or collision damage, the indentation depth should be measured, and one can get its residual strength ratio instantly with known slenderness of web. Still, some recommendation of using the formula can be proposed as following:

(1) The targeted cross-section should has a web slenderness between 60 and 165. Normally, it is found that web slenderness ratio is 75-150 in ship structures. When the slenderness ratio is not known, the upper bound (165) of web slenderness ratio in equation should be adopted, for a conservative design.

(2) The drop of bending strength due to indentation is quite significant. 20% indentation of total height may results in a strength reduction of 70% -85% (ULS) according to different web slenderness. In practical situations, once the web is considered to be indented more than 20% the cross-section should be totally removed from global bending strength calculation of ship hull structures.

(3) First little indentation related to a significant drop of bending strength, it should be keep in mind for bending strength calculation: when situation should be very conservative, a indented cross-section should be totally removed from global calculation of bending strength of ship hull structures.

(4) Rupture of material is not included in this study, when large rupture occurs on material, the residual bending strength should be calculated otherwise.

(5) When indentation occurs right on a transverse stiffener, the area of indentation should be considered as the area between two adjacent transverse stiffeners to the indented stiffener. The geometry of indented stiffener should not be included in calculation. This user rule is for conservative consideration of design.

(6) In ship hull structures, a number of other structural parts beyond transverse frame is I-shaped, namely secondary stiffener and longitudinal girder, etc. The analytical solution demonstrated above cannot only be used to calibrate residual strength of a damaged transverse frame but also to calibrate residual bending strength of damaged these structures.
Chapter 7

Summary and Recommendation

7.1 Summary

The article is oriented to quantify the residual strength of ship hull structures after grounding according to varied levels of plastic deformation. Literature study indicates that most of studies are focused on longitudinal residual strength of damaged ship structures, and transverse residual strength is rarely researched. Hence, it is target to study transverse residual strength after grounding damage, with implementation of a joint study of test and finite element simulation.

7.1.1 Numerical Simulation

Finite element package Ansys is adopted as tool of simulation. Selection is made due to its reproductive nature of using APDL language. As non-linear simulation is involved, the proper set-up of material non-linearity is first step to gain an accurate result. Boundary conditions, mesh quality, element type, load application method and choice and configuration of solver are also critical factors. Two material models based on DNV-RP-C208 are adopted for different purposes: one for test design and the other for data verification and formulation. As large strain simulations are involved, true stress-strain curve is adopted to reflect plastic flow in material. Additionally, working hardening is examined in terms of loading direction, and isotropic hardening model is adopted as simulation shows that direction of loading rarely changes throughout indentation process and four point bending. For a convergent solution, load application method and finite element solver are investigated, and displacement control is adopted for this particular simulation as it follows the load-displacement correctly in non-linear simulation such as bifurcation and buckling. Accuracy and computational time are discussed to achieve a balanced simulation.

Initial imperfection induced by fabrication has a considerable influence on strength of specimen, extensive literature are studied and concept of equivalent initial imperfection is introduced and incorporated into simulation to take this factor into account. Moreover, springback after offloading of indentation process is analyzed and its influence on geometry and residual stress distribution is inherited into four point bending simulation.

7.1.2 Specimen Design and Test

The design of specimen has been fulfilled in five aspects:

(1) Determine bending capacity of the middle cross-section.
(2) Shear buckling design of side webs.
(3) No pre-buckling at transverse stiffeners.
(4) Lateral torsional buckling design.
(5) Considering the slenderness of cross-section in real ship structure, make the design comparable to practice.
Design rules of Bureau Veritas, Det Norske Veritas and Eurocode 3 are examined and compared to results of simulation to guarantee a proper design of specimen. Test is carried out in five stages, and each stage contains a set of corresponding indentation process and four point bending procedure to measure residual bending strength of plastically damaged specimen. In terms of damage control, the loading of each four point bending process is conservatively designed to guarantee plastic deformation is only applied by indentation process which is comparable to grounding damage. Testing accessories are designed to provide proper boundary condition and load application of two testing procedures. Digital Image Correlation is adopted as test measurement which monitors lateral displacement of specimen throughout the whole testing procedures.

7.1.3 Data Verification and Analytical Solution

Experimental data and FEM data are compared for indentation process and four point bending. Testing data of four point bending contains large amount of systematic misalignment and a graphic subtraction method is proposed and validated to cancel these misalignment. Multiple simulation indicates that friction at contacts has a significant influence on the strength of specimen. A wide range of friction coefficient is therefore studied numerically. The influence of amount of imperfection on specimen is also discussed and proposition is given and validated that test results and numerical results becomes convergent when indentation is largely applied as the influence of fabrication imperfection drops dramatically due to the rise of the amount of indentation imperfection. The data comparison shows that test model and FE model has a reasonable agreement and therefore FE model is proved to be accurate.

A wide series of parametric studies have been conducted by simulation to formulate analytical solution of residual strength ratio. Two analytical solutions of residual strength ratio are studied: one is based on ultimate strength of structure (6-5) and the other is based on elastic strength of structure. The conclusion has been drawn that the residual strength ratio (ultimate/elastic) is only relevant with indentation percentage and web slenderness ratio. Generally, a compact cross-section has more residual strength when plastically deformed compared to a slender cross-section (Fig6-6). This difference becomes more significant when indentation becomes larger. Finally, the utilization of two analytical solutions is discussed and guidance is given for practical engineering design.

7.2 Discussion and Recommendation

Thesis is oriented to quantify strength reduction of a grounded ship structure according to varied levels of plastic deformation. According to American Bureau of Shipping, 1995 [5], current reliability analysis of grounded ship structure in most of organizations adopts total elemental removal method (Fig1-2), which did not take residual strength of a plastically deformed structure into account. To calibrate residual strength and improve accuracy in estimation, analytical solution of residual strength ratio according to varied plastic deformation is formulated in this article. It is based on the assumption that plastic deformation happens without occurrence of rupture and crack. In such way, damage is more confined in its appearance and forms so that a specified experiment can be applied to fulfil the study. Such type of damage is usually observed in grounding damage. Extensive experimental studies are designed to study its configuration and formation. H. Ohtsubo, 1994 [4] researched grounding damage by apply similar indentation process in a full scale test to study structural resistance to grounding damage, compared it dynamically and quasi-statically. Lin Hong, 2008 [2], compared different experiments of indentation carried out to formulate crushing resistance of web panel and established theory of web folding in terms of possible shapes.

However, grounding damage is random in its location and appearance. Although, plastic damage is one
single mostly occurred form, many grounding damages are accompanied with penetration of material, ruptures and cracks. Those articles can be referred to: Hagbart S. Alkos, 2009 [30], carried out an experiment to study penetration resistance of a stiffened panel. Anuar AbuBakar, 2013 [29], simulated ship grounding damage numerically, with plastic deformation accompanied with material penetration, and grounding resistance was studied. The confinement of these studies is that only grounding resistance is studied but without establishment of residual strength estimation based on those types of damage.

This article provides an accurate solution on residual strength of plastically deformed structures based on extensive parametric studies with experimentally verified numerical model. But to encapsulate grounding damage for different forms, above studies need to be incorporated. A list of recommendation is given:

- The experiment has been conducted in a quasi-static way, which did not take dynamic effect of impact due to grounding into account. A dynamic test needs to be conducted to fill the blank.
- Rupture is commonly seen in grounding damage. This should be studied and tested to incorporate into the analytical solutions derived in this study.
- Crack propagation is also a matter of concern when rupture of material is incorporated.

With experience of experiment in this article, two recommendation are given:

- Imperfection control is the key to successive test. It should be thoroughly studied in design procedures and rigidly controlled in the fabrication of specimen.
- When study of stress and strain is concerned, a tensile test should be conducted to determine an accurate stress-strain curve.
Reference


[26] Pre Young Bong Kwon, Gun Ho Seo, Prediction of the flexural strengths of welded H-sections with local buckling, Thin-Walled Structures, Volume 54, May 2012, Pages 126-139, ISSN 0263-8231.


[31] DNV-OS-C401, fabrication and testing of offshore structures, October 2010.
Appendix A

A.1 Design of Ultimate Bending Moment Resistance with Eurocode3

Three calculation are shown here. The calculating procedures for class1 and class2 are the same, so only calculation of class1 cross-section is demonstrated. Other two are the calculation for class3 and class4 cross-section. The spacing, a, of webs is 500mm long.

FigA-1: four point bending test and specimen profile. FigA-2: loading distribution.

Specification of cross-section

<table>
<thead>
<tr>
<th>Cross-section number</th>
<th>$t_w$ (mm)</th>
<th>$t_f$ (mm)</th>
<th>$h_w$ (mm)</th>
<th>$b_f$ (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6</td>
<td>8</td>
<td>400</td>
<td>100</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>8</td>
<td>400</td>
<td>100</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>8</td>
<td>400</td>
<td>100</td>
</tr>
</tbody>
</table>

Yielding stress of material is 235 N/mm$^2$. Young’s modulus is 210000 N/mm$^2$. Poisson ratio=0.3.

TableA-1: specimen specification.

Calculation for cross-section No.1

Classification of cross-section

$$\varepsilon = \sqrt{\frac{235}{f_y}} = \sqrt{\frac{235}{235}} = 1$$

The boundaries for classification are:

| Flange slenderness, $c/t_f$ | Web slenderness, $h_w/t_w$ |
### TableA-2: classification of cross-section.

<table>
<thead>
<tr>
<th>Class</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Class1</td>
<td>9</td>
<td>72</td>
</tr>
<tr>
<td>Class2</td>
<td>10</td>
<td>83</td>
</tr>
<tr>
<td>Class3</td>
<td>14</td>
<td>124</td>
</tr>
<tr>
<td>Class4</td>
<td>If the slenderness ratio exceeds class3 boundary, it is class4.</td>
<td></td>
</tr>
</tbody>
</table>

Actual slenderness for web:

\[
\frac{h_w}{t_w} = \frac{400}{6} = 66.67 < 72
\]

The web belongs to class1.

Actual slenderness for flanges:

\[
\frac{e}{t_f} = \frac{100}{2}/8 = 6.25 < 9
\]

The flanges belong to class1. By choosing the least favorable classification, it is a class1 cross-section.

**Section modulus**

Therefore, the cross-section modulus for ultimate bending moment calculation is plastic section modulus, which is:

\[
W_y = W_{pl} = A_c y_c + A_t y_t = 2 \cdot (8 \cdot 100 \cdot 204 + 6 \cdot 200 \cdot 100) = 566400 \text{ mm}^3
\]

**Ultimate bending resistance**

\[
M_{c,rd} = \frac{W_y f_y}{\gamma_{MO}} = \frac{566400 \cdot 235}{1.0} \cdot 10^{-6} = 133.104 \text{ KN} \cdot \text{m}
\]

**Calculation for cross-section No.2**

**Classification of cross-section**

Actual slenderness for web:

\[
\frac{h_w}{t_w} = \frac{400}{4} = 100 > 83
\]

The web belongs to class3.
Actual slenderness for flanges:
\[
\frac{c}{t_f} = \left(\frac{100}{2}\right)/8 = 6.25 < 9
\]

The flanges belong to class 1. By choosing the least favorable classification, it is a class 3 cross-section.

**Section modulus**

Therefore, the cross-section modulus for ultimate bending moment calculation is elastic section modulus, which is:

\[
I_{el} = \frac{t_w b_w^3}{12} + 2 \cdot \left[ \frac{b_y t_f^3}{12} + b_f t_f \cdot \left(\frac{b_w}{2} + \frac{t_f}{2}\right)^2 \right] = 98594133.33 \text{ mm}^4
\]

\[
W_y = W_{el} = \frac{I_{el}}{\Delta y} = \frac{98594133.33}{200 + 4} = 483304.5752 \text{ mm}^3
\]

Ultimate bending resistance

\[
M_{cr,d} = \frac{W_y f_y}{\gamma_{M0}} = \frac{483304.5752 \cdot 235}{1.0} \cdot 10^{-6} = 101.29 \text{ KN} \cdot \text{m}
\]

**Calculation for cross-section No. 3**

**Classification of cross-section**

Actual slenderness for web:

\[
\frac{h_w}{t_w} = \frac{400}{2} = 200 > 124
\]

The web belongs to class 4.

Actual slenderness for flanges:

\[
\frac{c}{t_f} = \left(\frac{100}{2}\right)/8 = 6.25 < 9
\]

The flanges belong to class 1. By choosing the least favorable classification, it is a class 4 cross-section.

**Section modulus**
Therefore, the cross-section modulus for ultimate bending moment calculation is effective section modulus. Procedure for this calculation is referred to NEN-EN 1995-1-5.

1) Elastic buckling stress

The boundary condition is taken as simply supported, thus $k = 23.9$.

$$\sigma_{E,1} = k \frac{\pi^2 E}{b^2 t} = k \frac{\pi^2 E}{12(1 - v^2)} \left(\frac{t_w}{h_w}\right)^2 = 23.9 \cdot \frac{3.14^2 \cdot 210000}{12(1 - 0.3^2)} \cdot \left(\frac{2}{400}\right)^2 = 113.3 \text{ N/mm}^2$$

2) Effective length

$$\overline{\lambda} = \frac{f_y}{\sigma_{E,1}} = \sqrt{\frac{235}{113.3}} = 1.44$$

$$\rho = \frac{\overline{\lambda} - 0.11}{\overline{\lambda}^2} = 0.641$$

$$b_{eff} = \frac{\rho \cdot h_w}{2} = 128.26 \text{ mm}$$

Upper effective length:
$$s_1 = 0.4b_{eff} = 51.3 \text{ mm}$$

Ineffective zone length:
$$s_2 = \frac{h_w}{2} - b_{eff} = 71.4 \text{ mm}$$

Lower effective length:
$$s_1 = 0.6b_{eff} = 76.9 \text{ mm}$$

3) Area of net effective section

$$A_{eff} = 2b_ft_f + t_w \left(\frac{h_w}{2} + s_1 + s_2\right) = 2256.52 \text{ mm}^2$$

4) Neutral axis of cross-section
\[ \gamma_{n.a} A_{\text{eff}} = \frac{t_f^2 b_f}{2} + \left( \frac{h_w}{2} + s3 \right) t_w \left( t_f + \frac{\left( \frac{h_w + s1}{2} \right)}{2} \right) + s1 \cdot t_w \left( h_w + t_f - \frac{s1}{2} \right) + t_f b_f \left( h_w + t_f + \frac{t_f}{2} \right) \]

\[ = 453167.3 \text{ mm}^3 \]

\[ \gamma_{n.a} = \frac{453167.3}{A_{\text{eff}}} = 200.83 \text{ mm} \]

The neutral axis is lowered:

\[ e_N = \frac{h_w}{2} + t_f - \gamma_{n.a} = 7.17 \text{ mm} \]

5) Effective moment of inertia

The calculation is done as to sum up different components’ second moment of inertia to the neutral axis (FigA2).

\[ I_{\text{eff}} = \sum_{i=1}^{5} I_{i,N} = 75248113.06 \text{ mm}^4 \]

\[ \Delta y_c = h_w + 2t_f - \gamma_{n.a} = 215.17 \text{ mm} \]

\[ W_{\text{eff}}^c = \frac{I_{\text{eff}}}{\Delta y_c} = 349708.0632 \text{ mm}^3 \]

Where \( \Delta y_c \) is the height of compression part and \( W_{\text{eff}}^c \) is the compression section modulus.

**Ultimate bending resistance**

\[ M_{c,rd} = \frac{W_{\text{eff}}^c f_y}{\gamma_{Mo}} = \frac{349708.06}{1.0} \cdot 10^{-6} = 82.18 \text{ KN} \cdot \text{m} \]

\[ P_{b,uls} = \frac{M_{c,rd}}{a} = \frac{82.18}{0.5} = 164.36 \text{ KN} \]

FigA-4: Effective section numbering.
A.2 Design of Shear Buckling Capacity

The specimen specification No.3 is chosen for sample calculation. The design generally consists of two sub-steps: 1. Determination of shear capacity of side webs. 2. Design transverse stiffeners to withstand tension field action.

As \( \frac{a}{h_w} = \frac{500}{400} \geq 1 \), the shear buckling coefficient is:

\[
k_T = 5.34 + \frac{4}{\left( \frac{a}{h_w} \right)^2} = 5.34 + \frac{4}{\left( \frac{500}{400} \right)^2} = 7.9
\]

The Euler elastic buckling stress of the plate is:

\[
\tau_E = k_T \frac{\pi^2 E}{b^2 t} = k_T \frac{\pi^2 E}{12(1-v^2)} \left( \frac{t_w}{h_w} \right)^2 = 37.44 \text{ N/mm}^2
\]

\[
\tau_F = \frac{\sigma_T}{\sqrt{3}} = \frac{235}{\sqrt{3}} = 135.68 \text{ N/mm}^2
\]

\( \tau_E < 0.5\tau_F \), so according to the Johnson-Ostenfeld formula, the critical shear buckling stress is:

\[
\tau_{cr} = \tau_E = 37.44 \text{ N/mm}^2
\]

The shear buckling strength of web is:

\[
V_{cr} = h_w t_w \tau_{cr} = 29958.015 \text{ N}
\]

The extra bearing stress due to tension field action is:

\[
\tau_{tf} = \frac{\sigma_T}{2} \frac{1 - \tau_{cr}/\tau_F}{\sqrt{1 + (a/h)^2}} = \frac{235}{2} \frac{1 - 37.44/135.68}{\sqrt{1 + \left( \frac{500}{400} \right)^2}} = 53.14 \text{ N/mm}^2
\]

Thus, contribution from tension field action is:

\[
V_{tf} = h_w t_w \tau_{tf} = 42513.9948 \text{ N}
\]
The ultimate shear is, by neglecting the contribution from flanges:

\[ V_u = V_{cr} + V_{tf} + V_f = 29958.015 + 42513.9948 + 0 = 72472.01018 \ N = 72.472 \ KN \]

Comparing with the ultimate bending strength calculated in the last section:

\[ P_{b,uls} = 164.36 \ KN > V_u = 72.472 \ KN \]

Thus shear buckling of side webs would occur before the local buckling of middle web.

### A.3 Design of Transverse Stiffener

Transverse stiffener force due to tension field action is:

\[
P_s = \frac{\sigma_{yw} h t_w}{2} \left( 1 - \frac{t_{cr}}{t_y} \right) \left( \frac{a}{h} - \frac{(a/h)^2}{\sqrt{1+(a/h)^2}} \right) = \frac{235 \cdot 400 \cdot 2}{2} \left( 1 - \frac{37.44}{135.68} \right) \left[ \frac{500}{400} - \frac{(500/400)^2}{\sqrt{1+(500/400)^2}} \right] = 18641.38 \ N
\]

Consider a transverse stiffener of 5mm thickness and 50mm half width:

\[ A_s = 250 \ mm^2 > \frac{P_s}{\sigma_{yw}} = 79.32 \ mm^2 \]

\[ I_s = 52083.3 \ mm^4 > \frac{P_s h^2}{\pi^2 E} = 1440.5 \ mm^4 \]
Appendix B

Manufacture proposal of specimen, includes:

1. Fabrication requirement.
2. Specimen drawings.
RAPID

Specimen Fabrication

Part of test, Master Thesis

You Zhou
Specimen Fabrication

1. Tolerances for Straightness
In accordance to DNV-OS-C401, fabrication tolerances in terms of specimen length, web and flange are subsequently defined.

1) Tolerances on the straightness over specimen length, bow shaped:

\[ e_0 = 0.0015L = 0.0015 \times 1700 = 2.55 \text{mm} \]

Fig1: Tolerance over specimen length.

The maximum out of straightness is:

\[ e_0 = 0.0015L = 0.0015 \times 1700 = 2.55 \text{mm} \]

2) Tolerances on the straightness of middle web, sinusously shaped:

\[ e_{ow} = M\text{m}(0.005a, 0.005h_w) \]

\[ = 0.005 \times 500 = 2.5 \text{mm} \]

Fig2: Tolerance on center web of specimen.

The maximum out of straightness is:

\[ e_{ow} = M\text{m}(0.005a, 0.005h_w) \]

\[ = 0.005 \times 500 = 2.5 \text{mm} \]

2. Welds
In accordance to DNV rules for steel ships, Pt.3Ch.1Sec.11 welding and weld connections, requirement for the welding and weld connections are hereby specified.

Welding Consumables
The material used for specimen fabrication featuring yielding stress of 235N/mm². In order to give strength reserve, the yielding stress of welding consumables should be at 355N/mm².

Fillet Welds
The fillet welds are adopted to fabricate the specimen. Due to varied loading (tension, compression and shear) at different location of weld, rule of DNV are adopted in the design of weld size, and conservative values are taken as possible high stress concentration in the test. The requirement of fillet weld is defined by requested minimum throat thickness of welds (Fig3).

Fig3: definition of weld size, T-joint (DNV, 2014).

Minimum throat thickness at different locations are specified as (Fig4, Table1):

<table>
<thead>
<tr>
<th>Location</th>
<th>Throat thickness (mm), Fig3</th>
</tr>
</thead>
<tbody>
<tr>
<td>L1</td>
<td>3.0</td>
</tr>
<tr>
<td>L2</td>
<td>4.5</td>
</tr>
<tr>
<td>L3</td>
<td>4.0</td>
</tr>
<tr>
<td>L4</td>
<td>4.0</td>
</tr>
</tbody>
</table>

Table1: minimum throat thickness requirement for fillet welds.

Fig4: location of different weld.

L1: weld web to the top flange;
L2: weld web to the bottom flange;
L3: weld transverse stiffener to web;
L4: weld shear stiffener to web.

Welding Test
Please follow the normal procedure of DAMEN ship yard to inspect welds.
TU Delft specimen production

355N/mm²

For fabrication tolerance and fillet weld size, please refer to file, specimen fabrication.
Appendix C

Manufacture proposal of testing accessories, includes:

1. Fabrication requirement.
2. Testing accessories drawings.
Residual strength of ship panel after grounding

Fabrication of Test Accessories (headers&supports)

You Zhou
Fabrication of Test Accessories

1. Summary
Fabrication of test accessories includes six objects: two end supports, loading connector, indenter and two loading headers. The specified drawings of those objects are attached. The connectable parts are attached to each other by spikes. Steel of yielding stress 235N/mm² is adopted for production.

2. Fabrication Tolerances
Fabrication requirement is necessary at the plating which have openings, so as to two objects can connect by screws through openings:
(1) The diameter of openings should not be less than the specified.
(2) The angular errors should not greater than 0.1 deg.
(3) The linear errors should not be greater than 0.1mm.
No other fabrication requirement is indicated if the openings fit each other.

3. Welds
In accordance to DNV rules for steel ships, Pt.3Ch.1Sec.11 welding and weld connections, requirement for the welding and weld connections are specified in the attached drawings. In order to give strength reserve to plating, the yielding stress of welding consumables should be at 355N/mm². Fillet welds are adopted for production. Please refer to procedures at DAMEN to inspect welding.

4. Surface Finish
Spherical surfaces are specified to a certain surface finish to create suitable boundary conditions for testing. The marks are made on drawing attached around the required parts.

5. Screws
Screws are used to connect two connectable parts, two configuration of screws are used in confirmation of ISO standards:
1. M38 tapping type, is used to connect loading connector to testing machine, provided by AE lab, TUD.
2. M16 tapping tape, 15.95mm in nominal diameter, is used to connect the loading connector to loading headers or indenter. The length of screw should not be less than 40mm. Please provide M16 screws with suitable bolts if applicable.
1. Two end supports are required for production.
2. Surface are required for spherical surface and fillets.
3. Continuous fillet welding are adopted for production.

End Support

COMMENT:
1. Yielding stress=235N/mm²
2. Welding consumable=355N/mm²
3. Continuous fillet welding are adopted for production.

WEIGHT: 25.9 kg
1. Surface are required for spherical surface and fillets.
2. Continuous fillet welding are adopted for production.
3. Openings at plates are required not less than specified diameter.

Yielding stress = 235 N/mm²
Welding consumable = 355 N/mm²

MATERIAL:

Comment:

Indenter

WEIGHT: 14 kg
Loading Connector

Comment:
1. Continuous fillet welding are adopted for production.
2. Openings at plates are required not less than the specified diameter.

Yielding stress: 235 N/mm²
Welding consumable: 355 N/mm²

MATERIAL:
13.11 Y.Zhou

WEIGHT: 26 kg
1. Surface are required for spherical surface.
2. Continuous fillet welding are adopted for production.
3. Openings or pins are required not less than specified diameter.

Yielding stress = 235 N/mm²
Welding consumable = 355 N/mm²

MATERIAL:

Comment:

WEIGHT: 9 kg
Loading Header- Left side

Comment:
1. Surface area required for spherical surface.
2. Continuous fillet welding are adopted for production.
3. Openings of plates are required not less than specified diameter.

Yielding stress = 235 N/mm²
Welding consumable = 355 N/mm²

MATERIAL:
13.11 Y. Zhou

UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH: Ra = 1 µm
TOLERANCES:
LINEAR: ±0.1 mm at openings
ANGULAR: ±0.1 deg at openings

WEIGHT: 9 kg
Connect loading headers to loading connector
Connect indenter to loading connector
Appendix D

Material certificate of specimen provides by DAMEN B.V.

*Acknowledge is given to DAMEN B.V. to produce specimen and testing accessories.*
### Tata Steel IJmuiden BV

**Kievitstraat 1**

**Postbus 10885**

**IJmuiden, The Netherlands**

**Telefoon:** 051-4951111 **Quality Department**

**Telefoon:** 051-4951101 **Quality Department**

**Trade Register:** 34040251

---

**INSPECTION CERTIFICATE 3.1, EN 10204**

**HOT ROLLED COIL S235JR+AR EN 10025-2**

**TOLERANCE EN 10051.**

---

**SERVICE CENTRE MAASRICHT B.V.**

**B.U. FEIJEN**

**POSTBUS 3040**

**6292 NA MAASRICHT**

**LIMBURG**

---

**ORDERNO.**

**51021 A**

---

**TREKPROEF**

**HARDHEID**

**KERSFLAGPROEF EN OVERIGE PROEVEN**

---

**CHEMISCHE SAMENSTELLING in %**

---

**STALBEREIDINGSWIJZE:** BD

---

**Het materiaal voldoet aan de in de bestelling gestelde eisen.**

---

**Stempel van de deskundige:**

---

**J.M. VAN DER WAL**

**MANAGER TESTING**

**IJMUIDEN, 6 JULI 2013**
Appendix E

!------------------------------------------------------------------------------------------Analysis Initiation
finish$/clear
/filname, sindentation25
/prep7
local,11,0,0,0,0,0,0,0,0
esys,11
!-------------------------------------------------------------------------------------------User Parameters
*SET,tw,5.5    ! Centre web thickness.
*SET,sidetw,5.5          ! Side web thickness.
*SET,h,500              ! Web height.
*SET,tf,8.1     ! Flange thickness.
*SET,bf,200             ! Flange width.
*SET,a,500     ! Side web Span.
*SET,a0,500     ! Centre web span.
*SET,ts,8.1     ! End transverse stiffener thickness.
*SET,ts0,8.1     ! Centre transverse stiffener thickness.
*SET,tl,10.3     ! Diagonal stiffener thickness.
*SET,div1,25    ! Horizontal mesh division.
*SET,div2,5     ! Vertical mesh division 1.
*SET,div3,5    ! Vertical mesh division 2.
*SET,e,2.5      ! Magnitude of initial imperfection.
*SET,F,-1     ! Unit force, eigenvalue buckling analysis.
*SET,dis,25    ! Indentation depth.
*SET,dis2,8     ! Machine loading displacement, four point bending.
*SET,Ra,50     ! Radius of Indenter and loading headers.
*SET,fc,0.25    ! Fiction coefficient.
!-----------------------------------------------------------------------------------------System Parameters
*SET,dis1,dis+10   ! Actual indenter displacement.
*SET,S,1               ! 1 for contact bending; 0 for nodal bending.
*SET,LS,1       ! Diagonal stiffener type: 1, compression stiffener; 2, tension
!-stiffener; 3, horizontal stiffener.
*SET,CD,1     ! Lateral restraints opts, Eigenvalue Analysis.
!---------------------------------------------------------------------------------------------Assigning real constant
R,1,tf, , , , , ,
R,2,tw,
R,3,ts, , , , , ,
R,4,tl, , , , , ,
R,5,sidetw, , , , , ,
!---------------------------------------------------------------------------------------------Element Type& Linearity of Material
et,1,shell181
!KEYOPT,1,3,2     ! De-comment for a more precise analysis
<table>
<thead>
<tr>
<th>Material temperature</th>
<th>!Material temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modules</td>
<td>!Young’s modules</td>
</tr>
<tr>
<td>Poison ratio</td>
<td>!Poison ratio</td>
</tr>
</tbody>
</table>

Model Build-up

<table>
<thead>
<tr>
<th>Layer Description</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>k,,,-h/2-tf/2,-bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-h/2-tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-h/2-tf/2, bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-h/2+tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-h/2+tf/2,-bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-h/2-tf/2,-bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-h/2-tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-h/2-tf/2, bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-h/2+tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-h/2+tf/2,-bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-a+0,-h/2-tf/2,-bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-a+0,-h/2-tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-a+0,-h/2-tf/2, bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-a+0, h/2+tf/2, bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-a+0, h/2+tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-a+0, h/2+tf/2, -bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-2*a+0,-h/2-tf/2,-bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-2*a+0,-h/2-tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-2*a+0,-h/2-tf/2, bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-2*a+0, h/2+tf/2, bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-2*a+0, h/2+tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-2*a+0, h/2+tf/2, -bf/2</td>
<td></td>
</tr>
<tr>
<td>k,,,-100,-h/2-tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-100,-h/2-tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-100,-h/2-tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-100,-h/2+tf/2,</td>
<td></td>
</tr>
<tr>
<td>k,,,-100,-h/2+tf/2,</td>
<td></td>
</tr>
<tr>
<td>a,12,18,17,11</td>
<td></td>
</tr>
<tr>
<td>a,11,17,16,10</td>
<td></td>
</tr>
<tr>
<td>a,7,13,14,8</td>
<td></td>
</tr>
<tr>
<td>a,8,14,15,9</td>
<td></td>
</tr>
</tbody>
</table>
a,11,17,14,8
a,5,11,10,4
a,6,12,11,5
a,18,24,23,17
a,17,23,22,16
a,1,7,8,2
a,2,8,9,3
a,13,19,20,14
a,14,20,21,15
a,5,11,8,2
a,17,23,20,14
a,5,2,3,4
a,6,1,2,5
a,12,7,8,11
a,11,8,9,10
a,17,18,13,14
a,17,14,15,16
a,24,19,20,23
a,23,20,21,22
a,1,2,11,12
a,2,11,10,3
a,17,20,21,16
a,18,19,20,17
FLST,2,4,3
FITEM,2,30
FITEM,2,6
FITEM,2,5
FITEM,2,29
A,P51X
FLST,2,4,3
FITEM,2,29
FITEM,2,5
FITEM,2,4
FITEM,2,28
A,P51X
FLST,2,4,3
FITEM,2,25
FITEM,2,1
FITEM,2,2
FITEM,2,26
A,P51X
FLST,2,4,3
FITEM,2,26
FITEM,2,2
Mesh Attributes

FLST,5,8,5,ORDE,2
FITEM,5,28
CMSEL,S,_Y1
AATT,  1,  2,  1,  0,
CMSEL,S,_Y
CMDELE,_Y
CMDELE,_Y1
FLST,5,8,5,ORDE,2
FITEM,5,16
FITEM,5,-23
CM,_Y,AREA
ASEL,, , ,P51X
CM,_Y1,AREA
CMSEL,S,_Y
CMSEL,S,_Y1
AATT,  1,  3,  1,  0,
CMSEL,S,_Y
CMDELE,_Y
CMDELE,_Y1
/VIEW,1,1,2,3
/ANG,1
/REP,FAST
FLST,5,4,5,ORDE,2
FITEM,5,24
FITEM,5,-27
CM,_Y,AREA
ASEL,, , ,P51X
CM,_Y1,AREA
CMSEL,S,_Y
CMSEL,S,_Y1
AATT,  1,  4,  1,  0,
CMSEL,S,_Y
CMDELE,_Y
CMDELE,_Y1
allsel,all
aplot
!----------------------------------------------------------------------------------------Mesh Size Control
allsel,all
lsel,none
lsel,s,loc,z,bf/2
lsel,a,loc,z,-bf/2
lsel,a,loc,z,0
lsel,r,loc,x,-1,1501
LESIZE,ALL, , ,div1, ,1, , ,1,
MSHKEY,0
allsel,all
lsel,u,loc,z,bf/2
lsel,u,loc,z,-bf/2
lsel,u,loc,z,0
lsel,r,loc,x,-1,1501
LESIZE,ALL, , ,div2, ,1, , ,1,
alls
lsel,s,loc,x,-50,-1
lsel,a,loc,x,1501,1550
lsel,u,loc,y,0
LESIZE,ALL, , ,div3, ,1, , ,1,
alls
lsel,s,loc,x,-50,-1000
lsel,a,loc,x,1501,2000
lsel,r,loc,y,0
LESIZE,ALL, , ,div1, ,1, , ,1,
alls
lsel,s,loc,x,-50,-1000
lsel,a,loc,x,1501,2000
lsel,u,loc,y,0
lsel,u,loc,x,-50
lsel,u,loc,x,1550
LESIZE,ALL, , ,div2, ,1, , ,1,
amesh,all
MSHKEY,0
CMDELE, _Y
CMDELE, _Y1
CMDELE, _Y2
Nummrg,elem !Emerge close elements
Nummrg,node !Emerge close nodes
allsel,all
eplot
allsel,all
numcmp,all !Compress node numbers
/eshape,1 !Plot elements in 1:1 size
!!-------------------------------------------------------------Boundary Condition Eigenvalue Analysis
allsel,all
nsls,s,loc,x,0
nsls,r,loc,y,h/2+tf/2
nsls,r,loc, z, 0
D,all, , , , ,,,UZ,
allsel,all
nsls,s,loc,x,3*a
nsls,r,loc,y,h/2+tf/2
nsls,r,loc, z, 0
D,all, , , , ,,UZ,
allsel,all
Nsel,s,loc,y,-h/2-tf/2
nsel,r,loc, x, 0
nsel,r,loc, z, 0
D,all, , , , ,,UZ,
allsel,all
nsel,s,loc,x,3*a
nsel,r,loc,y,-h/2-tf/2
nsel,r,loc, z, 0
D,all, , , , ,,UZ,
*if,CD,eq,1,then
allsel,all
nsel,s,loc,x,a
nsel,r,loc,y,h/2+tf/2
nsel,r,loc, z, 0
D,all, , , , ,,UZ,
allsel,all
nsel,s,loc,x,2*a
nsel,r,loc,y,h/2+tf/2
nsel,r,loc, z, 0
D,all, , , , ,,UZ,
allsel,all
Nsel,s,loc,y,-h/2-tf/2
nsel,r,loc, x, a
nsel,r,loc, z, 0
D,all, , , , ,,UZ,
allsel,all
nsel,s,loc,x,2*a
nsel,r,loc,y,-h/2-tf/2
nsel,r,loc, z, 0
D,all, , , , ,,UZ,
*elseif,CD,eq,2,then
allsel,all
nsel,s,loc,y,-h/2-tf/2
nsel,r,loc, z, 0
D,all, , , , ,,UZ,
allsel,all
nsel,s,loc,x,0
nsel,r,loc, z, 0
D,all, , , , ,,UZ,
allsel,all
Nsel,s,loc,y,h/2+tf/2
nsel,r,loc, z, 0
D,all, , , , ,,UZ,,
allsel,all
nsel,s,loc,x,3*a
nsel,r,loc, z, 0
D,all, , , , ,,UZ,,
*endif
allsel,all
Nsel,s,loc,y,-h/2-tf/2
nsel,r,loc, x, 0
nsel,r,loc, z, 0
D,all, , , , ,,UY,,
allsel,all
nsel,s,loc,x,3*a
nsel,r,loc,y,-h/2-tf/2
D,all, , , , ,,UY,,
allsel,all
Nsel,s,loc,y,-h/2-tf/2
nsel,r,loc, x, 0
D,all, , , , ,,UX,,

!!!!-----------------------------------------------------------------------------------------------Loading of Eigenvalue Analysis
allsel,all
nsel,s,loc,y,h/2+tf/2
nsel,r,loc,x,a
*Get,nnode,node,,count
F,all,fy,F/nnode
allsel,all
nsel,s,loc,x,2*a
nsel,r,loc,y,h/2+tf/2
F,all,fy,F/nnode
allsel,all

!!!!----------------------------------------------------------------------------------------------Nodal Components 1
alls
nsel,s,loc,x,750,770
nsel,r,loc,y,-tf/2-h/2
CM,LowX,NODE
alls
nsel,s,loc,x,750,770
nsel,r,loc,y,tf/2+h/2
CM,UpperX,NODE
alls
nsel,s,loc,x,-100
nsel,r,loc,y,-tf/2-h/2
CM,LowX1,NODE
alls
nssel,s,loc,x,1600	nssel,r,loc,y,-tf/2-h/2
CM,LowX2,NODE
allsel,all

Nssel,s,loc,y,-h/2-tf/2
nssel,r,loc, x, 0
CM,NL,NODE
allsel,all

nssel,s,loc,x,2*a+a0
nssel,r,loc,y,-h/2-tf/2
CM,NR,NODE
allsel,all

nssel,s,loc,y,h/2+tf/2
nssel,r,loc,x,a
CM,MS1,NODE
allsel,all

nssel,s,loc,y,h/2+tf/2
nssel,r,loc,x,2*a
CM,MS2,NODE
allsel,all

nssel,s,loc,x,0
nssel,r,loc,y,h/2+tf/2
nssel,r,loc, z, 0
CM,LS1,NODE
allsel,all

nssel,s,loc,x,2*a+a0
nssel,r,loc,y,h/2+tf/2
nssel,r,loc, z, 0
CM,LS2,NODE
allsel,all

Nssel,s,loc,y,-h/2-tf/2
nssel,r,loc, x, 0
nssel,r,loc, z, 0
CM,LS3,NODE
allsel,all

nssel,s,loc,x,2*a+a0
nssel,r,loc,y,-h/2-tf/2
CM,LS4,NODE
D,all, , , , ,UX,,UZ,

allsel,all

!--------------------------------------------------------------------------Solution, Eigenvalue Analysis
/solu
antype,static ! Static analysis
nlgeom,off !Geometric non-linearity off
pstress, on !Pre-stress effects
solve
finish
/solu
antype,buckle !Buckling analysis
nlgeom,off
bucopt,lanb,1 !Extract modal shape 1
mxpand,1
solve
finish
save
/post1
SET, LIST, 999
SET,. . . ,1
/EFACET,1
PLNSOL, U,Z, 0,1.0
/prep7
UPGEOM,e,. . . ,sindentation75,rst !Update modal shape to model
allsel,all
eplot
/prep7
lsclear,all !Clear boundary condition
! ---------------------------------------------------------------------------Define Material Non-linearity
/prep7 !Re-enter pre-processor.
!E    = 206000 !De-comment for Bilinear material.
!nu   = 0.3 !De-comment for Bilinear material.
!sy   = 235 !De-comment for Bilinear material.
!rho  = 0.0078 !De-comment for Bilinear material.
!mp,dens,1,rho !De-comment for Bilinear material.
!tb,BISO,1,1,2 !De-comment for Bilinear material.
!tbdat,1,sy,E/100 !De-comment for Bilinear material.
!tbpl,BISO,1 !De-comment for Bilinear material.
MPTEMP,1,0,20 !Define temperature-dependent EX.
MPDATA,EX,1,.,210000
MPDATA,PRXY,1,.,0.3
TB,MISO,1,2,23 !Activate multi-linear isotropic material table.
TBTEMP,20 ! Temperature = 20.
TBPT, DEFI, 0.001227, 257.7155
TBPT, DEFI, 0.0040, 286
TBPT, DEFI, 0.0198, 290.5174
TBPT, DEFI, 0.1823, 503.6690
TBPT, DEFI, 0.4, 504
TBPlot, MISO, 1

! Plot strain-stress curve.

! ------------------------------------------------------------------------------------- Create Indenter
alsel, all
CYL4,(2*a+a0)/2,h/2+tf/2+10+Ra,Ra,-180, , ,250
VGEN, ,P51X, ,0,0,-125, ,1
alsel, all
lsel,s,loc,y, h/2+tf/2+3,1000
FLST,3,1,6,ORDE,1
FITEM,3,1
VGEN, ,P51X, ,0,0,-125, ,1
alsel, all
aplot

! ------------------------------------------------------------------------------------- Create Ground
k,101,-200,-h/2-tf/2-2,250
k,102,-200,-h/2-tf/2-2,-250
k,103,2*a+a0+200,-h/2-tf/2-2,250
k,104,2*a+a0+200,-h/2-tf/2-2,-250
a,101,103,104,102
lsel,s,loc,y,-h/2-tf/2-2
alsel, all
aplot

! ------------------------------------------------------------------------------------- Contact Analysis Indentation Process
CM, _TARGET, AREA
/COM, CONTACT PAIR CREATION - START
CM, _NODECM, NODE
CM, _ELEMCM, ELEM
CM, _KPCM, KP
CM, _LINECM, LINE
CM, _AREACM, AREA
CM, _VOLUCM, VOLU
/GSAV,cwz,gsav,,temp
MP, MU, 1, fc
MAT, 1
R, 6
REAL, 6
ET, 2, 170
ET, 3, 174
KEYOPT, 3, 9, 1
KEYOPT, 3, 10, 2
R, 6,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
ASEL,S,,,38
ASEL,A,,,39
ASEL,A,,,40
ASEL,A,,,41
ASEL,A,,,42
CM,_TARGET,AREA
AATT,-1,6,2,-1
TYPE,2
AMESH,ALL
! Create a pilot node
N,5719, 750,500,0
TSHAP,PILO
E,5719
NSEL,S,,,5719
CM,PIDT,NODE
CMSEL,S,_NODECM
! Generate the contact surface
ASEL,S,,,1
ASEL,A,,,2
CM,_CONTACT,AREA
TYPE,3
NSLA,S,1
ESLN,S,0
NSLE,A,CT2 ! CZMESH patch (fsk qt-40109 8/2008)
ESURF
*SET,_REALID,6
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,2
ESEL,A,TYPE,,3
ESEL,R,REAL,,6
ASEL,S,REAL,,6
/PSYMB,ESYS,1
/PNUM,TYPE,1
/NUM,1
EPLOT
! Reverse contact normals
ESEL,NONE
ESEL,A,TYPE,,3
ESEL,R,REAL,,6
ESURF,,REVERSE
ESEL,ALL
ESEL,S,TYPE,,2
ESEL,A,TYPe,,3
ESEL,R,REAL,,6
ASEL,S,REAL,,6
/PSYMB,ESYS,1
/PNUM,TYPE,1
/NUM,1
EPLOT

ESEL,ALL
ESEL,S,TYPE,,2
ESEL,A,TYPE,,3
ESEL,R,REAL,,6
ASEL,S,REAL,,6
CMSEL,A,_NODECM
CMSEL,A,_ELEMCM
CMSEL,A,_KPCM
CMSEL,A,_LINECM
CMSEL,A,_AREACM
CMSEL,A,_VOLUCM
CMDEL,_ELEMCM
CMDEL,_KPCM
CMDEL,_LINECM
CMDEL,_AREACM
CMDEL,_VOLUCM
/GRES,cwz,gsav
CMDEL,_TARGET
CMDEL,_CONTACT
/COM, CONTACT PAIR CREATION - END
/MREP,EPLOT
alls
/DIST,1,1.08222638492,1
/REP,FAST
*CREATE,cwzplot,mac
/COM,
/COM,PLOT CONTACT PAIR(S)
~eui,`:apdl::noprIn 1'
~eui,`:apdl::nooutput 1'
!*
CM,_CWZ_EL,ELEM
CM,_CWZ_ND,NODE
CM,_CWZ_KP,KP
CM,_CWZ_LN,LINE
CM,_CWZ_AR,AREA
CM_,CWZ_VL,VOLU
ESEL,NONE
ESEL,A,REAL,,6
ESEL,R,ENAME,,169,177
NSLE
KSLN,S
LSLK,S,1
ASLL,S,1
/PNUM,TYPE,1
/NUM,1
/PSYMB,ESYS,0
EPLOT
CMSEL,S,CWZ_EL
CMDEL,CWZ_EL
CMSEL,S,CWZ_ND
CMDEL,CWZ_ND
CMSEL,S,CWZ_KP
CMDEL,CWZ_KP
CMSEL,S,CWZ_LN
CMDEL,CWZ_LN
CMSEL,S,CWZ_AR
CMDEL,CWZ_AR
CMSEL,S,CWZ_VL
CMDEL,CWZ_VL
!* /PSYMB,ESYS,0
/PNUM,0
/PNUM,TYPE,0
/PNUM,REAL,0
/mrep,cwzplot
~eui,'::apdl::nooutput 0'
~eui,'::apdl::noprint 0'
*END
cwzplot
!* /FOC,1,,-0.3,1
/REP,FAST
/FOC,1,,-0.3,1
/REP,FAST
!* /COM, CONTACT PAIR CREATION - START
CM_,NODECM,NODE
CM_,ELEMCM,ELEM
CM_,KPCM,KP
/GSAV,cwz,gsav,,temp
MP,MU,1,fc
MAT,1
R,7
REAL,7
ET,4,170
ET,5,174
KEYOPT,5,9,1
KEYOPT,5,10,2
R,7,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
ASEL,S,,,43
CM,_TARGET,AREA
AATT,-1,7,4,-1
TYPE,4
AMESH,ALL
! Generate the contact surface
ASEL,S,,,3
ASEL,A,,,4
ASEL.A,,,10
ASEL,A,,,11
ASEL,A,,,12
ASEL,A,,,13
ASEL,A,,,30
ASEL,A,,,31
ASEL.A,,,34
ASEL,A,,,35
CM,_CONTACT,AREA
TYPE,5
NSLA,S,1
ESLN,S,0
NSLE,A,CT2 ! CZMESH patch (fsk qt-40109 8/2008)
ESURF
*SET,_REALID,7
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,4
ESEL,A,TYPE,,5
ESEL,R,REAL,,7
ASEL,S,REAL,,7
/PSYMB,ESYS,1
/PNUM,TYPE,1
/NUM,1
E P L O T
ESEL,ALL
ESEL,S,TYPE,,4
ESEL,A,TYPE,,5
ESEL,R,REAL,,7
ASEL,S,REAL,,7
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
/GRES,cwz,gsav
CMDEL,_TARGET
CMDEL,_CONTACT
/COM,CONTACT PAIR CREATION - END
!!!-------------------------------------------------------------------------------Grouping Indenter Element Component
allsel,all
esel,s,cent,y,-h/2-tf/2-2
CM,grd,ELEM
allsel,all
esel,s,cent,y,h/2+tf/2+3,1000
CM,idt,ELEM
allsel,all
!-------------------------------------------------------------------------------Create Loading Headers
/prep7
CYL4,a,h/2+tf/2+1+Ra,Ra,-180, , ,250
CYL4,a+a0,h/2+tf/2+1+Ra,Ra,-180, , ,250
aplot
FLST,3,2,6,ORDE,2
FITEM,3,2
FITEM,3,-3
VGEN, ,P51X, , , ,125, , ,1
Create End Supports

/prep7
CYL4,0,-h/2-tf/2-Ra-2,Ra,180, ,250
CYL4,a+a0+a,-h/2-tf/2-Ra-2,Ra,180, ,250
alls
vplot
FLST,3,2,6,ORDE,2
FITEM,3,4
FITEM,3,-5
VGEN, ,P51X, , , ,125, ,1

Contact Analysis Four Point Bending

COM, CONTACT PAIR CREATION - START
CM, _NODECM, NODE
CM, _ELEMCM, ELEM
CM, _KPCM, KP
CM, _LINECM, LINE
CM, _AREACM, AREA
CM, _VOLUCM, VOLU
/GSAV, cwz, gsav, , temp
MP, MU, 1, fc
MAT, 1
R, 8
REAL, 8
ET, 6, 170
ET, 7, 174
KEYOPT, 7, 9, 1
KEYOPT, 7, 10, 2
R, 8,
RMORE,
RMORE, 0
RMORE, 0
! Generate the target surface
ASEL, S, , 44
ASEL, A, , 45
ASEL, A, , 46
ASEL, A, , 47
ASEL, A, , 48
ASEL, A, , 49
ASEL, A, , 50
ASEL, A, , 51
ASEL, A, , 52
ASEL,A,,,53
CM_TARGET,AREA
AATT,-1,8,6,-1
TYPE,6
AMESH,ALL
! Create a pilot node
N,6412, 750,600,0
TSHAP,PILO
E,6412
NSEL,S,,,6412
CM,PHRD,NODE
CMSEL,S_NODECM
! Generate the contact surface
ASEL,S,,,1
ASEL,A,,,2
ASEL,A,,,6
ASEL,A,,,7
ASEL,A,,,8
ASEL,A,,,9
CM_CONTACT,AREA
TYPE,7
NSLA,S,1
ESLN,S,0
NSLE,A,CT2 ! CZMESH patch (fsk qt-40109 8/2008)
ESURF
*SET_REALID,8
ALLSEL
ESEL,ALL
ESEL,S_TYPE,,6
ESEL,A_TYPE,,7
ESEL,R_REAL,,8
ASEL,S_REAL,,8
/PSYMB,ESYS,1
/PNUM,TYPE,1
/NUM,1
EPLOT
! Reverse contact normals
ESEL,NONE
ESEL,A_TYPE,,7
ESEL,R_REAL,,8
ESURF,REVERSE
ESEL,ALL
ESEL,S_TYPE,,6
ESEL,A_TYPE,,7
ESEL,R,REAL,,8
ASEL,S,REAL,,8
/PSYMB,ESYS,1
/PNUM,TYPE,1
/NUM,1
EPLOT
!
ESEL,ALL
ESEL,S,TYPE,,6
ESEL,A,TYPE,,7
ESEL,R,REAL,,8
ASEL,S,REAL,,8
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
/GRES,cwz,gsav
CMDEL,_TARGET
CMDEL,_CONTACT
/COM, CONTACT PAIR CREATION - END
CM,_TARGET,AREA
/COM, CONTACT PAIR CREATION - START
CM,_NODECM,NODE
CM,_ELEMCM,ELEM
CM,_KPCM,KP
CM,_LINECM,LINE
CM,_AREACM,AREA
CM,_VOLUCM,VOLU
/GSAV,cwz,gsav,,temp
MP,MU,1,fc
MAT,1
R,9
REAL,9
ET,8,170
ET,9,174
KEYOPT,9,9,1
KEYOPT,9,10,2
R,9,
RMORE,
RMORE,,0
RMORE,0
! Generate the target surface
ASEL,S,,,54
ASEL,A,,,55
ASEL,A,,,56
ASEL.A,,,57
ASEL.A,,,58
ASEL.A,,,59
ASEL,A,,,60
ASEL.A,,,61
ASEL.A,,,62
ASEL.A,,,63
CM_TARGET,AREA
AATT,-1,9,8,-1
TYPE,8
AMESH,ALL
! Generate the contact surface
ASEL,S,,,10
ASEL.A,,,11
ASEL.A,,,12
ASEL.A,,,13
ASEL.A,,,30
ASEL.A,,,31
ASEL.A,,,34
ASEL.A,,,35
CM_CONTACT,AREA
TYPE,9
NSLA,S,1
ESLN,S,0
NSLE,A,CT2! CZMESH patch (fsk qt-40109 8/2008)
ESURF
*SET_REALID,9
ALLSEL
ESEL,ALL
ESEL,S,TYPE,,8
ESEL,A,TYPE,,9
ESEL,R,REAL,,9
ASEL,S,REAL,,9
/PSYMB,ESYS,1
/PNUM,TYPE,1
/NUM,1
EPLOT
ESEL,ALL
ESEL,S,TYPE,,8
ESEL,A,TYPE,,9
ESEL,R,REAL,,9
ASEL,S,REAL,,9
CMSEL,A,_NODECM
CMDEL,_NODECM
CMSEL,A,_ELEMCM
CMDEL,_ELEMCM
CMSEL,S,_KPCM
CMDEL,_KPCM
CMSEL,S,_LINECM
CMDEL,_LINECM
CMSEL,S,_AREACM
CMDEL,_AREACM
CMSEL,S,_VOLUCM
CMDEL,_VOLUCM
/GRES,cwz,gsav
CMDEL,_TARGET
CMDEL,_CONTACT
/COM, CONTACT PAIR CREATION - END
!---------------------------------------------------------Create Loading Headers Element Component
alls
esel,s,cent,y,h/2+tf/2+1,1000
esel,u,cent,x,600,900
CM,ELoadingHeader,element
alls
!---------------------------------------------------------------Create End Support Element Component
alls
esel,s,cent,y,-h/2-tf/2-2,-1000
esel,u,ename,,grd
nsel,s,,
CM,EEndSupport,element
alls
!----------------------------------------------------------------------Enter Solution Indentation Process
/solu
PSTRES,on       ! Pre-stress effect
rescont,,all,all ! Save all substeps for possible restart
antype,static   ! Analysis type
nggeom,on        ! Non-linear geometry effects included
time,1           ! Loading step 1, to time step 1
allsel,all
!-----------------------------------------------------------------Load Application, Indentation Process
allsel, all
nsel, s, node, PIDT
D, all, UY, -dis1
D, all, UX, 0
D, all, UZ, 0
D, all, ROTX, 0
D, all, ROTY, 0
D, all, ROTZ, 0
! -----------------------------------------------------------------Solver Configuration

deltim, 0.025, 0.025    ! integration time step.
neqit, 100             ! maximum number of iterations in each load step.
outrres, all, all,     ! write data of every substep to file.
arclen, off, 1.0      ! Do not use the arc-length method.
ncnv, 1, 0, 0, 1000000, 0.0, 0.0 ! limit the total number of iterations.
cnvtof, f, 0.005, 2.1, 0 ! set convergence values.
nsel, all
outres, all, all
solve
save
!--------------------------------------------------------------Element Death, Kill Indenter and Ground
allsel, all
esel, s, ename, grd
esel, a, ename, idt
ekill, all
ESEL, S, LIVE
/out
!--------------------------------------------------------------Enter Four Point Bending Simulation
time, 6
!!!!---------------------------------------------------------------------Delete BC of Indentation Process
allsel, all
nsel, loc, y, -h/2-tf/2-2
DDELETE, all, ALL
ESEL, S, LIVE
!--------------------------------------------------------------------BC and Load Application, Four Point Bending
alls
Nsel, s, loc, y, -h/2-tf/2-2-Ra
nplot
D, all, , , , ,,UX, UY, UZ, ROTX, ROTY, ROTZ
alls
Nsel, s, ,,LowX
D, all, , , , ,,UX
alls
Nsel, s, ,,UpperX
D, all, , , , ,,UX
alls
nsel, s, node, PHRD
D, all, UY, -dis2
D, all, UX, 0
D, all, UZ, 0
!---------------------------------------------------------------------Solve Four Point Bending
allsel, all
solve
save
!--------------------------------------------------------------Load-displacement Curve, Indentation Process
/post26                          ! Enter time historic post processor.
nsel, all
nsel, s, ,,PIDT
*get, nn, node, , count      ! Get total number of nodes and save in variable 'nn'.
*get, mn, node, , num, min   ! Get minimum node number and save in variable 'mn'.
rforce, 2, mn, f, y, fy      ! Get reaction force in y-direction of node mn and save in
variable

*do,i,1,nn-1,1
  *get,j,node,mn,nxth
  rforce,3,j,f,y,fy
  !Get next node number and save in variable 'j'.
  rforce,3,j,f,y,fy
  !Get reaction force in y-direction of node j - save in variable 3.
  add,2,2,3,,fy
  mn=j
*enddo
abs,2,2,,
vardel,3
!Add variable 2 and 3 and save in 2 (overwrite).

mn=j
*enddo
abs,2,2,,
vardel,3
!Delete aux variable 3.

allsel,all
nsel,s,,,PIDT
*get,mn,node,,num,min
!get minimum node number and save in variable 'mn'
nsol,3,mn,u,y,uy
!read z-deflection of node mn and save into var 3
abs,3,3,,uy
!take the absolute value of the x-deflection, name: "Uz"

nsel,all
xvar,3
!make variable 3 the x-axis
plvar,2
!plot variable 2 on y-axis
/axlab,y,Load(N)
!change y label
/axlab,x,Machine Disp. (mm)
!change X label
lines,5000
!5000 lines on each page
/replot

!------------------------------------------------------Load-displacement Curve, Four Point Bending
nsel,all
nsel,s,,,PHRD
*get,nn,node,,count
!get total number of nodes and save in variable 'mn'
*get,mn,node,,num,min
!read z-deflection of node mn and save into var 3
abs,3,3,,uy
!take the absolute value of the x-deflection, name: "Uz"

nsel,all
*get,mn,node,,num,min
!get minimum node number and save in variable 'mn'
nforce,2,mn,f,y,fy
!get reaction force in y-direction of node mn and save in variable 2
*do,i,1,nn-1,1
  *get,j,node,mn,nxth
  rforce,3,j,f,y,fy
  !get reaction force in y-direction of node j - save in variable 3
  add,2,2,3,,fy
  mn=j
*enddo
abs,2,2,,
vardel,3
!delete aux variable 3
allsel,all
nsel,s,,,PHRD
*get,mn,node,,num,min
!get minimum node number and save in variable 'mn'
nsol,3,mn,u,y,uy
!read z-deflection of node mn and save into var 3
abs,3,3,,uy
!take the absolute value of the x-deflection, name: "Uz"
nsel,all
xvar,3        ! make variable 3 the x-axis
plvar,2       ! plot variable 2 on y-axis
/axlab,y,Load(N)     ! change y label
/axlab,x,Machine Disp.(mm) ! change X label
lines,5000     ! 5000 lines on each page
/replot
save
finish

!---------------------------------------------------------------END of Analysis