NTIS #PB2019-######

SSC-475

GUIDELINES FOR EVALUATION OF MARINE FINITE ELEMENT ANALYSES

Editors: Dr. E.D. Wang, Mr. J.S. Bone, Dr. M. Ma, and Mr. A. Dinovitzer

SHIP STRUCTURE COMMITTEE

2019

Member Agencies:

American Bureau of Shipping
Defence Research & Development Canada
Maritime Administration
Military Sealift Command
Naval Sea Systems Command
Office of Naval Research
Society of Naval Architects & Marine Engineers
Transport Canada
United States Coast Guard

Ship
Structure
Committee

Address Correspondence to:

Commandant (CG-ENG-2/SSC)
ATTN: Executive Director, SSC
US Coast Guard
2703 Martin Luther King Jr. Ave SE
Washington, DC 20593-7509
Website: http://www.shipstructure.org

SSC – 475
SR – 1481

17 December 2019

GUIDELINES FOR EVALUATION OF MARINE FINITE ELEMENT ANALYSES

Finite element analysis (FEA) is one of the most common structural analysis tools in use today. In marine industries, the use of this technique is widespread in the design, reliability analysis and performance evaluation of ship structures. Users of FEA have considerable freedom in designing the finite element model, exercising it, and interpreting the results. A consequence of this freedom is that significant variability in FEA results can be obtained depending on the assumptions and modelling practices adopted by the analyst. Unsatisfactory analysis is not always obvious and the consequences usually will not manifest themselves until the vessel is in service. In response to this, a systematic and practical methodology is necessary to assess the validity of FEA results based on the choice of analysis procedure, type of element/s, model size, boundary conditions, load application, etc.

The use of finite element analysis (FEA) techniques in ship design and analysis has grown since the original SSC-387 Guideline for Evaluation of Finite Elements and Results was published in 1996. This guide is an update to SSC-387 and includes current best practices for FEA application to ship structures and advanced analysis topic discussion and sample applications for the following: Impact and Plasticity, Fracture and Fatigue, Whole Ship Analysis, and Frequency Response Vibration Analysis.

We thank the authors and Project Technical Committee for their dedication and research toward completing the objectives and tasks detailed throughout this paper and continuing the Ship Structure Committee's mission to enhance the safety of life at sea.

R. V. TIMME
Rear Admiral, U.S. Coast Guard
Co-Chairman, Ship Structure Committee

L. C. SELBY
Rear Admiral, U.S. Navy
Co-Chairman, Ship Structure Committee

SHIP STRUCTURE COMMITTEE

RDML Richard Timme
U. S. Coast Guard Assistant Commandant
for Prevention Policy (CG-5P)
Co-Chair, Ship Structure Committee

RADM Lorin Selby
Chief Engineer and Deputy Commander
For Naval Systems Engineering (SEA05)
Co-Chair, Ship Structure Committee

Mr. Jeffrey Lantz
Director, Commercial Regulations and Standards (CG-5PS)
U.S. Coast Guard

Mr. Derek Novak
Senior Vice President, Engineering & Technology
American Bureau of Shipping

Mr. H. Paul Cojeen
Society of Naval Architects and Marine Engineers

Dr. John MacKay
Head, Warship Performance, DGSTCO
Defence Research & Development Canada - Atlantic

Mr. Kevin Kohlmann
Director, Office of Safety
Maritime Administration

Mr. Luc Tremblay
Executive Director, Domestic Vessel Regulatory Oversight
and Boating Safety, Transport Canada

Mr. Albert Curry
Deputy Assistant Commandant for Engineering and
Logistics (CG-4D)
U.S. Coast Guard

Mr. Eric Duncan
Group Director, Ship Integrity and Performance Engineering
(SEA 05P)
Naval Sea Systems Command

Mr. Neil Lichtenstein
Deputy Director N7x, Engineering Directorate
Military Sealift Command

Dr. Thomas Fu
Director, Ship Systems and Engineering Research Division
Office of Naval Research

SHIP STRUCTURE SUB-COMMITTEE

UNITED STATES COAST GUARD (CVE)
CAPT Robert Compher
Mr. Jaideep Sirkar
Mr. Charles Rawson

AMERICAN BUREAU OF SHIPPING
Mr. Daniel LaMere
Ms. Christina Wang

SOCIETY OF NAVAL ARCHITECTS AND MARINE ENGINEERS
Mr. Frederick Ashcroft
Dr. Roger Basu
Dr. Robert Sielski
Dr. Paul Miller

DEFENCE RESEARCH & DEVELOPMENT CANADA ATLANTIC
Dr. Malcolm Smith
Mr. Cameron Munro

MARITIME ADMINISTRATION
Mr. Todd Hiller
Mr. Todd Ripley

TRANSPORT CANADA
Ms. Veronique Bérubé
Mr. Bashir Ahmed Golam
Ms. Tayyebe Seif

UNITED STATES COAST GUARD (FLEET)
CAPT George Lesher
Mr. Martin Hecker
Mr. Timothy McAllister
Mr. Debu Ghosh

NAVSEA/ NSWCCD
Mr. David Qualley
Mr. Dean Schleicher
Dr. Pradeep Sensharma
Mr. Daniel Bruchman

MILITARY SEALIFT COMMAND
Ms. Jeannette Viernes

OFFICE OF NAVAL RESEARCH
Dr. Paul Hess

PROJECT TECHNICAL COMMITTEE

The Ship Structure Committee greatly appreciates the contributions of the individuals that volunteered their time to participate on the Project Technical Committee, listed below, and thanks them heartily. They were the subject matter expert representatives of the Ship Structure Committee to the contractor, performing technical oversight during contracting, advising the contractor in cognizant matters pertaining to the contract of which the agencies were aware, and performing technical peer review of the work in progress and upon completion.

Chair:

Dr. Paul Miller, US Coast Guard Academy

Members:

Dr. Roger Basu, Roger Basu & Associates Inc.
Mr. Jason Cordell, US Coast Guard
Mr. Paul Lara, US Navy
Dr. Miguel Núñez-Sánchez, Universidad Politécnica de Madrid, Spain
Mr. Earl Powers, US Coast Guard
Mr. Charles Rawson, US Coast Guard
Dr. Mahmud Sazidy, Defence Research and Development Canada
Dr. Pradeep Sensharma, US Naval Sea Systems Command
Dr. Robert Sielski
LT Braden Rostad, US Coast Guard

Ship Structure Committee Executive Director:

LT Braden Rostad, US Coast Guard

1. Report No. SSC-475	2. Government Accession No. NTIS #PB2019-######	3. Recipient's Catalog No.
4. Title and Subtitle GUIDELINES FOR EVALUATION OF MARINE FINITE ELEMENT ANALYSES		5. Report Date December 2019 6. Performing Organization Code
7. Author(s) Dr. E.D. Wang, Mr. J.S. Bone, .Dr. M. Ma, and Mr. A. Dinovitzer		8. Performing Organization Report No. SR-1481
9. Performing Organization Name and Address BMT Designes and Planners Inc.MIL Systems 2900 South Quincy Street, Suite 210 Arlington, VA, 22206, United States		10. Work Unit No. (TRAIS) 11. Contract or Grant No.
12. Sponsoring Agency Name and Address Ship Structure Committee COMMANDANT (CG-ENG-2/SSC) ATTN (ADMIN ASST/SHIP STRUCTURE COMMITTEE) US COAST GUARD 2703 MARTIN LUTHER KING JR. AVE SE MAILSTOP 7509 WASHINGTON DC 20593-7509		13. Type of Report and Period Covered Final 14. Sponsoring Agency Code CG-5P

15. Supplementary Notes

Sponsored by the Ship Structure Committee. The research completed by the above author for the Ship Structure Committee was reviewed by the Project Technical Committee for satisfactory completion of the objectives outlined in the Statement of Work developed and approved for funding by the Principal Members of the Ship Structure Committee.

16. Abstract

Commercial and open source finite element analysis (FEA) programs can easily be used to model structures and generate impressive looking results even when fundamental mistakes are introduced by engineers with little previous design experience or with improper modeling techniques. This can result in inadequate structures from the point of view of strength, fatigue, vibration, and other design or analysis criteria. Some structural failures have demonstrated that, if not appropriately used, FEA may mislead the designer with erroneous results. The original SSC-387 Guideline for Evaluation of Finite Elements and Results published in 1996 addressed this concern. The use of finite element analysis (FEA) techniques in ship design and analysis has grown since the original SSC-387 Guideline for Evaluation of Finite Elements and Results was published in 1996. This guide is an update to SSC-387 and includes current best practices for FEA application to ship structures and advanced analysis topic discussion and sample applications for the following: Impact and Plasticity, Fracture and Fatigue, Whole Ship Analysis, and Frequency Response Vibration Analysis. This document structure follows the original document structure. This document provides, in checklists and discussions, support for the review of FEA models and output to ensure that the analysis is prepared appropriately for the intended situation. The document is no substitute for a solid education, enhanced by the experience of the impact of modeling choices on results. The document is to be construed as a guideline to assist the analyst and reviewer in determining deficiencies or identifying good practice in an FEA; it is not a substitute for technical qualifications.

17. Key Words Finite Element Method, Finite Element Analysis, FEA, Ship Structure, Structural Analysis, Quality Assessment, Impact Analysis, Collision Analysis, Fracture Analysis, Damage Analysis, Fatigue Analysis, Whole Ship Analysis, Vibration Analysis	18. Distribution Statement Distribution Unlimited Available from: National Technical Information Service Springfield, VA 22161		
19. Security Classlf. (of this report) Unclassified	20. Security Classification (of this page) Unclassified	21. No. of Pages 404	22. Price

CONVERSION FACTORS
(Approximate conversions to metric measures)

To convert from	to	Function	Value
LENGTH			
inches	meters	divide	39.3701
inches	millimeters	multiply by	25.4000
feet	meters	divide by	3.2808
VOLUME			
cubic feet	cubic meters	divide by	35.3149
cubic inches	cubic meters	divide by	61,024
SECTION MODULUS			
inches^2 feet	centimeters^2 meters	multiply by	1.9665
inches^2 feet	centimeters^3	multiply by	196.6448
inches^3	centimeters^3	multiply by	16.3871
MOMENT OF INERTIA			
inches^2 feet^2	centimeters^2 meters^2	divide by	1.6684
inches^2 feet^2	centimeters^4	multiply by	5993.73
inches^4	centimeters^4	multiply by	41.623
FORCE OR MASS			
long tons	tonne	multiply by	1.0160
long tons	kilograms	multiply by	1016.047
pounds	tonnes	divide by	2204.62
pounds	kilograms	divide by	2.2046
pounds	Newtons	multiply by	4.4482
PRESSURE OR STRESS			
pounds/inch^2	Newtons/meter^2 (Pascals)	multiply by	6894.757
kilo pounds/inch^2	mega Newtons/meter^2 (mega Pascals)	multiply by	6.8947
BENDING OR TORQUE			
foot tons	meter tons	divide by	3.2291
foot pounds	kilogram meters	divide by	7.23285
foot pounds	Newton meters	multiply by	1.35582
ENERGY			
foot pounds	Joules	multiply by	1.355826
STRESS INTENSITY			
kilo pound/inch^2 inch^{½}(ksi$\sqrt{\text{in}}$)	mega Newton MNm$^{3/2}$	multiply by	1.0998
J-INTEGRAL			
kilo pound/inch	Joules/mm^2	multiply by	0.1753
kilo pound/inch	kilo Joules/m^2	multiply by	175.3

Contents

LIST OF ILLUSTRATIONS

LIST OF TABLES

ACKNOWLEDGMENTS

The authors gratefully acknowledge the discussions with Dr. Martin Storheim at Entail AS and his help in providing FE models for the impact analysis. The authors also wish to thank Canarctic Shipping Limited, and in particular, Mr. John McCallum, for permission to use the Arctic tanker example presented in Appendix B.

The authors also wish to thank Altair Software for providing free trials to aid in the preparation of this updated guide.

LIMITATIONS

The following document is intended to support the development and review of finite element modeling through a suggested review methodology. The methodology presented has been developed based on the experience and expertise of the authors. The use of this document does not replace education or experience and should be treated as good practice guidance but not a guarantee of acceptable performance for all design or analysis scenarios.

PART 1
PROJECT OVERVIEW

1/1 EXECUTIVE SUMMARY

Commercial and open source finite element analysis (FEA) programs can easily be used to model structures and generate impressive looking results even when fundamental mistakes are introduced by engineers with little previous design experience or with improper modeling techniques. This can result in inadequate structures from the point of view of strength, fatigue, vibration, and other design or analysis criteria. Some structural failures have demonstrated that, if not used properly, FEA may mislead the designer with erroneous results. The original SSC-387 Guideline for Evaluation of Finite Elements and Results published in 1996 addressed this concern. The use of finite element analysis (FEA) techniques in ship design and analysis has grown since the original SSC-387 Guideline for Evaluation of Finite Elements and Results was published in 1996. This guide is an update to SSC-387 and includes current best practices for FEA application to ship structures and advanced analysis topic discussion and sample applications for the following:

- Impact and Plasticity
- Fracture and Fatigue
- Whole Ship Analysis
- Frequency Response Vibration Analysis

This document structure follows the original document structure. This document provides, in checklists and discussions, support for the review of FEA models and output to ensure that the analysis is prepared appropriately for the intended situation. The document is no substitute for a solid education, enhanced by the experience of the impact of modeling choices on results. The document is to be construed as a guideline to assist the analyst and reviewer in determining deficiencies or identifying good practice in an FEA; it is not a substitute for technical qualifications.

1/2 INTRODUCTION

1/2.1 Background

Finite element analysis (FEA) is a common structural analysis method for advanced problems. Great strides have been made in theoretical and computational aspects of FEA. This has been accompanied by phenomenal advances in computer technology, both in hardware and software, together with a rapid reduction in the cost of this technology. A consequence of this is a dramatic increase in the affordability of, and accessibility to, finite element technology. In marine industries, the use of this technique is widespread in ship structure design, reliability analysis, and performance evaluation.

Finite element analysis is a powerful and flexible engineering analysis tool that allows the analyst considerable freedom in designing the finite element model, exercising it, and interpreting the results. Key components of this process include the selection of the computer program (e.g., OS – operating system and FEA software package), the determination of the loads and boundary conditions, development of the mathematical model, choice of elements, and the design of the mesh. The analyst needs to make numerous experience-, scenario- and modeling tool-based decisions during this process. Results from FEAs for the same structure performed by different individuals or organizations may differ significantly as a result of differences in the assumptions and modeling procedures employed.

An unsatisfactory analysis is not always obvious, and the consequences may not manifest themselves until the vessel is in service. Design changes and any structural modifications required at this stage are generally much more expensive to implement than would be the case if the deficiency was discovered earlier.

A particular difficulty is faced by those who have the responsibility for assessing and approving FEAs. The individual concerned may not be an expert in FEA, or familiar with the software package used, and would face a dilemma when coming to judge the acceptability, or otherwise, of the results of the FEA. This may require the evaluator to incur further cost and time in the attempt to ensure satisfactory FEA results.

In response to the difficulty faced by those who evaluate FEAs, a systematic and practical methodology was required to support the assessment of the validity of the FEA results efficiently. That need resulted in the original SSC-387 Guideline for Evaluation of Finite Elements and Results. SSC-387 included in support of this methodology a selection of finite element models that illustrated good modeling practice. Besides, SSC-387 included benchmark tests to allow the validation of new FEA software packages or packages that have undergone significant modification.

Since SSC-387 was issued in 1996, the use of finite element analysis has advanced in terms of:

- Tools available (e.g., automated meshing, interaction with drafting/solid modeling tools)
- Materials considered (e.g., steel, aluminum, plastic, composites, non-linear (post-yielding) behavior)
- Load conditions (e.g., fluid-structure interaction, collision, blast simulation)
- Analysis types (e.g., implicit versus explicit (time-domain) modeling)
- Element formulation (non-linear, hybrid, and contact elements)
- Structural geometries (crack-tip elements, connection, and weldments; contact/sliding component fit-up)
- Larger models being able to be analyzed more rapidly than in the past.

1/2.2 Scope

This guideline provides a systematic and practical methodology to support the rapid assessment of the validity of FEA results for ship structures. The methodology is an updated version of SSC-387, which includes current FEA best practice, and can be used for the following types of ship structure analysis problems:

- Linear static
- Non-linear (plastic) static
- Impact/Collision
- Fracture
- Fatigue
- Whole ship
- Natural frequency
- Forced vibration

The emphasis is on the structural assembly level, rather than on local details, such as weldments. Only FEA of structures composed of isotropic materials are addressed, thus excluding fiber reinforced plastics and wood. Despite these limitations, the guidelines apply to most ship structure FEAs.

1/2.3 Overview of Report

The report is structured in six parts, four appendices, and a reference section as follows:

Part 1: Project Overview

This part introduces the document and provides the background for the methodologies developed for assessing FEAs and FEA software, which are described in subsequent Parts.

Part 2: Assessment Methodology for Finite Element Analysis

This part presents a systematic methodology for assessing FEAs. Appendix A contains forms that can be used for the evaluation process. Appendix B presents an example of an FEA and its evaluation.

Part 3: Guidelines for Assessing Finite Element Models and Results

This part provides guidance in support of the methodology presented in PART 2. It is a comprehensive description of good FEA practices. As an aid to the assessment of FEA models and results, some FEAs, typical of ship structures, are presented in Appendix C. These examples are designed to illustrate the influence on the results of varying specific model parameters.

Part 4: Benchmark Problems for Assessing FEA Software

The assessment methodology described in PART 2 includes a requirement that suitable FEA software is used. In support of the assessment, new or significantly-modified FEA should be evaluated regarding its suitability for ship structure FEA. The benchmark problems and results presented in PART 4 are for this purpose. The benchmark problems are presented in Appendix D.

Part 5: Advanced Analysis Sample Applications

This part presents sample applications for the following analysis types:

- Impact and Plasticity
- Fracture and Fatigue
- Whole Ship
- Frequency Response Vibration

Part 6: Conclusions and Recommendations

This part summarizes observations and insights gained in the course of this project regarding the process of evaluating finite element models and results and FEA software. Also presented is a summary of where effort should be directed to further improve the methodologies in response to likely future trends in finite element technology.

Appendices

The following appendices are included:

- Appendix A Evaluation Forms for Assessment of Finite Element Models and Results
- Appendix B Example Application of Assessment Methodology
- Appendix C Examples of Variations in FEA Modeling Practices and Results
- Appendix D Ship Structure Benchmarks for Assessing FEA Software

References

The literature considered in preparing this guide, and that may be of interest to readers, is summarized in this section. The literature list includes books, reports, scientific papers, tutorials, and help documents used in this best practice. The list of references is also an excellent resource to help readers find more technical descriptions and solutions in the use of advanced FEA.

1/2.4 About the Guidelines

The purpose of the guidelines presented in this document is to provide support for evaluating finite element models and results, and also FEA software.

There are many attributes to any FEA, and it is difficult to assess quality unless the FEA has been comprehensively documented, and a systematic assessment methodology is applied. This volume presents such a methodology.

The methodology is presented in three levels:

1. Level 1 comprises a checklist of attributes of the FEA that need to be evaluated as part of the assessment process.
2. Level 2 comprises a more detailed breakdown of the checklist provided under Level 1. Level 1 can be regarded as a summary of the Level 2 assessment.
3. Level 3 contains guidelines on acceptable finite element modeling practices. The guidelines are cross-referenced with the Level 2 checklists. During the assessment process, the evaluator may, if required, refer to Level 3 guidelines for advice.

For simple FEAs, an experienced evaluator can probably perform the assessment without referring to Level 2 checklists. The methodology is structured to allow the evaluator to apply the methodology at the appropriate level of detail. The reader is referred to Figure A/0-1 in Appendix A for summary of the methodology.

In addition to presenting an assessment methodology and supporting material, this report presents benchmark problems for assessing the quality of the FEA software and its suitability for ship structural analysis.

1/2.5 Using the Guidelines

The primary audience for these guidelines is the evaluators of FEAs. The guidelines assume that the evaluator is trained in ship structural analysis and design, but is not necessarily expert in FEA.

Ideally, the guidelines would be provided to the analysts as part of the job specifications (such as a statement of work or statement of requirements). Level 1 and 2 guidelines could then be viewed as acceptance criteria for the work. The documentation requirements listed in the guidelines could then be used to stipulate the documentation required.

The methodology can be used for conducting reviews that could then be used to provide intermediate and final approvals. For this purpose, each of the five areas of an FEA shown in Figure A/0-1 would be treated as a phase in the project. Reviews could be held at the end of each phase, or less frequently for smaller projects. Depending on the outcome of the review, approval to proceed to the next stage could be given, or, in the case of severe deficiencies, rework would be required.

Most FEAs are iterative in character. This applies particularly to analyses performed in support of design tasks. The iterative nature also applies to certain aspects of the analysis itself. Some modeling decisions can only be validated during the evaluation of the results. To facilitate this, this document is presented as a step-by-step QA process, and therefore, can accommodate iterations where necessary.

The document may be considered as guidance and a memory aid supporting the review process. With this understanding, the analyst may use the guide to self-check or plan their work, thus developing or documenting their design or analysis process.

1/2.6 The Guidelines as Quality Procedures

The guidelines presented in this document incorporate several elements of a quality system as it pertains to FEA and, as such, could be incorporated in an organization's quality system for FEA.

The requirements for such a system have been developed under the direction of the National Agency for Finite Element Methods and Standards (NAFEMS) Quality Assurance Working Group. These requirements in [1] are intended as a supplement to ISO (International Organization for Standardization) 9001.

1/2.7 Where to Get Further Information

While the information provided in the guidelines is self-contained, there may be circumstances when more detailed information is required.

There are many texts that describe FEA and theory. The reader is referred to a comprehensive bibliography of books and monographs on finite element technology in [2]. Besides these texts, there are several publications more suited for engineering office use. These include the following guidelines and application-oriented texts that the reader may wish to consult:

- MORRIS, A, A Practical Gude to Reliable Finite Element Modelling, John Wiley & Sons, 2008 [3].
- MACDONALD, B.J., Practical Stress Analysis with Finite Elements (2nd edition), Glasnevin Publishing, 2011 [4].
- BRAUER, J.R., What Every Engineering Should Know About Finite Element Analysis (2nd edition), Marcel Dekker, Inc., New York, 1993 [5].
- MEYER, C. (Ed.), Finite Element Idealization for Linear Elastic Static and Dynamic Analysis of Structures in Engineering Practice, American Society of Civil Engineers, New York, 1987 [6].
- NAFEMS, Guidelines to Finite Element Practice, National Agency for Finite element Methods and Standards, National Engineering Laboratory, East Kilbride, Glasgow UK, August 1984 [7].
- STEELE, J. E., Applied Finite Element Modeling, Marcel Dekker, Inc., New York, 1989 [8].
- DNV-GL, DNVGL-CG—0127 Finite Element Analysis, 2015 [9].

PART 2
ASSESSMENT METHODOLOGY FOR FINITE ELEMENT ANALYSIS

The methodology developed for evaluating finite element analyses of ship structures is provided in Appendix A. The evaluation is carried out at two levels conducted in parallel. The highest level (Level 1) addresses general aspects of the finite element analysis (FEA) broken down into five (5) main areas:

1. Preliminary Checks
2. Engineering Model Checks
3. Finite Element Model Checks
4. Finite Element Results Checks
5. Conclusions Checks

These five groups are identified in each of the five (5) main boxes shown in Figure A/0-1. The Preliminary Checks cover background and objectives for the analysis. The Engineering Model Checks cover inputs and assumptions; the Finite Element Model Checks cover pre-processing; the Finite Element Results Checks cover post-processing for the analysis. The Conclusion Checks cover evaluations, recommendations, and reports for the analysis.

Evaluation of each of these five general aspects, in turn, requires that certain related detailed (Level 2) aspects be checked. The Level 2 aspects to be checked are listed within the main boxes and are presented in detail in separate tables in Appendix A that form the core of the evaluation process. The Level 2 tables contain many detailed questions regarding specific aspects of the FEA.

The methodology is intended to be used as follows. The evaluator will begin by assembling the analysis documentation and perhaps computer files of the finite element (FE) model and results. The evaluation then begins with the Preliminary Checks contained in Figure A/0-1 No 1 Preliminary Checks. The first of the Preliminary Checks involves assessment of the contents of the analysis documentation (Section A/1.1). To perform this assessment, the evaluator refers to the table entitled "A/1.1 Documentation Requirements", which is a systematic list of checks of the documentation for information that is essential for the FEA evaluation. The table also directs the evaluator to Section 3/1.1 of the guideline should further explanation or guidance be necessary.

- **Checkmark (√)**

  If an item is contained in the documentation, the evaluator should place a checkmark (√) in the corresponding box under the "Result" column.

- **Cross mark (X) or "*NA*" or "*?*"**

  If an item is not included with the documentation, the evaluator may enter a cross mark (X) in the result box, or "*NA*" (for Not Applicable), or "*?*" (for Further Information Required).

After checking off each item in the table, the evaluator is asked to answer Question 1.1 at the bottom of the page based on the evaluators' assessment of each item listed in the table. The evaluator should place the answer in the "Result" box to the right of the question, and then if acceptable check off the corresponding circle in Figure A/0-1. The table in Section A/1.1 also includes spaces for the evaluator to enter comments regarding specific and overall aspects of the document contents. At the end of the evaluation process, these comments will provide the evaluator with reminders of specific aspects of the FEA that were good, bad, or not explained well. The evaluator may refer to these comments to seek further explanation or clarification from the analyst performing the work before deciding on the final acceptability of the FEA.

Having completed the first of the preliminary checks, the evaluator then proceeds to the second set of checks entitled "A/1.2 Job Specification Requirements". In a manner similar to the previous checks, the evaluator will refer to the table in Section A/1.2 and perform checks 1.2.1 to 1.2.7, which are aimed at verifying that the analysis covers the main requirements and objectives of the job specification (or contract, or statement of work, etc.). Based on the results of these checks, the evaluator should answer Question 1.2 and then if acceptable check off the corresponding circle in Figure A/0-1. This procedure is repeated for the other Preliminary Checks (i.e., A/1.3 Finite Element Analysis Software Requirements, and A/1.4 Personnel Qualification Requirements).

Having answered all of the Level 2 questions for Prelliminary Checks and checked off the appropriate circles in Figure A/0-1, the evaluator should then move on to Engineering Model Checks. If the subject analysis does not meet these requirements then the evaluator may choose to terminate the evaluation.

The evaluation process continues as described above for each of the five main areas identified in Figure A/0-1. It should be noted, however, that the process is designed to proceed in the order provided. The evaluator should not move on to the next of the five major sections until the previous sections are satisfactorily completed.

Ideally, at the start of the job, the contractor would be given the assessment methodology as part of the job specification in order to encourage self-checking and ensure that the data provided by the contractor to the customer is complete.

In this document, consideration is given to both in-house FE analysts or contractors. In both cases, the quality of the FEA is highly dependent on the work processes, training, and tools employed.

The Appendix A forms can be adopted and converted into a spreadsheet format for if desired.

PART 3

GUIDELINES FOR ASSESSING FINITE ELEMENT MODELS AND RESULTS

The guidelines recommended below are structured to match the Assessment Methodology described in PART 2. Therefore, the guidelines are grouped under the same five sections:

1. Preliminary Checks
2. Engineering Model Checks
3. Finite Element Model Checks
4. Finite Element Results Checks
5. Conclusions Checks

3/1 PRELIMINARY CHECKS

This section describes the checks that need to be undertaken to ensure that the finite element analysis (FEA) satisfies certain basic requirements. The first requirement before evaluating an FEA is to ensure that there is sufficient documentation provided with the analysis. This step should ensure the analysis addresses the objectives, scope, and requirements of the work specification. It is necessary to establish that the tools the analyst uses in the FEA are adequate and appropriate to the analysis; this applies particularly to the software used. Finally, the analyst should be appropriately trained and should have sufficient experience.

3/1.1 Documentation Requirements

Proper documentation is an essential part of any FEA. The documentation submitted should be sufficient to allow a thorough evaluation of the FEA. The complete documentation package, which can be defined as that required by an independent party to reproduce the analysis, should be available and submitted if required by the evaluator. The complete documentation would typically include information from each of the five major groups being evaluated. The key information needed to evaluate the FEA analysis is as follows. Note that the items identified with a letter are included in the Section A/1.1 Documentation Requirements checklist.

Preliminary Checks

Job Specifications

a) Scope and objective of analysis.
   - The rationale for using FEA (e.g., types of solving problems and approaches in setup)
     - 2D or 3D problems
     - Static (or quasi-static) or Dynamic (with inertial effects)
     - Approaches in modeling
   - List of colors, if requested (for each property, each component, each material, and each thickness)

Reference documents

b) Reference codes, manuals, and/or standards required.
c) Physical problem documentation references.
   - drawings
   - CAD models

- geometric specifications and tolerances (e.g., 2D drawings/ sketches or 3D models) of the subject structure (with assembly if applicable)
- Identification of Initial imperfections (e.g., geometry imperfections, thickness reduction, crack profiles for fatigue or fracture analysis)
- gauging reports

FE Software Requirements

d) FEA software used.
- Finite element software, including its version and release date
- Pre- and post-processing software

Personnel Qualifications

e) Personnel qualifications.

Engineering Model Checks

Analysis type and assumptions

f) Analysis type(s).

Geometry Assumptions

g) Description of physical problem.

h) Description of engineering model.

i) Description of the FEA model.
- Contacts (e.g., linear or nonlinear)
- Loading (per load case section)
- Boundary condition (per load case section)

j) Plots of full FEA model and local details.

k) System of units.

l) Coordinate axis systems.

Material Properties

m) Material properties.
- Metallic (e.g., steel, aluminum, and magnesium)
- Non-metallic (e.g., polymer and rubber)
- Composite (e.g., GFRC, CFRC, foam, and piezoelectric)
- Analytical, semi-analytical, or empirical constitutive relationship
- With or without damage model (e.g., element erosion)
- High cycle or low cycle fatigue (e.g., SN or EN)
- Environmental assist corrosion-resistant (e.g., hydrogen diffusion)
- User-defined material models (e.g., UMAT or VUMAT in Abaqus)

Stiffness and Mass Properties

n) Stiffness and mass properties.

Loads and Boundary Conditions

o) Loads and boundary conditions.

- Loading history if applicable

Finite Element Model Checks

Element Types

p) Element type(s).

- Element formulations (e.g., 1D, 2D or 3D, full integration or reduced integration)
- The thickness for shell elements (with or without corrosion deduction)
- User-defined element formulation

Mesh Design

q) Meshing idealizations/assumptions/representations/simplifications.

- General structures (i.e., shell plates, deck plates, bulkhead plates, stringers, transverse webs)
- Connectors (e.g., spring, beam, and weldment)
- Girders
- Pillars
- Stiffeners
- Openings

r) Meshing criteria for 2D and 3D elements.

- Standard mesh size with maximum and minimum element sizes
- Finer mesh size with maximum and minimum element sizes
- Aspect ratios
- Warpage
- Skew
- Jacobian
- Maximum and minimum angle for quadratic and triangular elements
- Taper
- Percentage of triangular elements
- Mesh sensitivity study (if needed)
- Mesh and CAD overlay review

FE loads and boundary conditions

s) FE loads and boundary conditions.

Solution Options and Procedures

t) Solution options and procedures.

Finite Element Results Checks

u) Results.

- Critical state(s) and corresponding location(s)
- Maximum values (e.g., displacement, force, energy, stress, strain, or strain energy)
- Animations of interested states (e.g., deflections in modal or buckling analysis)
- Validation (e.g., comparison with analytical solutions)

Conclusion Checks

Results and Acceptance Criteria

v) Comparison of results with acceptance criteria.

Accuracy Assessment

w) Accuracy assessment.

Overall Assessment

x) Conclusions and recommendations for amendments.

The input and output data should be presented in graphical, tabulated, or textual form depending on what is the most convenient for evaluation purposes.

The documentation requirements listed in A/1.1 Documentation Requirements are the minimum required. In general, any additional information considered necessary for a complete evaluation should also be provided.

Plots should be annotated appropriately to show the location of the subject structure in the ship (e.g., frame numbers, deck numbers, etc.), axes to orient the model, location of equipment supported by the structure, and the position of major structural features that define boundaries (e.g., bulkheads). All symbols used in the plots should be defined either on the plots or in the body of the report.

3/1.2 Job Specification Requirements

The purpose of this check is to ensure that the analysis has been undertaken according to the requirements of the job specification. This can be done only if the documentation provided addresses every requirement of the job specification. It is not possible to list all such requirements, but at least the following items should be addressed:

- Definition of the problem
- Scope and objectives of the analysis
- All relevant documentation such as drawings, sketches, CAD, and reports to completely define the subject structure and loading
- Any previous analyses, service experience and experimental data related to the subject structure
- Acceptance criteria (e.g., allowable stress in an analysis in support of a design)

It is expected that the analyst has carefully read the job specifications and followed it as closely as possible. Deviations from the specifications, if any, should be identified and justified. All reference documents should be identified.

If the job specification does not explicitly call for an FEA, then the analyst should explain the rationale for using FEA in preference to another method of structural analysis, or preference to experiments. It is also expected that the analyst is aware of any previous related studies and their outcome.

The selection of FEA as the preferred method of structural analysis will depend on many features of the engineering problem. Features of the problem that should be discussed include, but are not limited to, the following:

- purpose of analysis;
- complexity of the structural form;
- redundancy of structural system;
- assessment of expected accuracy;
- accuracy of known input variables such as loads, material properties, etc.; and
- suitability, or otherwise, of hand calculation methods.

3/1.3 Finite Element Software Requirements

There are many finite element software systems available, i.e., commercial and open-source codes, which include pre-/post-processing software and solvers. Most are intended for general purpose FEAs (e.g., Ansys, Abaqus, LS-DYNA, and MSC.Nastran/Marc). There are some, however, that are known to be designed specifically for ship structure applications. Ship structure FEA is, to a certain extent, specialized in nature, and there are some efficiencies that can be gained from using a ship structure-specific application. It is not, however, necessary to employ ship structure-specific software, and a multitude of ship structure structural problems are analyzed using the general-purpose FEA software.

It is necessary to ensure that the software has been verified and validated. This may be less of a concern than when this guide was initially published. However, the FEA evaluator should still confirm that a verified and validated FEA software package is being used. This may be more of a concern for advanced analysis undertakings, such as fracture or impact analysis.

Commercial finite element analysis systems are large and complex. Developing and maintaining such systems require systematic methods to be applied to the design and development of the code, the testing, the verification and validation of the code, and the configuration management of the software system. Reputable software vendors rely on quality systems to ensure that the relevant processes that comprise the development and maintenance of the software are appropriately controlled. The evaluation of FEA software should include an assessment of the vendor's quality system.

In the past decades, several open-source FEA codes developed by reputable engineering companies (e.g., Code-Aster by EDF – Électricité de France) are being used and associated with verification and validation modules.

There are several ways in which FEA software can be validated. The methods for validating FEA software include:

- independent analysis

  in general, compared with analytical solutions or verified data

- experimental results

  Note that when it is impossible to compare the numerical output with experimental data. It is possible to validate the FEA software by validating relatively smaller-scale coupon tests. Additional attention needs to be considered, such as distributions of flaws or imperfections (e.g., the weakest link theory).

- service experience

Especially for those time-dependent problems, the service experience is extremely valuable to validate whether the numerical results are converged or reliable. For example, the fatigue lives of weldment are in the same scales or comparable.

- cross platforms (codes) analysis

  Results from well verified and validated FEA codes can be used as the benchmark. For example, the open-source code WARP3D specialized in macro- and mesoscale plasticity and fracture mechanics was validated by comparing with numerical results from Abaqus.

Many finite element software vendors publish verification examples. Generally, verification examples are based on problems with closed-form solutions. The analytical results are compared with those obtained by exercising the finite element code. While a comprehensive set of satisfactory verification examples is convincing evidence of high-quality code, it does not constitute proof. Verification examples based on problems based on closed-form solutions are necessarily simple, and the finite element models are generally not too demanding on the software. It is necessary, therefore, to employ additional methods to validate the software.

An additional validation method is to use benchmark problems that, while simple, are more representative of typical structure. In contrast to the type of verification example mentioned above, benchmark problems can be designed to use combinations of element types, element shapes that vary from the ideal, complex boundary conditions, multiple load cases, etc. to test the software. These problems are more closely related to how the software will be used in practice.

Closed-form solutions are generally not available for benchmark problems. However, results from other well-established FEA software could be regarded as an example of an independent analysis. If results from several other FEA software systems are consistent, or where any differences can be rationalized, then these results can be regarded as benchmarks. Any significant differences between benchmark results and those obtained from the candidate FEA software system would be an indication of unsatisfactory performance.

Depending on the size of the organization and the volume of FEA work, it may be useful to maintain a register of FEA software validated based on satisfactory performance using the methods outlined above. Similar logs or case-specific validations should be documented, where possible, supporting advanced analysis modeling scenarios. Alternatively, this function could be performed by a body representative of the industry, such as a professional society.

In the absence of such an arrangement at present, benchmark problems typical of ship structures have been formulated, and the results documented in Part 4 of this report. These benchmark problems could be used to evaluate candidate FEA software. If there is documented evidence (based on previous applications of the software to ship structural analysis problems) that the software can perform the required analysis, this requirement may be waived at the discretion of the evaluator.

Successful performance of the candidate FEA software on the benchmark problems is a necessary, but not sufficient, condition for approving the software. The software should also satisfy requirements outlined in the opening paragraphs of this section, particularly regarding requirements for the vendor's quality system.

In recent years, there is a trend that industries started making parametric studies by working with FEA codes, aiming to obtain the optimal designs (such as stiffness, weight, strength, fatigue life, and/or critical stress/strain). The FEA solvers adopted in the parametric studies should also be validated by the means mentioned above.

To date, there are a lot of FEA codes developed for solving engineering problems. To the author's knowledge (perhaps not an exhaustive listing) the widely used commercial and open-source FEA codes for general purpose applications as well as parametric analysis codes are listed as follows:

1. List of selected commercial FEA codes

a. Pre- and Post-processing software
    i. Altair series (e.g., HyperMesh, HyperView, and HyperCrash)
    ii. ANSYS series
    iii. BETA ANSA
    iv. LSTC LS-Pre/Post
    v. MSC series (e.g., Patran and Apex)
    vi. Siemens series (e.g., FEMAP)
b. Solvers
    i. ADINA
    ii. Altair series (e.g., Optistruct, Radioss, and partner alliance packages)
    iii. ANSYS series (e.g., Mechanical APDL, Workbench, Fluent, HFSS)
    iv. LSTC. LS-DYNA
    v. MAESTRO
    vi. nCode
    vii. MSC series (e.g., Nastran and MARC)
    viii. Siemens series (e.g., NX. Nastran)
    ix. SIMULIA. ABAQUS
    x. Strand7

2. List of selected open-source FEA codes
    a. Pre- and Post-processing software
        i. Cubit / Trelis
        ii. EDF Salome
        iii. Gmsh
        iv. ParaView
    b. Solvers
        i. EDF Code_Aster
        ii. Impact
        iii. Warp3D
3. Parametric analysis codes
    a. Altair HyperStudy
    b. SIMULIA Tosca
    c. VR&D Genesis

Recently, the vendors of commercial FEA codes start adopting the benchmark examples provided by the not-for-profit companies (e.g., NAFEMS) to demonstrate the accuracy of their codes [10]. As to the end-user, it is encouraged to independently examine the accuracy of the FE code by verifying the results of benchmark examples.

3/1.3.1 Reasons for Using A Particular FEA Software Package

It is recognized that the analyst will prefer to use FEA software packages that are readily available and that the analyst has experience with. However, the analyst should assess the suitability of the selected FEA software for the analysis under consideration. The items that should be discussed include the following:

- availability of required element types (e.g., general types, special types, or user-defined types)
- availability of required material types (e.g., general types, special types, or user-defined types)
- availability of required material behavior (e.g., yielding, plasticity, crack, creep, fracture, and fatigue)
- availability of required load types (e.g., forced-displacement, concentrated force, distributed force/pressure, gravity, cyclic loading, thermal, electric, fluid and multidiscipline)
- availability of required boundary condition types (e.g., slide, pin, clamp, and inertial relieve)
- availability of required contact types (e.g., linear and nonlinear)
- capability of the software to perform the required analysis
- preprocessing and postprocessing capabilities
- support from vendors (e.g., documentation and technical supports)
- Other concerns
    - Geometry effects (e.g., initial imperfection, stress stiffening, 2nd order load effects)
    - Temperature effects (e.g., material degradation, thermal expansion)
    - Nonlinear load effects (e.g., follower loads)

Note that vendors of FEA software may be able to provide the following [11].:

- recommended hardware specifications
- compatible hardware vendors/ manufacturers
- training resources or programs for analysts
- Technical direction and troubleshooting support
- correct driver versions

3/1.4 Personnel Qualification Requirements

The personnel performing and checking the analysis must meet minimum training and experience requirements. The following aspects of personnel background will need assessment:

- formal academic or professional qualifications
- engineering expertise in design and analysis of ship structures
- relevant experience in the modeling and analysis of design problems using the finite element method

- familiarity with, and appreciation of, the limitations of the software employed

Personnel are seperated into two categories: *analysts* and *checkers*. An *analyst* is a person who undertakes the FEA. A *checker* performs independent checks of the analyst's work and certifies the quality of the work. In some instances, there may be a lead analyst that provides direction and oversight to less seasoned analysts. In this instance, the judgment as the appropriateness of the lead analyst to act as the checker should be considered, and the qualifications of the less seasoned analyst may be considered to be augmented by the supervision provided by the lead analyst.

Neither these guidelines or the routines built into software and documentation provided for those routines can serve as a substitute fro knowledge of the principles involved. Ideally, an analysis should be backed by analysis of the results of experimentation on a similar structure.

3/1.4.1 Academic and Professional Qualifications

Ideally, the analyst and the checker should be qualified to first degree level in engineering and have taken at least one full course in FEA. Professional Engineer (or equivalent) status is good practice for the checker and desirable for the analyst. In addition, different FEA vendors and professional engineer committees are now providing professional training in helping understand both theoretical and practical aspects.

3/1.4.2 Training and Experience

The analyst and checker should have received training in the application of the finite element method. Either of the following is acceptable, in principle, as training:

- Training provided by various courses offered by educational establishments and software vendors. These courses are only acceptable if they are application-oriented. It is preferable that these courses also cover the theoretical background that could allow analysts and checkers to understand the limit of finite element methods.
- In-house formal or informal training provided by a supervisor capable of satisfying the requirements of a checker. The content of the training should be at least comparable to an equivalent period of the application-oriented training program. The training course(s) should be documented. In addition, it is recommended that the group has an internal best FEA practices (company-specific) as a reference that is continually updated.

The analyst or checker must be familiar with the design requirements, codes of practice, analysis, and design standards relating to ship structures. The checker must have, and the analyst should preferably have, experience with analyses of comparable size and complexity as the analysis under assessment.

The checker should be an experienced analyst with substantial experience in the application of the finite element method. This experience should include working as an analyst on finite element analyses that are comparable in complexity to the analysis the checker will be verifying. The documentation should include a brief outline of previous experiences.

The experience requirements for analysts recommended by NAFEMS [1] are summarized in Table 3/1-1. The experience required of the analyst depends on the criticality of the analysis. The criticality category depends on the consequences of the failure of the structure being analyzed. In addition, it should be noted that the requirements listed are just minimums. Finding more experienced analysts is encouraged.

Table 3/1-1 Minimum Recommended Experience Levels (adapted from NAFEMS, 1990)

Analysis Category	Engineering Experience	FE Modeling and Problem Solving	
	Design & Analysis Experience	*FE Experience After Formal Training for Each Analysis Type*	*Relevant Jobs Performed*
1. Vital: endanger human life, or property or the environment on a scale of a public disaster	5 years	6 months	2 x Category 1 under supervision or 5 x Category 2 properly assessed
2. Important: Category 1 problem however analysis is not an exclusive part of the integrity demonstration	2 years	2 months	1 x Category 1 or 2 under supervision or 3 x Category 3 properly assessed
3. Advisory: All analysis other than the ones covered in Categories 1 and 2	1 year	1 month	Prescribed Benchmarks [1]

[1] For example, see PART 4 of this report for benchmark problems

3/2 ENGINEERING MODEL CHECKS

The checks recommended in this section are generic in nature, and they form part of any engineering analysis. The engineering model is a simplified representation of the physical problem, and hence it is crucial that this modeling process is undertaken correctly since the finite element analysis (FEA) cannot improve on a poor engineering model. The aspects covered in this section include:

- Analysis Type
- Analysis Geometry
- Material Properties
- Loads and Boundary Conditions

The discussion here is restricted to an understanding of the physical problem as well as the corresponding impacts of different simplifications. Translating these aspects into a finite element model, in a format recognized by the software program, is covered in Section 3/3.

3/2.1 Analysis Type

The available FEA analysis types are as summarized in Table 3/2-1, and the conditions for the analysis types are presented in a flowchart (see Figure 3/2-1 Analysis Type Flow Chart)

The analysis targets for each analysis type have also been briefly summarized in Table 3/2-1. Note that the analysis targets are not limited to those listed in the table, and it is up to analysts to define the analysis type and choose the proper FE solvers. It is also recommended to carefully review the job specifications, and such that the analysis types and targets can be appropriately defined prior to the pre-processing of FEA works.

Table 3/2-1 FEA Analysis Types

	Analysis Types	Analysis Target(s)
1	Linear static analysis	Stiffness, etc.
2	Nonlinear analysis	Strength, remaining strength, permanent deformation, limit load, material/structural failure, pre-load, etc.

	Analysis Types	Analysis Target(s)
3	Buckling analysis (linear or nonlinear)	Critical load, post-buckling behavior, etc.
4	Impact analysis (quasi-static and dynamic)	Energy dissipation, critical stroke, failure modes, crash resistance, etc.
5	Fracture/damage analysis	Stress intensity factor, stress concentration, fracture driving force, crack propagation, etc.
6	Fatigue analysis (high cycle and low cycle)	Fatigue life, remaining life, etc.
7	Whole Ship Analysis	Hull girder stresses
8	Vibration Analysis (frequency/transient/random-spectrum response) (linear or nonlinear)	Eigenmode, modal/frequency response vibration, etc.
9	Thermal analysis (conduction, convection, radiation)	Temperature, temperature gradient, flux, etc.
10	Optimization / DOE (linear or nonlinear)	Weight reduction, strength, fatigue life, etc.
11	Multi-physics, electromagnetics, welding and/or co-simulations	Antenna design and placement, EMC, Structure-thermal-fluid interaction, etc.

An engineering model is a simplification and idealization of an actual physical structure or component. The analyst should include, as a minimum, discussion of the following analysis type topics in the body of the report:

- purpose of analysis (e.g., design, optimizations, failure investigation, integrity assessment, validation & verification, design of experiment, etc.)
- whether the problem is static or dynamic
- whether the problem is linear elastic, nonlinear elastic, or nonlinear elasto-plastic
- whether the problem is time-independent or time-dependent
- appropriateness of linear elastic analysis
- appropriateness of nonlinear elastic analysis
- appropriateness of nonlinear elasto-plastic analysis
- assumptions of material models (i.e., constitutive relationship), as well as damage models if applicable, for nonlinear problems

The loadings for all physical problems will be either static, dynamic, or a combination. Some dynamic loads can be treated as quasi-static, and this should be done where possible, and it is desirable to minimize the required computational power. However, special care should be taken before treating any inertia force related loads as quasi-static.

The analyst will need to consider the frequency range over which there is significant energy in the forcing function when a dynamic analysis is required. This will determine the number of modes to be extracted. Note that different algorithms may develop modal values with certain differences.

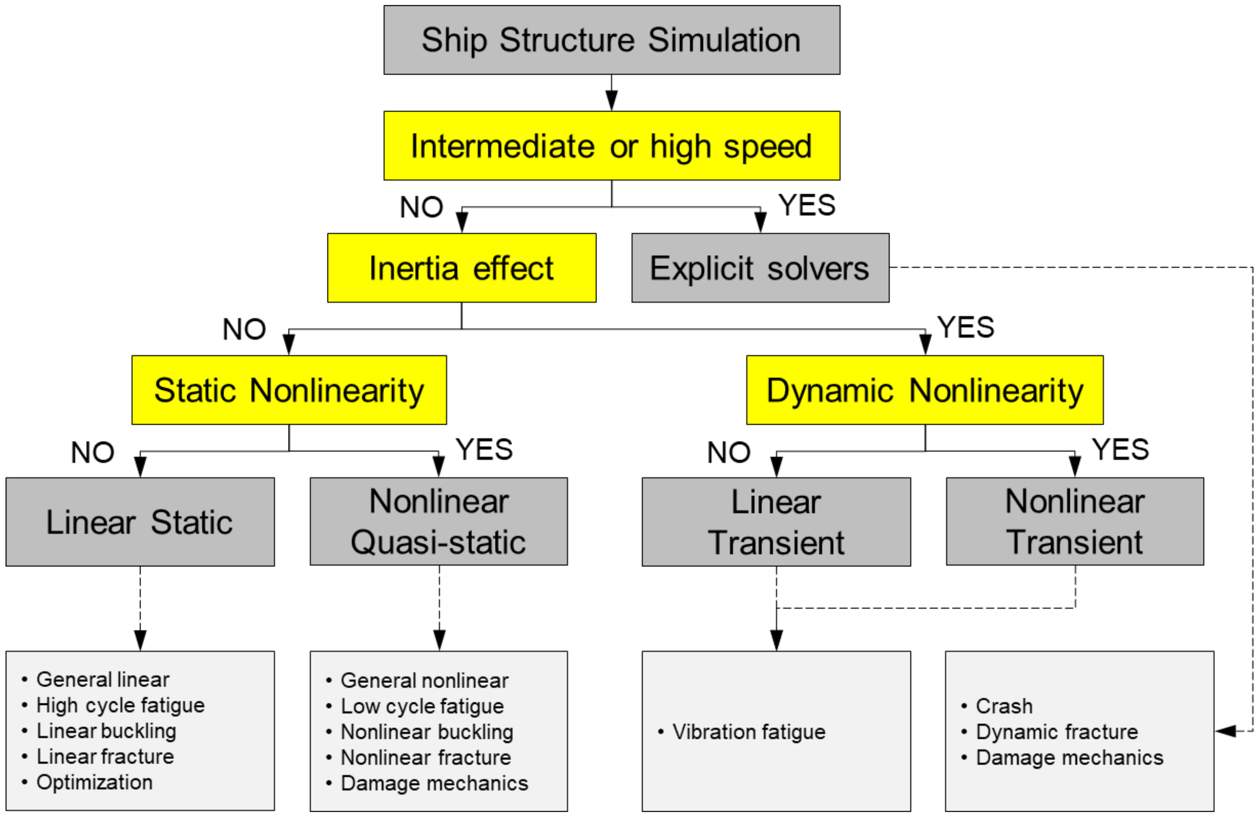


Figure 3/2-1 Analysis Type Flow Chart

The analyst can use the flow chart shown in Figure 3/2-1 to determine which type of analyses or set of simulation methods should be used for a given physical problem. The flowchart is a two-way lookup diagram:

(1) From ***top*** to ***bottom***: if limit information is available for the physical problem, the "YES"/"NO" logic helps the analyst understand which simulation method should be used. For example, if a ship structure is under a blast load, which is generally an intermediate or high-speed scenario, the explicit solver should be used for the simulation. Note that it becomes a time-dependent problem where the applied loads and boundary conditions should be representative of the physical observations.

(2) From ***bottom*** to ***top***: the analyst may directly find the analysis type, if available. The flow chart helps understand which set of simulation methods should be used in the analysis. For example, the high cycle fatigue analysis is one kind of linear static analysis, which means it is time-independent, no inertia effect, and no nonlinear phenomenon considered.

The remainder of this section provides brief descriptions for a select number of Table 3/2-1 analysis types.

3/2.1.2 Linear static analysis

A linear analysis has a straight-line relationship between force and deflection. This analysis type is suitable for evaluating structures before yielding (i.e., stress-strain relationship within the linear region) where buckling is not a concern. The calculation does not consider failures in nature (i.e., material models), and it is up to the analyst to conclude whether the structure meets the criteria (e.g., yield strength or ultimate tensile strength).

Static analysis has loads that are independent of time. Note that in order for this to be the case, the sum of the forces and moments must necessarily be in equilibrium, so they do not change with respect to time.

3/2.1.3 Nonlinear analysis

A nonlinear analysis does not have a straight-line relationship between force and deflection. This may be the result of one or more of the following nonlinearities:

- A material nonlinearity.
- A geometric nonlinearity.
- A contact or boundary condition nonlinearity

Material nonlinearity is caused by nonlinear stress and strain relationship (i.e., a nonlinear material model). There are several types of nonlinear stress and strain relationships, including nonlinear elastic, elasto-plastic, visco-elastic, and visco-plastic. For general metallic materials, a nonlinear analysis is needed when the stress or strain state exceeds the yielding point. Note that a low cycle fatigue analysis is an example of a material nonlinear analysis. Some non-metallic materials (e.g., rubber, plastic, and composite materials) may also show a nonlinear stress-strain relationship. Material nonlinearity may also be observed at a typical creep loading condition (i.e., elevated temperatures, external loads, and over a certain period).

A geometric nonlinearity indicates that the slope of force and deflection ($\boldsymbol{F} = \boldsymbol{Kd}$) curve is not a straight line, i.e., $\boldsymbol{K}$ is dependent on $\boldsymbol{d}$. For example, the value of $\boldsymbol{K}$ changes suddenly after reaching the bifurcation point for a buckling problem. Note that for geometric nonlinear analysis, small strain formulations are sufficient to capture the deformation/rotation and stress/strain states. It should be noted that the force direction may change and follow the instantaneous local coordinate (i.e., deformed state) – also known as the follower force.

In an analysis that includes contact, the contact stiffness may need to be defined to capture the contact and separation, where the contact stiffness is a function of displacement.

It should be noted that in the FEA codes, the analyst needs to define or enable the formulations used for solving nonlinear problems, such as large or small deformation and finite or small strain formulations.

Buckling analysis is a special form of nonlinear analysis. Buckling can occur when one or more of the following are true: (1) compressive load (or equivalent compressive load from bending), (2) high height-over-width ratios (e.g., slender beams or sheet metal), (3) bending stiffness is not comparable with axial stiffness in terms of scales, i.e., $K_{bending} << K_{axial}$, (4) large lateral deformation, and (5) geometric imperfection.

3/2.1.4 Impact analysis

The impact analysis can be either quasi-static (relative time-independent) and dynamic (time-dependent). Note that, in general, it is recommended to use the implicit and explicit time-integration methods to carry out quasi-static and dynamic impact analyses, respectively. The differences between the two integration techniques will be discussed in the following section (3/3.6.2.2). The current practice focuses on the time-dependent impact (crash/collision) analysis. The impact load is characterized by the kinetic energy, which is a function of the mass of the ship and the collision speed.

3/2.1.5 Fracture/damage analysis

Fracture and damage analyses are special FEA analysis types that investigate what will happen locally in the vicinity of high-stress concentration areas. In such an analysis, high-stress concentrations appear close to the tip of flaws/pores/voids/ cracks, and the magnitude of stress depends on the tip geometry and the shape of the structure. The fracture analysis, in general, is carried out to evaluate the structural integrity. The results may include critical load values, critical crack sizes, fracture toughness, and remaining fatigue life. A damage analysis, in general, is carried out to predict the material failure as well as the failure mode by evaluating the crack formation and propagation in a macro- or mesoscale without considering the failure mechanism of materials in the micro-scale.

In a linear fracture analysis, the stress intensity factor (*K*) and crack driving force (e.g., G – energy release ratio and CTOD – crack-tip opening displacement) is measured. In a nonlinear fracture analysis, the crack driving force (e.g., J-integral and CTOD) and crack-tip constraint (e.g., T-stress, Q-index, A_i-index) are measured. Note that a nonlinear fracture analysis includes both small-scale and large-scale yielding conditions. Therefore, formulations and solving options should be chosen with extra care.

In a damage analysis, different damage models have been developed for both metallic and non-metallic materials. In general, the damage criteria can be based upon the maximum principal stress, maximum principal strain, critical strain energy, cohesive strength, triaxiality, and so on. The FEA input parameters need to be calibrated from testing, and sometimes user-defined material models are needed. Note that extra cares are needed to carry out such kinds of special analyses.

3/2.1.6 Fatigue analysis

In a fatigue analysis, there are two different types of analyses: high cycle fatigue analysis and low cycle fatigue analysis. A high cycle fatigue analysis typically uses a stress-life approach with an SN curve as the input. A low cycle fatigue analysis typically uses a strain-life approach with an EN curve as the input. In general, a high cycle fatigue analysis applies greater than 1000 loading-unloading cycles to a subject structure, to initiate a crack or exhaust the fatigue life. In design codes (e.g., [12,13]), the fatigue life can be evaluated from fracture based power law (e.g., Paris equation) and a damage accumulative rule (e.g., Miner’s summation).

3/2.1.7 Vibration analysis

Vibration analysis has loads that are dependent on time. Dynamic analysis can be either linear or nonlinear. Dynamic analysis in FEA typically takes one of five forms:

1. a modal analysis (also known as a natural frequency of free vibration analysis),
2. a frequency response analysis (also known as a forced vibration analysis),
3. a spectral analysis (also known as a random vibration analysis),
4. a transient response analysis
5. a shock analysis (i.e., dynamic design analysis method [DDAM])

A modal analysis determines the modes and the natural frequencies of a system. A frequency response analysis determines the system response based on a single frequency input. The response values can include displacement, velocity, and acceleration at any given node in an FEA model. This type of analysis is discussed more fully in Section 3/2.8. A spectral analysis determines the statistical likelihood of a given response (displacement, velocity, and/or acceleration) based on an applied input spectrum. The input spectrum for such an analysis is a formula with frequency as the independent variable and energy density as the dependent variable. The three foregoing analyses are determined in the frequency domain. A transient analysis is a time-domain analysis that determines the response at a given time. Lastly, a shock analysis is a specialized vibration analysis typically used for naval vessels.

Ship structure vibration can cause structural fatigue, malfunction of machinery and equipment, passenger discomfort, and/or crew habitability issues. It is therefore vital for ship structural engineers to analyze ship structural vibration during the design phase and be able to correct it if necessary.

Ship structure vibration issues can be addressed by making a change to one or more of the three primary vibration variables. These are the excitation force, the structural stiffness, and the damping. Reducing or modifying the frequency of the excitation force can reduce or modify the vibration response. Increasing or decreasing the structural stiffness can move the natural frequency of the structure away from the resonant frequency. Increasing damping can reduce vibration response. None of these, however, are easy to modify in service; therefore, early detection of any vibration issues is extremely important to any given design.

The primary sources of vibration excitation on a conventional propulsion ship are the main engine and the propellers. The propeller induced excitations are the result of two phenomena. First, the hull wake effect that causes alternating thrust to be transmitted through the propeller shaft. Second, the hull-propeller clearance effect that causes alternating pressure pulses on the hull. These excitation forces can be difficult to obtain. The main engine excitation force information should be provided by the engine manufacturer. They should have information that provides both the amplitudes of the forcing and the axis in which the forcing acts. See [14] for more information regarding the main engine, hull wake, and hull-propeller clearance excitation forces.

Ship structural vibration responses can be grouped into two groups: whole-ship hull-girder responses and local responses. The hull-girder responses are the lowest natural frequencies of a vessel. They are the result of the bending and twisting of the hull girder. Large ships have hull girder frequencies of less than 5 Hz. Small ships can, however, have higher hull girder frequencies. Local responses tend to have higher frequencies and are the result of mode shapes specific to stiffened panels, stanchions, and other ship structures. As an approximate guide, the following may be used for the first few modes of the below structures:

1. Hull Girder 1- 5 Hz
2. Main Mast 5 - 10 Hz
3. Superstructure 10 - 20 Hz
4. Typical Stiffened Plate Decks 10 - 40 Hz

Modal analysis is relatively simple to perform using FEA. These are used quite often in ship structures problems such as determining the natural frequency of mast. Frequency response analyses and spectral analyses are more valuable because they determine the actual displacements, velocities, and accelerations of a structure based on a given cyclical force. These, however, require premium FEA software and expertise. They are used less often at present than modal analysis but are growing in use as the FEA software tools develop. They are a valuable part of today’s ship design process. Transient analysis requires more computational power than is practically available for most ship structure problems and is largely not used in practice today in most ship design processes.

3/2.1.8 Thermal analysis

Thermal analysis is used to evaluate the temperature and fluxes of structures under thermal loading. The temperature shows the amount of thermal energy, and the flux shows the flow of thermal energy. In heat transfer analyses, there are various types of mechanisms, for example: (1) thermal conduction, (2) thermal convection, and (3) thermal radiation. A thermal conduction analysis deals with the thermal energy exchange by molecular motions [15]. A thermal convection analysis deals with the thermal exchange between solid and fluids surrounded. A thermal radiation deals with electromagnetic radiation emitted from a material.

The thermal analysis may be coupled with structural analysis to evaluate the stress or strain responses of the structure in light of the thermal loading.

It is also worth noting that in thermal analysis, heat transfer may be steady or unsteady, and may be linear or non-linear. Thermal radiation, as an example, is a highly non-linear process (4th power of temperature).

3/2.1.9 Optimization and Design of Experiments (DOE)

The optimization analysis can be categorized into two different types: concept design optimization and revision (or tuning) optimization. Concept design optimization includes topology, topography, and free size optimization. Revision or tuning optimization includes size and shape optimization (see Figure 3/2-2). Before carrying out such optimization analyses, the design variables and corresponding targets should be predefined, including associated upper and lower bounds. For example, in a size optimization analysis where the thickness of a metal panel is the design target, the analyst would input a starting thickness and allowable thickness ranges.

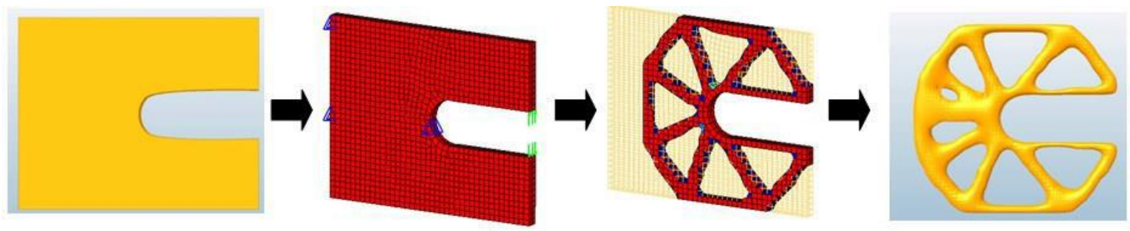


Figure 3/2-2 An example of topology optimization [10]

3/2.2 Analysis Geometry

3/2.2.1 General

Ship structures are usually large and complex in nature and can typically only be analyzed after idealization of the structure due to computing power limitations. It is, therefore, highly recommended to carry out FE analyses by conveying only the essential parts of the structure.

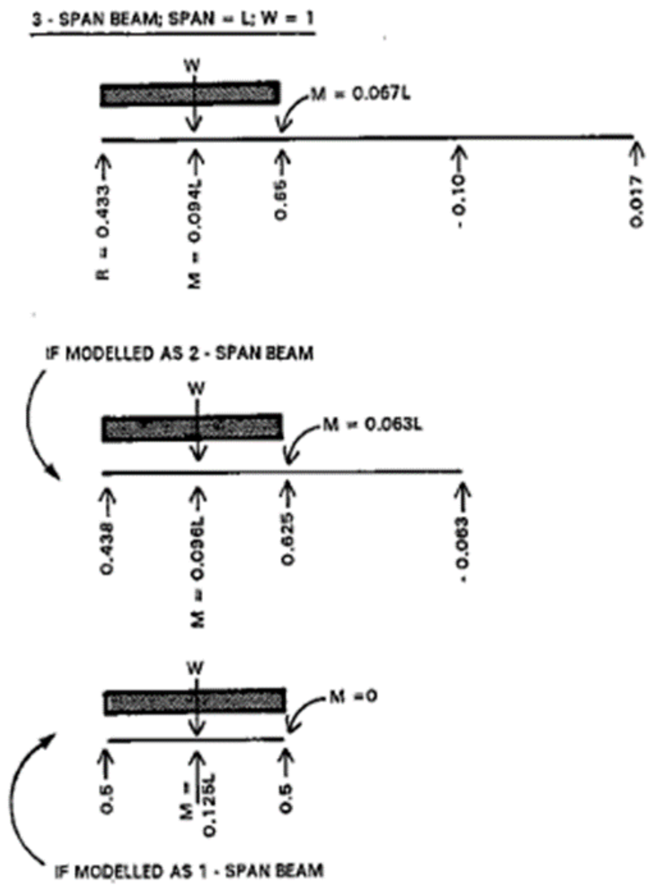
One of the first questions to arise during the planning phase of an FEA is "*how much of the structure needs to be modeled to yield answers of the required accuracy*." This is best approached by considering what the influence on the results of interest is by extending or reducing the extent of the model. If the influence is negligible, then the extent of the model can be established in advance. However, performing such an exercise on complex structures through intuition alone is difficult and these decisions can be aided by a discussion of the purpose of the analysis. In such cases, the analyst should ask the following questions before starting the FE modeling.

- "What is the purpose of this analysis?"
- Is it being done for the design or assessment of the existing structure?
- Is the purpose to analyze for stress, fatigue, vibration, dynamic response, or other purposes?

The answer to those questions will influence the appropriateness of the modeling performed. It will also help the analyst determine which structure needs to be considered in the analysis.

Several simplifying assumptions are made in the idealization process. In order to do this successfully, it is necessary to have a reasonable qualitative understanding of the expected response. This will allow the reduction of the complex response of the actual structure to its essentials. The elements that need to be considered in this idealization process are the character of loading, the primary loading paths, and the parts of the structure that participate in the response.

Consideration of the likely load paths, deformed shapes/patterns, failure modes, or critical areas will help establish the extent of the structure that should be modeled, and what boundary conditions might be appropriate. Once the main structural actions are identified, it is possible to apply simplified structural models to guide the analyst in deciding the extent of the structure to be modeled.



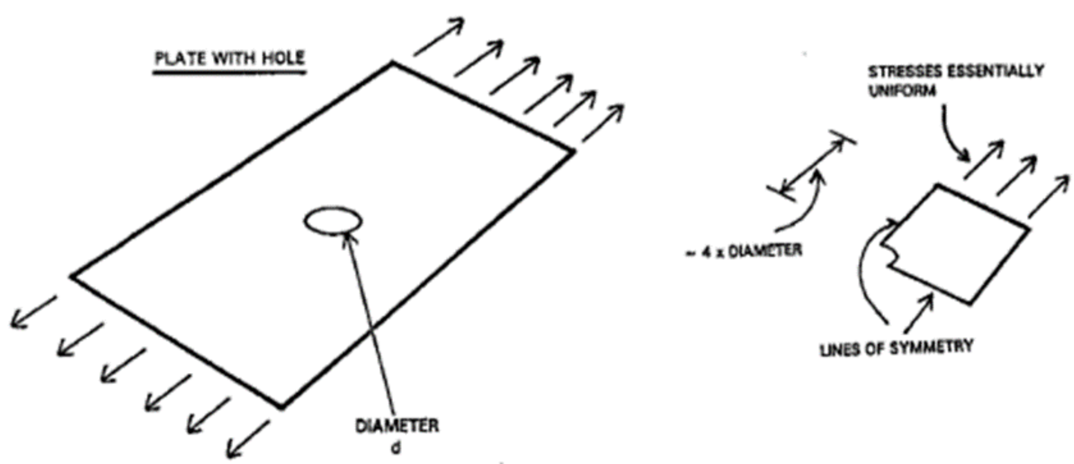


Figure 3/2-3 illustrates the concept with simple examples. The following general principles should be borne in mind when using this approach:

Drastic changes in stiffness are potential regions to end the model. Figure 3/2-4 presents an example in which the left-hand side of a beam is supported by a stiff structure. The bending stiffness of beams is proportional to (I/L^3), where I and L are the second moment of area about the neutral axis and the span/length of the beam, respectively. In this example, a difference in stiffness of, say, two orders of magnitude would be sufficient to justify the modeling approach shown in the figure. This general approach can be adapted for other more complex structures. It should be noted that different beam elements can be assigned with different cross-section types as well as I values.

Identification of load paths, deformed shapes/patterns, failure modes, or critical areas is a good indicator of which parts of the structure are best to model.

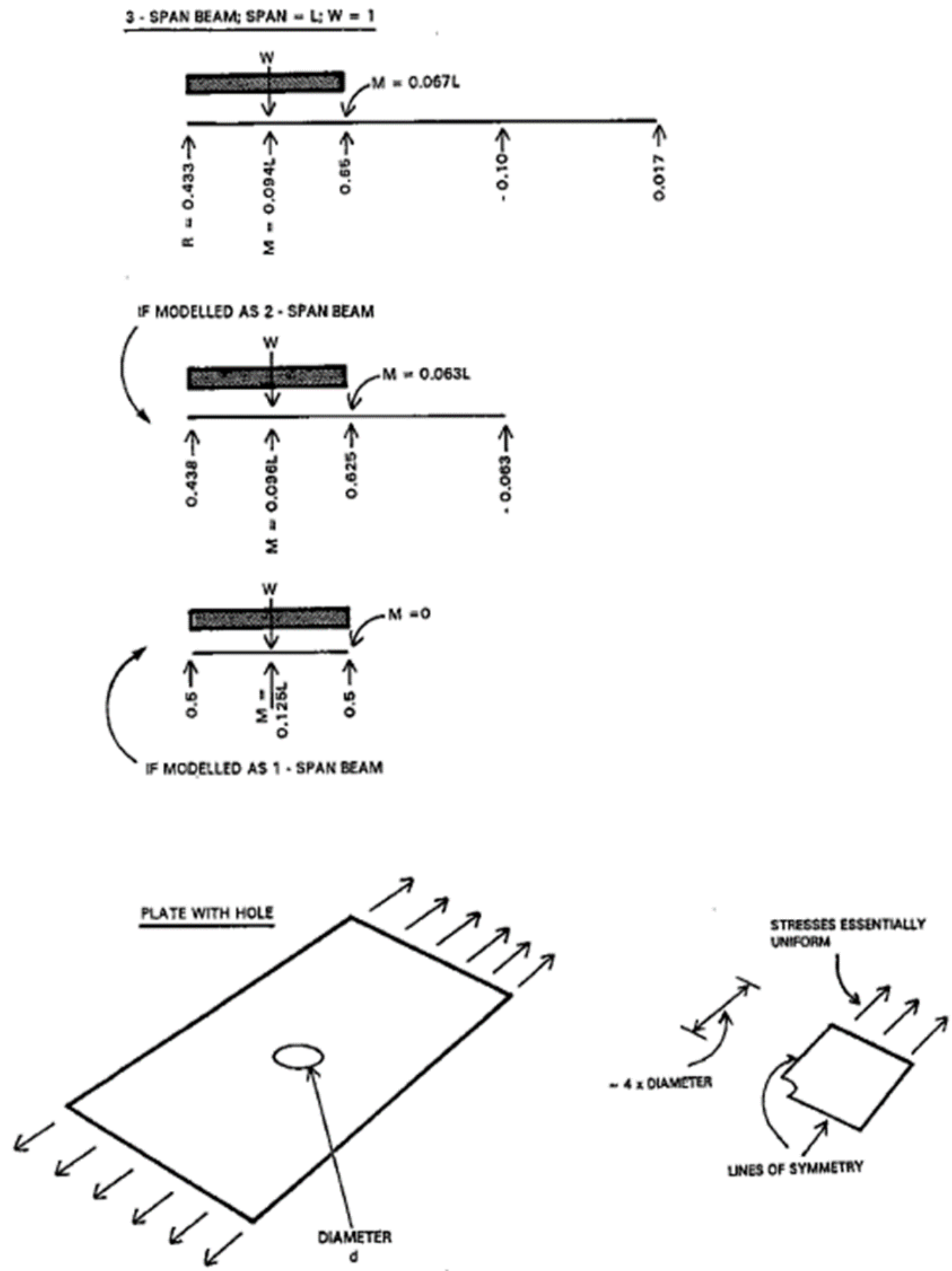


Figure 3/2-3 Examples of Simple Models that can Indicate Extent of Structure to be Modelled

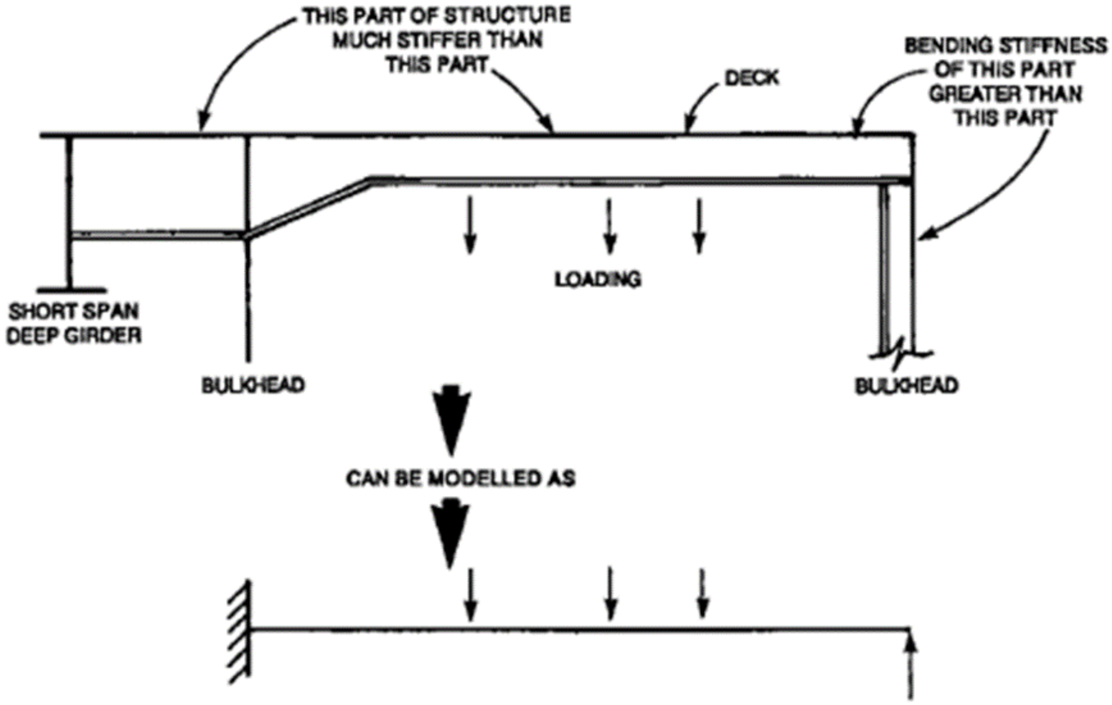


Figure 3/2-4 Large Changes in Stiffness to Indicate the Extent of Model

The actual extent of the finite element model depends on a tradeoff between the resources available for the analysis and the general requirements that all significant portions of the structure be modeled.

Most real structures are discontinuous and irregular at a local level. For example, it is likely that there will be brackets attached to the structure, openings, access holes, etc. The explicit modeling of these features is not practicable, and not necessary if only the global response is of interest, or such discontinuity has little impact on the response of interest area.

All structures are three-dimensional (with thickness). Depending on the configuration, it is often possible to reduce the number of dimensions to be considered. One basic application of this reduction is the typical reduction of a ship structure from a 3D solid model to a 3D model that uses 2D elements that have a mathematical means of representing their thickness, including the stresses across the thickness.

If a free or forced vibration analysis is being performed, additional cares should be exercised in determining the extent of the model. The PART 5 ADVANCED ANALYSIS SAMPLE APPLICATIONS frequency response application Section 5/4.5 Force shows that the extent of the model has a significant impact on the results. Specifically, a main engine forced vibration analysis may require the ship structure to be modeled from engine room bulkhead to engine room bulkhead or beyond depending on the ship's structural arrangement. Just modeling the main engine foundation down to the tank top would result in a non-conservatively stiff FEA model. Note also that in the current engineering practice, the substructure technique (e.g., super-element method [16]) is being widely adopted for such kinds of noise, vibration, harshness, and/or fatigue analyses.

If the FEA is concerned primarily with local effects, the concepts underlying Saint-Venant's principle can be helpful in establishing the extent of the model [17,18]. Essentially, this principle states that the replacement of a load (which could be caused by a restraint) by a different, but statically equivalent, load causes changes in stress distribution only in regions close to the change. Figure

3/2-5 illustrates the principle. Note that such kind of time-independent equivalent load technique can be achieved by using different load types or the combinations of them (e.g., force, displacement, pre-stress, pre-strain, thermal, etc.) [19–21].

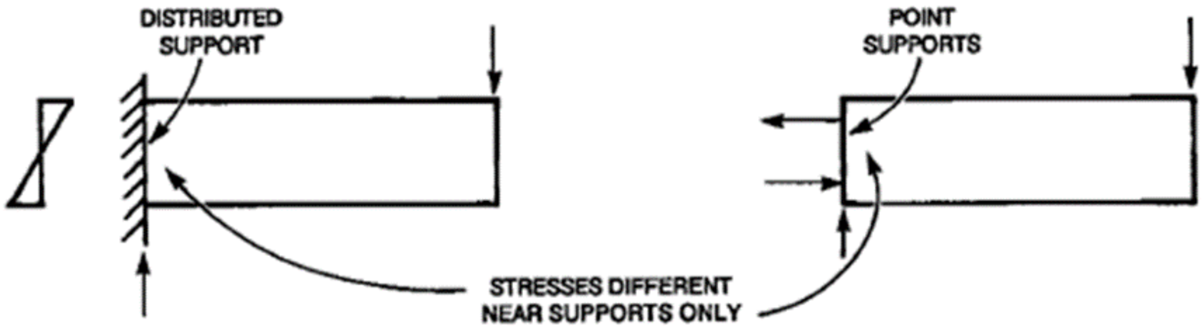


Figure 3/2-5 Illustration of Saint-Venant's Principle

The analyst should describe and justify the extent of the model. The justification statements should include a discussion of:

- all significant structural actions captured by the model.
- including and excluding parts of the structure
- taking advantage of symmetry, antisymmetry, axisymmetry, plane-stress, or plane-strain
- requirements to accurately predict stresses, strains, forces, deflections, and/or other parameters of interest.
- regions of the structure of particular interest
- whether the Saint-Venant's principle is satisfied
- obvious changes in structural stiffness that may suggest a model boundary
- very local application of the load to a large uniform structure
- for large models, can top-down analysis be used?
- for large models, can substructure (or super element) technique be adopted?
- whether the structure can be modeled with line elements (1D), area elements (2D), or volume elements (3D) or a combination of different element types (e.g., hybrid or user-defined)
- whether the formulation of the element is 1st- or high-order, or special type (e.g., crack-tip element to present singularity or cohesive element)

3/2.2.2 Mass and Added Mass

Certain problems in ship structures require that the interaction between the structure and the fluid be considered. The comments made here are limited to cases in which fluid displacements are small. The most common example is the vibration of plated structures adjacent to the fluid.

For vibrations of the plated structure adjacent to fluid, the practice is to account for the presence of the fluid by adding masses to the structure to represent the fluid. This mass is usually termed "added mass" and represents the part of the mass of fluid that the structure has to accelerate during vibrations. There are several sources for data on added mass appropriate to plate vibrations (see

ISSC, 1991- Report 11.2 for typical sources).

Hull girder vibrations can be treated similarly. Reference [1] provides guidance on approximate methods for computing added mass for the hull girder.

The use of added masses to account for fluid-structure effects is generally quite approximate. More rigorous methods require the finite element modeling of the surrounding fluid. Many general-purpose FEA systems include fluid elements that allow certain types of acoustics, sloshing, and fluid-structure analysis problems to be solved. This is a specialist area. For guidance, the reader is referred to finite element texts and the user manuals of the FEA system to be used in the analysis.

See also Reference [2] for additional guidance on the application of added mass to vibration analyses.

3/2.2.3 Shock Analysis Mass Modeling Reduction

Shock analysis, that would be otherwise too large, require a reduction in the number of dynamic degrees of freedom (DOFs). This can be done by changing the way the mass is modeled.

Certain techniques, such as Subspace Iteration, implicitly reduce the size of the problem. The degree of reduction depends on the number of modes that need to be extracted. The reduction process can also be accomplished more directly by a procedure known as condensation, and perhaps the best known such technique is Guyan reduction. While the condensation process is generally detrimental to accuracy, the loss of accuracy need not be significant if the appropriate guidelines are followed.

There are two alternative methods for mathematically modeling mass. The simpler of the two methods is the lumped mass method in which concentrated mass is located at nodes. The value of the mass represents the mass of the surrounding structure and equipment. This approach yields mass matrices that are diagonal. Rotational inertias may also be modeled in this fashion or can be condensed out. Rotational inertias are often ignored when this method is used. The alternative approach is called the consistent mass method. This is a theoretically rigorous method that results in a mass matrix with off-diagonal terms. The presence of these off-diagonal terms in the mass matrix is responsible for making dynamic analysis using consistent mass matrices more computationally demanding than when using lumped mass matrices. For large models, there does not appear to be much difference between the two methods in terms of the accuracy attained, at least for lower frequencies.

Whatever the technique may be for calculating natural frequencies and modes, the mass distribution needs to be accurately modeled.

Natural frequencies and modes are calculated for one of the following reasons:

1. to compare natural frequencies and modes of a structure with the frequency/ies of some source of vibration
2. as the first stage in the calculation of structural response.

In either case, it is necessary to anticipate the results to some extent. In the first case, the natural frequencies calculated must bracket the frequency of the vibration source. In the second case, the spectrum of the forcing function, for example, harmonic forces from the propellers or impulse loads from underwater shock, will suggest the range of natural frequencies of the structure that need to be calculated.

The higher the vibration mode, the more detailed the mass distribution needs to be. The general principle is illustrated in Figure 3/2-6. In the actual structure, the mass is distributed over the length. Hence, a reasonable number of lumped masses are required to represent the distributed mass. For higher modes, a more detailed representation of mass is required because the mode shape is more complex. In the example shown in the figure, essentially, a single mass is being used to represent the dynamics of one lobe of the third vibration mode. This is in contrast to the five masses used to represent the dynamics of the single lobe in the first mode.

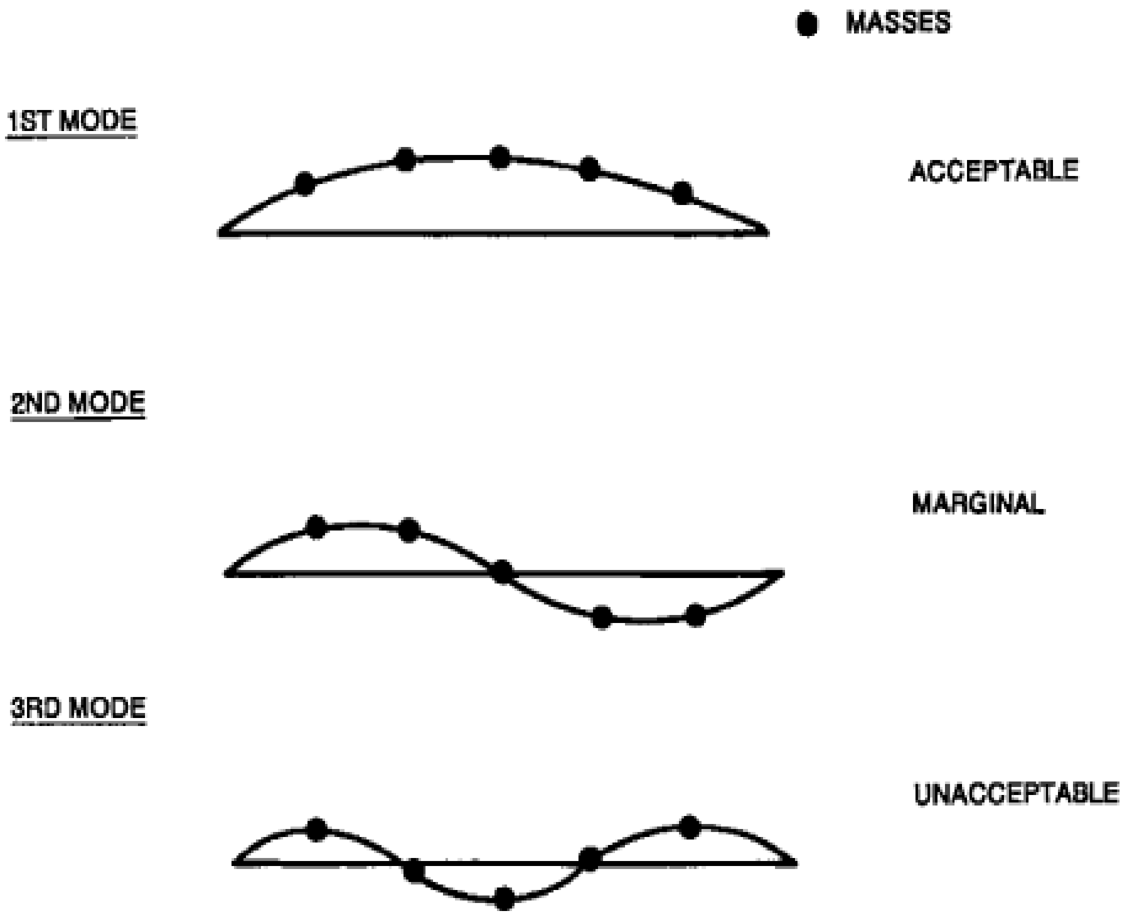


Figure 3/2-6 Mass Distribution Required for Accurate Determination of Natural Frequencies

Once the frequency range of interest is decided upon, the mode shape for the highest frequency in this range needs to be estimated. This will indicate the number of dynamic DOFs required to yield accurate results. Predicting a mode shape in advance is usually very difficult unless the structure is relatively simple. Therefore, it may be necessary to follow an iterative process in which the mass distribution is refined at each iteration.

Certain algorithms require any problem size reduction to be undertaken by the analyst. In this case, the analyst selects the number of dynamic DOFs to be used in the analysis. The selection of the dynamic dot's to be used in the dynamic analysis requires considerable skill except for the simplest structures. The selection of dynamic DOFs can be automated. The principle underlying the Guyan reduction process provides a guide on how this should be done if done manually. The most important dynamic DOFs are those that have the largest mass-to-stiffness ratio. This is because such masses are responsible for most of the vibration energy at lower modes. The concept underlying the selection of dynamic DOFs is shown in Figure 3/2-7. Viewing a plot of the mode shapes will allow an assessment to be made of the reasonableness of the selection of dynamic DOFs.

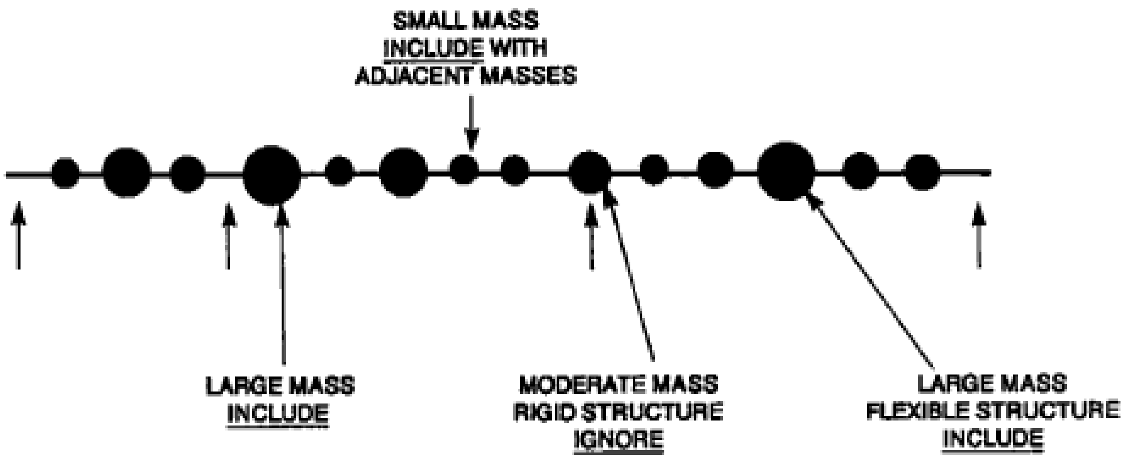


Figure 3/2-7 Selection of Dynamic DOFs

For most structural dynamics problems, translational masses are sufficient to define the problem. However, when components and equipment with large dimensions are being modeled, it is prudent to model their rotational inertia. If a single mass element is being used to model the component, then three rotational inertias should be input in addition to translational mass data. Alternatively, several masses can be input that approximately simulates the mass distribution. The procedures are summarized in Figure 3-2.6.

A summary of guidelines to be followed in selected in dynamic DOFs is given below:

1. The number of dynamic DOFs should be at least three times the highest mode required. For example, if thirty modes are required, at least ninety dynamic degrees of freedom should be specified.
2. Dynamic DOFs should be located in regions where the highest modal deflections are anticipated.
3. Dynamic DOFs should be located where the highest mass-to-stiffness ratios occur on the structure.
4. If a dynamic response computation is to be eventually performed, dynamic DOFs should be located at points where forces are to be applied.
5. For slender structures, such as masts, only translation dynamic DOFs need to be selected.
6. For stiffened plate structures, only dynamic DOFs at right angles to the plane of the structure need to be selected.
7. Enough dynamic DOFs should be retained such that the modeled mass does not differ from the actual mass by more than 10%.

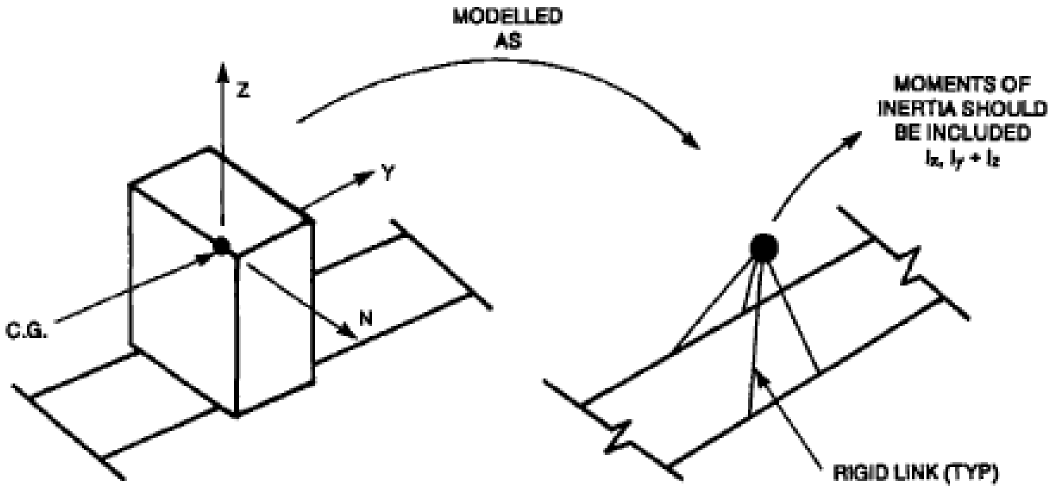


Figure 3/2-8 Modeling Rotational Inertia

3/2.3 Material Properties

The most common materials used in the construction of ships are metallic. Non-metallic materials are also used, for example, glass-fiber or carbon-fiber-reinforced plastic (GRFP or CRFP) composites, and other kinds of cellular materials (e.g., wood). The scope of these guidelines is confined to metallic materials, mainly working in the elasto-plastic range. Note that the properties of certain materials are loading rate-dependent, and this may need to be considered.

While Poisson's ratio for steel is not very sensitive to the increase of temperature, Young's modulus and yield strength do reduce significantly when the temperature starts to get above a few hundred degrees Centigrade [22]. Nuclear air blast explosions can cause thermal effects of sufficient magnitude to influence the values of Young's modulus and yield strength. High strain rates may increase the value of the yield and ultimate strengths of certain materials. For example, steel and aluminum show a strong rate-dependency, but magnesium can be regarded as rate-independent. However, these strain rates have to be very high to have a significant effect. Examples, where structures may be subject to high strain rates, include structural response to collision, armor-piercing, underwater explosions, and nuclear air blast. As a general guide, the effects of strain rate should be considered for strain rates over 0.1 s^{-1}.

3/2.3.1 Metallic Materials

The common inputs for metallic materials are yield strength, Young’s modulus, Poisson’s ratio, shear modulus, and density. For advanced analyses, more detailed material properties are required. Example metallic material properties are listed in Table 3/2-2.

Table 3/2-2 Example Metallic Material Properties

Material (reference grade)	**Yield strength [MPa]**	**Young’s modulus [GPa]**	**Poisson’s ratio [-]**	**Density [kg/m3]**	**Heat Capacity [J/(kg°C)]**	**Thermal conductivity [W/m·K]**
Steel (S235)	235	210	0.3	7.8*103	0.66	25
Aluminum (6061-T4)	145	70	0.33	2.7*103	0.90	154
Magnesium (AM60)	130	45	0.35	1.8*103	1.00	61

Note that these Table 3/2-2 values are reference values and strongly dependent on the grades as well as alloy compositions. It is highly recommended to get verified material properties from specifications or material vendors/suppliers for each FEA analysis.

3/2.3.2 Composite Materials

Modeling the behavior of composite materials is more complex than modeling isotropic materials such as steel. Composite materials are anisotropic and cannot always be regarded as a continuum. In cases where the global response is of interest, it may be reasonable to model composite materials using an anisotropic continuum model. More local analysis requires explicit modeling of the material.

Most general-purpose FEA software systems include the capability to compute the elastic properties of composite materials. This is done by defining the individual layers that comprise the composite. Alternatively, it is often possible to input the constitutive matrices that define the relationship between generalized forces and moments to generalized strains and curvatures.

The failure modes of composite materials are also more complex than those that typically apply to isotropic materials. To check the adequacy of a structure made from composite materials, it is necessary to define the failure criteria that must be applied. Whereas with isotropic materials, a single failure criterion (e.g., yield stress) is typically applied, with composite materials failure criteria are generally different for different directions and can be applied to strains, stresses, and combinations of stresses and strains.

There are other modeling issues that are particular to composite materials. Depending on the design of the composite, it may not be possible to apply symmetry conditions even when the loading and the overall geometry are symmetrical about one or more axes.

3/2.3.3 Stress and Strain Definitions

In general, the stress-strain relationship can be obtained by carrying out standard tests (e.g., uniaxial tensile test). For example, ASTM-E8 [23] is one of the most widely adopted standard test methods to obtain the stress-strain relationship of metallic materials. In the standard, the geometric configurations of standard coupons (e.g., cross-section area – A_0, gauge length – L), as well as the specifications of testing rigs, are well defined, aiming to obtain a set of reliable data.

The stress and strain from standard tests are *engineering* stress (σ_{ENG}) and *engineering* strain (ε_{ENG}), respectively:

$$\sigma_{ENG} = F/A_0$$

$$\varepsilon_{ENG} = \Delta L/L$$

Where F, A_0, ΔL, and L are measured force from load cells, initial cross-section area, length increment, and initial gauge length. A typical stress-strain curve is shown in Figure 3/2-9.

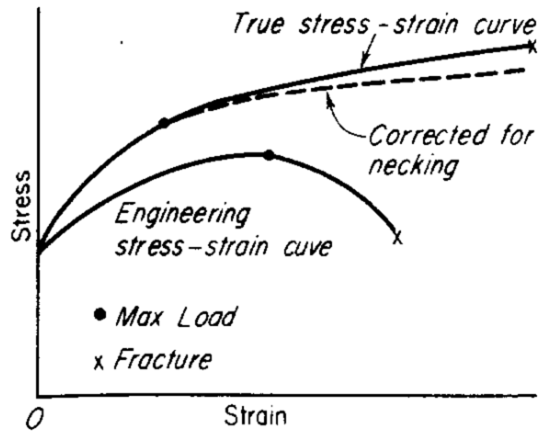


Figure 3/2-9 Comparison of engineering and true stress and strain curves. Reproduced from Fig.8.4 in Ref. [24]

The FEA software input is often required to be *true* stress (σ_T) and *true* strain (ε_T) calculated from updated geometry:

$$\sigma_T = F/A = \sigma_{ENG}(1 + \varepsilon_{ENG})$$

$$\varepsilon_T = \ln(\varepsilon_{ENG} + 1)$$

where A is the instantaneous cross-section area. From the engineering stress and engineering strain shown in Figure 3/2-9, the true stress and true strain are generated. It is clearly seen that there are pronounced differences between the two sets of curves. Note that nowadays, several commercial codes have the feature to automatically convert the engineering stress and engineering strain input to true stress and true strain, but additional care in the FEA setup is needed for end-users.

It should be noted that both engineering stress-strain and true stress-strain are widely mentioned and presented with definitions. However, what happens behind the scene in the solver – "solver Blackbox" involving other stress and strain definitions, e.g., Cauchy stress, 1st Piola Kirchhoff stress, 2nd Piola Kirchhoff stress, Green-Lagrange strain, or Euler-Almansi strain [25]. These stress and strain definitions are adopted for solving different problems, such as small deformation/strain (Green strain) and finite/large deformation/strain (Almansi strain). In addition, the limitations in formulations on the use of elements should be recorded and evaluated.

3/2.3.4 Material Models Stress-Strain Relationships

For metallic materials, the time-independent elasto-plastic material models are generally adopted. The following are several key components included in the material model:

3/2.3.4.1 Yielding Criteria

The stress-strain state when the material yields and generate plastic strain. von Mises criterion [24] is commonly used for metals. Some may also use the Tresca criterion [24], but extra care is needed in these cases.

3/2.3.4.2 Stress-Strain Curves

There are generally two types of stress-strain curves: discrete curves and continuum curves. Discrete curves are widely used in solving general engineering problems, which include (a) rigid ideal plastic, (b) ideal plastic, (c) bi-linear, and (d) multi-linear curves (see Figure 3/2-10). Continuum curves are used for solving special engineering problems (e.g., fracture, collision, and explosive). Several widely used continuum stress-strain formulations are Ramberg-Osgood, Power-law, and

Johnson-Cook [26,27].

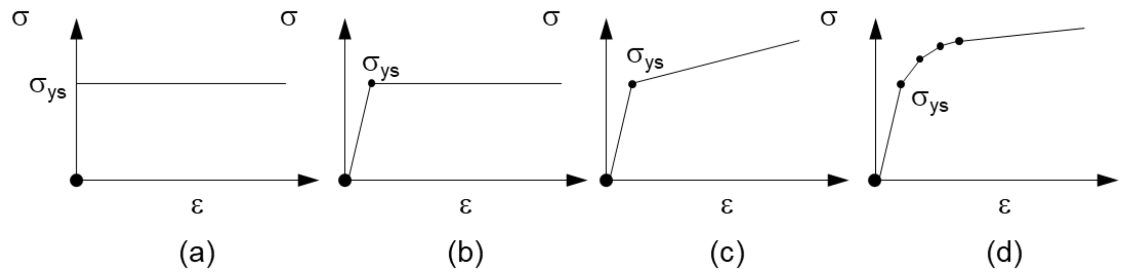


Figure 3/2-10 Discrete Stress-Strain Curves

3/2.3.4.3 Hardening Laws

Three common types of hardening laws are:

1. Isotropic hardening model (yield surface gradually expanded after each loading-reverse loading cycle)
2. Kinematic hardening model (yield surface translated after each loading-reverse loading cycle)
3. A combination of both isotropic and kinematic hardening models.

3/2.3.4.4 Flow Rules

In plasticity theory, deformation theory (i.e., equivalent to nonlinear elastic) and incremental plastic theory (J2 theory) are the two rules to define the response of plastic strain with the variation of stress. The J2 theory is commonly adopted in the FEA simulation. Note that in solving certain fracture mechanics related problems, deformation theory is still used to calculate the crack driving force (J-integral).

3/2.3.4.5 Rate and Temperature Effects

The material properties (such as yield strength and ultimate tensile strength) of some metallic materials depend on strain rate $\dot{\varepsilon}$ (at $\dot{\varepsilon}$ > 0.1 s^{-1}), e.g., steel and aluminum. A typical strain rate effect includes an increase of yield strength as well as the raise of the stress-strain curve after yielding and maybe a reduction of ductility (ultimate tensile strength and strain).

The Cowper-Symonds and Johnson-Cook models [28] are the models used frequently to simulate the strain rate effects:

- Cowper-Symonds model

$$\sigma = \sigma_{static}\left(1 + \left(\frac{\dot{\varepsilon}}{C}\right)^{\frac{1}{p}}\right)$$

  where *C* and *p* are user-defined input constants. This model is essentially based on a scaling term to predict the dynamic stress.

- Johnson-Cook model

$$\sigma = \left(A + B\left(\varepsilon_p\right)^n\right)\left(1 + Cln(\dot{\varepsilon}^*\,)\right)(1 - (T^*)^m)$$

  where A, B, C, n, and m are user-defined input constants; ε_p is the equivalent plastic strain,

$\dot{\varepsilon}^*$ (= $\dot{\varepsilon}_p$ /$\dot{\varepsilon}_0$) is the normalized strain rate based on $\dot{\varepsilon}_0 = 1s^{-1}$, and T^* (= $(T - T_{ref})^*$) reflects the temperature impact.

3/2.3.4.6 Failure and Damage Models

In general, a "failure" refers to either material failure or structural failure (e.g., buckling). The explanations of this sub-section will be restricted to material failure. A metallic material failure may include the following forms:

- Damage (e.g., pores/voids)
- Microcracks (e.g., coalescence)
- Fracture (e.g., separation)
- Reaching a high-stress level over-yielding (e.g., > ε_{UTS})

Failures can be predicted from the stress-strain state from the following hypotheses (but not limited to):

- Maximum principal stress hypothesis – In this hypothesis, failure occurs when the principal stress reaches a critical value.
- Maximum principal strain hypothesis – In this hypothesis, failure occurs when the principal strain reaches a critical value.
- Maximum strain energy hypothesis - In this hypothesis, failure occurs when the total deformation energy per unit volume reaches a critical value.

Failures can also be predicted from ductile damage models, such as the Lemaitre damage model [29,30] and the Gurson damage model [31].

3/2.3.4.7 Material Property Calibration

The basis of selecting a material model is to represent the material behavior (linear and/or nonlinear) under certain loading conditions. The material model should be carefully selected after adequately calibrating against experimental or empirical data such that the numerical output can represent the structural behavior accurately.

The stress-strain relationship is one such material property that requires calibration against experimental results. The below Figure 3/2-11 shows the difference in the stress-strain relationship between a non-calibrated FEA test and an experimental test of an ASTM E8 dog bone coupon. Calibration of the FE model parameters will result in better agreement between the model and physical trials, thus demonstrating that the model is capable of simulating reality.

Recently, several analytical methods have been adopted to calibrate the stress-strain relationship and to replace the traditional experiment requirement. These include the least-square curve fitting, parametric optimization, and design of experiments (DoE) [32]. These methods can reduce the calibration time and increase the calibration quality.

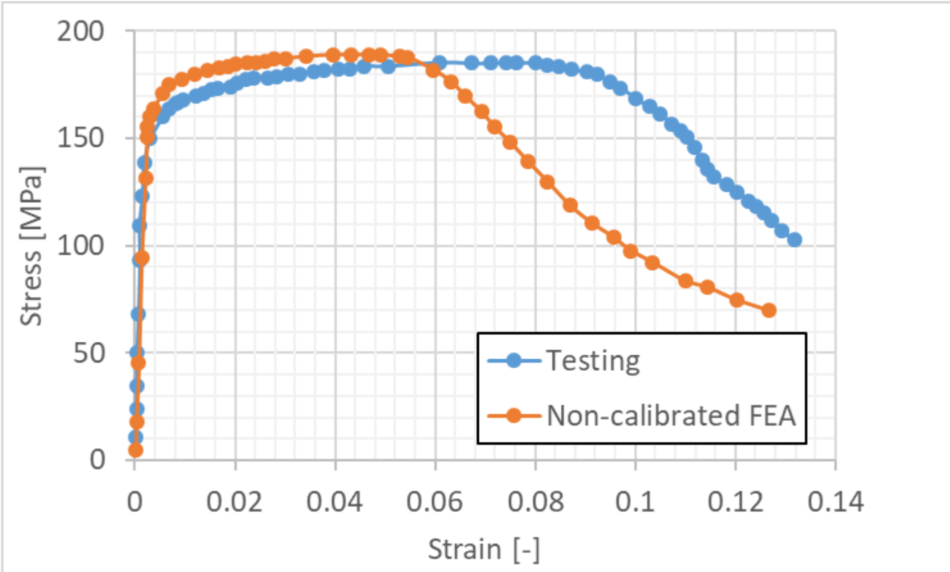


Figure 3/2-11 Stress-Strain Comparison of Non-Calibrated FEA Results with Experimental Results

3/2.3.5 Neuber Correction Method

The Neuber correction method has become popular in carrying out a fatigue analysis. The method is one of the scaling methods that can capture the nonlinear stress-strain relationship after the yielding point based on the linear elastic analysis. This method includes three steps:

1. Calculate the stress and strain values in the hot area(s), i.e., high stress or strain, from the linear elastic analysis;
2. Determine the strain energy of interest areas (Area-1 in Figure 3/2-12);
3. Estimate the stress and strain pair from the nonlinear stress-strain curve that has the same strain energy (Area-2 in Figure 3/2-12), i.e., Area-1 = Area-2.

The Neuber correction is depicted in Figure 3/2-12. This is a straightforward procedure to apply the correction, as long as the stress-strain curve is provided. More details and free calculation software are available in [33–39]. It should be noted that the Neuber correction method is only valid under relatively low loading levels. A rule of thumb when using this method in a durability analysis is that it is applicable if the load level is less than 1/3 of the limit load (full yield level). One of the advantages of adopting this method is a significant amount of calculation time can be saved in the analysis.

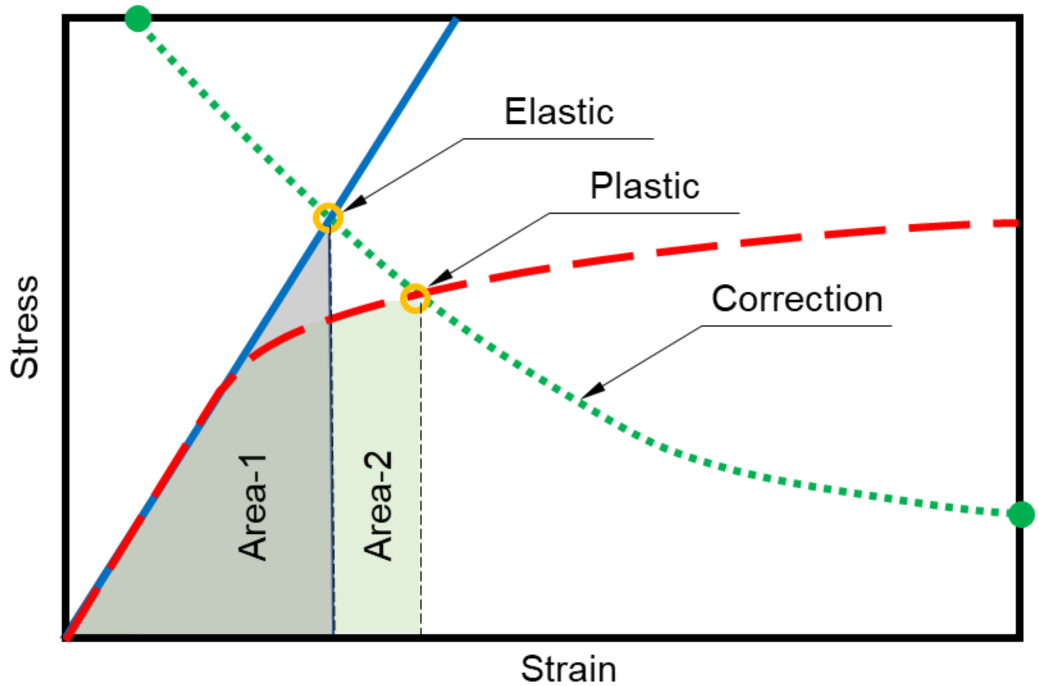


Figure 3/2-12 Neuber Correction from the Elastic Stress State to the Elasto-Plastic Stress State

3/2.4 Loads and Boundary Conditions

All loads that need to be considered should be described. The description should include a brief discussion of the accuracy level of the load.

Loads (compiled by Giannotti & Associates, 1984) typically applied in ship structural analyses include the following:

1. **Hull Girder Loads** consist of wave-induced and still water loads on the hull girder. This load should be considered for longitudinal structure in the main hull, and for the interaction of a long continuous deckhouse (superstructure).
2. **Hydrostatic Loads** are pressure loads due to fluids. The pressure could be either internal or external. Examples of hydrostatic loads are external pressure of the bottom and sides of shell plating, and internal pressure in tanks and on watertight bulkheads.
3. **Hydrodynamic Loads** consist of liquid sloshing in tanks, shipping of green water on the weather deck and impacting on the house front, and wave slap on all exposed structure and equipment above the waterline, etc.
4. **Live Loads** consist of uniform deck loading, concentrated loads such as forklift or aircraft landing and parking loads, support reactions from stanchions and equipment, cargo container reactions, etc.
5. **Dead Loads** consist of the weight of the structure.
6. **Ship Motion loads** consist of inertial forces that act on the entire ship and are important design loads for masts and topside foundations, such as topside cargo attachments. The effect of ship motion loads on the hull girder is to produce vertical and horizontal bending moments and torsion. A lengthy analysis is required to determine these values for a particular ship and service characteristics.

7. **Shock Loads** consist of displacements, velocities, and accelerations in all three directions. This load is important for naval ships in the design of vital equipment and their foundations, and ship structure in the vicinity of these foundations.
8. **Missile and Gun Blast Loads** consist of transient pressure and thermal load for all structures within the blast impingement area; usually, a static equivalent pressure is used.
9. **Nuclear Overpressure** consists of the transient traveling pressure waves from a nearby nuclear air blast; this is an important consideration in the analysis of deckhouses (superstructures).
10. **Vibratory Loads** consists of cyclic loading from rotating machinery, especially from propellers, low-frequency full girder response from slamming and springing can also be significant.
11. **Thermal Loads** are caused by heat inputs
    a. from solar radiation
    b. exhaust impingement from stack gases
    c. operation of machinery, especially combustion engines (important to deckhouses and exhaust ducting), diesel generator foundations, and condenser foundations
12. **Environment loads** consist of wind, snow, and ice loads.
13. **Impact loads** consist of displacement or velocity in all three directions.

A description of the boundary conditions applied to the model, and the reasons for the approach adopted should be described. The description should include, but not be limited to, a discussion of:

- model symmetry, antisymmetry, and axisymmetry
- material property changes at the boundary
- stiffness changes at the boundary
- assessment of influence on results of assumptions made concerning boundary conditions

3/2.5 Impact/Collision Analysis

An impact or collision problem is a highly nonlinear problem, which is generally simulated by using the explicit integration method (see 3/3.6.2). The impact problem is generally characterized by kinetic energy. The kinetic energy is governed by the impact speed and the mass of the ship (including hydrodynamic added mass).

If the ship structure collides with a relatively rigid object, kinetic energy is dissipated as strain energy in the ship structure. If the ship structure collides with a deformable object (e.g., ship to ship, ship to offshore structures, ship to ice, etc.) then the kinetic energy will be dissipated as strain energy into both the ship and the object, and there may be some remaining kinetic energy if both the ship and object are floating.

The impact problem involves large deformation and large plastic strain, and, therefore, significant structural damage may occur. The strain energy dissipation should be estimated from the force-displacement relationship. The structural damage can be captured by using different techniques. These include element erosions, cohesive elements, or other element formulations associated with material models with damage criteria.

In general, there are three different levels of impact analysis [40].

1. Local cross-section

2. Component and sub-structure
3. Total system

The analyst needs to decide which level of engineering model should be adopted for the study. Note that the interaction between three levels of energy dissipation should be considered and verified.

It should be noted that the elastic energy may contribute significantly on a global level, which may be driven by total mass, impact speed, structural flexibility, material properties, and other factors.

In the FEA software, there are generally two means to define the impact speed: velocity and displacement. Assigning a given velocity is a widely used method for impact analysis. The use of displacement, however, is also convenient for sub-structure and local cross-section analyses (a special type of submodeling analysis). This works especially well using the initial global model displacement outputs at the boundary areas of submodels.

3/2.6 Fatigue and Fracture Analysis

3/2.6.1 Fatigue Analysis

About 90% of service failures are caused by fatigue that leads to a fracture, and sometimes the failure locations predicted from the static or dynamic analysis are different from the test and field observations [41,42]. One of the reasons for this is that the failure is caused by fatigue damage.

Fatigue is defined as progressive and localized damage that occurs when a structure is subjected to cyclic loading [43]. There is a need to keep in mind that a fatigue failure is a probabilistic event, and even a good structural design may not be fatigue-free. A reasonable structural design against fatigue should involve both analysis and testing validation.

The fatigue analysis needs three key inputs that the analyst must review during the engineering model check: the material properties, the geometric profiles, and the loading history. Figure 3/2-13 shows these within a schematic representation of the standard procedure for fatigue analysis. The fatigue related material properties include the stress-cycle (SN) curves, strain-cycle (EN) curves, and cyclic stress-strain curves. Note that the SN or EN curve should be consistent with the cyclic loading conditions, such as R ratios, $R = \sigma_{min}/\sigma_{max}$, where σ_{min} and σ_{max} are maximum and minimum stresses (see Figure 3/2-14). It is the most common to test at an R ratio of 0.1. The loading history includes information such as the amplitude-frequency and loading path.

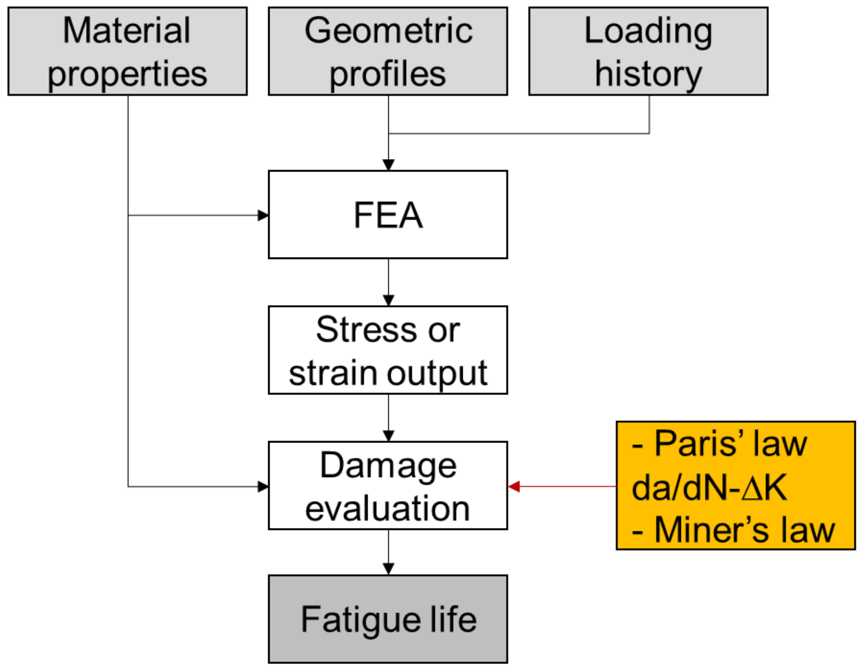


Figure 3/2-13 Fatigue Analysis Procedures

Note that in Figure 3/2-13, some factors that may affect fatigue life are not presented. These other factors include, but are not limited to, surface finish and treatment conditions, notch presence, and residual stress presence. The analyst needs to collect this information by reviewing the engineering model.

3/2.6.2 Basics of Fatigue Mechanics

In general, there are four different types of fatigue analysis:

1. **Stress life fatigue analysis:** This is also known as the high cycle fatigue (HCF). The data of stress vs. cycle numbers (SN curve) is used as FE input.
2. **Strain life fatigue analysis:** This is also known as low cycle fatigue (LCF). The data of strain vs. cycle numbers (EN curve) is used as FE input.
3. **Fracture mechanics fatigue analysis:** Either linear and nonlinear fracture mechanics theories are used to estimate the crack growth rate as a function of crack driving forces based on data from stress or strain life calculations.
4. **Vibration fatigue analysis:** This analysis can capture the resonance effect. Transient cyclical loadings and frequency-domain power spectral density functions are the possible FE input.

3/2.6.2.1 Cyclic Loading Definition

The cyclic loading can be characterized by using different methods such as a unit cycle associated with repeating numbers, a whole cyclic history, or a simplified loading history from Rainflow counts [44,45]. A unit cycle is one representative segment of the whole cyclic load history that can be regenerated by repeating the unit cycle, see Figure 3/2-14.

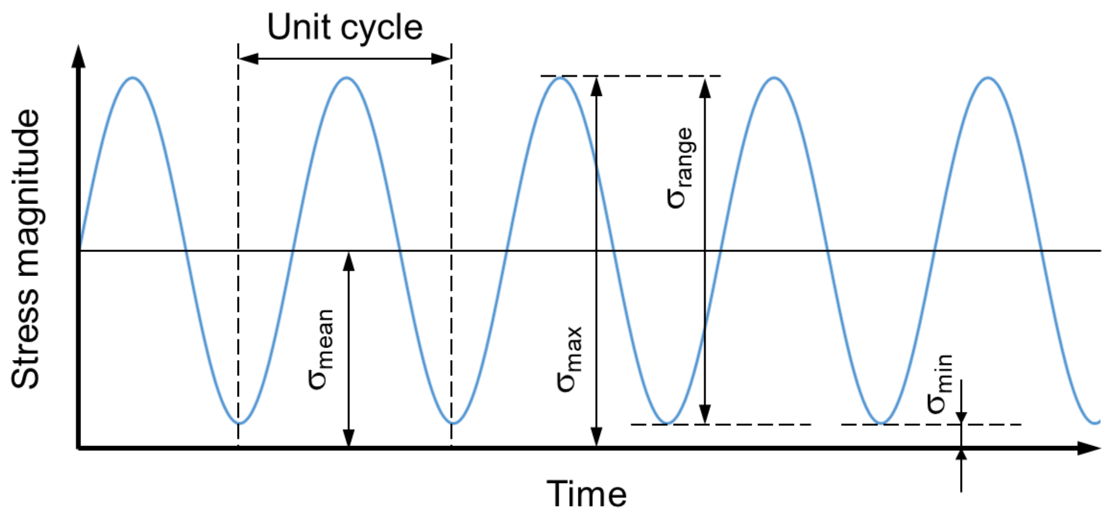


Figure 3/2-14 Cyclic Load Parameters

The unit cycle can be defined by the following parameters:

1. Mean stress, σ_{mean}

$$\sigma_{mean} = \frac{\sigma_{max} + \sigma_{min}}{2}$$

2. Stress range, σ_{range}

$$\sigma_{range} = \sigma_{max} - \sigma_{min}$$

3. Stress amplitude, $\sigma_{amplitude}$

$$\sigma_{amplitude} = \frac{\sigma_{max} - \sigma_{min}}{2}$$

4. Stress ratio, R

$$R = \frac{\sigma_{min}}{\sigma_{max}}$$

Since the stress amplitude is half of the stress range, the unit cycle load can be explicitly defined just by either the mean stress and the stress range, or the mean stress and the stress amplitude.

3/2.6.2.2 Fracture Mechanics Based Fatigue Analysis

The fracture mechanics-based fatigue analysis generally uses Paris' law to determine the crack growth rate from the stress intensity factor. A typical relationship between the crack growth rate and stress intensity range is shown in Figure 3/2-15.

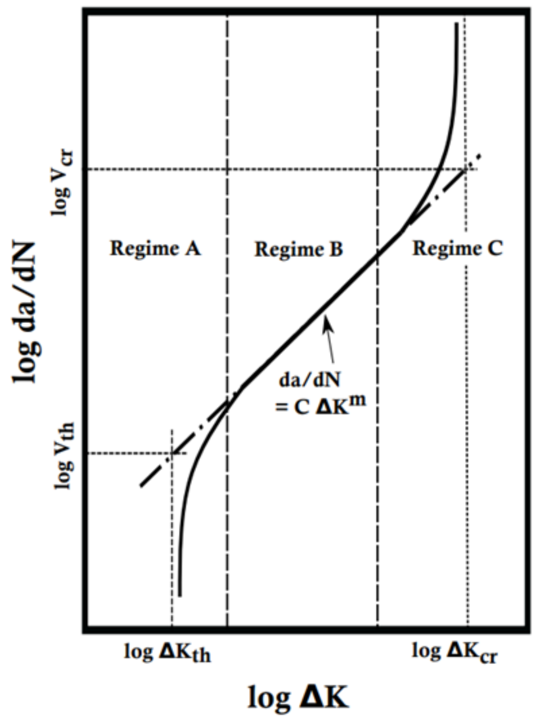


Figure 3/2-15 Crack Growth Rate vs. Stress Intensity Range

Paris' law is defined as follows:

$$\frac{da}{dN} = C(\Delta K)^m$$

where a is the crack length, N is the number of cycles, da/dN is the fatigue crack growth rate, and ΔK is the stress intensity factor range. The ΔK is the stress intensity range and calculated from the following equation.

$$\Delta K = K_{max} - K_{min}$$

where K_{max} and K_{min} are the maximum and minimum stress intensity factor for each load cycle, respectively. The C and m terms are non-dimensional curve fitting factors to fit the power-law relationship curve to the experimental test results. C and m are generally determined by fitting regime B in the figure.

3/2.6.3 Fracture Analysis

In the presence of geometric features with sharp radii such as notches, flaws, and /or cracks, the conventional approach of continuum mechanics would give erroneous answers. The fracture mechanics-based analysis is carried out, which can describe the behavior of solids and structures with geometrical discontinuity in a meso- or macro-scale. If required, a damage mechanics-based analysis can capture the behavior in the meso- or micro-scale. Figure 3/2-16 shows the differences among the classical continuum mechanics, fracture mechanics, and damage mechanics. In the figure, their applicable conditions are schematically shown:

1. In the continuum mechanics problem, the material is homogenous.
2. In the fracture mechanics, the material is homogenous but with cracks or flaws.
3. In the damage mechanics, the material is homogenous, but the effects of pores, voids, and cracks are considered.

Note that the current FEA practice mainly focuses on the fracture mechanics, and a limit amount of damage mechanics related to material models is discussed

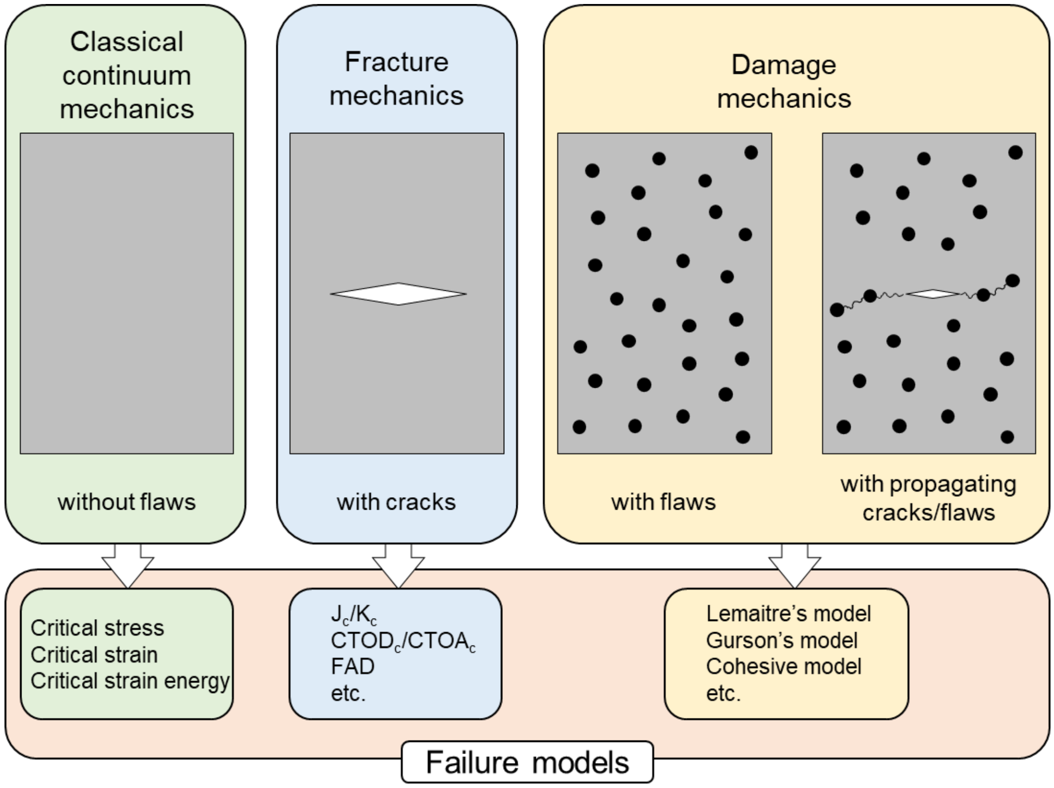


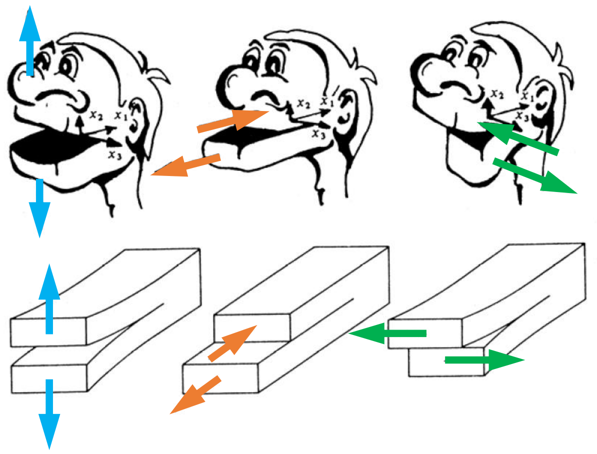
Figure 3/2-16 Differentiation of Continuum, Fracture, and Damage Mechanics

To perform a fracture analysis, an analyst needs to have a thorough understanding of basic local crack/flaw geometric profiles, fracture mechanics material models, how to apply loads, and other related information. Also, the fracture and damage parameters available have limits and are dependent on conditions such as test temperature and test duration. An analysis can only get an accurate evaluation with proper inputs. For example, if the J-integral (J) is of interest, small-deformation and small-strain formulations are sufficient to describe the crack-tip field. However, if the crack-tip opening displacement (CTOD) or crack-tip opening angle (CTOA) is of interest, the large-deformation and large-strain formulations have to be adopted to capture the crack blunting. Note the parameter J is a global fracture mechanics parameter, and the parameter CTOD/CTOA is a local fracture mechanics parameter.

It should be noted that the fracture analysis and fracture mechanics-based fatigue analysis (presented in the above two sections) are not widely used for designing the ship structures. These analysis types are more relavent to structural integrity assessment or maintenance. For illustation purposes, the fracture and fatigue example in Section 5/2 presents the approach of how to use these methods to find the root cause of a significant crack observed in the bottom tank. It may also help estimate the remaining service life when repairs have been undertaken.

3/2.6.4 Basics of Fracture Mechanics

In fracture mechanics, there are three deformed modes, and each type is shown in Figure 3/2-17.



(a) Mode-I (b) Mode-II (c) Mode-III

Figure 3/2-17 Basic Modes of Crack Extension: (a) Mode-I: Opening mode; (b) Mode-II: in-plane shear mode; (c) Mode-III: out-of-plane shear mode.

- Mode-I is the opening mode
- Mode-II is the in-plane shear mode (also known as a sliding mode)
- Mode-III is the out-of-plane shear mode (also known as tearing mode)

Fracture mechanics can be divided into linear elastic fracture mechanics (LEFM) and elasto-plastic fracture mechanics (EPFM). In general, LEFM is used for solving problems with brittle/elastic materials, and EPFM theories are used for solving problems with ductile materials. If the loading level is low enough, LEFM can also provide a reasonable approximation of physical reality for ductile materials.

3/2.6.4.1 Linear Elastic Fracture Mechanics

In LEFM, the fracture is under K-control [46], and the crack-tip field can be accurately described by using two fracture mechanics parameters,: K and $\mathcal{G}$. The parameter K is the stress intensity factor that is developed from the first term in the linear elastic crack-tip stress equation (Eq. 2.41 in [47]). The stress intensity factor characterizes the stress, strain, and displacement field in the vicinity of the crack tip. The parameter $\mathcal{G}$ is the Griffith energy release rate proposed by Irwin [48]. The energy release rate quantifies the net change in potential energy caused by the increase of crack extensions. Due to their natures, K is a local parameter, and $\mathcal{G}$ is a global parameter.

There are mathematical relationships between K and $\mathcal{G}$ that depend on whether the problems are linear elastic or not, and on what the mode of crack extension is. For problems that are linear elastic materials and are of the Mode I type, K and $\mathcal{G}$ are related by the following relationship:

1. Plane strain condition: $\mathcal{G} = K_I^2/E'$, *where* $E' = E/(1-v^2)$
2. Plane stress condition: $\mathcal{G} = K_I^2/E$

where E is the Young's Modulus, v is the Poisson's ratio, and K_I is the Mode-I (opening) stress intensity factor. The relationship between K and $\mathcal{G}$ in other failure modes (e.g., Mode-II or Mode-III) can be found in the literature [47].

The analytical and/or empirical solutions of K have been widely investigated in the past several decades, and the solutions have been reported in literature (e.g. Tada et al. [49]) and fitness-for-service (FFS) standards (e.g. BS-7910 [12], API-579 [13], ASME Boiler & Pressure Vessel Codes

[50]).

Note that the parameter K can also be used for predicting the fatigue crack growth using Paris's law mentioned in the previous section (2.8.2).

3/2.6.4.2 Elasto-Plastic Fracture Mechanics

The EPFM can be divided into two different categories based on the yielding conditions in the vicinity of the crack tip:

- small-scale yielding (SSY)
- large-scale yielding (LSY).

In some references, the full-scale yielding (FSY) condition is discussed separately from LSY. Here, FSY is regarded as a subset of LSY. SSY, LSY, and FSY also refer to the global loading levels that develop the crack tip with limit plastic zones, pronounced plastic zones, and fully plastic zones, respectively. The details of these definitions are discussed in Ref. [47]. In general, the fracture in both SSY and LSY problems of EPFM is controlled by J-integral (J), i.e., J-control. However, it should be noted that just as there are limits to K-controlled LEFM, J-controlled EPFM may become suspect when excessive plasticity or significant crack growth is presented (see Figure 3/2-18(c)).

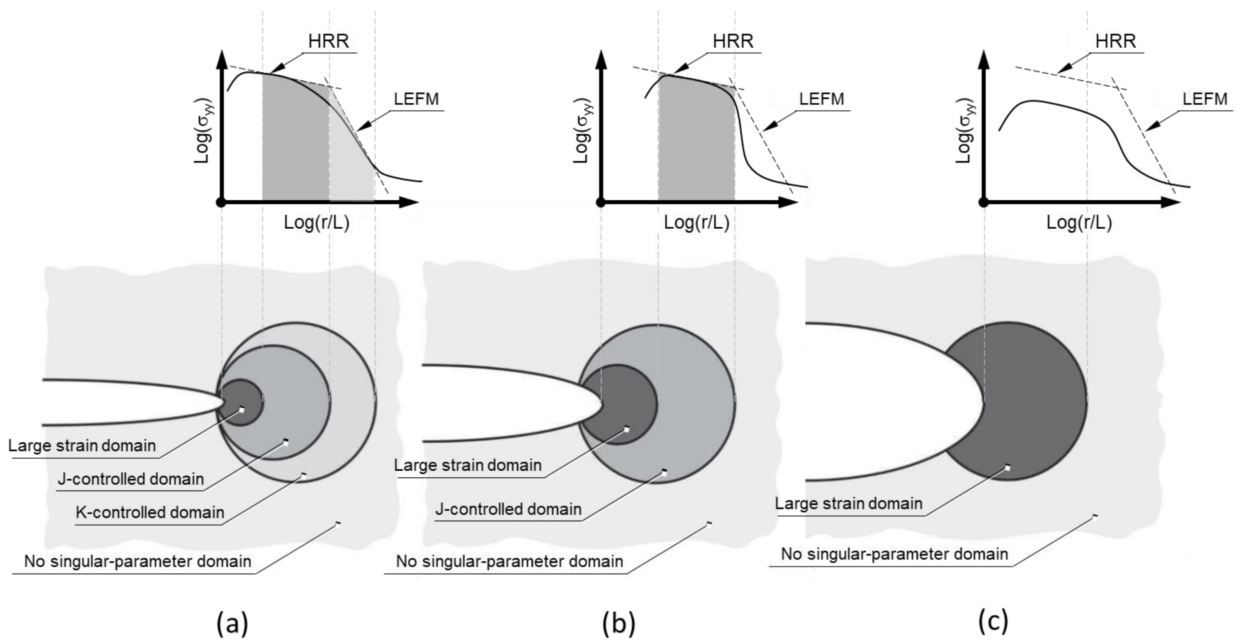


Figure 3/2-18 Size Effect of Plastic Zone in the Crack Tip Stress Field : (a) SSY (b) LSY with an intermediate amount of plasticity (c) LSY with the pronounced amount of plasticity

Figure 3/2-18 schematically presents the size effect of the plastic zone in the crack tip stress field. In SSY and LSY with an intermediate amount of plasticity (Figure 3/2-18 (a) and (b)), the crack tip stress field is under J-control and can be described using HRR singularity. The HRR singularity defines the stress variations in the plastic zone of the crack tip, which is proposed by Hutchinson, Rice, and Rosengren [51,52].

Figure 3/2-18 (c) schematically shows a pronounced crack blunting of crack surfaces and plastic zones under large scale yield. The one-parameter fracture mechanics theory breaks down in the presence of such extensive plasticity, and the fracture toughness may become dependent on the size and geometry of the test specimen. Note that there are strict conditions applied in measuring K and J resistant curves per standards (e.g. ASTM E1820 [53], ASTM E399 [54], ASTM E1290 [55], BS 7448 [56], and ISO 12737 [57]), aiming to minimize those impacts.

In the past decade, the two or multiple parameters fracture mechanics were proposed to describe the crack tip stress field with extensive plasticity, e.g., T-stress (Williams [58]), J-Q (O'Dowd and Shih [59,60]), and J-A_2 (Chao et al. [61,62]). Table 3/2-3 summarizes the fracture mechanics parameters being widely used in industry and adopted in FFS handbooks and failure access diagram (FAD) methods (e.g., BS 7910-2015 [12]). Note that the multiple parameter fracture mechanics recently presented in the scientific publications need further validations. Therefore, it is recommended to use either J-T or J-Q in the analysis, which has been validated by the industry (per BS7910 [12] or R6 [63]).

Table 3/2-3 Typical Fracture Mechanics Parameters

List of parameters	**LEFM**	**EPFM**	**EPFM**
	One parameter	*One parameter*	*Two or multiple parameters*
Local stress-based	K (stress intensity factor)	T (T-stress)	T-Q T-A2
Far-field energy based	G (Griffith energy release rate)	J (J-integral)	J-Q J-A2 J-Q-Tz / J-A2-Tz
Local geometric profile based	CTOD CTOA	CTOD CTOA	CTOD-Q CTOD-A2

3/2.7 Whole Ship Analysis

3/2.7.1 Introduction

The objective of a whole ship finite element analysis is to obtain a reliable description of the overall hull girder stiffness and to assess the global stresses and deformations of all primary hull members for specified load cases resulting from realistic loading conditions including the wave-induced forces and moments. Generally, the focus of the whole ship finite element analysis is not to judge local stresses due to stiffener or plate bending, but rather to assess the stiffness and strength of hull girder at the global level. A whole ship type model may be used to develop realistic boundary conditions for a local model or loading that is investigating a more localized behavior.

The whole ship analysis is generally used for design and scantling development. In certain cases, for example, commercial ships being designed under classification society rules and guidance, the class rules will specify the extent of the whole ship finite element model, or possibly an extent covering several midbody cargo holds. The class rules will also likely include guidance on key topics such as mesh density, boundary conditions, loading conditions, and evaluation of results. Other whole ship applications may be suited to using a finite element model that includes the entire ship structure, bow to stern, full beam, and hull plus superstructure. Whole ship analysis can be conducted using loading and structural design guidance or criteria from safety authorities such as class societies or from naval design authorities. Whole ship FEA is essentially a larger, more complete version of a standard FE model, and as such, solving it can be accomplished using many commercial finite element modeling and analysis codes such as Nastran, ANSYS, or MAESTRO.

Disclaimer

The authors of this section are most familiar with the capabilities and limitations of MAESTRO and have therefore used it for many illustrations. This does not constitute an endorsement of that computer code over any other. An important part of this report is justifying the particular computer code used for the finite element analysis, and any code should be acceptable if it meets those guidelines.

3/2.7.2 Whole Ship Finite Element Model Development

The modeling necessary for the whole ship structural analysis is that the structural model should provide results suitable for performing buckling, yield, fatigue and vibration assessment of the relevant parts of the vessel as required by safety authorities or owner's requirements. This is done in whole ship FEA by using a 3D model of the whole ship, supported by one or more levels of sub-models. The full-ship should be modeled, including all relevant structural elements as dictated by the requirements of the analysis from safety authorities or the design team itself. For large primary longitudinal girders and deep transverse frames, the web plate is best modeled by quadrilateral plate/shell elements and the flange plate can be modeled as beam elements or plate/shell elements. Stiffeners are often modeled by beam elements. Stiffened panels and grillages may be modeled as an assembly of plate-shell elements and beam elements. Structures not contributing to the global strength, and that has no influence on stresses in the evaluation area of the vessel may be omitted. For ships with a relatively small deckhouse, such as tankers, the deckhouse may be omitted from the model. For ships with a relatively large superstructure, such as naval ships, the decision of whether including superstructure in a finite element model depends on the purpose of the analysis, specific criteria, and the design requirements. A set of general guidelines for finite element modeling of ship hulls is given by safety and design authorities such as classification societies and naval design authorities [9,64,65]. The whole ship FE model and analysis can also be used to identify locations where local refined FE mesh models within the full-ship model should be used to provide more detailed evaluation to resolve high-stress levels or other design criteria or structural performance issues.

3/2.7.2.1 Hybrid beam vs. Offset beam

Plates are often reinforced by attached beams (stiffeners, frame, and girders) to one side of the plate (Figure 3/2-19). Beam nodes and plate nodes do not coincide but are typically separated by a distance that is small in comparison with other dimensions. Common finite element programs offer two formulations to address the problem.

1. "Hybrid" beam element (designation after Hughes [66], shown in Figure 3/2-19Figure 3/2-): In this formulation, the beam axial stiffness is determined by the cross-sectional area of the beam element, while the bending and shear stiffness is governed by the combined section of the beam and the effective part of the plate, b_e. The plate is additionally idealized by plate elements, normally taking only loads within their plate. The beam node is assumed to coincide with the plate node.
2. "Offset" or "Eccentric" beam (shown in Figure 3/2-): In this formulation, the beam nodes, which locate at the beam neutral axis, are connected to the plate nodes by rigid links. The displacements of the beam nodes can be derived from the displacements of the plate nodes. Thus the plate mid-surface becomes the reference plane of the assembled structure. Stiffness matrices of beam elements are transformed.

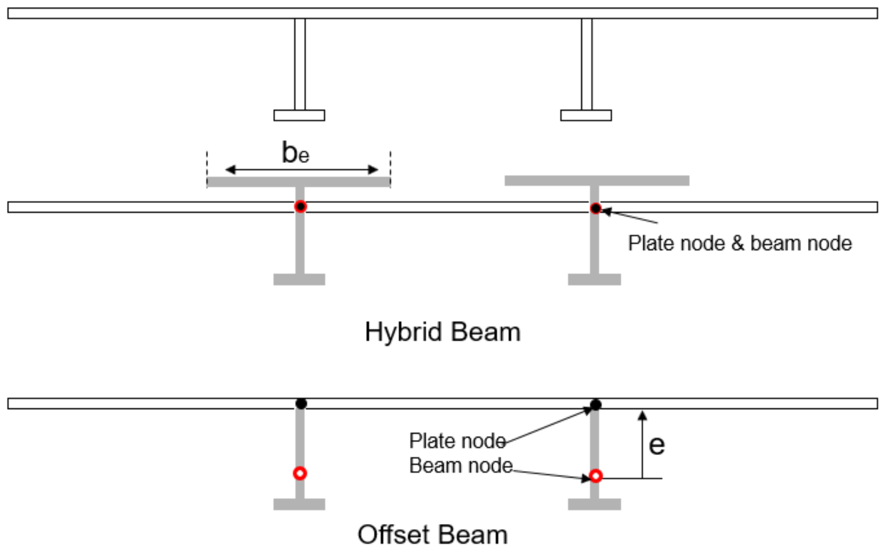


Figure 3/2-19 Modelling of Stiffened Plate

Both types of formulation are sufficient to evaluate hull girder primary stresses. However, they suffer from disadvantages when evaluating hull girder secondary stresses. For hybrid beams, the effective breadth of the plate, b_e, has to be specified by the user as data input to consider it in the sectional properties, which becomes a problem to review and validate because there are different formulations on how a plate effective breadth is defined. Furthermore, the hull girder global stiffness is slightly affected by the assumption of shifting the beam neutral axis. For ships with stiffeners below the deck, an increased moment of inertia and section modulus is obtained. For the offset beam formulation, the data input is easy, and their global behavior is better than that of the hybrid beams due to the correct location of their cross-sectional area, but their stiffness is underestimated when the assembly is subjected large shear forces [66,67]. The error may become nontrivial in a coarse mesh model where a stiffener between frames is represented by a single beam element. The error in displacements and stresses decreases with increasing mesh refinement because the rigid links at the additional nodes distribute the shear forces further between stiffener and plate [67]. In the early days of full ship model finite element analysis, the mesh density was quite coarse. The hybrid beams gave better results than offset beams. As computational power grew, the results using offset beam formulation got better with the increasing mesh density. It should be noted that although the offset beam has become industry standard in the full ship finite element analysis [65,68] because of its simplicity, the hybrid beam concept is still used in the limit state beam-column buckling analysis.

3/2.7.3 FE Model Mass Properties and Hydrostatic Loading

Ship design is a multi-disciplinary engineering process, and the whole ship analysis must be carefully integrated with the overall ship design. The full-ship FE model and analysis must reflect accurate mass properties in order to generate accurate finite element analysis results. As a floating structure, the overall ship mass properties will affect both hydrostatic and hydrodynamic loading. The whole ship FEA will require hydrostatic loading and may also require hydrodynamic loading, as discussed in Section 3/2.7.6. By the time a ship design reaches the structural finite element analysis stage, the full-ship total weight distribution at 21 stations is often available from a weight report. In addition, other weight items, such as significant equipment weights, miscellaneous or distributed lightship weights, tank liquid weights, and cargo weights, may also be defined. The structural self-weight can be automatically calculated using the finite element geometry and the associated material density. For liquid tank loads, it is recommended to apply hydrostatic/hydrodynamic pressure on the tank boundary. Note that for boundaries shared by two tanks, the pressure direction and the element normal need to be carefully synchronized. Some FEA software[69], simplify this process by grouping tank surfaces within the FE model and providing tank fill inputs in a format that enables using traditional ship loading data such as stability model input.

The applicable significant weight items are usually modeled as nodal mass loads. Many of these nodal masses are connected to the surrounding structure using rigid spline elements (RBE2/REB3) in order to connect a single node at the item's center of gravity to the expected footprint of the weight item. This permits a more accurate distribution of forces when the load case includes accelerations. Cargo masses can be distributed to a group of elements or nodes. For lightship weight, it is recommended to smear the mass throughout the model. For general finite element packages, this is often done by adjusting material densities. For ship specific finite element packages [69], the program may include built-in routines to simplify this process.

To simulate hydrostatic buoyancy in a finite element model, the elements likely in contact with seawater are grouped as "wettable" elements. The pressure side of these "wettable" elements is also set to ensure the hydrostatic pressure is applied on the appropriate side of the elements. The hydrostatic buoyancy forces are calculated by user-defined equations. Due to the nature of the finite element faceted surfaces, partially immersed shell elements, and the model simplification (such as omitting propeller and rudder), the buoyancy forces calculated using a hydrostatic tool (such as GHS or NAPA [70]) are going to be different from the forces evaluated in the finite element model. Consequently, to get equilibrium in a finite element model, the user needs to either use inertia relief or adjust heel, trim, and height of the waterplane free surface to re-balance the finite element model. Applying a waterplane free surface directly from another hydrostatic analysis tool on a finite element model usually will result in imbalanced forces and moments and produce corresponding FEA errors. An example is given in Section 5/3.8.

For ship specific finite element packages, a hydrostatic analysis is often built-in and will automatically adjust the waterplane free surface to reach equilibrium using a finite element model. The hydrostatic pressure is then computed and applied to the wetted elements after the model is in hydrostatic balance or equilibrium. Figure 3/2-20 shows a nominal frigate stillwater hydrostatic pressure distribution under the full load condition. To check whether a model is in equilibrium, it is often useful to examine whether the hull girder longitudinal bending moment distribution and the shear force distribution are in closure (or 0), as shown in Figure 3/2-21. The closure of the vertical bending moments at ends is equivalent to $\sum M_y = 0$, and the closure of the vertical shear force at ends is equivalent to $\sum F_z = 0$.

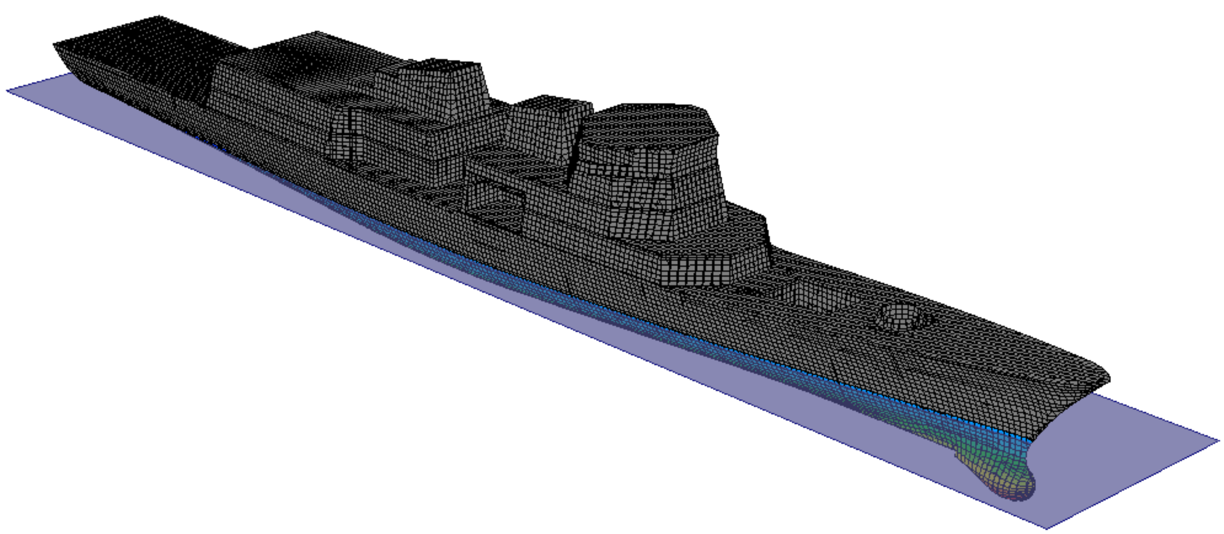
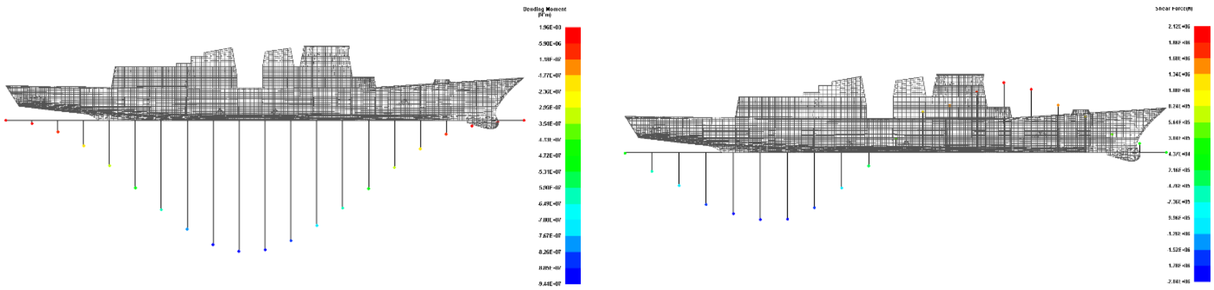


Figure 3/2-20 A Nominal Frigate Full Load Stillwater Hydrostatic Pressure Distribution



(A)Bending Moment Distribution (B)Shear Force Distribution

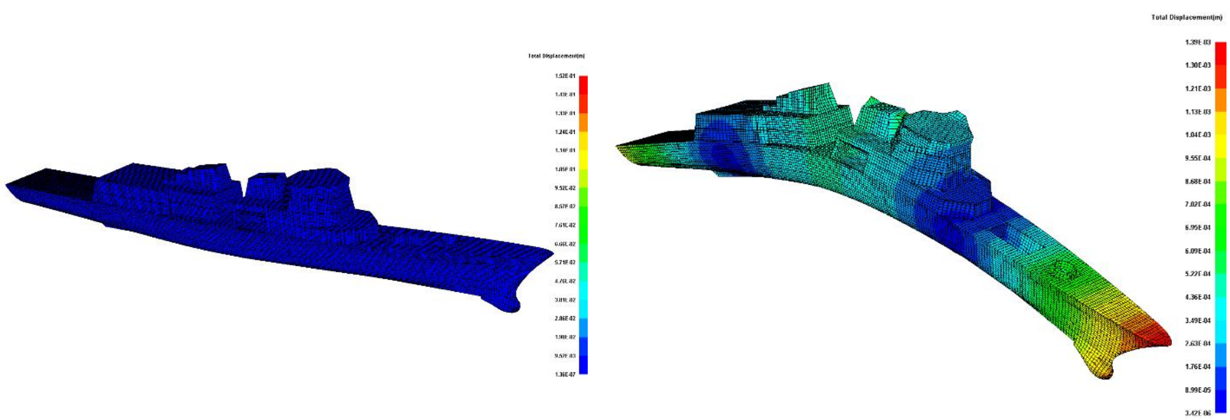
Figure 3/2-21 A Nominal Frigate Vertical Bending Moment and Shear Force Distribution

3/2.7.4 Modal Analysis Check of the FE Model

While it may not be necessary to perform checks in free-free modal analysis for a pure static model, it is recommended the check be performed if time and budgets allow. The model should be run unconstrained with 7~12 modes. The first six modes should be rigid body modes, and the frequencies are virtually zero. If the next couple of modes are not hull girder global elastic modes, it often indicates

- Some of the elements may not be connected properly
- Certain structure stiffness is not properly modeled
- Some nodal mass is too large

In the following free-free modal analysis example, the first six modes represent rigid body motions of the model. The first elastic mode (mode 7) did not display hull girder bending mode. By reviewing the internal structure, it was found that a few beam elements were not connected correctly. After fixing those modeling errors, mode 7 displayed a mode shape as expected. Figure 3/2-22 showed the eigenmode before and after fixing element connection modeling errors. Figure 3/2-23 showed the mode shape of mode 8. The local deflection at the sonar dome indicated either the stiffeners were missing in the area or large nodal masses were incorrectly placed at those nodes.



Before fixing element connection modeling errors *After fixing element connection modeling errors*

Figure 3/2-22 Mode 7 eigenmode

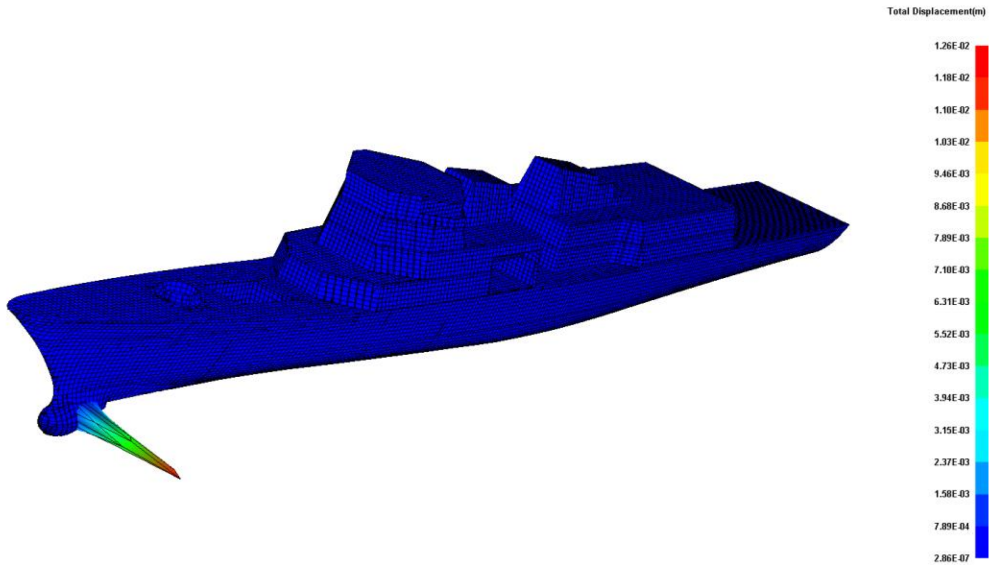


Figure 3/2-23 Mode 8 eigenmode

For large models, it is recommended that checks are performed as the model is being built. While this may not always be possible, checking as-you-go is significantly easier than assembling an entire large model and then debugging.

3/2.7.5 Boundary Conditions

Because ships are floating structures, the constraints are needed to remove ships' rigid body motions for static analysis. Applying appropriate boundary conditions is important and can be complex. If the boundary conditions are not properly applied, high restraint forces and other factors can easily render the analysis inaccurate. Constraints should be minimally applied to the structure resulting in a minimal number of locations where boundary conditions are enforced.

3/2.7.5.1 Traditional Rigid Body Motion Constraint

For sagging and hogging load cases, it is suggested that rigid body motion constraints are to be applied at the bow and stern locations, at the height of the closest continuous deck near the vertical location of the neutral axis, as shown in Figure 3/2-24. These locations should be evaluated and updated accordingly to minimize reaction forces and artificial stresses around constraints. For other load cases, these constraints may also be applicable, as long as the constraint reaction forces are small. If a constraint reaction force is greater than 1% of the whole ship weight, consider changing a constraint location. A multihull vessel may require a different set of rigid body motion constraints depending on the vessel proportions and number of hulls. These would need to be selected by the analyst based on the given situation. In such cases, the constraints chosen should seek to follow the same principle of ensuring that the constraint reaction forces are small.

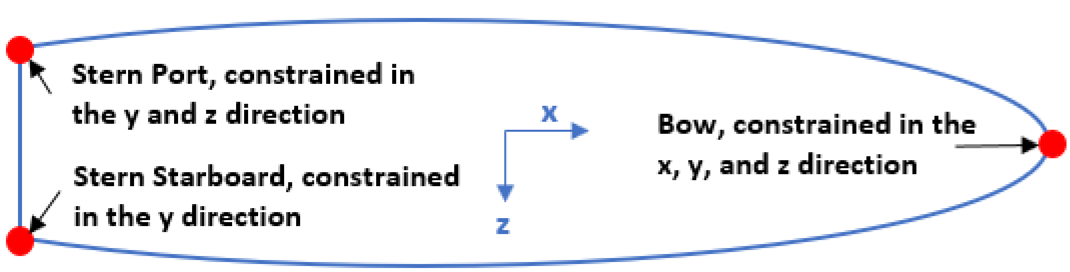


Figure 3/2-24 Typical full-ship boundary constraints

Alternatively, a remote displacement can be used to restrain the model against rigid body motion if the correct settings are available. A remote displacement creates a master node and slave nodes that are connected by rigid beam elements. Some FEA software allows the user to set the behavior of the slave nodes to "deformable," which allows the shape of the slave nodes to change relative to each other. This type of remote displacement, with a master node roughly located on the hull girder neutral axis at midship and the slave nodes located along the length of the vessel on a rigid line, may better distribute any non-real residual forces along the length of the vessel in a way that does not impact the results. Such a rigid line may be the main deck to side shell intersection or a hard chine on a box-shaped vessel (such as a dry dock). This approach should be executed with caution, however, and a good understanding of how the deformable remote displacement functions within the given software. The analyst should also pay just as much attention to minimizing the residual forces and moments in this approach as in the three-point constraint approach.

3/2.7.5.2 Inertia Relief Constraint

The technique of inertia relief has been a well-known approach for the analysis of unsupported systems such as air vehicles in flight, automobiles in motion, or satellites in space [71]. The sum of forces and moments are calculated and applied to achieve an equilibrium state in inertia relief analysis. In inertia relief calculation, the unconstrained structure or system is assumed to be in a state of static equilibrium. Acceleration is computed to counterbalance the applied loads. A set of translational and rotational accelerations provide distributed body forces over the structure in such a way that the sum of applied forces and the sum of moments are zero. Since rigid body motions are restrained, conventional static analysis can be performed. The inertia relief feature is available in most general finite element packages. It should be noted that the accelerations calculated to balance the applied loads should be small. If one of the acceleration components becomes relatively large, it often indicates the model is not properly loaded, and a review of loads and the model is required.

3/2.7.6 Hull Girder Design Waves

The prediction of the behavior of the ship in waves represents a key aspect in the quantification of both global and local loads acting on the ship. The solution of the seakeeping problem yields the loads directly generated by external pressures and also provides ship motions and accelerations. The latter is directly connected to the quantification of inertial loads and provide inputs for the evaluation of other types of loads, such as slamming and sloshing.

A traditional analysis for the evaluation of wave-induced loads is represented by a quasi-static design wave approach. The ship is statically positioned on a wave of given characteristics in a condition of equilibrium between weight and static buoyancy. The scheme is analogous to the one described for still water loads, with the difference that the waterline upper boundary of the immersed part of the hull is no longer a plane, but it is a curved surface. By definition, this procedure neglects all types of dynamic effects. One of the static design wave approaches is based on investigations during the 1930s and 1940s that determined that the trochoidal profile appropriately represents sea waves and that $0.61\sqrt{L_{BP}}$ in meters ($1.1\sqrt{L_{BP}}$ in feet) represents the steepest stable wave height to length ratio for ships under 150 meters in length [72]. The profile or form of the wave is chosen as "trochoidal," with the crest/trough of the standard wave centered amidships. Using the prescribed wave profiles, the hydrostatic pressure to the wettable surfaces can be calculated and applied to the corresponding finite elements. Note that the wave profile needs to be placed properly on the finite element model to ensure equilibrium. For ship specific finite element packages [69], the sinkage, trim, and heel of the model can be automatically adjusted into a quasi-static equilibrium between the defined weight and the buoyancy.

The second method for applying hull girder design bending conditions is to prescribe a hull girder bending moment distribution, then to assign artificial nodal forces in the FE model to generate the prescribed bending moment distribution. The hull girder bending moment distributions are usually given in the form of empirical equations by safety authorities. For U.S. Naval ships, SPECTRA [73] is often used to generate hull girder bending moment distributions. Sikora [74,75] published significant papers and reports detailing a reliability-based design method for primary seaway loadings on surface ships. Typically, the lifetime operational profile of the ship is considered. The objective is to form, on a reliability basis, estimates of global, primary hull girder loads. These include vertical and lateral bending and longitudinal torsion, in addition to criteria for lifetime fatigue strength. Low-frequency, wave-induced bending, and high-frequency, impact-induced whipping are also included. Sikora et al. [74]incorporated empirical data and analysis into a computer program called SPECTRA [73]. SPECTRA computes lifetime exceedance, histogram, and reliability information of hull girder bending moments for a specified hull section. One way to apply SPECTRA lifetime extreme bending moments to a 3D finite element model is to assume the longitudinal distribution of the dynamic vertical bending moment as follows,

$$M_x = \frac{M_{mid}}{2}\left(1 - \cos\frac{2\pi x}{L}\right)$$

where M_{mid} is the dynamic wave bending moment value at midship. To simulate the bending moment on a 3D finite element model, a large set of vertical nodal forces are placed on the model such that the resulting hull girder longitudinal bending moment distribution matches the prescribed bending moment distribution.

Another method to derive hull girder design dynamic wave loads is using potential flow based hydrodynamic seakeeping codes. Most seakeeping tools are capable of exporting hull girder loads, such as bending moments and shear forces. For tools with only sectional forces and moments available, a method based on quadratic programming can be used to transfer the sectional loads to 3D finite element models [76]. For tools with panel pressure available as an output, such as VERES (strip theory) [77], WAMIT [78], HydroStar [79] and PRECAL [80], it is desirable to map panel pressure loads from a hydrodynamic model to the corresponding 3D finite element model to get a more realistic structural response. However, because meshes for hydrodynamic analyses are often much coarser than the corresponding finite element model meshes, the structural model often becomes imbalanced after the panel pressure is mapped from the hydrodynamic mesh to the structural mesh.

Although various interpolation methods have been proposed and used in design practice, transferring panel pressure loads from a seakeeping model to a finite element model while maintaining equilibrium still remains challenging. Often, the “inertia relief” method [81] has to be used for the final adjustment to balance the structural model. The “inertia relief” technique is very powerful and can correct any imbalanced model. However, there are two shortcomings to this approach. First, the additional inertial forces cause a change in the hull girder response (such as bending moment). Second, the change of the accelerations has to be relatively small to ensure the fictitious inertial forces do not significantly distort the original structural response. This often requires visual inspection and engineering judgments.

Malenica et al. [82] proposed a method that mapped the panel source strength instead of the panel pressure from a hydrodynamic mesh to the structural mesh and then formulated the equations of motion using the structural mesh. This concept can be adopted to achieve a balanced structural model without using “inertia relief” (Ma et al. [83]), Some ship / marine-specific FEA software provides the ship designer with an integrated frequency-domain/time-domain computational tool to predict the motions and wave loads of floating structures. When this tool is integrated within an FEA toolset, the hydrodynamic loading process learning curve is greatly reduced, and the issue of mapping data between a seakeeping code and an FEA program is eliminated. This approach takes advantage of the existing structural mesh and defined loads to formulate the equations of motion, and results in perfect equilibrium for the structural model, so no inertia relief or artificial loads are required to balance the model. Bending moments, shear forces, and torsional moments are all automatically in closure and will not result in the distorted FE results associated with the excessive inertial balancing of a whole ship FE model.

Regardless of the toolset or computer codes used, the procedure must first compute the response of ship motions, hull girder loads, and panel pressures for incident waves of unit amplitude based on wave frequency, ship speed, and heading. The response amplitude per unit wave amplitude is often referred to as the response amplitude operator (RAO). RAOs are effectively transfer functions which give the proportion of wave amplitude "transferred" by the ship system into ship response. The short-term and long term most probable extreme wave loads can be generated by the: hull girder dominant load parameters (DLP) RAOs, operational profile, wave scatter diagram, wave spectra, and the return period. In principle, the base assumptions of the linear wave theory are valid only for small wave excitations, small motion responses, and low speed of the ship. In practice, the field of successful applications extends far beyond the limits suggested by the preservation of realism in the base assumptions; the method is used extensively to study extreme loads and high-speed vessels.

The typical hull girder dominant load parameters (DLPs), which represent the hull girder loading conditions that are used as a set of design conditions for the hull structure, are:

- Vertical bending moment
- Vertical shear force at ¼ and ¾ of the vessel length
- Vertical acceleration at bow
- Roll motion
- Relative vertical velocity at the bow
- Longitudinal torsional moment
- Horizontal bending moment
- Horizontal shear force

The corresponding unit wave for each DLP is then scaled up to the extreme value for an equivalent design wave load based on extreme ocean wave theory [84]. For each of these DLP load effects, an equivalent regular wave, defined by its wave height, wavelength, heading angle and position along the ship length, is determined so that the maximum response for the selected load effect is equal to its value given by the rules or design criteria for the probability of exceedance considered. The amplitude of the other effects is obtained from a ship motion analysis assuming the ship to be positioned on the equivalent regular wave. The superposition of the dynamic design wave load and the still water load is used for whole ship structural analysis. This procedure, illustrated in Figure 3/2-25, can be fully integrated into some software toolset [69].

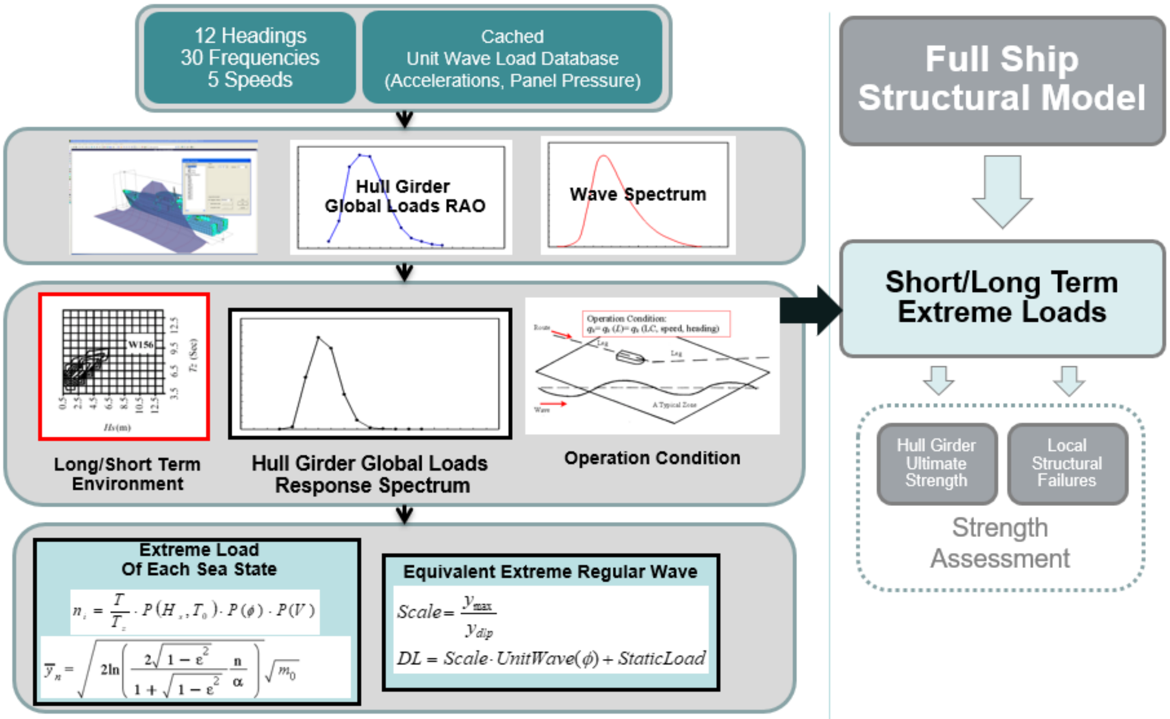


Figure 3/2-25 Extreme Load Analysis Procedure

Due to the level of detail in whole ship finite element models and the computational power required to solve the source strength for each hydrodynamic panel, limiting the number of wetted panels in the potential flow computation is an important consideration. Most potential flow codes have a specified upper limit, typically of 3,000 to 5,000 panels. In the case of some software tools [3], three different approaches can be identified to map the hydrodynamic mesh to the finite element mesh. The first method is to compute the loads on a one-to-one ratio between the finite element and hydrodynamic meshes. This method gives more accurate results but is computationally intensive, and it is also limited in computational time, not the number of panels. The second method is to use the limit state evaluation patches as hydrodynamic panels if they are available. The third method is to leverage a NURBS surface model and map the surface mesh to the "wettable" finite element elements. The number of hydrodynamic panels can be further reduced by using panel merging algorithm, which automatically merges adjacent evaluation patches into larger panels while maintaining a user-specified aspect ratio. Using the evaluation panel hydrodynamic mesh discretization means that each panel is made up of several finite elements.

3/2.7.6.1 Loads Due to Nonlinear Seakeeping Analysis

The ABS Guide for Dynamic Loading Approach [81] Subsection 6/1 requires that a non-linear seakeeping analysis is to be performed to establish the instantaneous design loads at a specific time when each Dominant Load parameter (DLP) reaches its maximum. The frequency-domain extreme load analysis results are used to identify the expected magnitude of each DLP's most probable extreme value and the 'critical' speed, heading, and sea state where it occurs. Time-domain simulations associated with each DLP's critical speed, heading, and sea state are run to capture the nonlinear effects for each DLP and create what is considered to be a more physics-based representation of the ship's inertial conditions and associated hydrodynamic loading. A number of separately seeded (independent) time-domain runs are computed for each DLP in order to create a sampling of maximum conditions to select from. Each time-domain simulation uses a unique speed, heading, and sea state combination. The panel pressures, point forces, accelerations, and hull girder loads are typically recorded at every 0.5 seconds, and the time-step associated with the maximum

DLP value will be identified for use as the design load case condition.

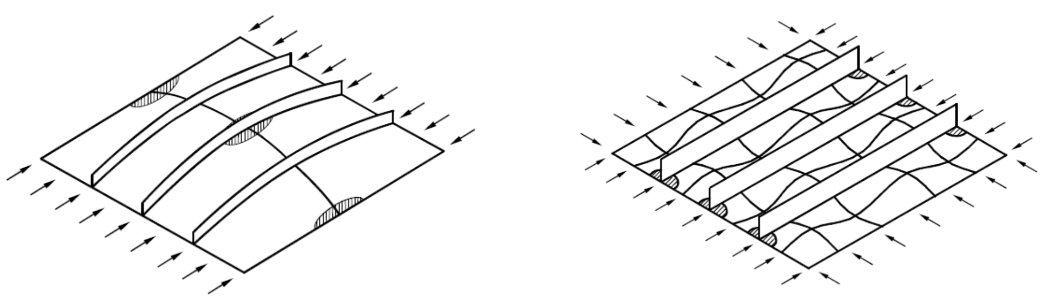
A load scale factor can be determined using the ratio of the peak time-domain hull girder loads (DLPs) and the frequency domain hull girder loads. The panel pressures, point forces, and accelerations at the peak response time-step of each time-domain simulation are scaled such that the magnitude of the DLP is equal to that predicted from the frequency domain analysis. If the peak of the time domain run exceeds the frequency domain extreme value, the peak of the time domain run is used for that particular DLP. To get an improved physics-based simulation of ship motions and hydrodynamic loads while preserving a process that is practical in a design context, a weakly-nonlinear time-domain simulation in a sea state is often used. The term weakly nonlinear refers to the use of potential flow analysis to predict hydrodynamic loading in which the radiation and diffraction forces are computed on the ship's stillwater line in order to remain computationally practical, while the Froude-Krylov and hydrostatic forces are computed using the actual wave profile on the ship's hull associated with each time step in the simulation.

3/2.7.7 Limit State or Failure Criteria

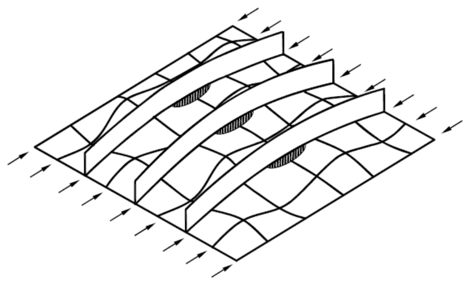
To assess the strength of whole ship primary structures, a set of acceptance criteria is often used. The acceptance criteria include not only stress limits but also buckling rules and other limit states. While the stress limit can be readily evaluated by extracting element stresses directly from finite element analysis, evaluating structural component buckling is more complicated because a buckling event occurs at the stiffened panel level but not at the finite element level. Currently, the use of an empirical or semi-analytical approach for buckling assessment is an attractive strategy due to its effectiveness in terms of engineering accuracy and related computational time. This aspect becomes even more important when dealing with highly nonlinear analyses and in the context of optimization procedures, in which repeated analyses are required.

The primary modes of overall failure for a stiffened panel under predominately compressive loads can be categorized into the following six types [85]:

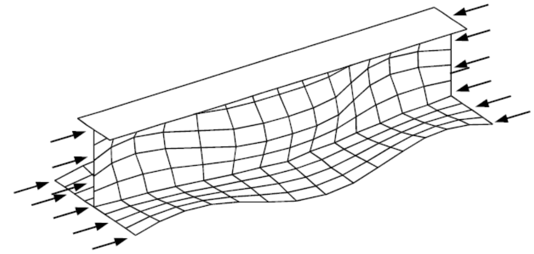
- Mode I: overall collapse after overall buckling
- Mode II: collapse of the plating between stiffeners without the failure of stiffeners
- Mode III: beam-column type collapse of a stiffener with attached plating
- Mode IV: local buckling of stiffener web (after buckling collapse of attached plating)
- Mode V: lateral-torsional buckling (tripping) of a stiffener
- Mode VI: gross yielding



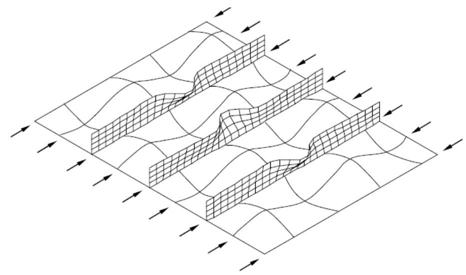
Mode I Mode II



Mode III



Mode IV



Mode V

Figure 3/2-26 Typical buckling failure modes of stiffened panel

Class societies and safety authorities have published acceptance rules for the above buckling failure modes, shown in Figure 3/2-26. While the rule-based strength criteria equations, also called “limit states.”, are relatively simple, they are not integrated into general FEA programs. It often becomes the responsibility of the engineering analyst to use internal organizational processes (e.g., spreadsheets and macros) to first extract structural response results (i.e., deformations/stresses) and to then perform external strength calculations. Typically, when using whole ship FEA for hull structure design, the engineering analyst post-processes these results within the context of the global FEA model, which can also be difficult, time-consuming, and error-prone. In some finite element programs [69], the limit state analysis is automated.The user can select from so-called first principles engineering algorithms including limit states as defined in Hughes’ “Ship Structural Design” [66], and ULSAP as defined by Hughes and Paik in “Ship Structural Analysis and Design, Ultimate Limit State Design of Steel-Plated Structures” [85], or from implementations of U.S. Navy or ABS criteria including HDBK-MIL-519 [72], DDS 100-4, ABS-DLA [81] and ABS’s High-Speed Naval Craft rules (HSNC) [86].

This limit state evaluation automatically defines evaluation panels, which are a collection of finite elements within the whole ship model used to define a stiffened panel or the true span of a beam, and compute the panel’s and beam’s loads and load effects from the finite element analysis results. Using this automation, the code evaluates the entire global structure, at the level of limit state evaluation panels or structural entities, for every finite element analysis load case. Figure 3/2-27 shows two types of evaluation panels automatically defined for the entire ship’s structure, one for stiffeners and plates, and one for stiffened panels.Each colored group of elements represents a single evaluation patch, which is evaluated against each of the selected limit states for all load cases defined.

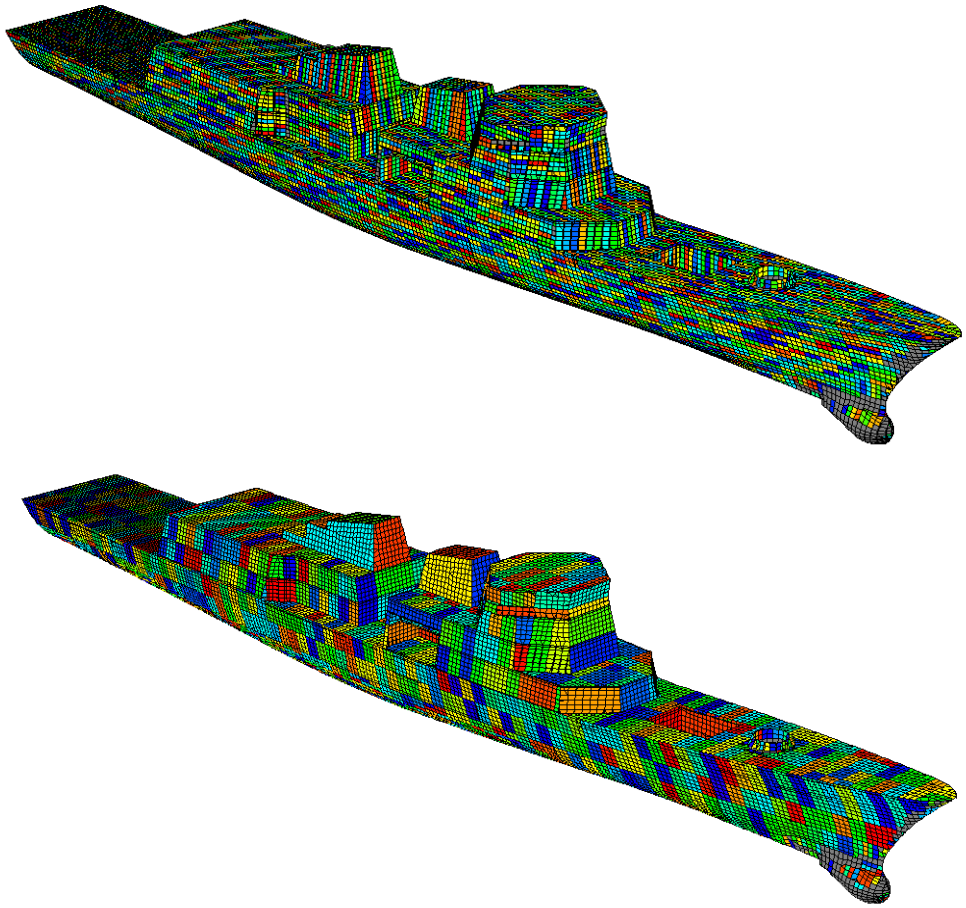
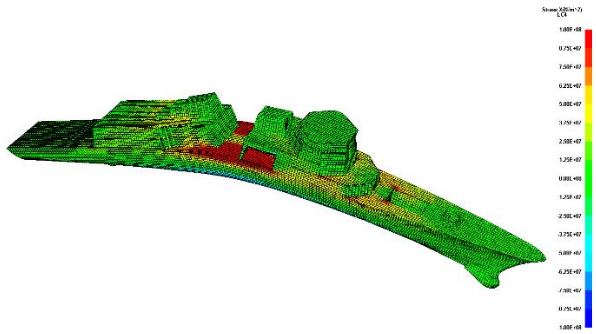


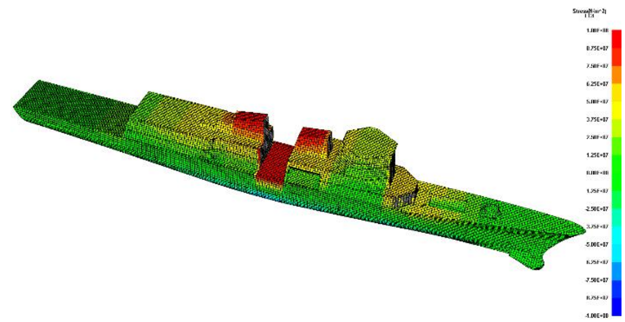
Figure 3/2-27 Frigate Evaluation Panels

3/2.7.8 Finite Element Analysis and Results

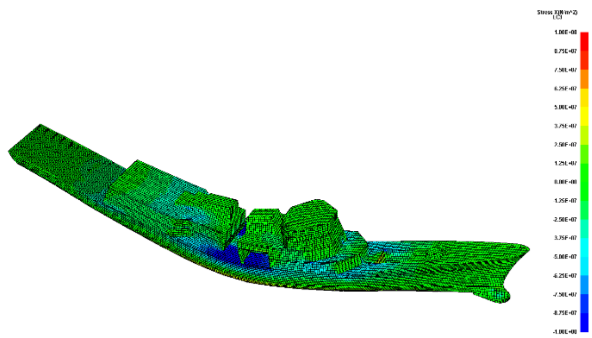
Before performing all analyses of various load cases on the full-ship model, obvious modeling errors, such as materials, properties, units, weights, warped elements, element connections, and free edges, should be checked and corrected. In addition, results under simple loads need to be examined qualitatively to see if they "look right." For example, for a global full-ship analysis, it is often prudent to check and review the stresses and deflections of still water, and quasi-static sagging and hogging load conditions. For these load cases, the stresses obtained from the finite element analysis and from traditional beam theory should have a reasonable correlation. A stress distribution comparison of finite element analysis and a simple beam analysis under hogging and sagging waves is shown in Figure 3/2-28. This figure illustrates that the longitudinal hull girder stress distribution of the finite element analysis tracks well with the simple beam analysis, but presents more realistic stress behavior such as shear lags and corner stress concentrations. It should also be noted that the stresses calculated using simple beam theory depended on the structural longitudinal effectiveness assigned by the user. In this example, some of the deckhouse structure members and the structures near the opening should be assigned as longitudinally non-effective and do not make any contribution to the section modulus.



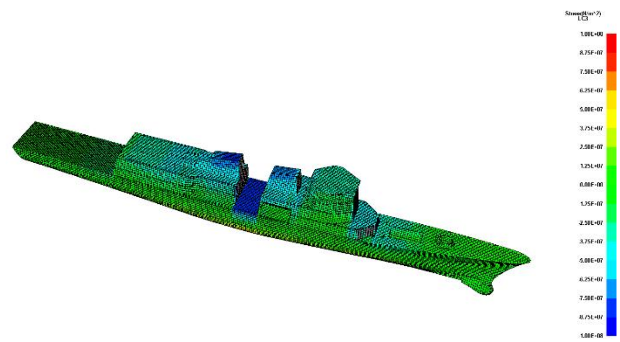
FEA (Hogging wave)



Simple beam theory (hogging wave moment)



FEA (Sagging wave)



Simple beam theory (sagging wave moment)

Figure 3/2-28 Longitudinal stress distribution comparison between FEA and simple beam theory

For full-ship global finite element analysis, longitudinal strength under vertical bending moment and vertical shear forces is the first important strength consideration. The hull strength under other load conditions, including horizontal bending, horizontal shear, and longitudinal torsion, should also be considered. For multi-hull models, load conditions of transverse bending, transverse shear, transverse torsion, squeezing, and prying need to be evaluated. Whole ship FEA typically includes these major hull girder load cases, and FE results will be generated and reviewed for these conditions. The reactions of structural components of the ship hull to external loads are usually measured by stresses and deflections. Stress is related to gradients of the field quantity, and gradients in a given element depend on field quantities at nodes attached to that element only. Stress contours are discontinuous across element boundaries. Strong discontinuities indicate too coarse mesh density or mesh transition, whereas practically continuous stress contours suggest unnecessarily fine discretization. For each load case, the element stresses extracted from the FE model are used to check against design acceptance criteria of yield, buckling, and fatigue, as discussed in the previous section. Excessive deflection may also limit the structural effectiveness of a member, even though material failure does not occur, if that deflection results in misalignment or other geometric displacement of vital components of the ship's machinery, navigational equipment, etc., thus rendering the system ineffective.

Finally, it should be noted that finite element analysis is only the mathematical approximation of a real-world system. Modeling decisions are influenced by what information is sought, what accuracy is required, the modeling and computational expense of FEA, and its capabilities and limitations. Also, initial modeling decisions are provisional. It is likely that the results of the first FEA will suggest refinements of the model and loads.

3/2.8 Frequency Response Vibration Analyses

3/2.8.1 Basics

A frequency response analysis requires less computational power because the FEA equations can be solved in the frequency domain once the frequency is assumed to be constant. A frequency response analysis determines the steady-state response of a structure that is loaded at a single frequency. This type of analysis ignores any initial transient motion and assumes the structure is linear.

A frequency response analysis determines the frequency response at a single frequency independent of all other frequencies. The results are how the structure would respond at that single frequency. A frequency response analysis, however, also typically produced results for a series of frequencies. The results for each frequency are independent of the other frequency results.

FEA programs often include post-processing tools that can convert the frequency domain results into time-domain results for visualization purposes. This feature, however, is not required because the frequency domain results contain all the necessary response information in terms of displacement, velocity, and acceleration.

If an engineering model has more than one load and the loads occur at the same frequencies, then these can be evaluated in the same frequency response vibration analysis. The loads also can have different phase relationships with one another.

If an engineering model has more than one load, and the loads occur at different frequencies, then multiple frequency response vibration analyses will need to be run. The maximum response values can then be summed together to get the total maximum response because the structure is assumed to behave linearly (as previously stated).

The interested reader may also refer to [87] for an excellent primer on frequency response analyses.

3/2.8.2 Modeling Requirements

The model should be large enough to model all ship structural vibrations of concern. If the hull girder vibration is a concern, then the whole ship should be modeled. If the superstructure fore and aft vibration is a concern, then the superstructure should be modeled as well. If the analysis is concerned with only local vibrations, then considerable care should be exercised in deciding where the boundaries of the model should be. Reference [88] recommends that if a local structure is being evaluated, then the model should extend one major transverse bulkhead forward and aft from the area of interest. The 5/3 FREQUENCY RESPONSE VIBRATION ANALYSIS sample application is a local analysis. It includes a comparison of modeling the full engine room bulkhead to bulkhead versus just modeling the structure above the tank top within the engine room.

The model should include all relevant masses. This will certainly include the structural weight. Equipment masses should be modeled separately in the FEA, where they may have an impact on the results. The remainder of the loading condition mass should also be included.

ABS [14] recommends that added mass be accounted for and provides some different options. See Section 3/2.4 for an additional discussion on the inclusion of the fluid mass surrounding the structure.

ABS [14] recommends that damping be accounted for in the model using a damping coefficient of 1.5% of the critical damping and that this is applicable to the entire range of typical propeller and main engine frequencies.

Recent use of ANSYS for a local frequency response vibration analysis did not require any need to reduce the model size using the dynamic degrees of freedom reduction methods discussed in Section 3/2.5. These techniques may be needed for large whole-ship models depending on the mesh density.

3/2.8.3 Frequency Range Requirements

Reference [88] recommends that a frequency response analysis be formed for a range of frequencies +7 and -3% of the excitation frequencies. This is recommended for “relatively high mesh density models” and that “it may be beneficial to increase this range in some cases, such as when a lower density mesh is used.” This is recommended, “because the prediction at any single frequency is likely to be erroneous.” The worst-case result of this range should be reported in the conclusion of analysis to be conservative.

3/3 FINITE ELEMENT MODEL CHECKS

The subject of this section is the checks that should be performed to ensure that the physical problem is appropriately translated into the finite element model. Guidance is provided on various aspects of a finite element model, such as appropriateness of the element types used, the density of finite element mesh used for plated structures, substructuring, and submodeling used to optimize the problem size, loads and boundary conditions application, and the solution setup process. There is also a short subsection on graphical checks using the software's pre and post-processors to scrutinize the finite element model and results.

Since access to the software is essential to perform many of these checks, it is the responsibility of the analyst to ensure that these checks are performed. However, documentation, in the form of plots and graphs, should be available for audit.

Several examples illustrating finite element modeling practices are presented in Appendix C. The purpose of these examples is to show the effect of varying certain finite element modeling parameters on the results. The main modeling parameters addressed in this appendix are element type and mesh density.

3/3.1 CAD Importing

3/3.1.1 Geometry Check and Cleanup

The very first step of pre-processing a model is to deal with CAD geometry. The pre-processing starts with the import of CAD data. The data can be in different formats (e.g., CATIA v4/v5, STEP, UG, IGES / IGS, Parasolid, Pro-E, STL, ACIS, DXF, JT, etc.). Note that it is highly recommended to have the CAD model in a format that is compatible with the FEA code being used. However, it is common that issues or errors may occur when importing CAD data. Some of the “issues” are for FE meshing but not for CAD designing, and it is up to the analyst to clean up these geometric issues for meshing [15].

Several CAD issues that need to be cleaned up are listed as follows:

- Not stitched surfaces, i.e., gap(s) between surfaces
- Overlapped surfaces
- Broken surfaces
- Misaligned surfaces
- Redundant surfaces
- Not connected surfaces, i.e., penetrated each other without connecting (see Figure 3/3-1)
- Small surfaces not possible to mesh with a reasonable quality
- Other detail will result in meshing with very poor-quality elements

A more detailed list of issues can be found from Question 3.1.3 of PART 2.

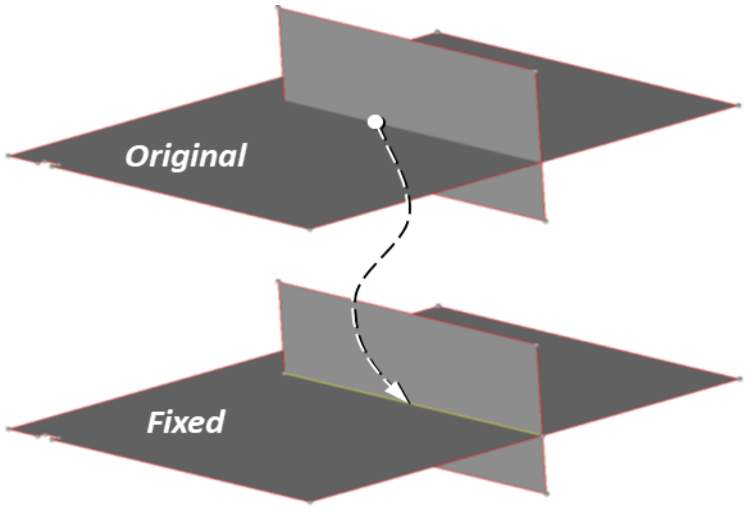


Figure 3/3-1 Not Connected Surfaces Reproduced from Ref. [15]

All the geometric issues and errors in the imported CAD model need to be resolved and cleaned up by carrying out topology repair. Reference [15] recommends geometric cleanup be performed based on the following strategy:

1. Define the global element size from the model scale
2. Determine the geometric cleanup tolerance (e.g., 15~20% of global element size) from the global element size
3. Review the topology of CAD data to find the issues to be fixed
4. Clean up CAD models
    a. Find and delete duplicate surfaces
    b. Stitch surfaces with free edge pairs (e.g., equivalence to combine)
    c. Toggle or untoggled issue edges
    d. Fill missing surfaces

Many FEA software vendor user manuals and tutorials provide more detail steps and explanations of how to find and fix geometric issues.

3/3.1.2 Geometry Defeaturing

Once accomplishing the geometry cleanup, the analyst needs to make a further geometric assumption to eliminate unnecessary CAD information. This is the second step of pre-processing and generally known as “geometry defeaturing.” In this step, the geometric details (e.g., fillets or corners with a small radius and tiny holes) that may not contribute to the overall performance of the component can be simplified or removed. The geometric profiles recommended to be defeatured are listed in Question 3.1.3 of PART 2. Reference [15] recommends geometric defeaturing be performed according to one of the following three algorithms:

- Preserve boundaries between components
- Recognize and preserve major feature edges
- Recognize and suppress construction edges

If the 2D mesh type is adopted, it is necessary to extract the mid-surface from the 3D geometry data. The details of choosing a mesh type and mesh design will be discussed in the next section. The selected variables for the mid-surface extraction are listed as follows:

- Algorithm for topology extraction
- Maximum thin solid thickness to width ratio
- Maximum thick solid thickness
- Minimum feature angle between the solid's edge and its faces
- Thickness transition as well as the transition increment (e.g., ± 0.5mm)

After geometry cleanup and geometry defeaturing, the analyst needs to carry out another iteration of the CAD data reviewing. This process will result in the model being free from topology violations, especially in the area of interest.

3/3.2 **Element Types**

To some extent, all finite element types are specialized and can only simulate a limited number of types of responses. An important step in the finite element modeling procedure is choosing the appropriate element type(s). The elements best suited to the particular problem should be selected while being aware of the limitations of the element type. A good guide to the suitability of an element type is their performance in other similar situations. Also, it is rare to use only one type of element to solve engineering problems. Typically multiple element types are used in combination (e.g., 1D, 2D, 3D, and others).

Element performance is generally problem-dependent. An element or mesh that works well in one situation may not work as well in another situation. An understanding is required of how various elements behave in different situations. The physics of the problem should be understood well enough to make an intelligent choice of the element type. As a reasonable guideline, Cook [89] considers elements of intermediate complexity work well for many problems. According to this reference, the use of a large number of simple elements or a small number of very complex elements should be avoided. The selection of element type is mainly dependent on four factors: (1) geometry size and shape, (2) type of analysis, (3) structure actions to be modeled, and (4) time allocated for the project per references [15,90].

3/3.2.1 Geometry Size and Shape

In each FE simulation, a model must be defined in 3D properly with discrete meshing (e.g., nodes and elements). The global geometry can be divided into representative local geometries (in 1D, 2D, and 3D), and then the element type for each part can be selected accordingly.

- For a geometry with one dimension significantly larger than the other two dimensions (e.g., X >>> Y, Z), it is recommended to use 1D elements. For example, beam-like structures, long shaft, rods, columns, bolt joints, fasteners, or connectors. The 1D elements can include 2-node, 3-node, and 4-node elements.
- For a geometry with two dimensions significantly larger than the other one dimension, it is recommended to use 2D elements. The 2D elements, in general, include the following kinematic assumptions:
    - Plane stress
    - Plane strain
    - Axisymmetric
    - Generalized plane strain
    - 3D plane stress (membrane)

  The typical 2D element (i.e., SHELL) may have the following formulations:

- 3-node triangle (1st order)
- 6-node triangle (2nd order)
- 7-node triangle (2nd order with additional center integration point)
- 4-node quadrilateral (1st order)
- 8-node quadrilateral (2nd order)
- 9-node quadrilateral (2nd order with additional center integration point)

- For a geometry with comparable sizes in three dimensions, the "3D element" should be adopted. Some typical 3D elements are listed as follows:
  - Hex / Brick elements
    - 8-node (1st order)
    - 20-node (2nd order)
    - 21-node (2nd order with additional body-center integration point)
    - 27-node (2nd order with an additional body- and surface-center integration points)
  - Tetra elements
    - 4-node (1st order)
    - 10-node (2nd order)
    - 11-node (2nd order with additional body-center integration point)
  - Penta / Wedge elements
    - 6-node (1st order)
    - 15-node (2nd order)
  - Pyramid elements
    - 5-node (1st order)
    - 13-node (2nd order)
    - 14-node (2nd order with additional body-center integration point)
  - Transition elements

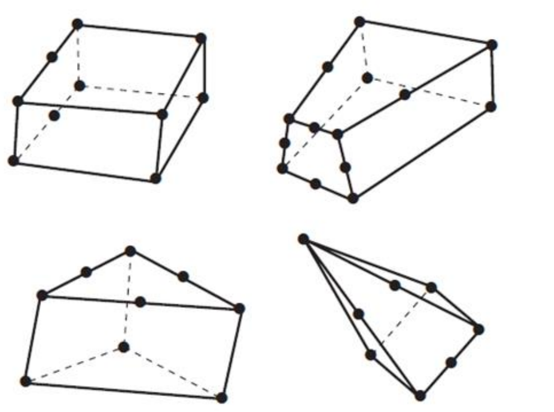


Figure 3/3-2 Elements for Transition Zones [91]

Elements with different numbers of nodes along each geometric feature line are used for transition zones (see Figure 3/3-2)

The choice of elements in current FEA codes, in general, is not limited to the above-mentioned types, and there are other types available for meeting special requirements. For example, pipe elements, spring/damper/mass elements, alignment elements, cohesive elements, and connector elements.

3/3.2.1.1 Truss/Beam Elements

Truss elements are the simplest in form. The only physical property required is the cross-sectional area. Beam elements, on the other hand, are considerably more complex. The various sectional properties needed to define beam elements are discussed in the following paragraphs.

The basic sectional properties required to define beam elements are cross-sectional area, shear areas in two orthogonal directions normal to the longitudinal axis of the element, torsional constant, and the second moments of the area about two orthogonal axes. The axes are usually chosen to coincide with any axes of symmetry that may exist. While this definition of beam properties is complete for the vast majority of cases, there are circumstances in which additional factors need to be considered.

The torsional stiffness is based on the torsional constant alone, and therefore no account is taken of warping effects. Warping is most relevant for open sections. The error introduced by ignoring warping is, fortunately, usually not serious because of the circumstances in which open sections are generally used in structures. However, in situations where the main structural force acting on an open-sectioned beam is torsion, this shortcoming should be considered in calculating rotations and torsional stresses. Structures modeled using standard beam elements in most general-purpose FEA software would yield incorrect results. Some FEA software does offer beam elements that account for warping effects.

Shear flexibility is vital for deep short beams. Ignoring shear effects for this configuration would result in an overestimate of flexural stiffness.

3/3.2.1.2 Shell Elements

The input data required for plate and shell members is thickness. Most finite element computer programs can accommodate the non-uniform thickness and have the facility to input different thicknesses at each node.

Type of Analysis

Linear stress field elements are the most commonly used. Almost all finite element analysis (FEA) software have families of elements that include elements with linear stress capabilities. For many portions of structures, a mesh of linear stress elements can provide a good description of the stress and strain state. In areas of discontinuities, high thermal gradients, fatigue/fracture studies, or nonlinear material problems, where there is an interest in evaluating more than just a linear stress state, linear elements associated with a relatively fine mesh can sometimes produce reasonable results. However, it is recommended to adopt elements with quadratic and higher-order stress fields, which employ cubic or higher-order displacement functions. Note that for certain engineering problems, only high order elements can capture the variations of stress or strain gradients, e.g., stress singularity of a crack tip [47,92]. These elements have either more nodes per element and/or more degrees of freedom per node. These can be more expensive in terms of computational effort to form the element stiffness matrices.

Complex structures (e.g., ship deck structure with openings) require relatively fine meshes to model the geometrical discontinuities adequately. According to Kardestuncer [93], higher-order elements are also practical when modeling areas of the high-stress gradient with a relatively coarse mesh. Note that reference [93] points out that the quadratic or higher-order fit may over or underestimate the stresses at free surfaces. The order of the stress function must match the gradient properly. The behavior of a linear stress element is easy to visualize, which is one reason for their popularity and simplicity. Another limitation higher-order elements suffer is the limited availability of companion elements (i.e., meshing compatibility). Lower order element families have a complete range of elements, and therefore it is easier to use these element types when it is necessary to mix different elements (e.g., plates and beams).

Several recommendations [15], on the selection of elements for different analysis types, are summarized as follows.

- **Structural and fatigue analysis:** Quadrilateral and Hexahedral elements are preferred over Triangular, Tetrahedral, and Pentahedral.
- **Crash and nonlinear analysis:** Quadrilateral and Hexahedral are preferred over Triangular and Tetrahedral, and priority to mesh flow lines (including topology features).
- **Dynamic analysis:** 2D elements are preferred over 3D. Shell elements can capture the modes with fewer nodes and elements.
- **Fracture and damage analysis:** Quadrilateral and hexahedral elements are preferred over triangular and tetrahedral elements, and high order elements are preferred over low order elements.
- **Mold flow analysis:** Triangular elements are preferred over quadrilateral elements.

3/3.2.2 Structural Action to be Modelled

When a finite element model of a structure is being planned, it is necessary to have a clear concept of the main structural actions. Each element type has limitations and is designed to model a single or limited number of structural actions.

Before modeling a structural problem, it is useful to have a general idea of the expected behavior of the structure. This knowledge serves as a useful guide in several modeling decisions that need to be made in building the model. In an ideal situation, the first model will yield adequate results. However, the first model is seldom adequate. Hence, one or more revisions will usually be necessary.

In triangulated framed structures, if the members are relatively slender, then the main action is axial with limited bending action. In this case, the use of truss elements would be justified, and the use of beam elements may introduce an unnecessary complication. In some instances, a mixed approach may be appropriate. Consider a lattice mast, as shown in Figure 3/3-3. The main legs, which are continuous, should perhaps be modeled using beam elements, whereas the bracing members would be better modeled using truss elements.

Similarly, deck structure in ships that are subject primarily to in-plane loadsare better modeled using plate/shell elements. In some cases, transverse shear effects may be significant. Certain element formulations do not account for shear. Some FEA software provides plate bending elements in which the ability to model transverse shear is optional and has to be selected by the analyst.

If through-thickness stresses are considered to be important, then the use of solid elements is prudent.

3/3.2.3 Time Allotted for Project

In general, the selection of elements, mesh designs, and reasonable mesh quality is highly recommended for FEA simulations. However, the project sometimes needs to be accomplished under a very tight schedule. For such situations, it is recommended to perform the simulation work with reasonable assumptions and approaches. For example, the following approaches may help analysts to accomplish the project with reasonable accuracy and also in a timely manner.

- Use automatic or batch meshing tools embedded in the FEA code.

- Perform preliminary studies by adopting tetrahedral elements that are generally meshed using the automatic meshing function with quality control criteria.
- Mesh the critical or interesting parts with finer meshing and high-order elements if the assembly of several components is involved. Mesh the rest of the parts with either coarse mesh or representative 1D elements.

3/3.2.4 Additional Concerns

From references [15,42,90,94–97], there are some additional useful comments and concerns regarding selections of element types. Most of them can be categorized as empirical engineering experiences (i.e., Rules of Thumb). They are:

- If possible, do NOT overly use triangular elements or mix triangular and quadrilateral elements, especially close to areas of interest. The use of triangular elements can lead to a stiffer structure compared with reality.
- If the model is meshed with either triangular or quadrilateral elements, carry out the mesh sensitivity study based on one element type, i.e., coarse vs. refined meshed model for the shell elements.
- After preliminary or linear FEA studies that generally are based on tetrahedral elements, compare results from shell elements with that from tetrahedral elements. For example, the comparisons of modal analysis results in terms of eigenvalues and hot spots can properly indicate the qualities of mesh design and mesh sensitivity.
- Whenever possible, control the element size consistently in the global model or at least in the areas of interest.
- It is recommended to add a layer of ultra-thin shell element on top of the 3D elements in areas of discontinuity for mesh transitions and fatigue analyses. This procedure will capture the high stress and strain states on the skin by eliminating solid element smoothing to its free surface.
- In an assembly model, it is acceptable to have a combination of different mesh types (1D, 2D, and 3D). However, the analyst should first understand all the assumptions and implications of the problem.

3/3.2.5 Miscellaneous One-Dimensional Element Comments

From the references of NASTRAN [95] and OPTISTRUCT [98], the 1D element types are summarized as follows. It should be noted that other FEA codes (e.g., LS-DYNA) have different 1D element types, and the analyst should choose the 1D element type after checking the corresponding references. In general, there are two types of 1D elements: rigid-body elements and physical elements.

Rigid-body elements have the following characteristics:

- They do not need material properties and cross-section profiles.
- One master node drives multiple slave nodes, i.e., RBE2 [95]. It is used as a shortcut to model another body or control multiple nodes through one node.
- One master node transfers distributed loading (e.g., force or displacement) to multiple slave nodes, i.e., RBE3 [95]. This type of element can simplify the modeling process and mimic the distributed loading.

Physical elements have the following characteristics:

- They need material properties and cross-section profiles.

- Several widely used 1D physical elements are listed as follows:
  - Spring elements (i.e., CELAS1 [95])
  - Bushing elements (i.e., CBUSH [95])
  - Rod elements (i.e., CROD [95])
  - Bar elements (i.e., CBAR [95])
  - Beam elements (i.e., CBEAM [95])
  - Pipe elements (i.e., CPIPE [95])

By properly using the 1D element, the simulation time can be reduced significantly. However, extra care is needed in model development, and the analyst needs to fully understand the geometries, meshing approaches, and structural responses.

3/3.3 Mesh Design

Mesh design, the discretization of a structure into a number of finite elements, is one of the most critical tasks in finite element modeling. The following parameters need to be considered in designing the layout of elements: (1) mesh size & mesh density, (2) mesh transitions, (3) stiffness ratio of adjacent elements, (4) joint modeling, (5) advanced analysis mesh parameters, (6) element shape limitations, (7) mesh quality control, and (8) numbering. This subsection provides tips on these aspects of mesh design.

It should be noted that most of the FEA codes nowadays are using two different algorithms in designing mesh: geometry-based and FE-based. The former type of mesh design (geometry-based) is associated with the CAD geometry, that means the mesh design will be updated (sometimes automatically) as the input FE model geometry is modified. Some of the FEA codes have the feature to control or lock the mesh design when the CAD geometry is updated. The latter type of mesh design (FE-based) is independent of the geometry. Note that some of the state-of-the-art techniques can generate geometry in a mesh-based format, for example, laser scanning, 3D printing, and DIC (digital image correlation). It is recommended to use the geometry-based means to design the meshed model.

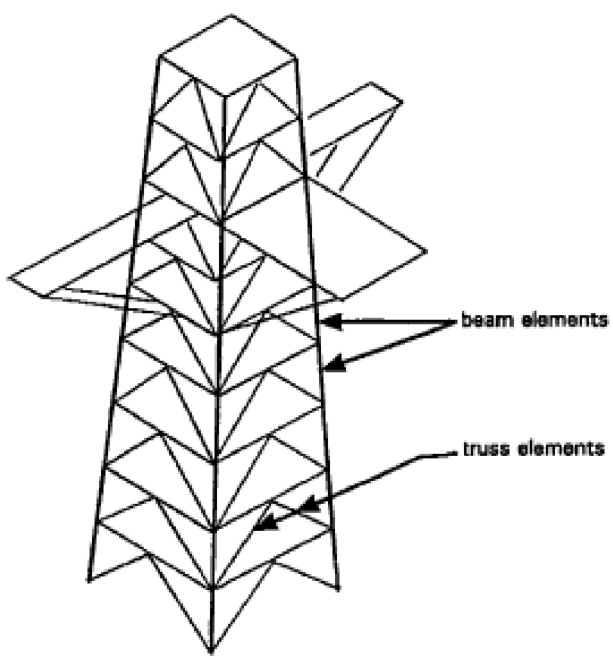


Figure 3/3-3 Typical Lattice Structure

3/3.3.1 Mesh Size & Mesh Density

The accuracy of the FEA results in local areas within an FEA model is dependent on the finite element size. An engineer may create an FE model with a finer mesh (smaller element size) to generate highly accurate local results, but this will require more time for modeling and computing. Alternatively, an engineer may use his engineering judgment to create an FE model with coarser mesh (larger element size) that excludes details that are judged to be unnecessary. For example, lightening holes and cutouts are usually not modeled in a global full-ship analysis. These coarse mesh models may lead to less accurate results in the areas of these holes, but the engineering decision saves time on modeling, computing, and postprocessing. In generating FEA models, the foremost problem is to choose appropriate element size based on the analyst's understanding of what features are important or unimportant such that the created models will yield accurate FEA results for the specified analysis while saving as much modeling, computing and postprocessing time as possible.

As a general rule, a finer mesh is required in areas of high stress or strain gradient. It is possible, of course, to use a fine mesh over the whole model, but this cost more in terms of computational power and time. This is undesirable on three counts: (1) economy, (2) the greater potential for manipulation errors, and (3) accumulated errors caused by an overly fine-meshed model. Hence, the meshes of variable density are usually used. Care is required in transitioning of mesh density. Abrupt transitioning introduces errors of a numerical nature.

The size of the mesh depends upon several factors, such as the type of analysis. The factors are briefly summarized as follows:

- Previous experience - At first, it is upon the analyst to decide the right element size. The analyst's experience plays an important role in striking a balance between economy and adequate mesh size. Analysis of similar structures under similar loading conditions in the past can help in the identification of stress concentrations and regions of rapid changes in stress patterns. In addition, the experience from experimental observations may also help in detecting sensitive areas. This issue is extremely important in the case of novel designs.
- Analysis type - For the linear elastic problems, it is relatively easier to solve the FE model with a large number of elements and nodes (i.e., large degree of freedom – DOF). However, for nonlinear problems, it is more difficult to solve and requires pronounced iterations to obtain converged results. In addition, it may depend upon the type of FEA solver (e.g., implicit vs. explicit) and formulations (e.g., small strain vs. finite strain) adopted.
- Hardware and software limits - One of the major bottlenecks in numerical simulations is the hardware and/or software, e.g., CPU/GPU architecture, CPU/GPU clock speed, and I/O bandwidth. The analyst should understand the upper bound of DOF numbers that the hardware and software can handle properly. In addition, the analyst should also consider the available time (i.e., cycle time and total project time), which can be impacted by the DOF numbers.
- Interested FEA results (e.g., stress, strain, fracture driving force, etc.) - As the interested FEA output may vary in different analysis types, it is recommended to choose the mesh size based upon the minimum requirement. For example, if the deflection of a beam or column is of interest, it is recommended to use 1D element instead of 2D or 3D elements; if the eigenvalues of a structure are of interest, it is suggested to adopt high order 2D or 3D elements with fewer DOF numbers instead of 1st order 2D or 3D elements with more DOF numbers. However, the analyst should know that for projects with special requirements (e.g., fracture and damage), the mesh size is strongly dependent on the problem investigated and sometimes has a physical meaning (e.g., comparable with grain sizes in the micro- or mesoscale). [31,47,99–103]

- Sensitivity of results - Compared with the global response (e.g., displacement/ deflection), the local response (e.g., stress and strain states) can be extremely sensitive to the mesh size and quality, especially around the hot spots. The analyst should use his/her engineering sense to predict the high stress/strain areas that may be prone to trouble and refine the mesh. Some typical geometric features with high stress/strain concentrations are listed as follows:
    - Profiles with sharp radius, for example:
        - Holes
        - Threats
        - Fillets
        - Chamfers
        - Flaws/defects or 2nd phase particle
    - Contact surfaces
    - Loading or fixing areas/points
    - Sudden cross-section changes along the load paths
    - Abrupt material changes
    - Crack or damage paths

The mesh density directly depends upon the mesh size. The basic rule is that the mesh is refined most in the regions of hot spots with steep stress or strain gradients. Therefore, if such regions can be identified during mesh design, the probability of developing an economical mesh with sufficient refinement is high.

In cases where the experience of a particular configuration is lacking and where it is difficult to anticipate the nature of the stress gradients, an iterative approach is necessary. Where stress or strain values show a sharp variation between adjacent elements, the mesh should be refined, and the analysis rerun. As a general practice, it is recommended to refine the meshes around the hot spots (e.g., areas with high stress/strain gradient) and resolve again after the preliminary run. This process may take several iterations until the difference of stress/strain values meets the convergence criteria (e.g., 5 – 10% in Ref. [15]).

3/3.3.1.1 Meshing Application

Once the element size and element density are defined, the analyst needs to mesh the geometry using the pre-processing tools of the selected FEA software. Essentially, there are three different means to mesh a model: (1) automatic meshing with predefined parameters, (2) mapped meshing with more predefined parameters and pre-trimmed partitions, and (3) manual meshing section by section.

Automatic meshing requires minimum user involvement but generates more elements and nodes that the other meshing means. A clean CAD geometry is needed to carry out automatic meshing accurately.

Mapped meshing requires more time and user involvement compared with automatic meshing, but mesh quality can be significantly improved, especially around the areas of interest. This technique also needs clean CAD geometry as the basic input.

Manual meshing requires the most time and user involvement but allows the analysis of full control over the mesh design and quality. This technique can be independent of geometry and results in the least elements and nodes.

The analyst has to specify all the parameters (e.g., minimum hole diameter, minimum fillet radius, average and minimum element length, quality parameters, etc.). The selected parameters are presented in 3/3.3.7, and the recommended values are shown in Table 3/3-1 General Problems with Average Element Size of 50mm Table 3/3-1, Table 3/3-2, Table 3/3-3, Table 3/3-4, and Table 3/3-5. The software can generate meshing automatically after several iterations by fulfilling all or most of the specified instructions. However, the analyst still needs to review the mesh quality and revise/refine areas that violate the criteria.

3/3.3.2 Mesh Transitions

If the mesh is graded, rather than uniform, as is usually the case, the grading should be done in a way that minimizes the difference in size between adjacent elements. Figure 3/3-4 presents several examples of transitions using quadrilateral elements. These examples attempt to keep within the guidelines for element distortion discussed in Section 3/3.3.6.

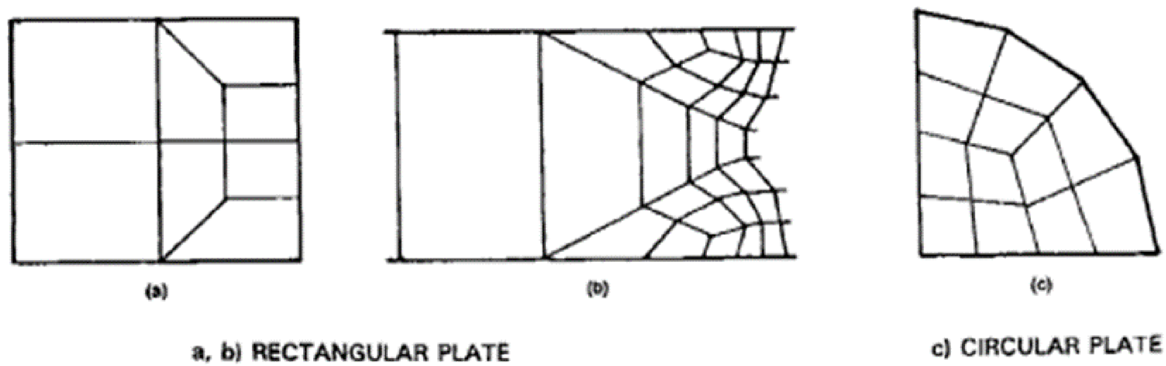


Figure 3/3-4 Transitions from Coarse to Fine Meshes

Another way of viewing good transitioning practice is to minimize large differences in stiffness between adjacent elements. A useful measure of stiffness is the ratio E/V_e, where E and V_e represent the elastic modulus and the element volume, respectively. As a working rule, the ratios of E/V_e for adjacent elements should not change by more than a factor of two [104].

Sometimes transitions are more easily achieved using triangular elements. Transitions of this type are illustrated in Figure 3/3-5. Most FEA programs will allow two nodes of a quadrilateral element to be defined as a single node in order to collapse the element to a triangular shape.

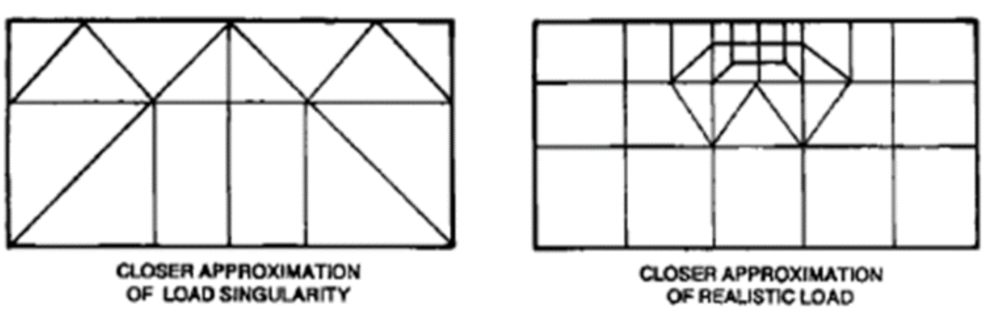


Figure 3/3-5 Transitions Using Triangular Elements

Figure 3/3-6 presents a summary of using either triangular or quadrilateral elements in designing mesh transitions [15]. These six recommended types can be used as a basis to design mesh without violating the element shape limitation criteria. The mesh transitions of 3D elements can be designed in a similar fashion.

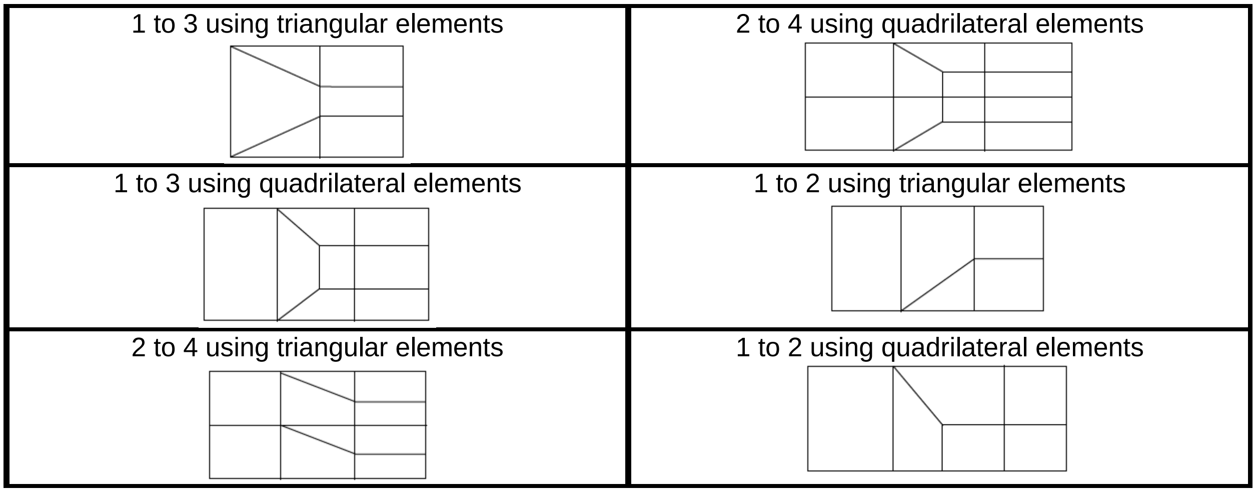


Figure 3/3-6 Six Element Transitions

In modern FEA codes, most analysts rely on preprocessors to develop the finite element mesh. In general, automatic mesh generators yield adequate meshes. However, in very demanding configurations, the mesh generator may produce a poor mesh. In such situations, the mesh should be manually improved to meet the guidelines. Note that some preprocessors have the feature that can highlight the poor mesh areas.

In regular rectangular meshes, there are two basic types of transition. One is the change in element density in the direction of the stress gradient. The second is transverse transitioning, which is used between areas with different element sizes and densities across a transverse plane, as shown in Figure 3/3-7.

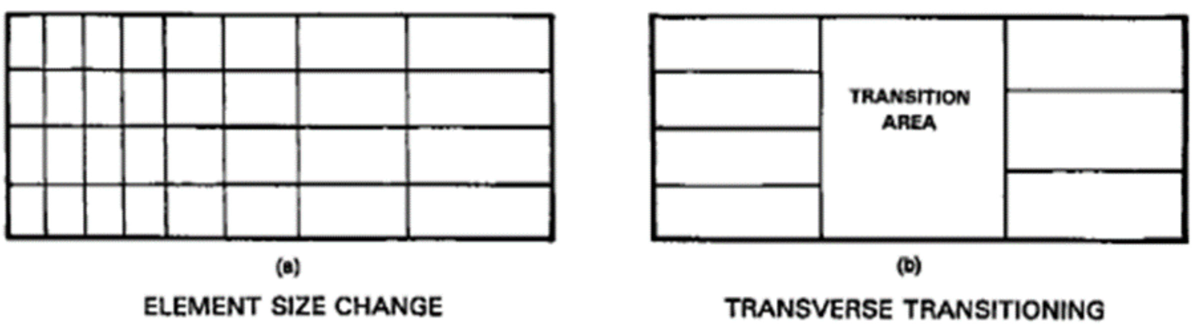


Figure 3/3-7 Mesh Transitions

Many rules of thumb for transitioning of elements are based on element strain energy and strain-energy density calculations. The ideal finite element model should have a mesh with constant strain energy in each element. To achieve constant strain energy of elements, the volumes must be relatively small in regions of high stress or strain and large in regions of low stress or strain. Transverse transition regions should be used only in areas of low-stress gradient and never near regions of maximum stress or deflection.

Improper connections between elements of different types can cause errors. Solid element types, for example, have only translational nodal degrees of freedom. If solid elements are interconnected with beam or plate/shell type elements, which have rotational degrees of freedom, in addition to translational ones, care must be taken to allow for the transfer of moments if that is what is intended. If this is the case, then it is best accomplished with linear constraints or multipoint constraints. In case the program does not offer such options, the beam (or plate) can be artificially extended through the solid elements. Figure 3/3-8 illustrates the problem and a solution for the sample problem.

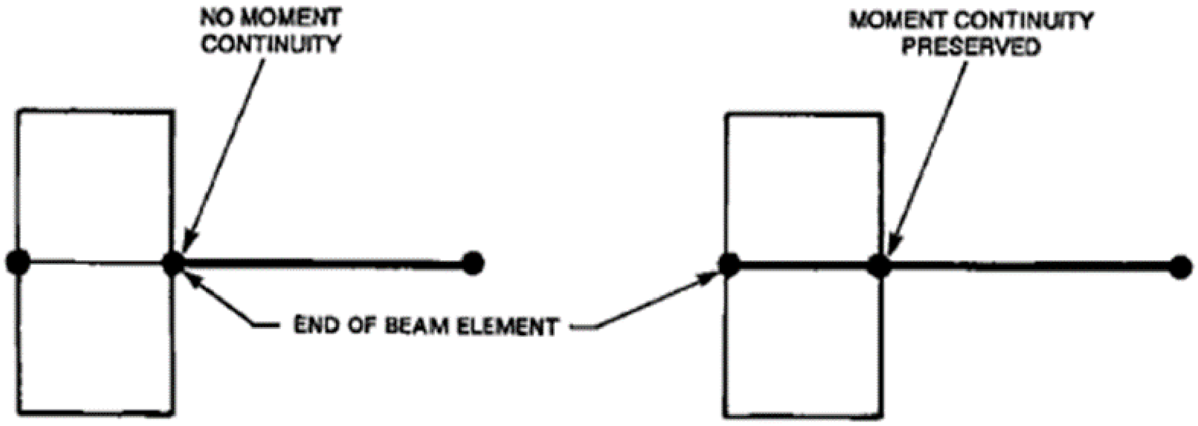


Figure 3/3-8 Connecting Elements with Different Nodal Degrees of Freedom

Most flat plate/shell element formulations do not have a shape function for the rotational degree of freedom about a normal to the surface of the element. Hence, in- plane rotational stiffness is not modeled. Some programs provide a nominal rotational stiffness to prevent free rotation at the node. Other programs use certain formulations to improve this aspect of performance but at the cost of the presence of spurious modes. The user should be aware of the possible limitations in the program that is being used when modeling situations in which moments are to be transferred into the plane of assemblages of flat plate/shell elements. The problem and one possible solution are illustrated in Figure 3/3-9.

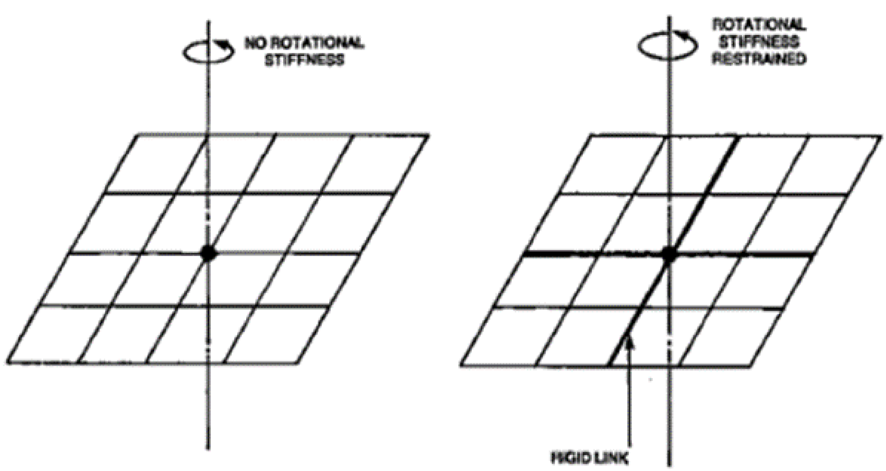


Figure 3/3-9 Modeling In-Plane Rotational Stiffness of Membrane Elements

3/3.3.3 Stiffness Ratio of Adjacent Structure

In modeling complex structural assemblies, there is a possibility of constructing models where adjacent structural elements have very different stiffnesses. These types of stiffness combinations can cause ill-conditioning of the equilibrium equations, which can severely degrade results. The transitioning guidance given above avoids this problem in models that use two or three-dimensional elements. For truss and frame structures, a different approach is required. To prevent large numerical errors in these cases, stiffness ratios of the order of 10^4 and more between members making up a model should be avoided. This is admittedly a conservative number. More realistic guidance can be obtained by undertaking tests.

The problem of stiffness mismatch is most severe in structures where a relatively rigid portion of the structure is supported on the flexible structure. In such cases, the deflections in the rigid portion are due more to rigid-body movement rather than elastic distortion. In these cases, it is suggested that the stiff portion be treated explicitly as a rigid body using rigid links, rigid regions, constraints, or combinations of these approaches.

3/3.3.4 Joint Modeling

Modeling of joints (e.g., fasteners and welds) in FEA simulation is a debatable procedure. In general, joints can exhibit highly nonlinear behaviors. Different design codes recommend different means to mimic the joint in the model. The analyst needs to decide which modeling approach should be adopted for different types of joints. The following is a discussion of a few key topics in joint modeling.

3/3.3.4.1 Mechanical joints

The mechanical joints are one of the most widely used in assemblies. They include fasteners such as bolts and rivets. In FEA modeling, mechanical joints are assumed to carry the full shear load between the surfaces as long as it is under the peak load. In some FEA codes, the element type for mechanical joints may also include other parameters such as tensile or shear failure strength. Note that it is needed for the analyst to understand the basic engineering principles and verify the input variables manually. The tensile or shear loads can be included in the FEA output by adopting a cross-sectional evaluation feature [28,105].

3/3.3.4.2 Welds and adhesive joints

Welds are widely used to connect sheet metal components (e.g., spot or seam welds).

Rigid or beam elements can be used to model the spot welds. The former is more straightforward in modeling but may cause the singularity (ill condition) of the stiffness matrix, due to the lack of drilling degrees of freedom. The latter naturally eliminates the singularity and can record the variations of forces by assigning length, material properties, and cross-section profiles.

Rigid, beam, or shell elements can be used to model seam welds. If the rigid or beam elements are used, there is no degree of freedom between neighboring welds. The number of rigid or beam elements needs to be calibrated based upon the test data with the same geometric profiles and weldment properties. If the shell elements are used, extra cares are needed to make the modeling meet meshing quality criteria. Note that sometimes it is difficult to capture the peak stress caused by irregular geometry of the actual welds. The nominal values can be adopted to compare with the testing data for validations. If the sharp notch is presented, it is recommended to carry out the structural integrity analysis associated with complicated meshed welds.

3/3.3.4.3 Pre-loaded joint

In general, joints are pre-loaded either from the mechanical assembly or residual stress/strain. There is a need to properly account the pre-load in a joint. In a common way, it requires a detail solid model in the joint area to represent the joint. However, in the recent development of FEA codes, there are novel element types that allow the analyst to adopt that can consider pre-loads from a thermal shrink or bolt pretension on a segment or cross-section of the fastener [28]. The magnitude of thermal or pretension can be adjusted to achieve the desired pre-load after the preliminary run. It should be noted that in some nonlinear analysis or fatigue analysis, the detail solid model is required aiming to capture the high stress or strain gradient.

3/3.3.5 Advanced Analysis Mesh Parameters

3/3.3.5.1 Impact/Collision Problems

In the studies of impact or collision problems, large deformations and/or failures may be presented, and therefore the mesh design should be adequately detailed to capture them. In general, the aspects of the mesh design for these problems are consistent with those described in the previous sections. However, extra cares are needed for specific problems. For example,

1. For ductility evaluations, several elements should be modeled in the yield zone to capture the strain gradient [106];
2. For stability evaluations, minimum 3 to 6 elements per expected half wave are required to capture the buckling modes [106];
3. For the application of the automatic mesh refinement, the mesh design and element type should be compatible with the algorithm of mesh refinement;
4. For failure analyses, the mesh design should be refined along with the expected failure or crack paths, and the element type should be compatible with the material models associated with damage formulations;

For the mesh refinement study, the analyst needs to verify that the mesh design is adequate for representing all the failure modes except the above-mentioned aspects. In Ref. [40,106,107], it is recommended to rerun the analysis with half the element size and check the difference of results. The identical results indicate a reasonable convergence of the mesh design.

3/3.3.5.2 Fracture Analysis

Different from the above-mentioned engineering problems, FEA studies on fracture and damage mechanics use unconventional continuum mechanics with singularities and discontinuities included. In these problems where cracks or flaws are included, it is required to have a relatively more profound engineering knowledge and analyst intervention regarding the mesh design.

This section reviews some techniques in constructing the mesh for crack-related analysis. However, it should be noted that such kind of analysis sometimes applies user-defined materials and user-defined elements. It is challenging to cover all the aspects that the analyst may face, and it is recommended to consult the published literature that covers numerous theoretical knowledge and practical aspects with respect to mesh design for crack problems [47,90,96,99,101,108].

In general, there are two different sets of modeling techniques regarding the presence of crack propagating, i.e., stationary cracks and propagating cracks. When modeling the crack faces (a pair), the nodes along each crack face are usually matched or coincide in terms of coordinates, see Figure 3/3-10(a). Note that some FEA preprocessors may merge these coincide nodes automatically, and one technique to avoid such a limit is to model a small gap between faces, see Figure 3/3-10(b). Figure 3/3-14 schematically shows an example of the half single-edge notched tension (SENT) specimen under tension, where two crack faces with a gap of 25 micros are modeled [109]. It should be noted that if the FEA output is related to crack mouth or crack tip opening indexes, the analyst should use the relative values instead of the absolute values.

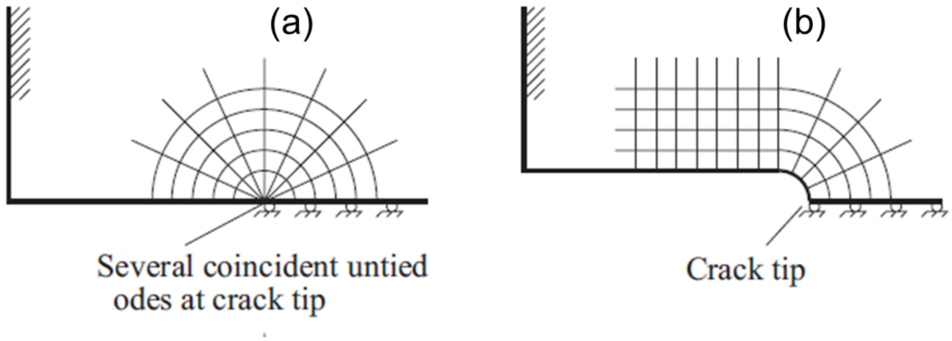


Figure 3/3-10 Meshes for the Analysis of Blunting and Local Crack Opening Criteria

In the stationary crack problems, the crack driving forces (e.g., stress intensity factor – K and energy release rate – G, J, CTOA, CTOD) are of interest for the subsequent engineering analysis, e.g., fatigue life and structural integrity assessment [12,63,110–116]. When modeling the vicinity of the crack tip, it is recommended to use either quadrilateral or brick elements for 2D and 3D problems, respectively, [47]. At the crack-tip, the quadrilateral or brick elements are usually collapsed and degraded to triangle or wedge, respectively (see 3-color highlighted circles in Figure 3/3-11).

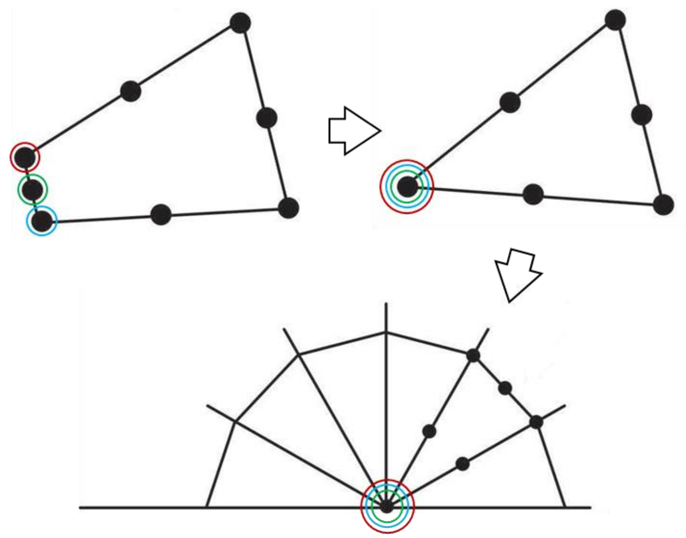


Figure 3/3-11 Nodes of Quadrilateral Element Collapse Coincident to Model Crack Tip [47]

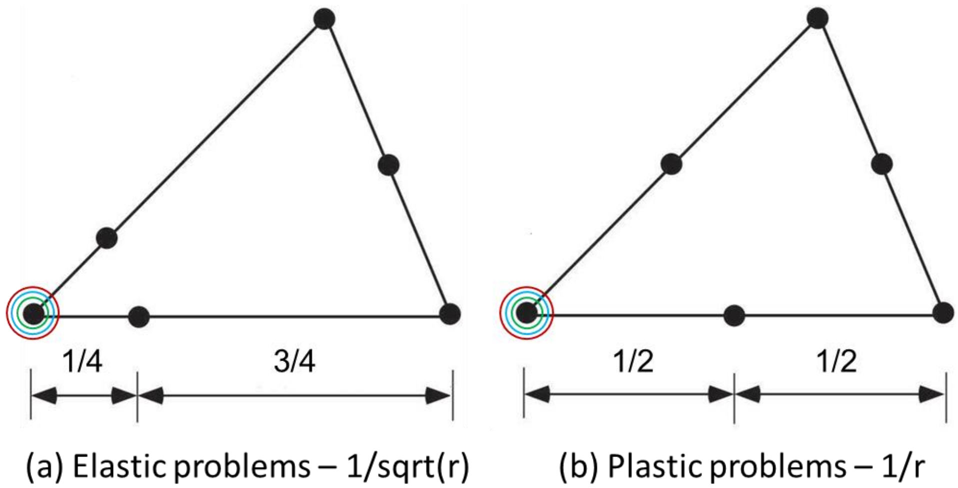


Figure 3/3-12 Crack Tip Elements

In elastic problems, the high order element should be adopted, and the nodes of the crack front are moved to the ¼ points (25% of element length from the crack tip), as shown in Figure 3/3-12(a). Such modification of elements can mimic the singularity strain field in the vicinity of the crack tip (i.e., *f*(1/sqrt(*r*))) and enhance the numerical accuracy. On the other hand, if plastic zones are formed in the vicinity of the crack tip, the singularity of the strain field becomes *f*(1/*r*), and there is no need to adjust the location of mid-side nodes (see Figure 3/3-12(b)). Note that in [47], the author pointed out that the side benefit of plastic singular element design is that it can assist in capturing the crack-tip opening displacement by computing the intersection of 90 vertices with crack faces. Figure 3/3-13 schematically shows the CTOD calculation.

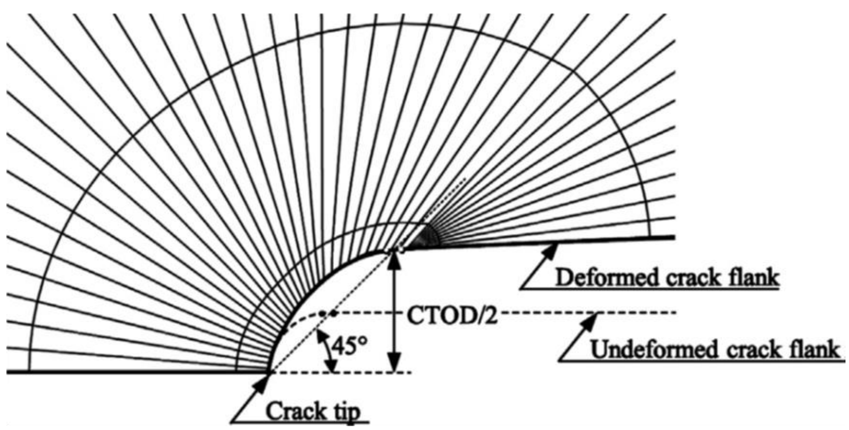


Figure 3/3-13 Crack Tip Opening Displacement (CTOD) At the Intersection of 90° vertex with Crack Faces

The most efficient mesh design for the crack tip region is the "Spider Web" configuration (see Figure 3/3-14 – SENT), which consists of concentric rings to quadrilateral elements focusing toward to the crack tip. Because the stress and strain field of crack tip regions vary drastically, it is recommended to use the finest mesh close to the crack tip and gradually decrease the mesh density towards the boundary of the spider-web. Note that the analyst should carry out a sensitivity study on the mesh design and verify the FEA output by comparing it with the benchmark examples.

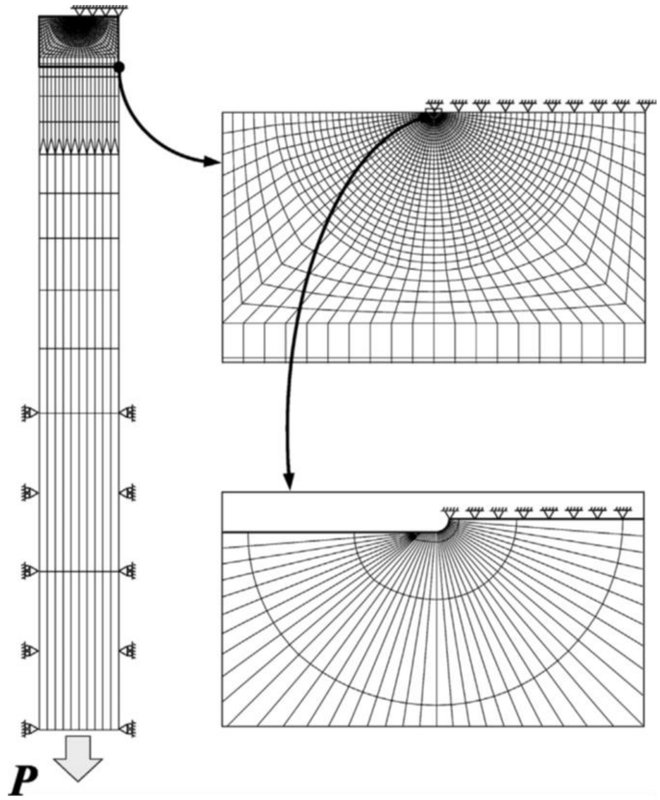
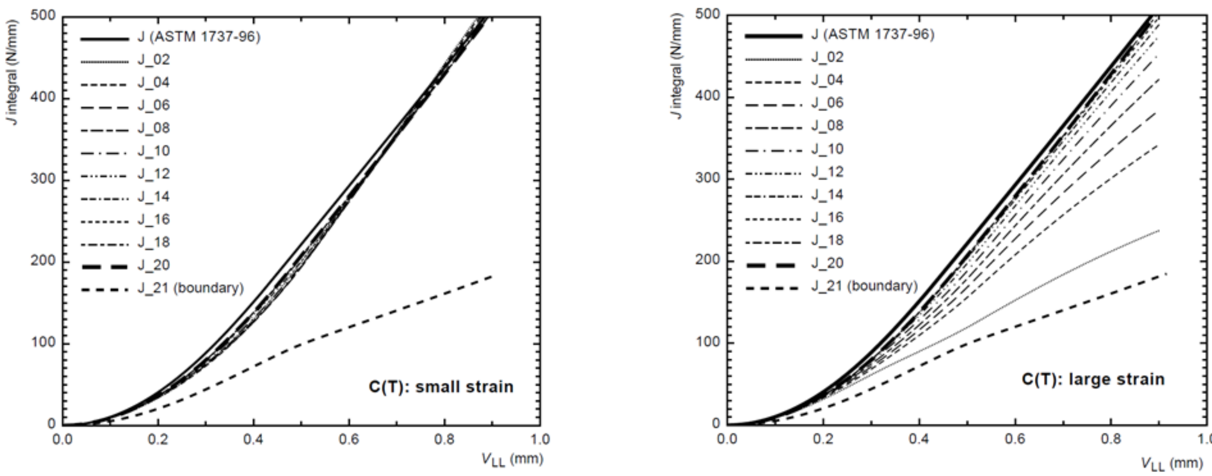


Figure 3/3-14 Blunt Crack Face of A Single-Edge Notched Tension (SENT) Specimen with Spider-Web Meshes [117]

In general, the variable of elastic problems (e.g., K and G) can be accurately captured based upon relatively coarser mesh design. It is mainly because of the use of advanced numerical approaches, such as contour integral and virtual crack extension methods, which is a domain-based approach and eliminates the requirements of solving the local crack-tip field. On the other hand, the variables of elastic-plastic problems (e.g., J and CTOD) can be mesh depended, and extra care is needed. First, the mesh design must be sufficiently refined in the vicinity of the crack tip, and proper strain formulations (i.e., small or finite/large strain) should be adopted, aiming to capture the stress, strain, and deformation with the presence of the large or full yielding. The analyst needs to have strong engineering knowledge regarding the theoretical definitions of the variables investigated and understand the limits of FEA codes adopted. For example:

1. The J-integral is essentially a far-field term to capture the energy release rate. The J values should be estimated from rings far from the first ring, and those from far-field rings are also independent of strain formulations adopted. Figure 3/3-15 shows the variations of J results as the increase of loading, where J is evaluated from different rings and different strain formulations [118]. In the figure, J values from ring No.20 are almost identical to the benchmark values (e solid line) calculated from the ASTM standard [53].

2. CTOD is strongly depended on the mesh design and strain formulations because of its definitions (e.g., 90° intersection and GKSS – $CTOD_5$). Aiming to capture the deformation of crack faces, such as crack blunting, the finite/large strain formulations have to be employed.



(a) Small strain (b) Finite strain

Figure 3/3-15 J-integral vs. Load Line Displacement for a C(T) Specimen [118]

In the propagating crack problems, the crack path(s) and failure mechanism(s) are examined. Note that as discussed in Section 3/2.6, there are different variables to quantify the critical value of crack initiation and propagation. If the critical value is reached or exceeded, a crack starts to grow. There are generally three different techniques to simulate the crack growth, and the mesh should be designed accordingly [119]. These techniques are:

- Node release method based on crack driving forces (e.g., J_c, G_c, $CTOD_c$, and $CTOA_c$) [101,120–122]. Figure 3/3-16 schematically shows this concept and mesh design [68,123,124].
- Cohesive element method [125–127]. Figure 3/3-17 shows the mesh design [96,101,128].
- Constitutive equations based on damage mechanics concepts [129–131].

Note that in such simulations, user-defined elements, and user-defined materials may be employed. It is recommended to design the mesh as well as to control the mesh quality by the following guideline in the FEA code vendor documentation.

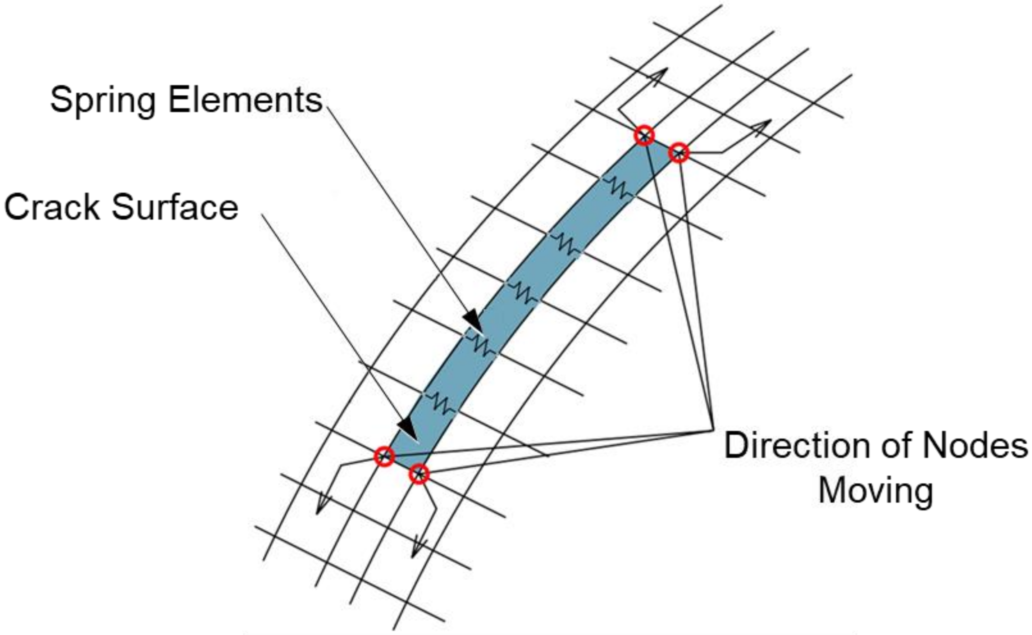


Figure 3/3-16 Spring Elements [123]

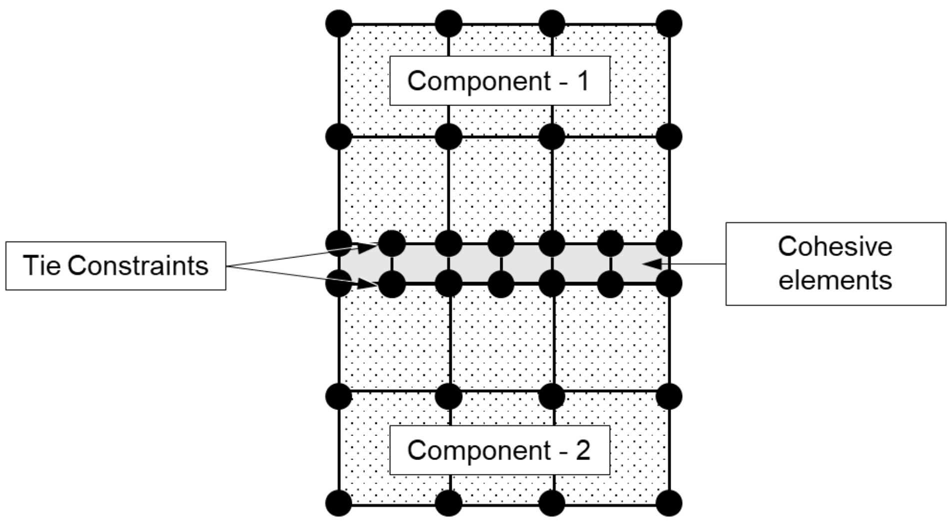


Figure 3/3-17 Cohesive Element [96]

3/3.3.5.3 Fatigue Analysis

In the fatigue analysis of a ship structure, there are three levels of FE models:

1. Global model
2. Partial ship model
3. Local structure model

The mesh design of the local structure model is discussed in the remainder of this section.

The local structure models are used for the investigation of hot spots with high stress or strain gradients, e.g., weldment and free plate edges. The typical geometric details considered in the standards [9,132–135] are as follows:

1. Bracket toe and flange terminations of stiffeners
2. Slots and lugs in the web frames at the intersection with stiffeners
3. Bracket and flange terminations of girder systems
4. Panel knuckles

It is recommended to evaluate the hot spots based on the following rules:

1. Refined meshing with an element size of t × t or 10 × 10 mm, where t refers to plate thickness
2. Minimum 10 elements in all directions from the refined (hot spot) zone to the coarse zone
3. Mesh transition from refined zone to the coarse zone should be done gradually per Section 3.3.4.

A typical example of mesh transition is shown in Figure 3/3-18 [132]. In the figure, an FE model of longitudinal hatch coaming end bracket to deck plating in the hot zone is presented, where the hot spot is designed with t × t mesh to capture the high-stress gradient.

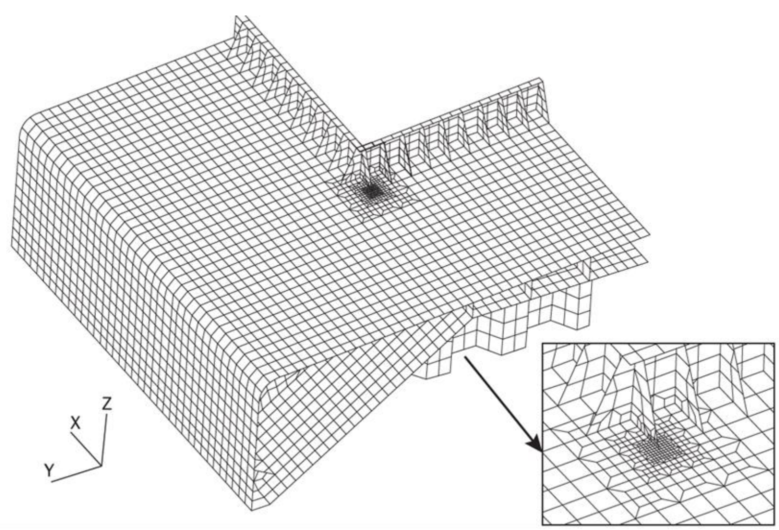


Figure 3/3-18 Mesh Transition Example from DNVGL-CG-0129 [133]

If the stress concentration factors are required in the fatigue life calculation per [134], it is recommended to carry out two levels of analyses:

1. A 50 × 50 mm meshed model to evaluate the semi-nominal stress.
2. A t × t meshed model to capture the stress gradient at the hot spot.

Note that the purpose of evaluating the semi-nominal stress (1) is to determine the local geometric stress flow and effect of cut-outs, web frame-toes, and tripping brackets [134].

An example of a meshed model is shown in Figure 3/3-19 [134]. In Figure 3/3-19 (b), the hot spots are modeled with the mesh density of t × t and web-mesh with four elements in all directions at all relevant hot spots. Figure 3/3-20 shows the stress distributions of the two levels of the meshed model. From Figure 3/3-20, it is clearly seen that the stress variations in the hot spots can only be accurately captured by using the t × t model, where the fine mesh design with 4 layers of elements is used.

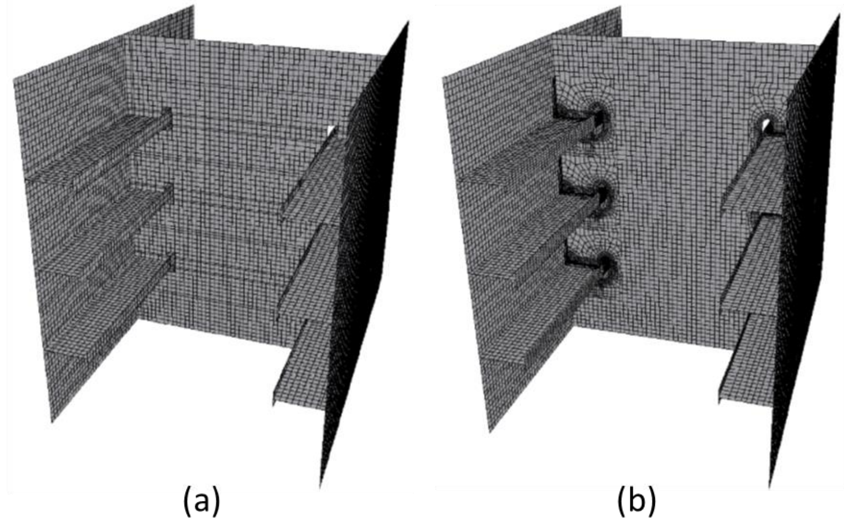


Figure 3/3-19 Two Levels of Meshed Models for Stress Concentration Analysis 50 x 50 mm (b) t x t refinement [134]

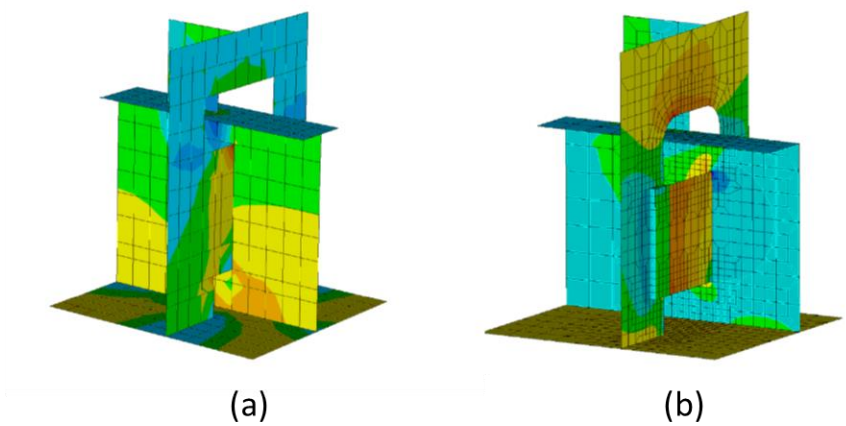


Figure 3/3-20 Stress Distributions of Two Levels of Meshed Models for Stiffener and Lug Details (a) 50 x 50 mm (b) t x t refinement [134]

The analyst may also adopt the submodeling technique to accomplish such kind of multi-level fatigue analyses. Note that it is an approximate solution by assuming that the displacement values from the global model are sufficiently accurate and can be used for the local model.

3/3.3.5.4 Whole Ship

The level of discretization of the global ship model depends on the complexity and size of the model. Ship structure is generally modeled by assembling structural elements such as stiffened panels and stiffeners. A stiffened plate is composed of a plate and its stiffeners, considered as secondary stiffeners. Traditionally in whole-ship mesh design, the nomenclature of the "coarse mesh model" and "finer mesh model" is separated by how a stiffened panel is modeled, as given in [136]. A coarse mesh model referred to an orthotropic plate, equivalent plate, or lumped stiffener model, where stiffeners are integrated into a plate element. A finer mesh model is a model where stiffeners are explicitly modeled as beam elements at their exact geometric location. It is noted that some documents (for example, NSWCCD's "Modeling Guidelines Full-ship FEM Development" [137] and ABS's Rules for Building and Classing Marine Vessels [64]) referred to the traditional "finer mesh model" as coarse mesh model.

In a whole-ship coarse mesh model, the most common and easiest way to represent the equivalent plate element is by lumping the plate internal stiffeners to plate element edges. In the lumping process, shell plates are modeled as plate elements, and all secondary stiffeners are modeled as beam elements. The stiffeners located inside the plate element are put at the edges of the plate element to reduce the number of nodes, as illustrated in [136] and shown in Figure 3/3-21.

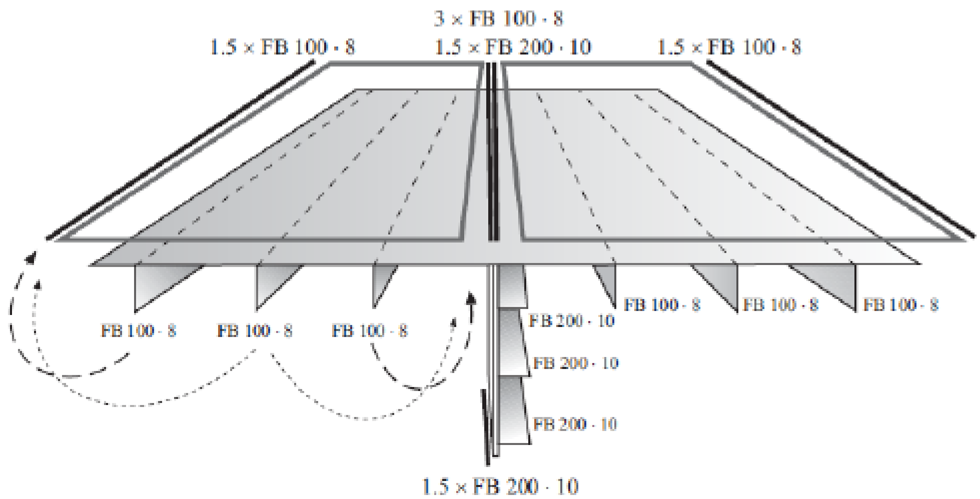


Figure 3/3-21 Coarse Mesh Stiffener Lumping

Also, in a whole-ship coarse mesh model, each stiffener causes an increase in the cross-sectional properties of the lumped beam elements. Since the stiffeners' cross-sectional area and direction are preserved, the actual hull girder cross-sectional properties are well represented. Therefore, the hull girder global response, especially under sagging and hogging conditions, should be close to the response of a finer mesh model. However, it is difficult to post-process the limit states of plates and stiffeners for a lumped model because many of the plate-stiffener geometry properties, such as the space between the stiffeners, are lost during the lumping process. In addition, accurately counting the number of stiffeners in a plate element and properly splitting the stiffeners to the edges of the plate element are also tedious and error-prone. Finally, it is difficult to review and check the lumped model graphically. Hughes [66] suggested the orthotropic plate technique, where the stiffeners are blended with the plating so that the plate has different stress-strain properties in two directions. Since the stiffened panel is represented as one orthotropic plate, the approach does not allow analyzing plate and stiffener separately, and the obtained normal stress is the average of the plate and the stiffener. Avi et al. [138] proposed a three-layer laminate element to model an equivalent stiffened panel, where the first layer represents the plate, the second layer the stiffener web, and the third the stiffener flange. Both the laminate equivalent plate and orthotropic plate method address most deficiencies of the lumped stiffener method and are more accurate. However, these methods are not readily available in general finite element packages such as Nastran and ANSYS.

A whole-ship finer mesh model is a model where stiffeners are explicitly modeled as beam elements at their exact geometric location. Transverse frames and longitudinal girders are generally modeled as beams, but in the case of larger beams such as web frames or longitudinal girders in the way of propulsion foundations, they might be modeled with their webs as quad elements and their flanges as either quads or flat bar beams. The longitudinal mesh is generally 2-4 elements between transverse frames. The finer mesh model approach is well suited to today's general tendency in both ship design offices and shipyards, which is to develop a model for the unified design process and to share the model information for numerous purposes. A finer mesh model provides higher fidelity for a full-ship structural analysis. However, it takes more resources to model, analyze, and post-process.

3/3.3.5.5 Vibration

The recommended model mesh size is dependent on the type of analysis being performed and the size of the ship. Hull-Girder vibration analysis on a large ship can have a mesh size of 3 to 4 stiffener spacings per ABS in [14]. However, a hull girder vibration analysis on a much smaller 54 ft catamaran is recommended to have a mesh size of 4 to 8 elements between each stiffener per SSC in [139].

A local model should have a finer mesh. As a general rule, the element size should be sufficiently small to capture the mode shapes expected within the structure.

The curvatures in structural mode shapes in higher modes are more severe than at lower modes, and several nodes (and associated masses) are required to represent the kinetic energy accurately at higher modes. The mesh density of a vibration analysis may require some iteration as natural frequencies are determined, and then the mesh density refined to make sure the structural response is being modeled accurately. Figure 3/3-22 illustrates the difference between a good mesh density and a poor mesh density for a given mode shape.

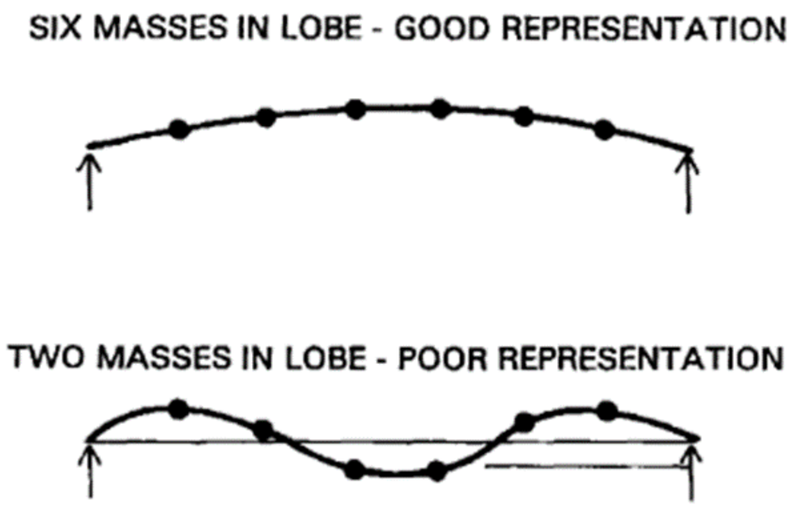


Figure 3/3-22 Assessing Accuracy of Higher Modes

3/3.3.6 Element Shape Limitations

The following is a discussion of element shape aspect ratio, skewness, and warpage.

3/3.3.6.1 Aspect Ratio

The element aspect ratio in a 2D element or a face of a 3D element is the ratio between the longest and shortest element dimensions, as shown in Figure 3/3-23.

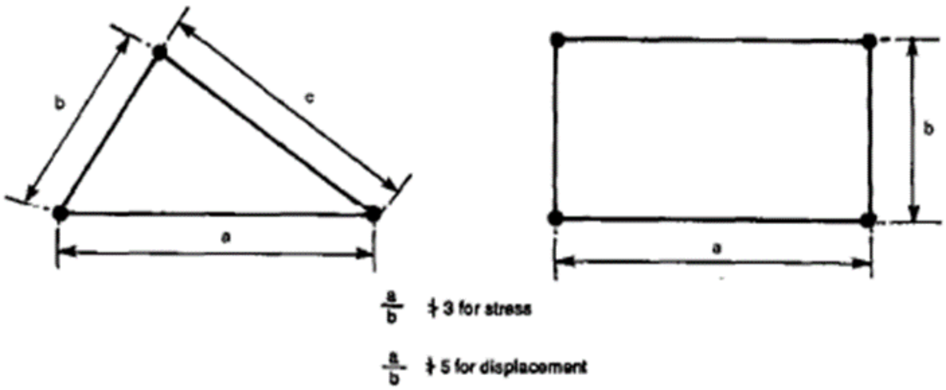


Figure 3/3-23 Aspect Ratio of Plane Elements

A crude rule of thumb that can be used is to limit the aspect ratio of the membrane and bending elements to three for good stress results, and to five for good displacement results. The ideal shape for quadrilateral elements is square and equilateral for triangular elements. Hence, the use of ideally shaped elements is particularly desirable in areas of high-stress gradients. In general, higher-order elements are less sensitive to departures from the ideal aspect ratio than lower-order elements. This observation also applies to solid elements.

Since an element's sensitivity to aspect ratio is dependent upon both element formulation and the nature of the problem, general tests and problem-dependent checks may be justified in cases where element performance is not well known.

3/3.3.6.2 Skewness

Generally, the performance of elements degrades as they become more skewed. Skewing is defined as the deviation of vertex angles (β) from 90° for quadrilateral shaped elements, and from 60° for triangularly shaped elements, as shown in Figure 3/3-24. For quadrilateral elements, angles greater than 135° and smaller than 45° are not recommended, i.e., $45° \leq \beta \leq 135°$. The limiting range recommended for triangular elements is 45° and 90°, i.e., $45° \leq \beta \leq 90°$. Skewed quadrilateral elements shaped more like parallelograms generally perform better than more irregularly shaped ones. The check of skewing angels on all faces of a 3D element is performed in the same fashion.

3/3.3.6.3 Warpage

When element nodes are not in the same plane, the element is warped, as shown in Figure 3/3-24. This is undesirable, and the degree to which this impairs the performance of plate elements depends on the element formulation. The warpage in a 2D element can be calculated by splitting a quadrilateral element into two triangular elements and evaluating the angle between two planes of triangular elements, where the maximum angle refers to the warpage of an element. For 3D elements, the warpage value on all faces of an element is evaluated in the same fashion. The best guidance in regard to limiting levels of warping is contained in the particular FEA program's user manual. The use of triangular elements is an option where the curvature of the structure is high.

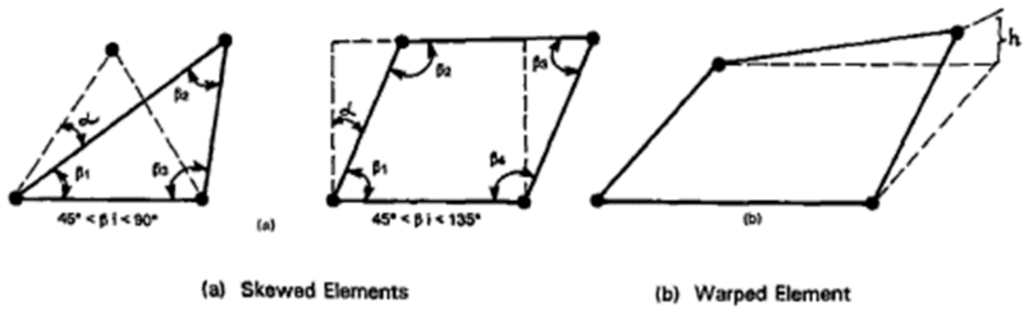


Figure 3/3-24 Element Shape Limitations

3/3.3.7 Mesh Quality Control

The mesh quality is one of the critical subjects in FEA simulations, which directly affects the results of the calculations. For example, a model with poor element quality may need unexpected iterations and time to obtain converged results. This is because, as discussed in Section 3/3.6.2.3, the time-step is directly calculated from the minimum length of elements in an explicit time-integration based analysis. Although there are typically discussions about this in FEA training and lectures, it is typically not fully understood by junior engineers. That is mainly because the element quality is a complex topic, and in nature, it is a ‘relative’ approximation that needs analyst’s engineering judgments. Therefore, it is recommended to have in-house validated mesh quality criteria that can meet the requirements of different analysis problems. The analyst should always check the mesh quality in the meshing process. The key aspects related to the mesh quality and the corresponding definitions are summarized as follows:

1. Penalty value - A weight factor to evaluate whether the element meets the criteria.
2. Max / Min length - Maximum and minimum dimension of the element. Note that the minimum length of the element is crucial to determine the time-step using the explicit solver.
3. Max / Min Angle Quadrilateral / Triangular - Individual maximum and minimum angles of the element.
4. Aspect Ratio - See Section 3/3.3.6.
5. Warpage Angle - See Section 3/3.3.6.
6. Skewness -See Section 3/3.3.6.
7. Jacobian Ratio - A scale factor to describe the transformation of the coordinate system.
8. Distortion - Evaluate the element topology from determining of Jacobian Ratio and areas in local/global coordinates.
9. Stretch - A ratio to describe the maximum topology dimension in terms of the minimum topology dimension.
10. Chordal Deviation - Determine how well curvatures have been modeled.
11. % of trials - Percentage of triangular elements in the mesh design.
12. Taper - Describe the angular distortion of a quadrilateral shell element.
13. Tetra collapse - The normalized height of a tetrahedron element.
14. Volumetric skew (for tetrahedron elements) - The normalized difference of volumes between actual and ideal tetrahedron elements for a given spherical domain passing all corner nodes.

In modern FEA codes, the pre-processor may include some or all of the above-mentioned aspects. In addition, the pre-processor may also provide a series of values for each aspect and divided them into four or five different commentary categories, e.g. (1) **Ideal**, (2) **Good**, (3) **Warn**, (4) **Fail**, and (5) **Worst**. A set of criteria examples for different analysis problem has been presented by Altair HyperMesh [15] and shown as follows:

1. For general problems with an average element size of 50 mm, the recommended criteria are shown in Table 3/3-1.
2. For vibration problems with an average element size of 50 mm, the recommended criteria are shown in Table 3/3-2.
3. For fatigue problems with an average element size of 50 mm, the recommended criteria are shown in Table 3/3-3.
4. For plasticity and crash problems with an average element size of 50 and 100 mm, the recommended criteria are shown in Table 3/3-4 and Table 3/3-5, respectively.

Table 3/3-1 General Problems with Average Element Size of 50mm

#	Criterion	Ideal	Good	Warn	Fail	Worst
0	penalty value	0	0	1	2	10
1	min length	50	41.65	33.2875	24.9375	22.85
2	max length	50	66.0625	82.125	87.5	125
3	aspect ratio	1	2	4	5	10
4	warpage	0	5	15	20	40
5	max angle quad	90	110	140	150	165
6	min angle quad	90	70	40	30	15
7	max angle tria	60	80	120	130	155
8	min angle tria	60	50	30	20	10
9	skew	0	10	50	60	75
10	jacobian	1	0.9	0.6	0.5	0.35
11	chordal dev	0	0.3	0.8	1	2
12	% of trias	0	4	6	8	12
13	taper	0	0.2	0.5	0.6	0.9

Table 3/3-2 Vibration Problems with Average Element Size of 50mm

#	Criterion	Ideal	Good	Warn	Fail	Worst
0	penalty value	0	0	1	2	10
1	min length	50	38.335	26.665	15	12.085
2	max length	50	62	73	90	113
3	aspect ratio	1	2	4	5	10
4	warpage	0	4.5	13.5	18	36
5	max angle quad	90	110	140	150	165
6	min angle quad	90	70	40	30	15
7	max angle tria	60	80	120	130	155
8	min angle tria	60	50	30	20	10
9	skew	0	10	50	60	75
10	jacobian	1	0.9	0.6	0.5	0.35
11	chordal dev	0	0.3	0.8	1	2
12	% of trias	0	4	6	8	12
13	taper	0	0.2	0.5	0.6	0.9

Table 3/3-3 Fatigue Problems with Average Element Size of 50mm

#	Criterion	Ideal	Good	Warn	Fail	Worst
0	penalty value	0	0	1	2	10
1	min length	50	38.5	26.7	15	12.1
2	max length	50	65	79	100	128.6
3	aspect ratio	1	2	4	5	10
4	warpage	0	3.75	11.25	15	30
5	max angle quad	90	110	140	150	165
6	min angle quad	90	70	40	30	15
7	max angle tria	60	80	120	130	155
8	min angle tria	60	50	30	20	10
9	skew	0	10	50	60	75
10	jacobian	1	0.9	0.6	0.5	0.35
11	chordal dev	0	0.3	0.8	1	2
12	% of trias	0	1.5	2.25	3	4.5
13	taper	0	0.2	0.5	0.6	0.9

Table 3/3-4 Plasticity and Crash Problems with Average Element Size of 50mm

#	Criterion	Ideal	Good	Warn	Fail	Worst
0	penalty value	0	0	1	2	10
1	min length	50	43.33	36.66	30	28.33
2	max length	50	62	73.2	90	112.9
3	aspect ratio	1	2	4	5	10
4	warpage	0	5	15	20	40
5	max angle quad	90	110	140	150	165
6	min angle quad	90	70	40	30	15
7	max angle tria	60	80	120	130	155
8	min angle tria	60	50	30	20	10
9	skew	0	10	50	60	75
10	jacobian	1	0.9	0.6	0.5	0.35
11	chordal dev	0	0.3	0.8	1	2
12	% of trias	0	3	4.5	6	9
13	taper	0	0.2	0.5	0.6	0.9

Table 3/3-5 Plasticity and Crash Problems with Average Element Size of 100 mm

#	Criterion	Ideal	Good	Warn	Fail	Worst
0	penalty value	0	0	1	2	10
1	min length	100	80	60	40	35
2	max length	100	124	146	180	226
3	aspect ratio	1	2	4	5	10
4	warpage	0	5	15	20	40
5	max angle quad	90	110	140	150	165
6	min angle quad	90	70	40	30	15
7	max angle tria	60	80	120	130	155
8	min angle tria	60	50	30	20	10
9	skew	0	10	50	60	75
10	jacobian	1	0.9	0.6	0.5	0.35
11	chordal dev	0	0.3	0.8	1	2
12	% of trias	0	3	4.5	6	9
13	taper	0	0.2	0.5	0.6	0.9

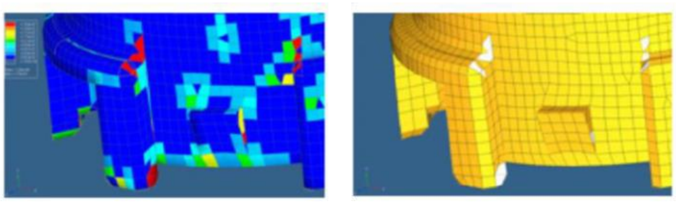
In the above five tables, five columns represent the boundary of ideal, good, warn, fail, and worst, respectively.

1. The ideal set of mesh quality is the optimal value of an element that can achieve.
2. The good set of mesh quality is slightly worse than the ideal set, but it can still progenerate reliable FEA results.
3. The warn set is the intermediate level of mesh quality and should be regarded as the baseline. Extra cares are required from the analyst, especially close to the area of interest.

4. The fail set indicates that the current mesh design and quality are unacceptable for the analysis. There is a need to fix these elements before performing any analysis.
5. The worst set denotes the elements that fail the criteria by a large margin. They have to be fixed immediately.

It should be noted that the criteria presented in Table 3/3-1, Table 3/3-2, Table 3/3-3, Table 3/3-4, and Table 3/3-5 can be used as a reference to prepare criteria for analysis problems with other element sizes. Except the key aspects mentioned above, there are still other factors may drive the mesh quality, such as the all the geometric issues (mentioned in Section 3/3.1), the presence of duplicate elements / nodes, inconsistent element normal directions, the presence of temporary / free nodes, the boundary conditions (e.g. contact pair faces, loading node(s), and fixture node(s)). In addition, it is also recommended to review the mass as well as the center of gravity of the FEA model.

The modern FEA pre-processors may provide advanced features to visually assist analysts in checking the mesh qualities. For example, different contour colors may represent a different level of mesh qualities, e.g., red depicting worst [15] (see Figure 3/3-25).



(a) (b)

Figure 3/3-25 Reviewing Mesh Quality
(a) Contour Plot (for example, red color indicates element fails to meet criteria)
(b) Highlighted and Selected Fail Elements for Subsequent Fixing [15]

3/3.3.8 Numbering

After preprocessing, the FE model should be carefully rechecked before generating the solver input files. Sometimes, due to the frequent import and export operations (especially coping with included files or super element), the ID numbers (e.g., nodes, elements, components, materials, properties, contacts, boundary conditions, and loadings) can be very large, and they might be conflicted or over the digit limit. It is recommended that a project has the numbering rules, and all the analysts use the same numbering rules in the pre- and post-processing.

3/3.4 Substructures and Submodeling

3/3.4.1 Substructuring

The primary reason for using substructuring (also known as super element technique) is to reduce computational effort in the solution process. Substructuring can result in an over 50% decrease in computational time. Substructuring can also reduce the risk of human errors in an iterative design process. Substructuring can also allow a project team to keep certain confidential component information secret from the larger team. This can be done because modern FE codes allow the transfer of boundary matrices (e.g., mass, stiffness, damping, and loads) without showing any geometric information [140]. However, these foregoing benefits have to be weighed against the cost of certain other computations that substructuring requires, which a normal analysis would not entail.

Irons and Ahmed [141] identify three circumstances in which substructuring might be attractive:

1. The same substructure is used repeatedly in the structure.
2. A relatively small portion of a structure may behave nonlinearly.

3. In a major design effort, different teams may be developing different parts of the structure. The use of substructuring would allow substructures of different versions of parts of the structure to be analyzed together. This feature could be very useful during the exploratory and concept design phases of large structures.

The use of substructuring in the FEA of ships is only likely to be attractive for models involving a substantial portion of the ship. If a general-purpose FEA system is used, it is essential to have an understanding of the substructuring technique. Even in the case of design-oriented FEA programs, it is useful to have an appreciation of the technique.

The ease with which substructuring can be undertaken depends on the features available in the FEA code being used. This section will be confined to a broad description of the steps necessary to undertake successful FEA using substructuring, guidelines in using substructuring techniques, and structural configurations where such techniques might be considered.

The basic steps in FEA using substructuring are:

1. Review of the global model and identification of portions of the structure that repeat. Sketch of the global model indicating substructure boundaries. Design of mesh in substructures and determination of boundary nodes.

2. Enter input data. Undertake condensation of substructures and develop substructure stiffness and load matrices.

3. Generation of global stiffness matrix, which, in general, will require combining the reduced substructure matrices with portions of the structure not modeled as substructures. At this point, all the elements of the system equilibrium equations are available.

4. Solve the system equilibrium equations. This run will only yield displacements at substructure boundaries and portions of the model that were modeled in the usual way.

5. The displacements from the global model can be back substituted into the substructure equations, as described below, to yield displacements and stresses within the substructures. This will be repeated for each substructure since, in general, the boundary displacements for identical substructure models will be different.

The following guidelines for substructure analysis are adapted from Steele [142]:

1. Substructures can be generated from individual finite elements, from other substructures, or both.

2. Master nodes to be retained must be identified and specified as input when the stiffness matrices for substructures are calculated. Master nodes include boundary nodes and nodes subject to loads.

3. Nodes on substructure boundaries that will be used to connect the substructure to the rest of the global model must be retained as master nodes.

4. Nodes constrained in substructures when substructure stiffness matrices are calculated will be constrained in subsequent stages of the analysis. These constrained nodes cannot be released in later stages. However, master nodes can be restrained during the analysis of the global model.

5. For a substructure to be cost-effective, it should be used at least three times (i.e., replicated twice).

The following paragraphs contain a description of static condensation, which is a technique fundamental to substructuring. Also discussed is the two-stage analysis technique, which has found favor with many analysts. This is followed by a summary of recommendations.

3/3.4.2 Static Condensation

In the condensation technique, the number of DOF in a portion of the structure is reduced by condensing out the internal DOF that the remaining active ones being on the boundary. The process is illustrated in Figure 3/3-26. This substructure can be regarded as a special type of finite element, and, indeed, is sometimes referred to as a superelement. The mathematics of the process is relatively simple. The equilibrium equations of the substructure with all its DOFs intact is partitioned as follows:

$$\begin{bmatrix} k_{rr} & k_{rc} \\ k_{cr} & k_{cc} \end{bmatrix} \begin{Bmatrix} \delta_r \\ \delta_c \end{Bmatrix} = \begin{Bmatrix} f_r \\ f_c \end{Bmatrix}$$ ***Equation 3/3-1***

in which the subscripts *r* and *c* refer to DOFs to be retained and condensed out respectively. An expression for δ_c can be extracted from the lower partition, which can then be substituted in the upper partition to yield:

$$\left([k_{rr}] - [k_{rc}] [k_{cc}]^{-1} [k_{cr}] \right) \{\delta_r\} = \{f_r\} - [k_{rc}] [k_{cc}]^{-1} \{f_c\}$$ ***Equation 3/3-2***

or in more compact form:

$$[K_C]\{\delta_r\} = \{F_C\}$$ ***Equation 3/3-3***

Where

$$[K_C] = [k_{rr}] - [k_{rc}] [k_{cc}]^{-1} [k_{cr}]$$ ***Equation 3/3-4***

and

$$[F_C] = \{f_r\} - [k_{rc}] [k_{cc}]^{-1} \{f_c\}$$ ***Equation 3/3-5***

The equilibrium equations given by Equation (Equation 3/3-4) can be solved in the usual way. If required, displacements internal to the substructure can be recovered by static condensation of Equation (Equation 3/3-1) using the Gaussian reduction procedure. Static condensation amounts to eliminating selected variables using the Gaussian reduction procedure. It is important to note that no approximation is involved in this process. The condensed-out DOFs are often called slave DOFs, and the retained DOFs are called master DOFs.

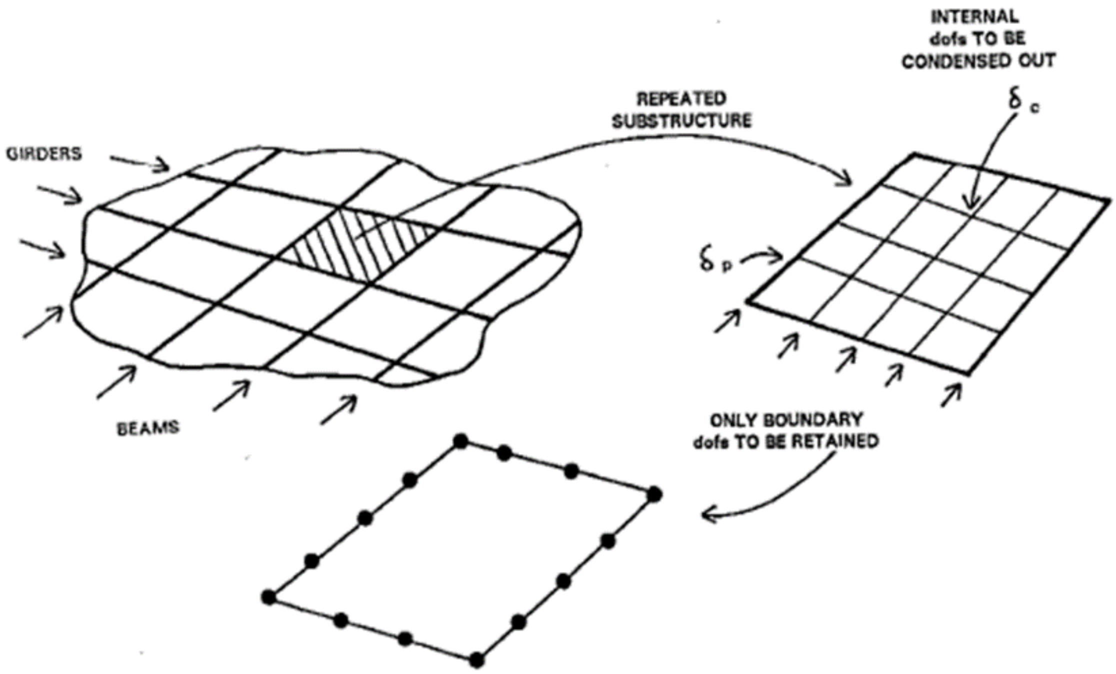


Figure 3/3-26 Schematic Illustration of The Static Condensation Process

3/3.4.3 Two-Stage Analysis

In cases where local mesh refinement is required, a two-stage analysis may be justified (see [142] for practical aspects of two-stage analyses). The first stage of this technique involves the analysis of a coarsely meshed global model. The local area of particular interest is remeshed using a finer mesh and reanalyzed using prescribed displacements at the boundary of the refined model as boundary conditions. The prescribed displacements are taken from the global analysis. The process is illustrated in Figure 3/3-27. The applied loading, i.e., stresses from the global analysis translated into pressure loading for the refined model, can also be used as boundary conditions. However, the use of displacements as boundary conditions is a more common practice since it eliminates the need to provide additional restraints for sufficiently supporting the model.

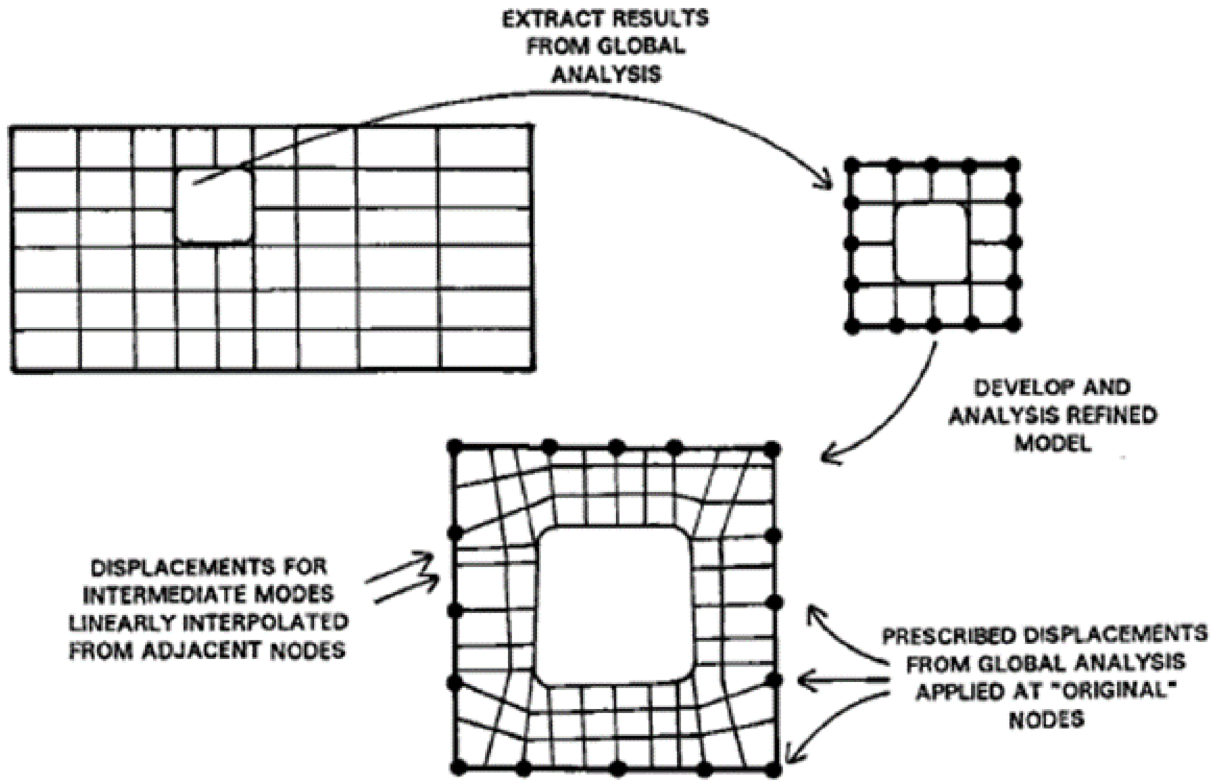


Figure 3/3-27 Two-Stage Analysis

Design-oriented FEA programs, which model the whole or a substantial part of a ship, suit this technique. The displacements from a model developed employing such programs can be used as prescribed boundary conditions for a local fine-mesh model.

In general, there will be several nodes on the boundary of the refined mesh model that are not modeled in the global model. Therefore, prescribed displacement values are only available for boundary nodes that exist in the global model. The practice is to assume a linear variation in displacement, interpolated from the displacements from the global model, for intermediate nodes. This observation is suggestive of where the appropriate position for the boundary might be. Ideally, boundaries should be placed in areas where gradients in displacement are small. A comparison of undeflected and deflected plots of the global model will yield this information.

A finer finite element model is generally more flexible than its coarser equivalent. Hence, there will be a tendency to underpredict the stresses in the refined model when using displacements generated in the global model. It is possible to correct approximately for this tendency using a procedure described by Cook et al. [89]. The procedure requires the computation of the nodal loads produced by the prescribed boundary displacements. The nodal loads for the local area in the global model are given by:

$$\{F_g\} = [K_g]\{\delta_g\}$$

in which K_g, δ_g, and F_g are the stiffness matrix, displacements, and calculated forces pertaining to the degrees of freedom associated with the nodes on the boundary of the local area. The corresponding expression for the refined model is:

$$\{F_r\} = [K_r]\{\delta_r\}$$

The subscript "r" refers to the refined model. Note that only the nodes common to both, the local area in the global model and refined model, are included in the above expressions.

Once the forces for both cases have been derived, the vector norms for these quantities are calculated. The norm is a measure of the "size" of vector or the size of the nodal loads. There are many types of norms, but for present purposes, the following version is recommended:

$$|F| = \left(\sum_{i=1}^{n} |F_i|^2\right)^{1/2}$$

where F_i refers to the value of the nodal load and n is the number of degrees of freedom on the boundary that is common to both the local area of the global model and the refined model. The ratio of the norms for both the cases is calculated to yield a factor as follows:

$$Factor = \frac{F_g}{F_r}$$

This factor, which usually exceeds unity, when applied to all stress results from the refined model, approximately corrects for the overstiffness of the global model results.

The convenience with which this technique can be applied will depend on the FEA software being used.

3/3.4.4 Superelement Generations

The FEA codes Nastran (e.g., MSC and NX) and Optistruct have three different methods to generate superelements [95,140]:

1. Static condensation superelement - This method reduces the elastic FE model and load vectors (e.g., points and pressure) to the interface degrees of freedom. Note that from this method, the stiffness matrix is accurate, but that of the mass matrix is obtained based on an estimated approach [140].
2. Dynamic reduction superelement - This method reduces the elastic FE model to the interface degrees of freedom and a set of normal modes, which are generated from static modes and normal modes analyses.
3. Component dynamic superelement - This method is efficient for models under iterative loading

The analyst should plot elements to schematically show the boundaries and size of FE models. Figure 3/3-28 shows a satellite model represented by using the superelement technique [140], where the RED and BLUE beams are plot elements to show the connectivity between components.

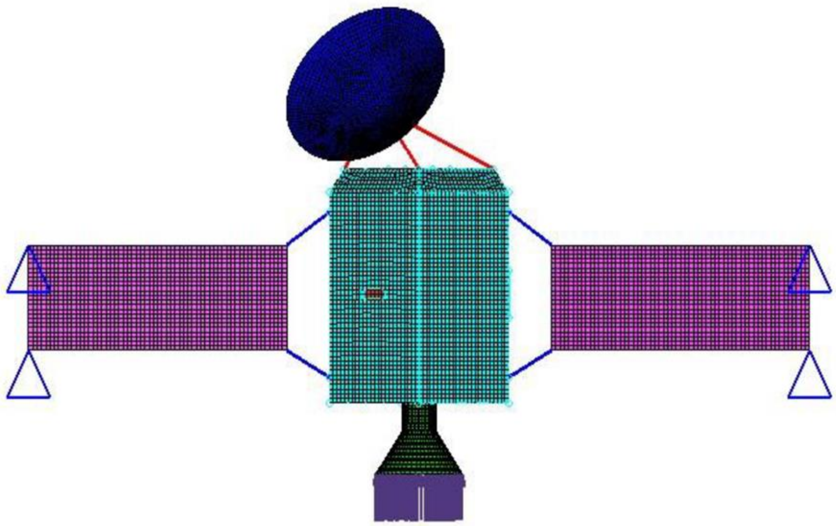


Figure 3/3-28 Use Superelement Technique to Represent a Satellite [15]

3/3.5 Loads and Boundary Conditions

The task of selecting appropriate boundary conditions for the model is often challenging. Generally, the support condition assumed for the degree of freedom concerned is idealized as completely rigid or completely free. In reality, the support condition is usually somewhere in between, and in some special cases, there is no fixture (e.g., satellite with the inertia relief condition).

Several techniques are used to minimize the impact on the analysis of the assumptions made in boundary conditions. The most popular is to develop models large enough such that the area of interest is sufficiently remote from the boundary. It is also the practice to make conservative assumptions so that the results will represent upper bound solutions.

The best guide for determining the extent of structure to model and determining the locations for boundaries are natural structural restraints or rigid or stiff supports such as major structural bulkheads, vertical pillars, and columns, or other structural components such as deep fabricated beams and girders.

It is possible to simulate various types of symmetry, antisymmetry, and axisymmetry by applying the appropriate boundary conditions. These and other topics related to boundary conditions are discussed in greater detail below.

3/3.5.1 Minimum Support Conditions

For certain models, it is necessary to provide minimum support for the structure. A good example of this is hull girder modeling in which the structure is, in reality, supported by the pressure distribution on the hull. In FEA modeling, a structure with self-equilibrating forces, without any supports, is not admissible except inertia relief cases. Without proper support or with the presence of rigid body motion, the equilibrium equations would be singular and, therefore, not solvable.

Models in a plane have three DOFs, and hence need to have two translations and a rotation constrained. Care is needed to avoid the possibility of rigid body motion. These principles are illustrated in Figure 3/3-29. Models in three- dimensional space need three translations and three rotations constrained, i.e., 6 DOFs. Examples to illustrate the minimum support conditions required are provided in Figure 3/3-29.

3/3.5.2 Boundary Conditions for Simulating Symmetry

Many structures have one or more planes of symmetry. It is possible to take advantage of this in FEA, and model just one portion of the structure. Through various devices, it is possible to analyze structures with a plane of symmetry but subjects to nonsymmetric loads. Such approaches are used to reduce modeling and computational effort.

In engineering applications, the most commonly encountered types of symmetry are reflective symmetry, rotational symmetry, and inversion symmetry, as shown in Figure 3/3-30.

In engineering problems, the characterization of symmetry requires not only geometrical symmetry, but also symmetry with respect to other factors (e.g., material properties, loadings, fixtures, and contacts (if applicable).

When only part of a symmetric structure is modeled, the symmetric or antisymmetric boundary conditions must be applied at artificial boundaries introduced because of symmetry. If the y-z plane is the plane of symmetry, and Ux, Uy, Uz, and Rx, Ry, Rz are assumed as the x, y and z components of displacement and rotation respectively, the following boundary conditions have to be applied to the nodes on the plane of symmetry or antisymmetry:

Ux = Ry = Rz = 0 - for symmetry

Rx = Uy = Uz = 0 - for antisymmetry

In the case of symmetry, the points lying in a plane of symmetry can suffer no translation out of the plane and no rotation about the in-plane axes. For antisymmetry, the complementary set of degrees of freedom is constrained.

The above discussion has been devoted exclusively to static problems, but free vibration problems (eigenvalue problems) can also exploit symmetry. The calculation of all-natural frequencies and mode shapes of a symmetric structure would require one modal analysis for each unique combination of symmetric and antisymmetric boundary conditions. Note that when only symmetric boundary conditions are applied to the plane of symmetry, antisymmetric frequencies and mode shapes are not calculated.

The conditions for static problems discussed above apply equally to linear (time- dependent) analysis. In addition, if the load is not symmetric or antisymmetric, it will be necessary to decompose the load into symmetric and antisymmetric components and run the problem twice for each case and combine the results.

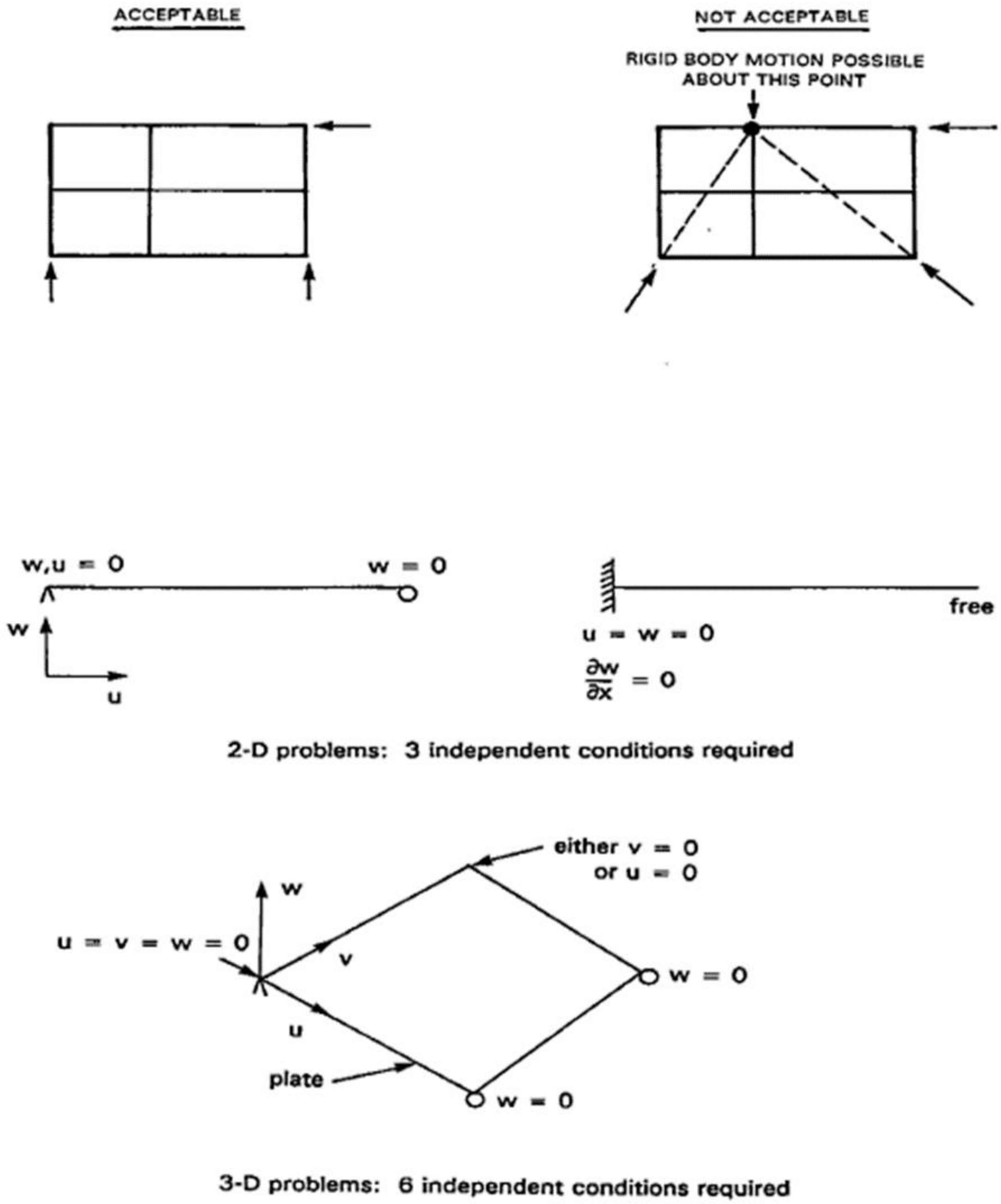


Figure 3/3-29 Minimum Support Conditions for Models

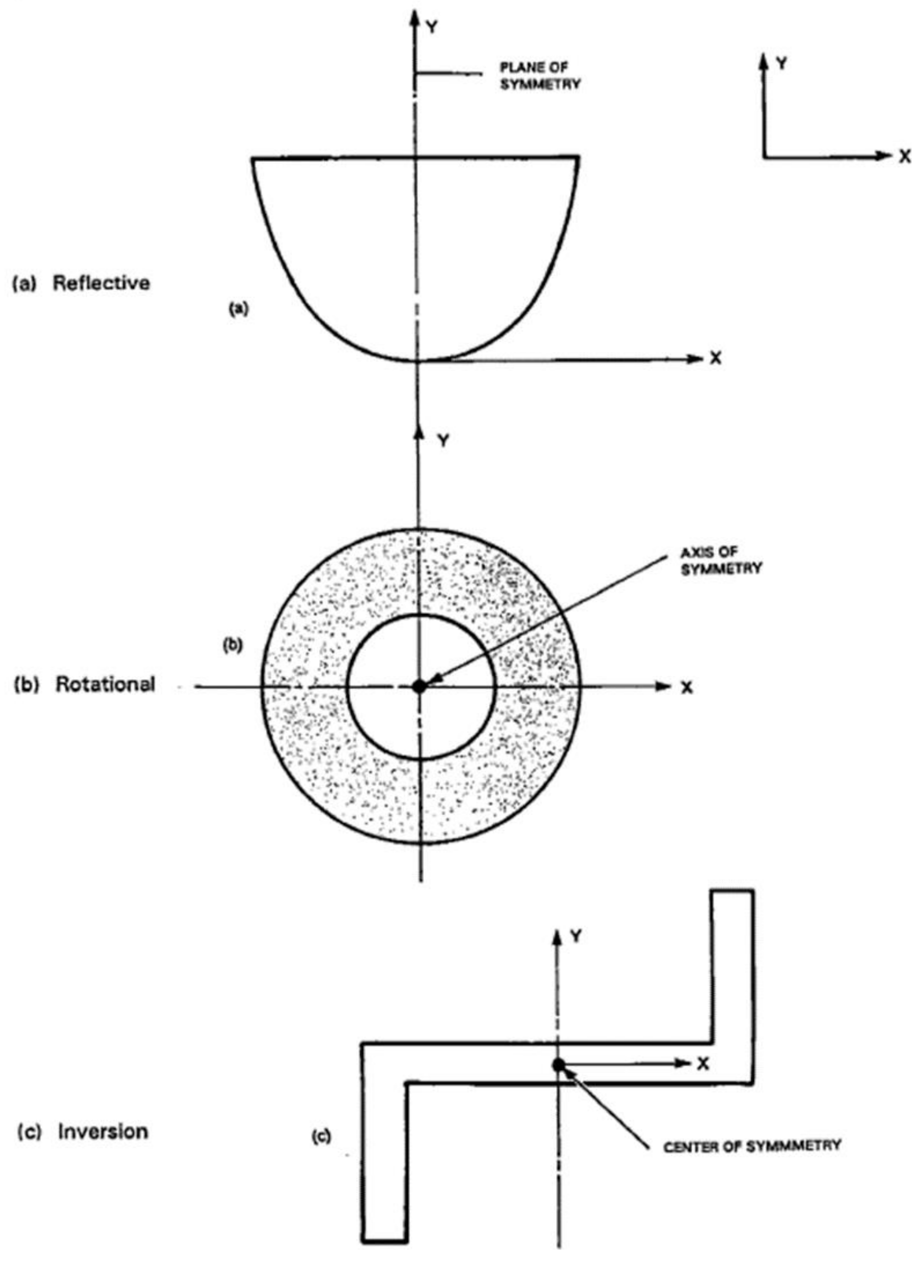
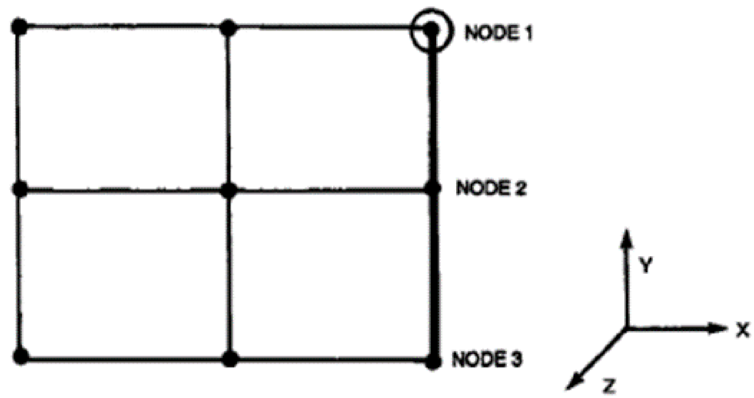


Figure 3/3-30 Different Types of Symmetry

3/3.5.3 Constraints

Constraints are enforced relationships between the DOFs of several nodes. There are many situations in which constraints can be useful modeling devices. Various types are discussed below and illustrated using simple examples. The circumstances in which they may be applied, and limitations in their application, are also discussed.

The simplest form of constraint is when certain DOFs of different nodes are coupled. The coupling can be used to enforce symmetry and to release forces and moments. A simple example is presented in Figure 3/3-31. During analysis, if the independent node is displaced in the y-direction and/or rotates about the y-axis, the dependent nodes are automatically displaced by the same magnitude in the same directions.



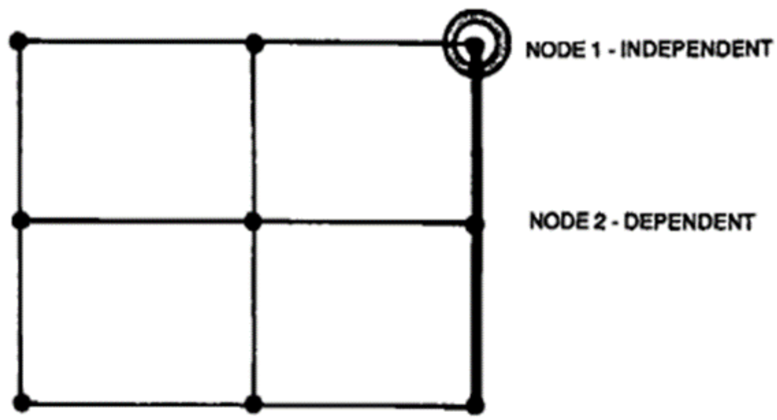
Node 1 is independent

Figure 3/3-31 Coupled DOF: Nodes 1, 2 and 3 Coupled in the y-Direction and About the y Axis

Releases can be introduced conveniently using coupling. For example, a pin can be introduced at mid-span in a continuous beam by coupling translational degrees of freedom of two coincident nodes. In certain circumstances, the coupling can introduce apparent violations of equilibrium.

A more powerful and general method for introducing constraints is by using constraint equations: A constraint equation is a linear equation that relates the displacement or rotational DOFs of nodes. These are sometimes referred to as multi-point constraints (MPC). Constraint equations may be used for many purposes, such as coupling of nodes by rigid members, rectifying small geometric discrepancies, and coupling adjacent nodes representing locally offset supports and attachments. Rigid regions in structure may be defined using constraint equations. Figure 3/3-32 illustrates the use of constraint equations using the example shown in Figure 3/3-31. In this case, the equation ensures that there is no relative movement between Nodes 1 and 2 in the x-direction.

Note that in certain analyses (e.g., crash problems), there is no specific constraint defined, and the calculation is based on the equilibrium between kinetic energies (external) and strain energies (internal).



MPC: (1)X1-(1)X2 = 0.0

Figure 3/3-32 Constraint Equation

3/3.5.4 Loads - General

Loading in finite element modeling may be applied in a variety of ways. Typical structural loads are forces, pressure load, gravity, body forces, and temperatures applied at nodes and on elements of the model. The load can be applied to:

1. nodes (e.g., nodal forces and body forces)
2. element edges or faces (e.g., distributed line loads, pressure, contact)
3. entire model (e.g., gravity loads and remote loads)

Generally, the load types and methods of its application to the model are specific to a particular FEA software package. However, descriptions of typical load types are provided in the following paragraphs.

3/3.5.5 Loads - Nodal Force and Prescribed Displacement

A nodal force is the combination of forces applied to the six nodal DOFs. A nodal force consists of:

1. force magnitudes in X, Y and Z directions
2. moment magnitudes about X, Y, and Z axes (for structural elements)

Nodal forces are usually applied in Nodal Coordinate System, as shown in Figure 3/3-33.

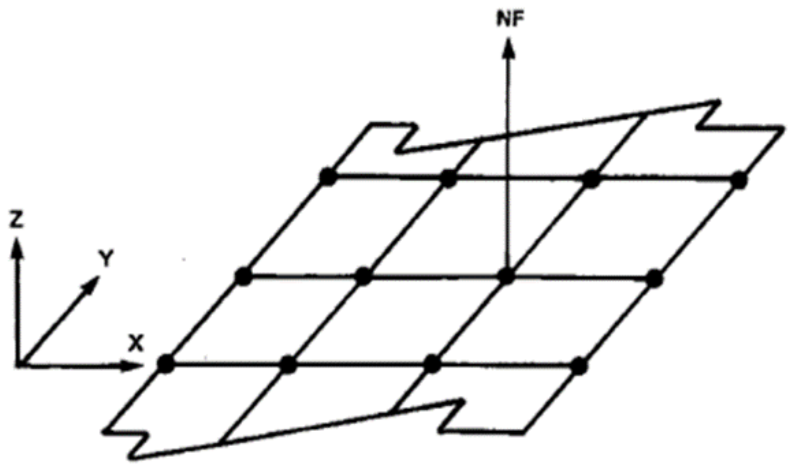


Figure 3/3-33 Definition of Nodal Force

Applied nodal loads must be compatible with the element type used. For example, a model consisting of only solid elements has no rotational degrees of freedom. Any nodal moment loads would have to be applied in such a case as a force couple with the forces acting at different nodes.

Also, forced or prescribed nonzero displacement may be input directly to nodes as a load case. This displacement should be prescribed with precision because small changes can cause large differences in stress response.

In fracture mechanics related problems, the load needs to be applied at the boundaries of the body. For example, a single-edge notched bend (SENB) specimen is loaded in a setup of three-point bending, where the force is applied at mid-span and restrained at two ends. In the three-point bending setup, the mid-span of the specimen is loaded by using a rigid pin. However, in the FEA setup, it is not recommended to mimic the setup by only applying load to a single node (see Figure 3/3-34(a)). That will cause local stress and strain concentration, and the element connected to the applied node will yield almost immediately. It is recommended to apply a locally concentrated load through a finite domain (3 nodes in Figure 3/3-34(b)) or to apply the load by using a rigid pin associated with friction or frictionless contacts [47].

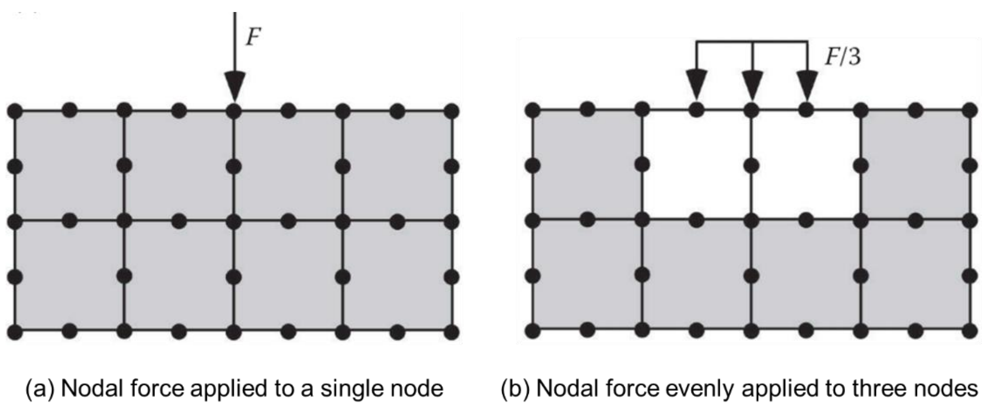


Figure 3/3-34 Examples of How to Applied Nodal Forced

3/3.5.6 Loads - Nodal Temperature

A nodal temperature is a single temperature value or pair of values applied to a node, as illustrated in Figure 3/3-35. A pair of values may represent the shell top and bottom surface temperatures. Some programs allow the specification of a pair of values representing the shell mid-plane temperature and a gradient.

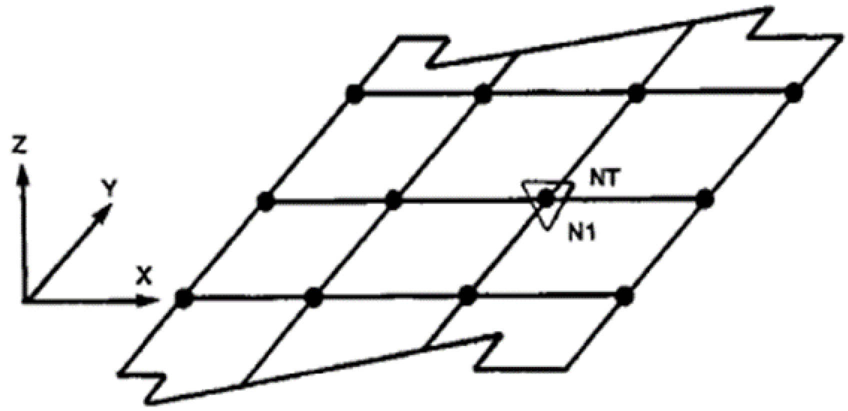


Figure 3/3-35 Definition of Nodal Temperature

3/3.5.7 Loads - Face Pressure

A face pressure is a single pressure value applied to selected faces of elements, as shown in Figure 3/3-36. The units of pressure value are force per unit area. The pressure is applied to each selected element face across the entire face and acts in a direction perpendicular to the face. Some FEA programs allow the user to specify pressure at nodal points. A variation of pressure over an element surface can thus be defined. Constant pressure is then a special case corresponding to all element nodes having the same pressure.

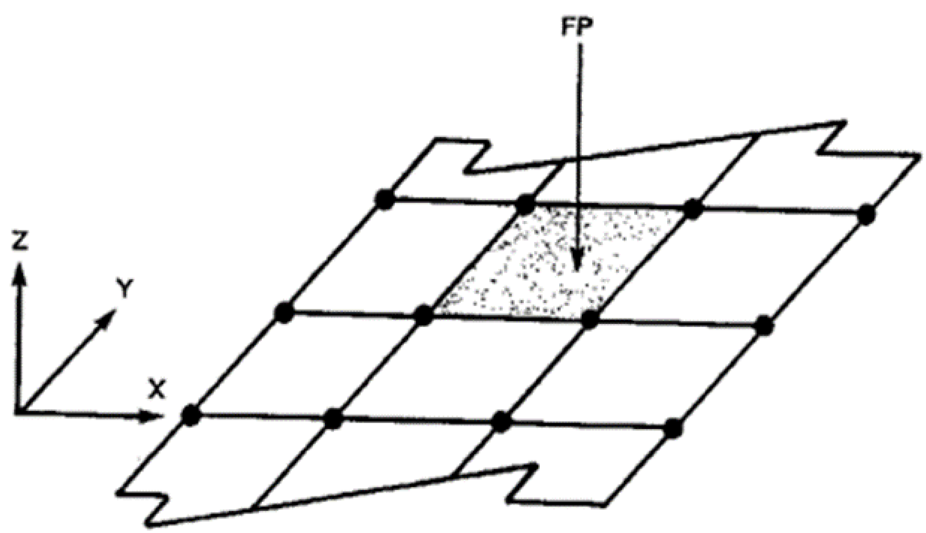


Figure 3/3-36 Definition of Face Pressure

3/3.5.8 Loads - Edge Loads

An edge load is the combination of the forces and moments that can be applied to the edge of an element, as shown in Figure 3/3-37. The types of edge loading depend on the type of element. An edge load can be applied to beam elements as:

1. axial force

2. shear force
3. torque
4. bending moment.

Uniformly distributed loads on beam elements can be handled exactly, and no further subdivision of the beam element is required to improve the representation of the load.

For membrane elements, edge loads can be applied as in-plane forces, and for plate bending elements both in-plane, and out-of-plane forces can be applied along with bending moments.

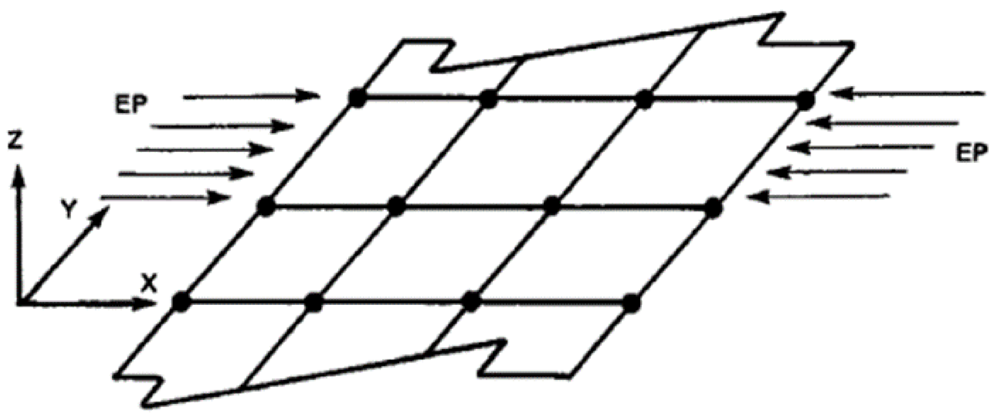


Figure 3/3-37 Definition of Edge Pressure

3/3.5.9 Loads – Thermal

A beam temperature is the temperature at the centroid of the beam's cross-section and is applied as temperature, Y-axis gradient, or Z-axis gradient in degrees, as shown in Figure 3/3-38.

Most programs allow for input of thermal loading directly on elements. Others permit, in addition, specified nodal temperature and temperature-dependent material properties.

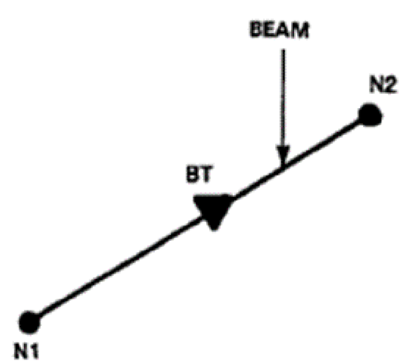


Figure 3/3-38 Definition of Beam Temperature

3/3.5.10 Gravity and Acceleration

Inertial loads are generated as a result of the body accelerating. A special case is the self-weight of a structure, or body, which is generated by the acceleration due to gravity. Inertial loads are generated as a result of one or more of the following:

1. translational acceleration
2. angular velocity
3. angular acceleration

FEA software systems treat weight data in different ways. It is important, therefore, particularly for dynamics problems, to be aware of the way in which the system treats mass and gravitational forces.

3/3.6 Analysis Control and Solution Options

The analyst must select the type of solver, the solver solution method, and the solver solution method settings (analysis controls/settings). These are discussed in turn in the sections that follow.

In the analysis control, the parameters specify the overall aspects for the analyzed FE model, e.g., degree-of-freedom, number of time steps and type of time functions, displacement and strain formulations, convergence tolerance, printout types. It should be noted that different analysis types may require different settings of analysis control. The analysis types have been presented in Section 3/2.1.

Aiming to run the jobs accurately and efficiently, it is highly recommended to use and control arguments of solution options instead of the default values. All the arguments of solution options for a given FEA code are generally available in the help documents.

This topic needs (1) lots of experience and understanding of what we want to accomplish, (2) a good understanding of analysis capabilities of the software adopted, and (3) extra cares in setting up models. This is a big topic, and it is recommended to read through the FEA and continuum mechanics books [25,28,99,143–151] for additional information.

3/3.6.1 Types of Solvers

In the FEA codes, the analyst may have several types of equation solvers in the nonlinear analysis, such as spare solver, 3D-iterative solver, and non-symmetric spare solver. In general, the **spare solver** is the most effective, the 3D-iterative solver is effective for solving models with high-order 3D elements or contact, and the non-symmetric spare solver is suitable for solving the model with Mohr-Columb or Drucker-Prager material models.

Note that in some FEA solvers (e.g., NASTRAN), there is an option to allow the job to continue execution, although the stiffness matrix is not positive definite. The forced accomplished solution may produce misleading results, and the analyst should review the model errors. In general, the non-positive definite stiffness matrix indicates an insufficient restraint condition.

3/3.6.2 Solver Solution Methods

3/3.6.2.1 Selection of Implicit or Explicit (or Co-simulation) Methods

There are mainly two different types of solution methods in solving equations of FEA simulations: the implicit method and the explicit method. The difference, essentially, between the implicit and explicit method is the incremental nature in nonlinear analysis. For a nonlinear analysis (e.g., geometric and/or material), it is required to gradually apply the load (or displacement) with an increment. After each increment, the global stiffness matrix needs to be updated due to the change of geometry and/or material. In both the implicit and explicit schemes, the stiffness matrix is updated in the incremental procedure, but the primary difference between the two methods is the implicit method has an additional step to enforce equilibrium of the internal loads with external loads by using the Newton-Raphson methods. This additional step makes the implicit method unconditional stable. In general, both methods can be used to solve the same nonlinear problem. However, as to the question of which method to use (implicit or explicit?), the answer is that – it depends. The following sections describe the difference between the two methods and help the analyst make decisions on which technique to use.

3/3.6.2.2 Integration Methods: Implicit vs. Explicit

Both implicit and explicit methods can be used to solve the partial differential equations (FEA). The general form of the equilibrium equation is as follows:

Equation 3/3-6

$$\boldsymbol{F_{inertia}(t) + F_{damping}(t) + F_{internal}(t) = F_{external}(t)}$$

Equation 3/3-7

$$M\ddot{u}(t) + C\dot{u}(t) + Ku(t) = F_{external}(t)$$

where M, C, and K are mass, damping, and stiffness matrixes, respectively; u and F are displacement and force matrixes.

Both explicit and implicit solvers can be adopted in solving equations Equation 3/3-6 and Equation 3/3-7. However, these two different solvers have their pros and cons regarding the equilibrium conditions and computational cost. For example, the explicit solvers are more computationally economical for solving dynamic problems (time-dependent) because the solution scheme does not require matrix inversion (unlike implicit solvers). However, the explicit solver solution is not unconditionally stable (unlike implicit solvers), and it is stable only if the time step is sufficiently small.

The time step calculation equations from element size or nodal mass are as follows:

Equation 3/3-8

$$\Delta \boldsymbol{t} = \frac{\boldsymbol{l}}{\boldsymbol{c}} = \boldsymbol{l}\sqrt{\frac{\boldsymbol{\rho}}{\boldsymbol{E}}}$$

Equation 3/3-9

$$\Delta \boldsymbol{t} = \sqrt{\frac{\boldsymbol{2m}}{\boldsymbol{k}}}$$

where l, c, ρ, and E are the dimension of the smallest element, speed of sound, the density of the material, and modulus; m and k are nodal mass and equivalent nodal stiffness, respectively. In practical applications, we generally add a scaling factor of 0.9 in Equation 3/3-9 to make sure that the time-step estimated is sufficiently small.

Note that interfaces (contact problems) also control the time step during the simulation because they are essentially stiff spring elements. In addition, it should be noted that the mesh quality is another key factor that controls the time steps, i.e., a model with higher mesh quality (especially by controlling smallest element sizes properly) has a reasonable time step.

It is interesting to know that nowadays, some FEA preprocessors have the ability to evaluate the time steps automatically.

Figure 3/3-39 schematically shows where implicit and explicit solvers fit in simulating an engineering problem. For example, a short-time transient problem (e.g., impact or explosion response analysis) is preferable to adopt explicit schemes, but a long-time transient problem is preferable to adopt implicit schemes. Figure 3/3-40 summarizes the cost of using implicit or explicit solvers in terms of the complexity of the solution. From two figures, the analyst may choose which solver to use in solving the engineering problem. A discussion of solution controls settings to improve accuracies is found in the following Section 3/3.6.2.3.

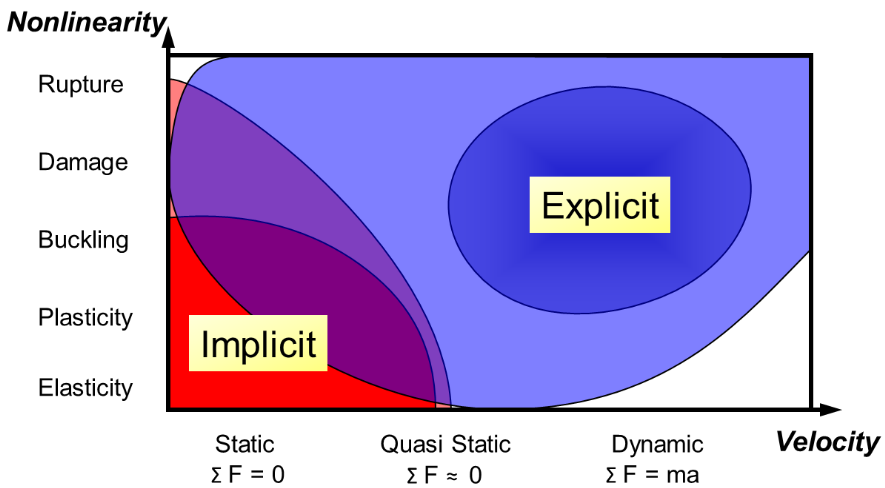


Figure 3/3-39: Implicit and Explicit Solver Applications Chart

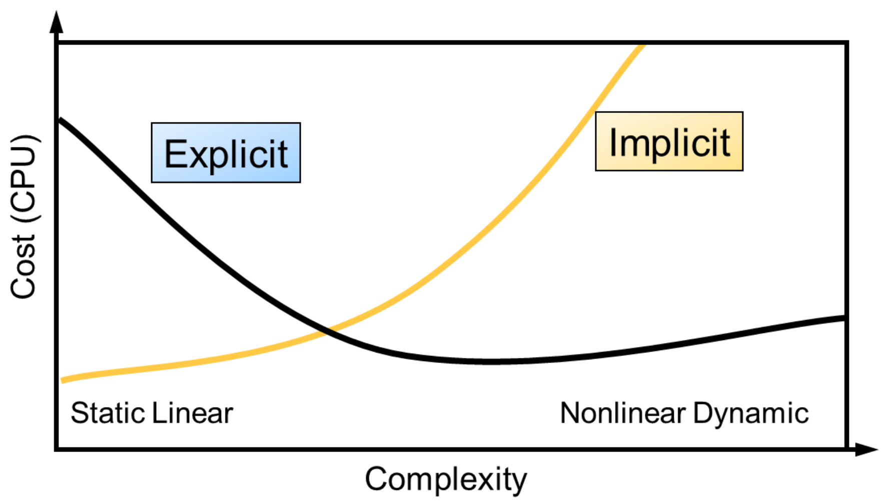


Figure 3/3-40 Implicit and Explicit CPU Requirements

3/3.6.2.3 Explicit Solution Controls

In the explicit analysis, the number of calculating steps depends on the time step determined based on Equation 3/3-8 and Equation 3/3-9. For a problem with a relatively longer time duration, analysts may want to save the calculation time and reduce the computational cost by (1) accelerating the event (to reduce the total duration time), and/or (2) adding mass to increase density ρ – (mass scaling to increase the time step). Note that although these methods are useful and can significantly reduce the calculation time, extra care is needed. Several recommended actions are listed as follows.

- Effect of inertia force is less pronounced (such as crack driving force J-integral calculation in quasi-static problems [28]);
- The increase of velocity and/or mass must be documented;
- If automatic mass scaling is enabled, the increase of mass at each time must be documented;

- The ratio of kinetic energy (increased) to the deformation energy should not exceed 1% for the quasi-static problem or 5% for the dynamic problem [28].
- It is recommended to use 'double precision' in the simulation to control the error accumulation and keep the stability of the solution.

3/3.6.2.4 Implicit Solution Controls

In the implicit analysis, the Newton Raphson or its derivatives methods are adopted to calculate the stiffness matrix for each iteration. For dynamic implicit analysis, the time integration procedures (e.g., Newmark family of methods and the α-method) are used with Newton methods. As a rule of thumb, the minimum number of time steps (tstep_dynamic_implicit) should be no less than 10, i.e., tstep_dynamic_implicit ≥ 10. The controlling parameters can be tuned based on the experience as well as the preliminary runs. For the α-method [28,106,152,153], there are three parameters α, β, and γ, and the method is unconditionally stable if having the following relationship:

Equation 3/3-10

$$\boldsymbol{\beta = \frac{1}{4}(1-\alpha)^2}$$

Equation 3/3-11

$$\boldsymbol{\gamma = \frac{1}{2} - \alpha \; for - \frac{1}{3} \leq \alpha \leq 0}$$

when α is less than 0, it gives a numerical damping. Such an approach can help reduce the noise of results from high-frequency modal analyses. The current FEA software may provide some recommended values of numerical damping that are unconditionally stable.

- Without numerical damping

$\alpha = 0$, $\beta = 0.25$, and $\gamma = 0.5$

- With numerical damping

$\alpha = -0.05$, $\beta = 0.2756$, and $\gamma = 0.55$

$\alpha = -0.1$, $\beta = 0.3025$, and $\gamma = 0.6$

Note that some codes may provide different recommended numbers of numerical damping parameters, e.g., $\beta = 0.38$, and $\gamma = 0.6$ [152].

In the static implicit analysis, the dynamic effects are less pronounced, and the equilibrium equation becomes

Equation 3/3-12

$$\boldsymbol{Ku(t) = F_{external}(t)}$$

In such a static or quasi-static analysis, the load or displacement (forced-displacement) is applied gradually with a user-defined increment (i.e., time steps). It should be noted that in the implicit method, the time steps generally have no physical meaning, whereas, in the explicit method, the time steps should be defined from the physical observations. Similar to the experimental tests, there are two kinds of control algorithms, i.e., displacement-control and force-control. The selection of control algorithm should be based on the expected responses as well as the interested results (e.g., post-buckling). There can be three feature points (i.e., limit point, bifurcation point, and turning point) regarding the force-displacement relationship (see Figure 3/3-41).

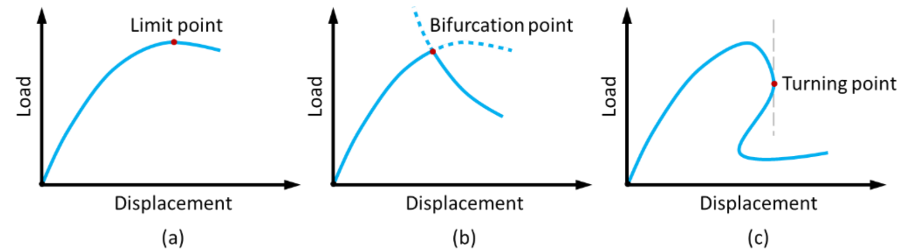


Figure 3/3-41 Three Feature Points in Load vs. Displacement Curves Reproduced from DNV GL C208

Three feature points correspond to three different ways to choose the control algorithm:

- **Force-control algorithm:** The load and displacement relationship can be captured up to the limit or bifurcation point, i.e., the tangent stiffness is greater than 0 or no decrease of force.
- **Displacement algorithm:** The load and displacement relationship can be captured up to the turning point, i.e., no decrease of displacement.
- **Displacement algorithm associated with the Arc-Length method:** The full force-displacement relationship with the presence of snap-back (turning point) can be captured.

In the implicit analysis, the use of the Newton Raphson method (with user-defined tolerance) tends to be more accurate sometimes with bigger increment steps. This method is robust and can handle highly nonlinear problems such as cyclic loading, snap-through, and snap back as well as the proper control algorithm is used. However, one of the major drawbacks of this method is that the global stiffness matrix needs to be updated and reconstructed in each iteration, which can be costly in terms of time and computational power. An alternative method – Modified Newton Raphson method can be adopted to decrease the cost.

This type of analysis tends to be more accurate and can take somewhat bigger increment steps. Also, this type of analysis can handle problems better, such as cyclic loading, snap-through, and snap back so long as sophisticated control methods such as arc length control or generalized displacement control are used. One drawback of the method is that during the Newton-Raphson iterations, one must update and reconstruct the stiffness matrix for each iteration. This can be computationally costly. (As a result, there are other techniques that try to avoid this cost by using Modified Newton-Raphson methods.) If done correctly, the Newton-Raphson iterations will have a quadratic rate of convergence, which is very desirable. The Newton-family methods have a quadratic convergence if done correctly.

3/3.6.2.5 Memory Management

In solving large problems, the usage of memory can often become an issue, especially when using the implicit method-based solver. In most FEA codes, the implicit solver has two different ways to manage and allocate the memory: in-core (physical memory) and out-of-core (virtual memory on the hard drive). Noted that it is more efficient to run simulations in the in-core mode than out-of-core mode if there is sufficient memory. If there is not sufficient memory, the simulation will be run in the out-of-core mode. The following three parameters are generally needed in the setup.

1. **Maximum memory for solution:** This parameter specifies the maximum memory to be used for the solution. It mainly includes the memory for allocating the model data and solving the equations. The rule of thumb to determine the maximum memory (MAX) is 80% of the physical memory (RAM), i.e., MAX = 0.80 × RAM.
2. **Memory for storing model data:** As above-mentioned, this parameter is part of the maximum memory and specifies the amount of memory storing stiffness and/or mass matrix and element information.

3. **Automatic memory allocation:** In modern FEA codes, there is an option to allocate the maximum memory automatically that may provide optimal value for the analyst, e.g., 90%RAM in ABAQUS (User Manual 3.4.1) [96] and 85%RAM in ADINA [128].

Note that the above-described contents are reasonable for SMP (shared memory parallel) based processing and that for DMP (distributed memory parallel) based processing will be different. In addition, the memory management is directly related to the number of processors (or cores) used in a run, as well as the precision of solvers (i.e., single or double precision). Note that the setup for 16-bit and 32-bit computers is not considered here.

3/3.6.2.6 Batch Mode

Most of the FEA codes on the workstation are run in an interactive mode. However, sometimes, it is more convenient and efficient to run the simulation jobs in the batch mode without user input. The analyst needs to write the batch command to run jobs per FEA references. Nowadays, there may be a job manager and third-party software with GUI (graphic user interface) available for the analyst to submit and run jobs in the batch mode, for example, LS-RUN for running LS-DYNA jobs in Windows OS. Several key input parameters are listed as follows:

- Input and output
    - Input and output file directory
    - Input and output file name(s)

  Note: some solvers have strict file naming rules.
- Solver
    - Solver directory
    - Solver version and precision
- Hardware
    - Number of CPU(s) and/or GPU(s)
    - Number of thread(s)
    - Size of RAM
    - Local workstation of the computer cluster
- Other arguments
    - Check run
    - Kill job
    - Stop job
    - Create Information file(s)
    - Create restart file(s)

3/3.6.2.7 Nonlinear Solution Convergence Monitor

For the nonlinear analysis, there is a need to monitor the convergence of simulation results at each time step. At a typical time step, it may need several iterations to achieve an equilibrium state where all the key indexes (e.g., force, energy, and contact) meet the criteria/tolerances. It should be noted that there are exceptions when special algorithms are adopted in the calculations, and a higher tolerance value (i.e., easier to converge) may be needed. The history of solution convergence is generally recorded in the result output file. Note that the recommended parameters (for example, max/min iterations per time step, convergence tolerance, and algorithms) are available in software manuals and benchmark examples. Some ship-structure-focused publications [154–158] also provide the best approaches and examples for convergence practice.

3/3.6.3 Static Analysis

Static analysis is used to determine the displacements, stresses, strains, and forces in structures due to loads that do not induce significant inertia and damping effects. The loads and the structure's response are assumed to vary slowly, if at all, with respect to time. The primary application of FEA in ship structures is in support of design, and this usually involves static analyses. These may range from global models encompassing the whole ship, to very detailed local models. Apart from FEA performed in support of design, static analysis is also used in the investigation of certain types of structural failures. Static FEA analyses can be categorized into two groups: linear static and nonlinear static.

3/3.6.3.1 Linear Static Analysis

In general, the linear static analysis is the easiest one to obtain converged simulation results. Therefore, before simulating complex problems, an approximated linear static analysis is carried out to check the degree of nonlinear behaviors in terms of stress levels above the yield strength. As indicated in Section 3/2.1.2, this type of analysis is related to stiffness (e.g., Young's modulus and Poisson's ratio), and there is no need to define material density. Note however that if the effects of mass are required (e.g., inertia relief), the material density needs to be considered.

One of the key solution options is to define the time step quantity for each load step. The time step quantity determines how many increments the calculation needs to reach the predefined loading. For example, some FEA codes assume a one-time step for a load step in the linear static analysis. Another key solution option is to define the load steps/sequences if there are multiple loads included.

Note that the linear static analysis mentioned here refers to general analysis and excludes special problems such as high cycle fatigue.

3/3.6.3.2 Nonlinear Static Analysis

The implicit integration method is generally used for solving nonlinear static analysis because such types of analyses can be regarded as being time-independent or infinitely slow. The time independence results in the partial differential equations being stable (unconditionally stable). This is discussed further in Section 3/3.6.2. The key non-linear solution options are the following:

1. **Load steps or load sequences:** A loading history may consist of several load steps (e.g., loading, unloading, reverse loading, and unloading). The solution of nonlinear static analysis is always history-dependent. Therefore, the load steps need to reflect the actual scenario defined in the problem description.

2. **Time steps or time increment:** The time step or time increment describes how many increments it takes to reach the predefined load at a given load step. Note that the number of time steps defined in the implicit method does not have physical meaning. For example, if "time step = 10", it means that the loading level increases 10% at each time step. By breaking the simulation into a number of steps, it may help find the approximate equilibrium configuration at the end of each time step, and the results of interest can be exported to output database files.

3. **Iterations for each time step:** The number of iterations is trials or attempts at finding the equilibrium solution in an increment. If the iteration process diverges, the iteration process can be terminated or recalculated with a smaller increment size.

4. **Finite or large strain formulations:** If the problem involves large geometric nonlinearity (for example, crack-tip opening displacement calculations), the large/finite strain formulations need to be adopted in the calculation. Note that the analyst should examine whether the formulations of shell and material are compatible with the finite strain formulations.

5. **Newton Raphson:** Newton Raphson method and its derivatives are one of the most widely used numerical methods to find the solution of static equilibrium equations. Most of the FEA software provides two options: (i) Full Newton Raphson method and (ii) Modified Newton Raphson method. The major differences between the two methods are that the modified Newton Raphson method uses the stiffness matrix from the first iteration to find the converged solutions, whereas the full Newton Raphson method uses the updated stiffness matrix after each iteration. For highly nonlinear problems, the Newton Raphson method can also use the line search method, which can assist in obtaining the converged solution. Note that the analyst may also use other methods to solve the equations, such as the load-

displacement incrementation method and Broyden-Fletcher-Goldfarb-Shanno (BFGS) method.

6. **Others:** Except the above-mentioned five major aspects, the analyst may also need to determine other aspects, such as direct or iterative equation solver, symmetric or unsymmetric matrix storage, full newton, or quasi newton solution technique.

It is crucially important to choose a proper time step. In some FEA codes (e.g., ANSYS, ABAQUS, LS-DYNA, ADINA, Optistruct/NASTRAN), the automatic time-stepping method is available and can be adopted to obtain the converged solution with fewer time steps during the equilibrium iteration. It is a reliable and suitable method for a wide range of nonlinear static problems. The time step estimated from this method is a dynamic value that keeps updating based upon the size of the time step where convergence was achieved previously. For example, if the original time step (Δt) cannot attain convergence, the approach reduces the time step (e.g., $0.5\Delta t$); if the new time step cannot attain convergence, the approach reduced the time step again (e.g. $0.25\Delta t$). The time step will be automatically revised until the convergence is achieved, or the run will be terminated if the number of trials reaches the limit.

3/3.6.4 Buckling Analysis

Depending on the structural element, the estimate of buckling load can be very sensitive to the inevitable presence of discontinuities, imperfection, residual stresses, and material nonlinearity. The application of FEA techniques to solving buckling problems should be approached with caution. The results can be very sensitive to assumptions made in regard to deviations from the ideal, more so than is typical for linear static analysis. The usual practice in design situations is to adapt classical solutions to the problem.

There are two different types of analyses for evaluating the buckling resistance of a ship structure, i.e., linear and nonlinear buckling analyses. The buckling resistance measured from the nonlinear analysis is generally higher than that from the linear analysis. However, the difference is dependent on the actual model.

The buckling behavior may be observed on a global ship structure or the local element(s), e.g., stiffened/unstiffened panels, girder webs, bulkheads, etc. The presence of elastic buckling of a local element may not be critical due to the redundancy characteristic.

3/3.6.4.1 Linear Buckling Analysis

For the linear buckling analysis, the analysis control is similar to a linear static analysis (see Section 3/3.6.3.1), because it is essentially an eigenmode analysis. The number of eigenmodes should be defined as well as the type of solving algorithms (e.g., Lanczos, AMSES, or AMLS). Note that if the effect of gravity is considered, the pre-stress state needs to be included in the analysis. Since this is a linear elastic analysis, the stress level should be less than the yield strength. The first (and sometimes second) eigenmode is the dominant one and should be used for buckling resistance calculations.

3/3.6.4.2 Nonlinear Buckling Analysis

The nonlinear buckling analysis includes nonlinear factors, such as imperfections, misalignments, discontinuities, residual stresses, material, and geometric nonlinearities. Most FEA codes have approaches with built-in or third-party functions to represent the profiles of imperfections, misalignments, discontinuities, and residual stresses in the analysis control (e.g., a geometric perturbation in LS-DYNA [28]).

The analysis controls of material and geometric nonlinearity are similar to those of nonlinear static analysis (see Section 3/3.6.3.2). The nonlinear stress-strain relationship is adopted to reflect the material nonlinearity. The large deformation formulations need to be activated to capture the geometric nonlinearity. Note that there are two different ways to control the load, i.e., force control or displacement control, and the differences between the two methods have been presented in Section 3/3.6.2.4. If the force control is employed in the analysis, the magnitude of the load must be higher than the limit load (i.e., failure load).

Note that in Section 5.4 of DNVGL-RP-C208 (2016 edition) [106], a hybrid method for determining the buckling resistance is proposed, which is based on both nonlinear buckling analysis and the calibration against testing results or empirical formations.

3/3.6.5 Impact Analysis

The existing explicit solvers for solving impact (or crash) analyses are historically from the same open-source research code – DYNA3D developed by Lawrence Livermore National Laboratory [159]. Therefore, the analysis controls (e.g., control decks/cards) are quite similar among those solvers.

As discussed in Section 3/3.6.2, the explicit solver is NOT unconditional stable, and extra care is required to obtain reliable results. Except material models and mesh designs/qualities discussed in Sections 3/3.2 and 3/3.3 of this part, respectively, the following parameters and criteria in the FEA settings [28,91,105,160,161] are also predominant factors in the analysis control:

1. **Timestep**
   a. critical time step $\Delta t_{critical}$
   b. scale factor, e.g., 0.9 in the time step calculation $\Delta t = 0.9\Delta t_{critical}$
2. **Hourglass**

   The hourglass describes a weird type of deformation mode that shows deformation in the "hourglass" shape without generating any stress when using reduced integration elements. An example of the hourglass effect is shown in Figure 3/3-42 [105,162]. In general, there are several hourglass formulations available in the FEA codes.

   A rule of thumb in monitoring the hourglass is that the hourglass energy should be no greater than 10% of the internal energy.

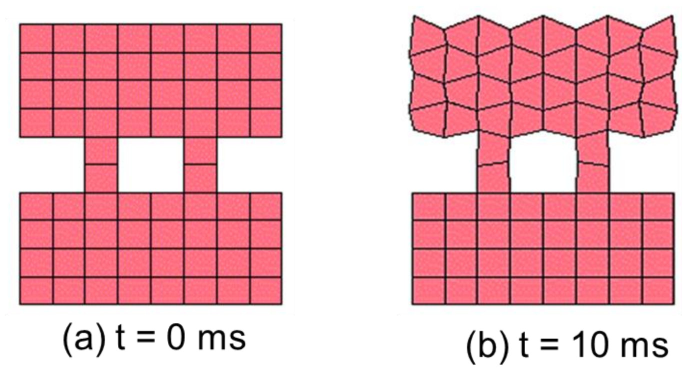


Figure 3/3-42 Test of Hourglass Control [28]

3. **Result output time step**

   It includes two sets of time steps: plotting results (e.g., d3plot files in LS-DYNA) and exporting information of solving equations (e.g., ASCII text files where energy, contact force, stress, strain, SPC force, status, etc.).

   The time step of plotting results (T_{plot}) defines the number of interest frames from the initial time to the termination time, and it is recommended to have at least 10 frames/time steps. For example, if the termination time is 100 ms, and the time step is 10 ms, the plotting result files will be generated at each 10 ms increment. In total, there will be eleven (11) result files, including that at the initial time step.

The time step of exporting information of solving equations ($T_{I/O}$) refers to the number of calculation information recorded, which are critical when reviewing the simulation results. It is recommended to store at least 10 sets of calculation information between each time step of plotting results, e.g., $T_{I/O} = 0.1 \times T_{plot}$.

4. **Initial contact penetration**

   The non-zero initial contact penetration will induce calculation errors. The analyst can either review the penetration in the pre-processor or run a simulation with a finite time step (e.g., 1 ms). In general, the initial penetration will be highlighted in the pre-processor (e.g., ANSYS and HyperMesh [15]). From the trial run, the ASCII text files will indicate whether there is a non-zero initial contact force or contact energy, which indicates an initial contact penetration.

5. **Contact models and compatibilities**

   The modern FEA codes have become smarter and more robust in terms of definition contacts, e.g., automatic single/multiple surface contacts in LS-DYNA [28]. However, it is still recommended to review the contact models to check whether the definition (e.g., linear or nonlinear formulations) is compatible with physical problems.

6. **Compatibility between element formulations and material models**

   Similar to the usage of implicit solver, the analyst needs to review the compatibility between element formulations and material models and understand the overall limits of the element and material adopted. The solver theory and keyword reference manuals generally include the checklist of compatibilities. In addition, it is recommended to have a verification run before the formal run.

7. **Mass scaling**

   As mentioned in Section 3, the analyst has to check the weight of the FE model after assigning thickness and properties to meshes. However, because of small-time step size from the usage of small-size or fine elements, the analyst needs to activate the mass scaling feature, which can increase the time-step size in each cycle and reduce the cost of computation power. This feature is extremely useful in solving quasi-static or low-speed problems, where the inertia force (kinetic energy) does not play a predominant role. The analyst may also increase the mass locally on several critical elements (e.g., the smallest element size) such that the time step increases per Equation 3/3-9. Figure 3/3-43 shows the time step size with and without applying the mass scaling technique.

   There are several ways to activate the mass scaling feature [163–165], for example:

   1. Increase the mass density of the component(s);
   2. Add mass automatically to elements invoking small time steps (with predefined criteria);
   3. Similar to (2) but added mass with the time-step scaling factor (generally default value is 0.9).

   It is up to the analyst to make an engineering judgment on gauging the effects of mass scaling on simulation results. As a rule of thumb, a simulation with a mass increase of less than 5% can be regarded as a reliable and effective run. Again, it should be noted that a pronounced increase in mass can cause overly severe penetration problems and increase the dynamic effects significantly. The penetrations and kinetic energy have to be monitored closely by the analyst after each run.

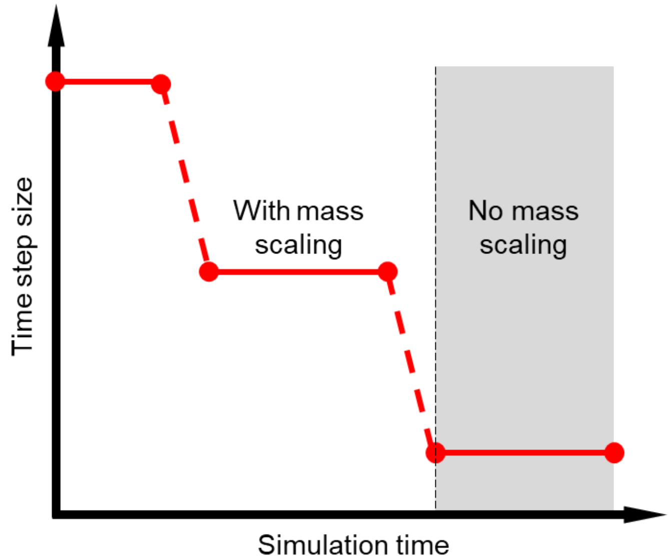


Figure 3/3-43 Relationship between Simulation Time and Time Step Size With and Without Mass Scaling Features

3/3.6.6 Fracture Analysis

3/3.6.6.1 Fracture Analysis

In general, the fracture toughness indexes (e.g., $K_{I, II, III}$, $G_{I, II, III}$, $J_{I, II, III}$, CTOD, CTOA, C*, etc.) are of interest for the fracture analysis. The analyst needs to choose the method for calculating these fracture toughness indexes. For example:

1. The linear contour method is good for calculating J and K (linear elastic condition) the 2D problems with the stationary crack and the propagating crack. The method is compatible with both blunt and straight crack front profiles.
2. The virtual crack extension (VCE) method with nodal virtual shifts (NVS) method can be used for both 2D and 3D problems with the stationary crack and the propagating crack. The VCE-NVS method is compatible with both blunt and straight crack front profiles.
3. The VCE with station virtual shifts (SVS) method is not applicable for 2D problems but 3D problems. The VCE-SVS method is valid for both mapped mesh and free-form mesh near the crack front. In addition, this method can deal with more than one crack in the analysis where linear contour and VCE-NVS methods cannot.

As discussed in Section 3/2.6.4 Basics of Fracture Mechanics, the extra cares are needed in determining the number of contours for J calculations, and which one should be used for J value output. The following bases should be noted:

1. If the small deformation and small strain formulations are adopted, the J values are essentially contour-independent, except the 1st contour, and the last one touches the model boundary.
2. If the large/finite deformation and large/finite strain formulations are adopted, the J values are strongly contour-independent. It is recommended to use the value that represents the far-field works.

The general analysis control is the same as those presented in Section 3/3.6.1. Most of the conventional constitutive material models (e.g., stress-strain relationship) can be adopted in this analysis, and it is up to the analyst to verify the compatibly between the material model and solvers per the user manual of adopted FEA solver.

Note that the averaged J values should NOT be used, which has no physical meaning. If the CTOD values are of interest, the analyst needs to understand which type of CTOD definition is adopted and how to interpret it from FEA results.

3/3.6.6.2 Fracture Analysis with Damage Criteria

Similar to Section 3/3.6.6.1, general analysis control is the same as those presented in Section 3/3.6.1. If the damage model adopted is not stress- and/or strain-indexes based, it is up to the analyst to check the compatibility between the damage model and FEA setups (e.g., element types and strain formulations). The micromechanical parameters in the damage models require a hybrid methodology to be determined and calibrated from both testing and numerical simulations.

If the Gurson model is adopted to detect the local damage, the user-defined material model is required, which is generally not provided by the FEA software.

If the cohesive model is adopted, it is recommended to choose the multi-polynomial law (i.e., four parameters) where the model can fulfill the following requirements [118]:

1. The initial stiffness of the cohesive element can be varied;
2. The initial thickness of the cohesive element has a minimum or no impact on the results;
3. A region can be defined, where the traction is kept constant;
4. The curve of the cohesive law is steadily differentiable and reason for numerical implementations.

3/3.6.7 Fatigue Analysis

Fatigue analysis includes two types of problems: linear elastic FEA based and nonlinear elastic-plastic FEA based. The major differences between them in the analysis control are the loading levels and material models. The loading levels are higher or comparable to the limit load, where the corresponding stress and strain relationship is beyond or close to the proportional limits (i.e., beyond yield strength). If both types of fatigue damage cases included, the damage from linear fatigue analysis should be added to that from nonlinear fatigue analysis, and the total damage should be the summation of both, i.e.

$$\text{Damage}_{\text{Total}} = \text{Damage}_{\text{Linear}} + \text{Damage}_{\text{Nonlinear}}$$

The following parameters affect the fatigue life and should be considered in the analysis control.

1. Cyclic loading (e.g., type, stress range, mean stress, stress amplitude, R ratio)
2. Cyclic stress and strain relationship (e.g., uniaxial or multiaxial, base metal or weldment, yield strength, ultimate tensile strength, and plastic strain level)
3. Stress/strain vs. number of cyclic loading at interest R ratios (e.g., SN or EN)
4. Thickness effect
5. Temperature effect
6. Surface finish (e.g., polishing, grounding, machining, hot rolling, and forging)
7. Surface treatment (e.g., nitriding, shot peening, and cold rolling)
8. Residual stress effects
9. Notch presence (e.g., fracture mechanics-based fatigue analysis)
10. Safety factors (e.g., Dang Van Criterion)

Note that the FEA code may have selected parameters mentioned above, and it is up to the analyst to review the calculation results and determine whether there should be a subsequent correction by considering the impacts of other factors.

If the rainflow cycle counting is adopted, it is recommended to define a threshold value (also known as gate value) for load peaks simplification.

If the shell element is used, the stress values from nodal corners and skin layers under tension should be used for fatigue life calculations.

The certainty of survival of SN or EN curves should be defined or recorded because the data curve is a statistical approximation of experimental results. For example, if the value of survival certainty is defined to be 90%, the SN or EN curves offset downward such that 90% of the experimental data lays above the offset curves (see Figure 3/3-44) [166].

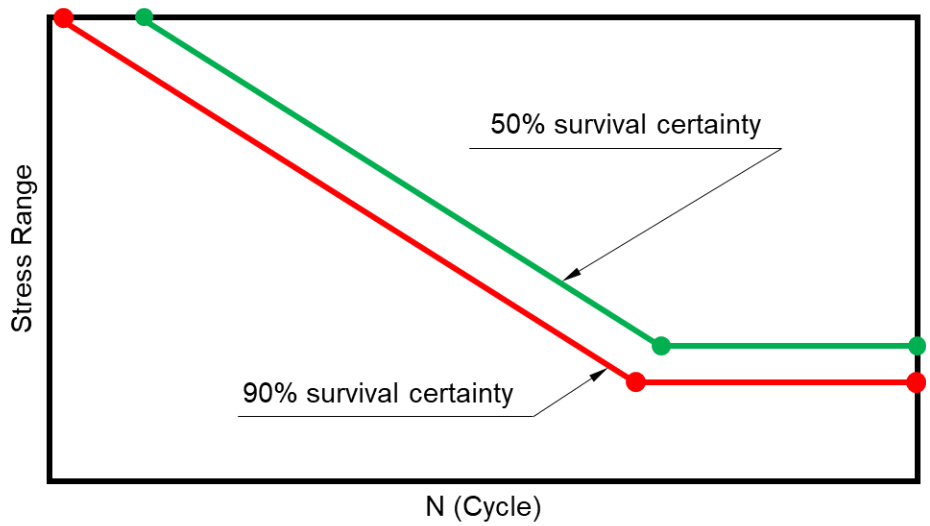


Figure 3/3-44 SN Curve with Different Survival Certainty

It is recommended that the analyst has a sense of the potential fatigue failure mechanisms (e.g., uniaxial or multiaxial) as well as the failure modes (e.g., Mode-1, 2, 3, or Mix-mode).

3/3.6.7.1 Linear Fatigue Analysis

Except for the parameters presented in the above section, the mean stress correction approaches should also be determined in the analysis. The three widely used mean stress correction approaches in FEA analysis control are the Goodman method [167], the Gerber method, and Soderberg method [168]. The Goodman method (a bi-linear line plotted in Haigh diagram, see Figure 3/3-45) is recommended for analyzing brittle materials, and the Gerber method (a polynomial curve, see Figure 3/3-45) is recommended for analyzing ductile materials. Note that the Goodman, Gerber, SWT, and Morrow correction methods may have alternative or modified formulations presented in FEA codes.

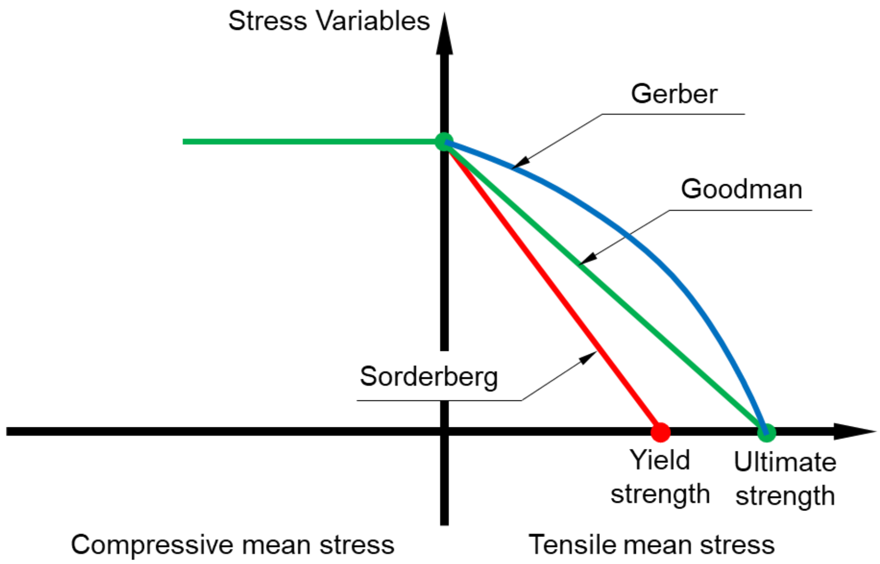


Figure 3/3-45 Mean Stress Correction Methods in Haigh Diagram

Three approaches of discrete SN curves are generally available in FEA codes, and the SN curves should be defined accordingly based upon the availability of parameters or experimental data.

1. One segment SN curve in log-log scale with fatigue life defined (Figure 3/3-46(a)).
2. One segment SN curve in log-log scale with the cycle limit of endurance defined (Figure 3/3-46(a)).
3. Two segment SN curve in log-log scale (Figure 3/3-46(b))

In Figure 3/3-46, the abbreviations SR1, B1, NC1, B2, and FL refer to fatigue strength coefficient, first fatigue strength exponent, the cyclic limit of endurance / transition point, second fatigue strength exponent, fatigue limit, respectively [166].

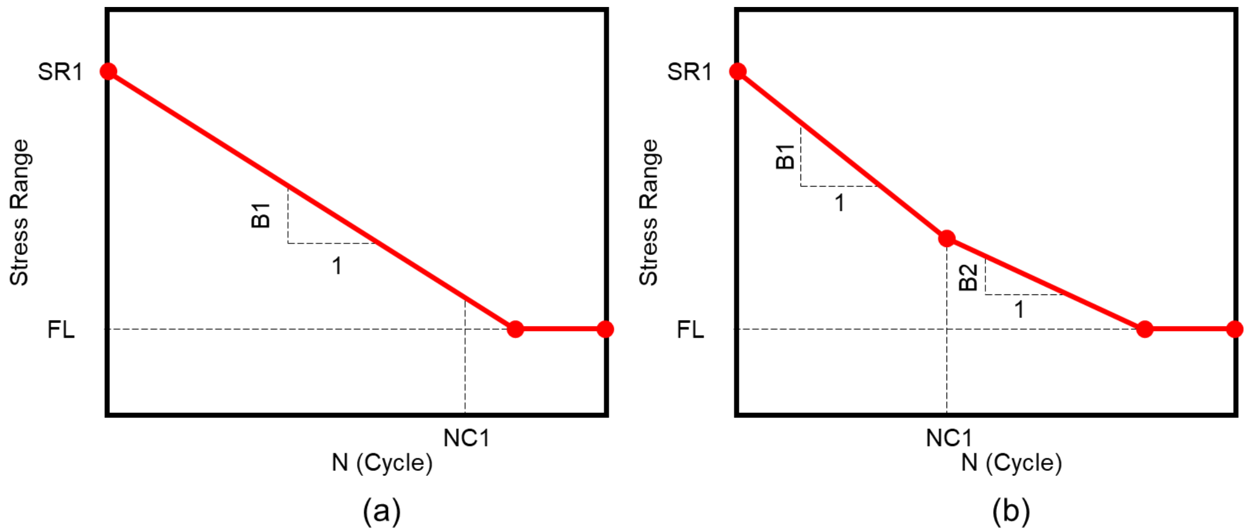


Figure 3/3-46 Different SN Curve Approaches

3/3.6.7.2 Nonlinear Fatigue Analysis

The two widely used mean stress correction approaches in FEA analysis control are the Morrow method and Smith-Watson-Topper (SWT) method. The Morrow method (the red-line curve in Figure 3/3-47) is recommended for analyzing brittle materials, and the SWT method (the green-line curve Figure 3/3-47) is recommended for analyzing ductile materials.

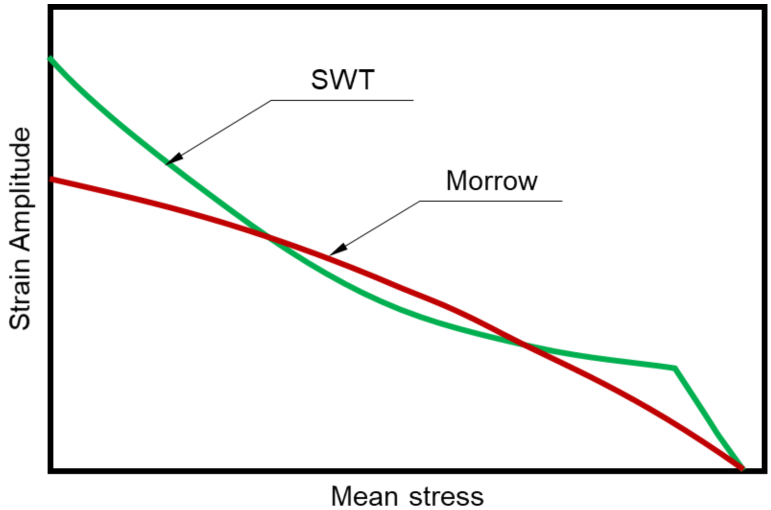


Figure 3/3-47 Morrow and SWT Methods for Nonlinear Fatigue Analysis

The settings of material nonlinearity analysis are similar to that of nonlinear static analysis. However, the fatigue life with the presence of plastic strain is dependent on the loading magnitudes and sequences, and extra care required in defining load steps.

A typical discrete EN curve in the FEA code is shown in Figure 3/3-48. The abbreviations SF, B, EF, and C refer to fatigue strength coefficient, fatigue strength exponent, fatigue ductility coefficient, and fatigue ductility exponent, respectively [166].

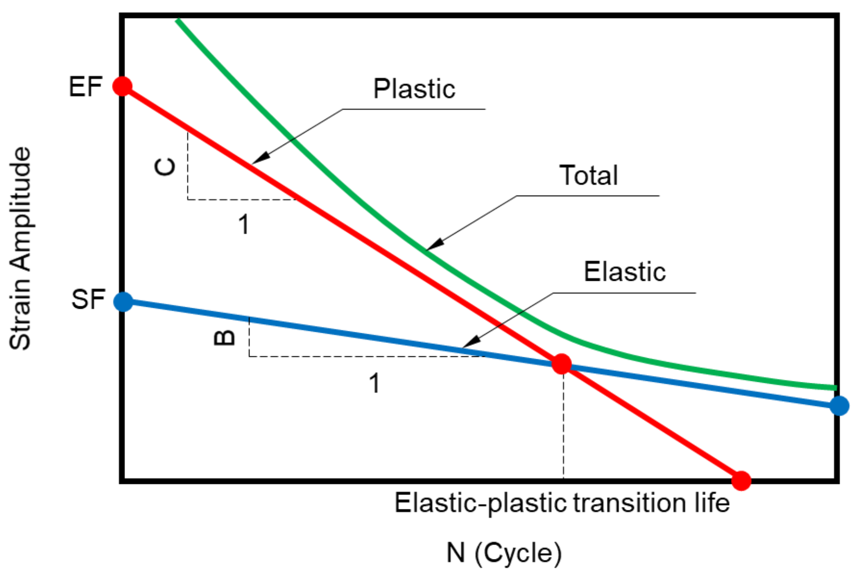


Figure 3/3-48 EN Curve Approach

3/3.6.8 Vibration Analysis

Vibration analyses in ship structures are usually performed for the following reasons:

1. To ensure that the natural frequencies of sensitive structures and components do not coincide with those of the hull girder or with the forcing frequencies associated with propellers and other mechanical sources of vibration energy.
2. In preparation for dynamic response computations.

Several quasi-static design procedures have been developed for design against dynamic load conditions. For some of these procedures, for example, the Design Response Spectrum Method used for shock analysis, it is often necessary to compute several tens of natural frequencies of the subject structure or component. In complex structures, such as a large ship’s mast, the natural frequencies and modes can usually only be calculated using FEA.

As an alternative to quasi-static procedures, a more rigorous dynamic response calculation may be used. Two methods are available: direct integration of the equations of motion or the superimposition of modal responses. For nonlinear behavior, such as that associated with large deflections and/or plasticity, only the former is appropriate.

Transient dynamic response analysis is used primarily for computing response to suddenly applied loads and/or short-duration loads. Examples include forces due to collisions, wave slamming, and shock, and blast. In these cases, the loading is very uncertain. Various procedures have been developed to compute loads from these types of loading. For example, procedures are available to model the shock forces generated as a result of underwater explosions. The procedure models the underwater explosion, the pressure-induced on the hull, and finally, the transmission of the dynamic forces through the hull structure to the structure or component in question. Many transient dynamic problems involve fluid-structure interaction phenomena where the structural response affects the loading on the structure. Sometimes it is possible to treat such phenomena very approximately by adding a certain amount of fluid mass to the elements adjacent to the fluid.

3/4 FINITE ELEMENT RESULTS CHECKS

The results obtained from a finite element analysis (FEA) should always be verified, and their validity established. To make sure that the results are devoid of any errors in modeling or analysis, it is necessary to perform the checks outlined in this section. These checks ensure that the FEA results are calculated, processed, and presented consistently with the analysis requirements.

3/4.1 General Solution Checks

Many of the following checks can be performed using the graphical display features available with most FEA software systems. Where such features are not available, these checks will have to be performed by examining the modeling results output.

3/4.1.1 Errors & Warnings

Well established finite element software systems generally have several built-in checks to identify poor modeling and analysis practices. A warning or an error message is issued when built-in criteria are violated. The correct practice is to resolve any such messages and take the appropriate remedial action. If the warning/error message is not applicable to the analysis, proper justification should be provided.

An example could be a warning message for the angle between adjacent edges in a quadrilateral shell element. The generally recommended range is between 45°and 135°. If this rule is not followed, valid justification could be that the element in consideration is located well away from the area of interest.

3/4.1.2 Mass and Center of Gravity

It is good practice to verify the mass of the model and the location of the model's center of gravity of the model. Several programs provide the mass without the need for a full analysis. If this option is unavailable, the analysis could be run with a 1G loading (with no other applied loads).

3/4.1.3 Self-Consistency

The results should be checked for 'self-consistency.' For example, displacements at fixed supports should indeed have zero displacements, and any symmetries in the model should be reflected in the stress and deflection results.

3/4.1.4 Static and Dynamic Balance

This is a fundamental check. The applied loads should be compared with the reactions. The check should include moments, inertial forces, residual forces/stresses, pre-tension, or other types of source loadings where appropriate. This check ensures that the applied loads and reactions are in balance and ensures that the user-specified loading definitions are appropriately interpreted by the program. When the applied loads and reactions are not in balance, this is an indication of a severe error.

Checking the forces and reactions also ensures that the results are actually for the intended load. In the case of pressure loads, due to possible discrepancies in arriving at nodal forces from pressures, the actual load level could be different from that intended.

In addition, the energy indexes (e.g., kinetic energy, hourglass energy, etc.) should be checked in both static and dynamic analysis. This check ensures that all the energy indexes are on the right scale and ensures that the loading and boundary conditions are properly defined in the model. For example, the energy indexes related to contacts (i.e., slave energy, master energy, friction energy) can be useful indicators to evaluate whether the contact is defined correctly.

3/4.1.5 Defaults

All FEA software packages have built-in defaults. For certain input parameters, default values or options are assumed if a value has not been input, or if an option has not been selected. Hence, checks should be performed to ensure that where defaults have been used, they are consistent with the assumptions of the analysis.

3/4.1.6 Checklist

The following is a list of checks to ensure the quality of the FEA. The checklist covers both pre-run and post-run checks. A reference checklist is shown as follows. The analyst can adjust the content of the list based on the different load cases. For example, in the 2D analysis, “Plot thickness of shell elements by color” is an efficient way to review whether the thickness is appropriately assigned.

1. Pre-Run Checks - Graphical:
    a. Extremities of the model - global dimensions OK
    b. Free edges - look for element connectivity
    c. Shrunken elements - no missing elements
    d. Duplicate nodes
    e. Duplicate elements
    f. Size of adjacent elements - avoid ill-conditioning
    g. Mesh density
    h. Mesh transitions

i. Plot material properties by color

j. Plot physical properties by color

k. Loads applied to correct elements

l. Direction of loads correct

m. Boundary conditions applied to correct nodes

In addition to the graphical pre-run checks, it is good practice for the analyst to complete numerical -pre-run checks that may identify common issues such as key geometric dimensions, magnitudes of loads and material properties, as well as, verification of the consistency of the units of measure for each of the numeric values.

2. Post-Run Checks:
   a. Static or dynamic balance
   b. Comparison
      i. classical results
      ii. simple finite element model
   c. Numerical accuracy
      i. residuals
      ii. stiffness ratio
   d. Model response sensibility, e.g., Does the deformed shape make sense? (see 3/4.4.2)
   e. Numeric response sensibility, e.g., Are the stresses produced by the model in the realm of possibility?

3/4.2 Postprocessing Methods

Methods used for postprocessing of derived quantities from an FEA should be explained. The derived quantities include parameters such as stresses, design margins, factors of safety, etc.

The need and justification for applying correction factors for FEA results should be explained. The need for applying correction factors may arise due to the necessity to compare FEA results with design codes. Note that sometimes, the same FEA results can be interpreted and plotted differently in different post-processors. That is mainly because the default setting for the result visualization is different, e.g., presentation of averaged, non-average, nodal, or elemental values.

3/4.2.1 Result Accuracy

Since FEA is an approximate technique, the accuracy of simulation results is always dependent on the engineering approach applied and solver adopted. Where the analytical solution is not available, the analyst can use experimental data or verified numerical data as the benchmark to check the accuracy of the results. There are two major aspects to check:

1. computational accuracy
   a. strain energy including nominal and residual
   b. reaction forces and moments
   c. convergence test including mesh sensitivity study
   d. average and unaverage stress difference
2. correlation with experimental testing, analytical solution, or verified numerical data

a. strain gauge – stress, strain, and displacement comparison
   Note: The gauge “sensor” element is available is some FEA codes

b. natural frequency comparison

c. dynamic response comparison including modal or deformed shape comparison

d. temperature and pressure distribution comparison

In general, FEA results that have not greater than 10 to 15% difference from the benchmark results are a reasonable correlation and accurate calculation. This is according to automotive standards. If the deviation is more than 15%, one or several of the following input may not be correctly set up.

1. Geometric idealization of the structure
2. Idealization of the applied loading
3. Boundary conditions and/or contact definitions
4. Material properties and/models
5. Presence of residual stress or strain
6. Local geometric effects (e.g., weldment)
7. Local force effects (e.g., bolt with pre-tension)

It should be noted that there is a chance that the benchmark value may also have errors. In such a case, it is recommended that the analyst review the results together with the project technical supervisors or senior engineers.

3/4.2.2 View and Interpret Results

The displacement and animation for deformation should always be the first to view, and then any other output can be viewed. It is recommended [15] that the analyst should visually imagine how the object would deform for the given loading condition before seeing the result. If an excessive displacement or illogical movement of the object is observed, it may indicate errors in the FEA setup.

The exaggerated displacement or deformation can be plotted by controlling the scale factor in the post-processor. It is recommended to set a scale factor a scale higher than the default value (= 1), e.g., scale factor = 10.

To animate the result is another useful visualization technique. It can help in interpreting results from both static and dynamic analyses. The animated motions mimic the structural deformation due to the applied loads and constraint, and also provide insight into the overall structural response.

3/4.3 Displacement Results

In the design of ship structures, the primary result parameter of interest is stress. Most design criteria are expressed as allowable stresses. Although deflection criteria are not as numerous as stress criteria in design codes and standards, they can be just as critical. Stiffness requirements for various components of navigation and combat systems are often quite onerous. Stiffness requirements are often related to dynamic requirements in which the coincidence of equipment operating frequencies and those of the equipment-support structure system is to be avoided. As noted elsewhere, modeling for dynamic analysis is considerably more difficult than modeling for static analysis. This is particularly true for higher modes of vibration.

In interpreting displacements, it is essential to have an understanding of the accuracy of the FEA, how they vary for different response parameters, and the influence on the accuracy of modeling decisions made earlier.

In general, displacements are more accurately determined by FEA than stress.

The methods used for plotting the displacements of framed structures and certain plated structures in many FEA software packages may understate the actual accuracy. Beams are often plotted as straight lines. In reality, the displacement function for beam elements is a cubic polynomial. The same observation applies to plate bending elements.

In general, displacements in structures composed of beam and truss elements are accurately predicted within the limitations of the engineering model. In terms of the finite element model doubling the number of beam elements in, say, a grillage will not improve the accuracy of the result.

The response of two and three-dimensional structures is much more complex, and hence, in general, displacement results are sensitive to mesh refinement level. Therefore, interpreting displacement results in plated and solid models require more care. Gross errors are generally uncovered by the application of intuition and knowledge of previous analyses and physical experiments. More subtle errors are more difficult to uncover.

3/4.4 Force Results

As mentioned in Section 3/4.1.4, the check of force and moment balance helps in estimating the numerical accuracy. If the critical or limit load is the design criteria, the summation of force results should be interpreted and reviewed. Note that there are different approaches to interpreting the force results. For example, if distributed loads or fixation are applied, the beam elements (e.g., RBE-2 and RBE-3) can be employed to automatically output the force summation. Note that RBE stands for Rigid Body Elements. Note also that in general, RBE-2 and RBE-3 elements are used for transferring displacement and force loadings, respectively. For example, the RBE-3 element should be used for the multi-points constraint problems. The information of force results is also available in the output text file.

If the global or local stiffness is of interest, the force results should be extracted at the same location where the displacement results are extracted.

If the boundary condition nonlinearity (e.g., contacts and following force) is defined in the model, extra care is needed to check the global or local balance of forces and/or moments. For example, if the frictional contact is defined, the frictional forces on the master side (e.g., point or surface) should be equilibrium to the slave side.

As above mentioned in 3/2.3.5, the force results also indicate the level of plastic deformations in a Neuber correction-based fatigue analysis.

In a fracture analysis, the force results can be used to estimate the J values based on empirical equations in standards, e.g., [53]. Note that the J values from force results can be used as a benchmark value to review the accuracies of J values from methods such as contour integral or virtual crack extension.

3/4.5 Stress Results

As noted earlier, stresses are more difficult to predict accurately than displacements. Limitations in the finite element method are such that stresses are not normally continuous across boundaries between elements. For ease of interpretation of results, most FEA software averages stress in some fashion before presenting the results.

These results are presented attractively as stress contours in color plots, and the underlying discontinuous nature of the stresses may be obscured as a result of averaging processes, thus engendering a false sense of confidence in the results.

These problems can be compounded by misunderstandings in regard to the type of stress being plotted.

The analyst should review the magnitudes of the stresses presented to confirm that they are in the realm of possibility (e.g., are the stresses significantly higher than the material yield stress). Small areas of high-stress remote form the area of interest may be developed due to low mesh refinement or geometric simplifications and may be justified by the analyst. If the magnitude of the stresses can not be appropriately described, they may be an indication of modeling data or geometry inconsistencies.

Stress contours provide a good qualitative indication of the adequacy of the density of the mesh. Smoothly changing contours usually indicates that the mesh is suitably fine. Alternatively, stresses in adjacent elements can be compared. It is difficult to give firm qualitative guidance since the accuracy required depends on the nature of the analysis. A change in stress of more than + /- 20% would be regarded as unsatisfactory for design purposes. A discontinuity or abrupt change in the stress pattern across the elements, especially in the vicinity of the maximum stress, indicates that there is a need for local mesh refinement. In a fatigue analysis, the submodeling technique can be adopted to capture the high-stress gradient with finer meshing.

In addition to the default option of average stress (generally known as “simple average”), the current post-processor may also offer other plotting options, such as nodal value, elemental value, corner value, centroidal value, gauss point value, unaverage value, etc. The stress values plotted from the nodal, corner, and unaverage options are usually higher than those from default average and elemental values. If the analysis is being performed to meet a specific regulatory body rule set, then that regulatory body rule set should be reviewed to confirm see if average stress results (or some other stress results) are to be reviewed.

In a linear analysis or a unit cycle fatigue analysis, the analyst has to check whether the maximum stress level is higher than the yield strength.

In a fatigue analysis, it is recommended to use corner stress values rather than the nodal stress values.

In a fracture analysis, the calculation of J integral is based on the stress values from gauss integration points. In the literature, the nodal stress of crack front is generally used for calculating the crack-tip stress.

If the element erosion or damage failure algorithm is activated in the simulation, it is a case by case scenario. For example, if the maximum principal strain is the criteria index, the strain values of shell elements are calculated from integration points for both in-plane and through-thickness in each element.

Appendix E of DNVGL-CG-0129 (2018) standard [88,132] proposed an approach on how to interpret stress values close to the hot spot for the ship fatigue analysis.

Note that the stress index to check should be made on a case-by-case basis. However, in general, von Mises stress or equivalent stress should be reviewed or checked for linear and nonlinear analysis, except special FEAs (e.g., fatigue, fracture, damage, and vibration). In addition, the check of stress results (1st and 2nd principal stresses, and von Mises stress) sometimes can be used to understand how load transfers in a local or global structure. That may assist the analyst in revising the structural design.

3/4.5.1 Stress Components

The unknowns solved for in FEA are displacements (translations and rotations). These displacements are then used to calculate strains in the element, and hence the stresses. For some element types, intermediate steps are involved. The nature of inter-element stress discontinuities depends on the element type concerned.

In one-dimensional elements such as truss and beam elements, there are no discontinuities because the displacement functions are sufficiently detailed. For example, the standard beam element is based on cubic displacement and hence can represent linear variations of bending moment.

Two and three-dimensional lower-order elements generally have discontinuities in the stress field at element boundaries unless they are in a constant stress field. For plane and solid elements, stresses depend on displacement derivatives, and on curvature for plate bending elements.

The stress state at a point is defined by several stress components depending on the element type. These are summarized in Table 3/4-1.

Table 3/4-1 Stresses Represented by Element Type

ELEMENT TYPE	STRESSES
Truss	σ_x
Beam	σ_x, τ_y, τ_z
Plane Element	σ_x, σ_y, τ_{xy}
Plate Bending	σ_x, σ_y, τ_{xz} (Top & Bottom)
Solid	σ_x, σ_y, σ_z, τ_{xy}, τ_{yz}, τ_{xz}

The state of stress in plated and solid structures is generally quite complex and has to be combined in some way for design situations. Many failure theories have been developed wherein "failure" is said to have occurred when some equivalent stress exceeds the yield stress. The equivalent stress combines all the stresses acting at a point in the material. The most popular of these is the von Mises stress which is given by:

$$\sigma_e = \frac{1}{\sqrt{2}}\left[\left\{(\sigma_x-\sigma_y)^2 + (\sigma_y-\sigma_z)^2 + (\sigma_z-\sigma_x)^2\right\} + 6\ (\tau_{xy}^2+\tau_{yz}^2+\tau_{zx}^2)\right]^{1/2}$$

The use of the equivalent stress for checking the critical buckling stress is not appropriate. For buckling checks, normal stress (σ_x, σ_y) and shear stress (τ_{xy}), as appropriate, should be used. Generally, normal stresses will not be uniform across the panel. Where this is the case, it will be necessary to approximate the stress by a linear distribution for which there are standard buckling formulae. In some cases, the stress state may be biaxial and/or there may be significant shear stresses. To check these situations, it is usual to calculate the ratios of actual stress and critical stress for individual stress states and combine the effects using interaction formulae.

In a high cycle fatigue analysis, the maximum principal stress is usually of interest, and the key input for durability life calculations. Note that it depends on the failure modes to determine which principal stress index to use in the calculation. For example, if the stress ratio R = 0.1 (only tension), the 1st principal stress (i.e., maximum tensile stress) is used in a fatigue life estimation. On the other hand if the failure mode is caused by compression, the 3rd principal stress (i.e. maximum compressive stress) is used.

In a fracture analysis, all the stress components are used for stress intensity factor or toughness calculations. In a stress-based damage analysis, the ultimate tensile strength, together with the maximum principal stress, is used for estimating the critical state.

In a structural optimization analysis (e.g., topology and shape optimizations), the stress components or scalar derivatives (von Mises stress) can be used as the constraint condition in the objective functions.

The von Mises stresses should be reported for ductile materials, and the maximum principal stresses should be reported for brittle materials. Extra care is needed in the nonlinear analysis, where the true and engineering stresses are used. Some FEA software does allow the use of engineering stress and strain as input, but some software does not. The definitions of true and engineering stress are presented in Section 3/2.3.3.

3/4.5.2 Average and Peak Stresses

Except for the one-dimensional elements, each stress component for each element meeting at a node will be different. In FEA programs, various techniques have been developed to average stresses. The stresses in four adjacent membrane elements may look something like the distribution depicted in Figure 3/4-1.

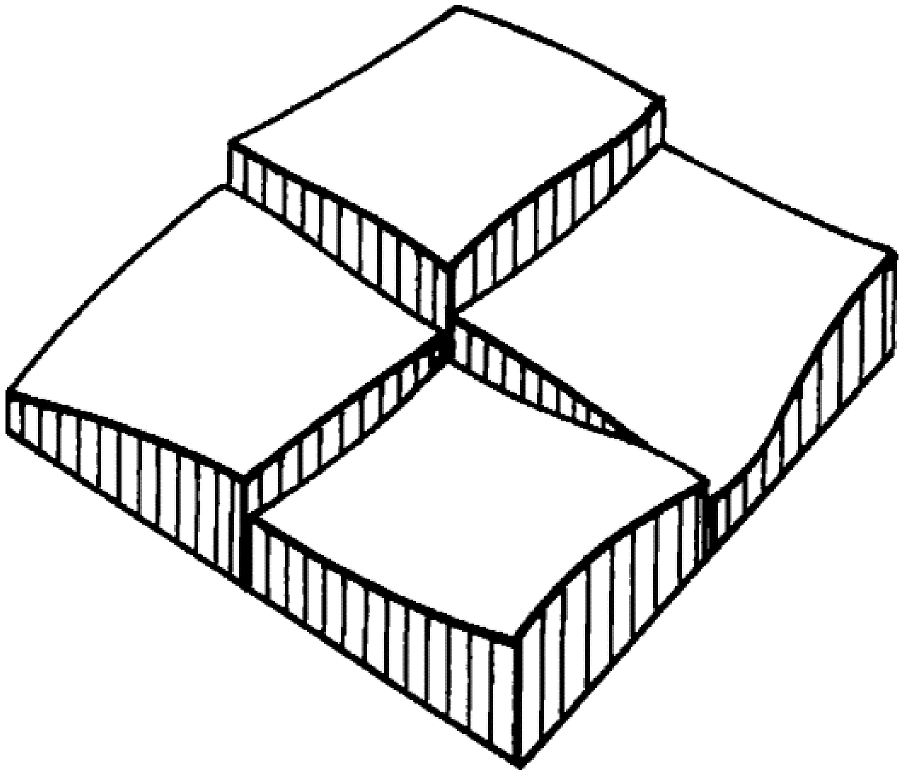


Figure 3/4-1 Distribution of Element Stresses

Stresses can be calculated at any point in the element. It has been shown, however, that depending on the element formulation, there are optimal points for computing stresses. In general, stresses are least accurate at corners, more accurate at mid sides, and most accurate at certain interior points. For two and three-dimensional elements based on the isoperimetric formulation (by far the most popular), these interior points are the so-called Gauss points (integration points). One popular method is to extrapolate the stresses calculated at the Gauss points to the nodes using a more suitable formula than the actual interpolation functions such as least squares. However, in some FEA software, the values at the Gauss points are copied to the nearest node without extrapolation, unless otherwise instructed. There are yet other methods for estimating nodal stresses.

Once the nodal stresses have been calculated for all elements contributing to the node, they can be averaged to yield average nodal stress. This will be done for all appropriate stress components. Averaged nodal stresses are much more reliable than element nodal stresses, although the extent of the stress discontinuity at the nodes should decrease with mesh refinement.

The different methods used by FEA software systems for extrapolating Gauss point stresses to the nodes is perhaps the main reason analyses of the identical problem, using different systems, can yield identical displacement results yet differing stress results. One technique used to overcome this problem is to employ dummy line elements in critical regions of the structure. In this technique, a dummy truss element is included in the model in the area of interest. An example of such a situation is the placement of such an element at the edge of an opening. The stress results from the truss element are directly calculated and are not dependent on extrapolation. The area of the truss element should be small enough to have a negligible influence on response. An area of t 2/100, where t is the thickness of the plate, is a reasonable upper bound. The use of such elements in the interior of plated structure, or indeed any structure, should be undertaken with caution. Line elements will yield only normal stresses in the direction of the axis of the element. In general line elements will not be aligned with the direction of principal stress.

The current popularity of producing smoothed stress fields in stress plots have hidden dangers. It hides large disparities in stress in adjacent elements. Large disparities indicate too coarse a mesh. A more revealing plotting technique is stress contours.

These should be smooth and not jagged. It is evident from Figure 3/4-2 that the contours in the coarse mesh are not smooth. This might be regarded as an unacceptably coarse mesh. An even more revealing method with modern postprocessing systems is stress isoband / isosurface plots. These plots will show a "checkerboard" type of distribution for unacceptable stress distributions.

The stress results from an FEA undertaken in support of design are often plotted in terms of von Mises stresses, although principal stresses and component stresses are. also sometimes plotted. There are two potential pitfalls that should be guarded against in interpreting stresses:

1. At nodes on boundaries between membrane elements of different thickness stresses, of course, cannot be simply averaged. A check should be made to ensure that the software does not perform averaging blindly in such a configuration.
2. Care should be taken in interpreting stresses at nodes where two-dimensional elements are not in the same plane. Clearly, simple averaging is not appropriate.

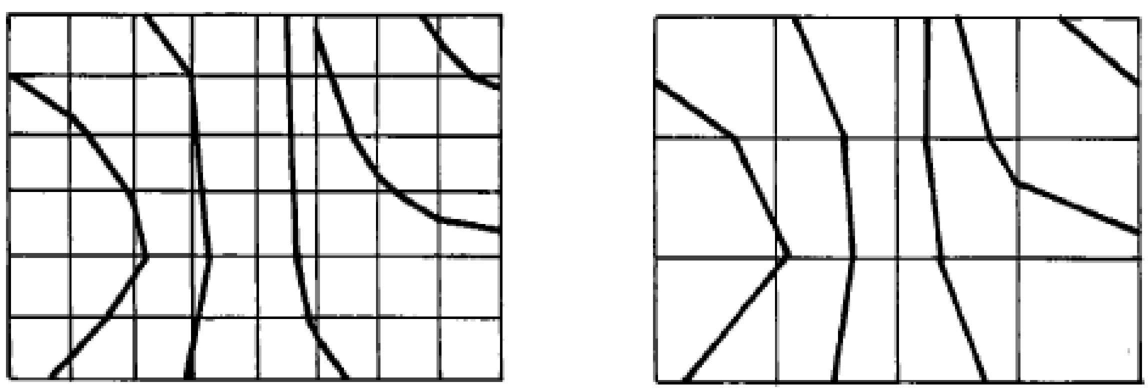


Figure 3/4-2 Stress Contours in Coarse and Fine Meshes

3/4.6 Strain Results

The strain results are usually viewed along with the stress results in a nonlinear analysis, especially for ductile materials. They can be plotted and interpreted in the same fashion as stress results. The strain levels can indicate the ductility and degree of deformation. Similar to the stress indexes, some FEA software allows the use of the engineering strain as input, and some only allow us to use the true strain. The calculated results are generally from the true strain in the nonlinear analysis.

In a linear analysis, if the stress level is greater than the yield strength, either a Neuber correction analysis or an elasto-plastic nonlinear analysis should be carried out to evaluate the degree of plastic deformations. The stress and strain curves from testing or analytical material models (e.g., Ramberg-Osgood) can be used in the simulation.

In the low cycle fatigue analysis, the strain results of both in-plane and through-thickness are examined, assuming 2D shell elements are employed.

In the analysis has an excessive plastic deformation (e.g., forming and crash), the strain results may become unreliable once the strain level exceeds the ultimate tensile strain. That is mainly because the predominant deformation mechanism changes from the grain dislocation to the void nucleation and void coalescence. It is recommended to use the material model with damage criteria if there is a need to capture the local failure modes. Note that the strain results are the key input to calculate the damage parameters (e.g., Johson-Cook model and Gurson model).

In “strain-based design,” the strain results at the critical fracture toughness level are used in the structural integrity analysis[169,170]. Note that the strain-based design method allows the structure from high strength steel to sustain a certain level of plastic deformation. It is different from the stress-design concept and can reduce the conservatism in the structural design.

The following four-strain indexes are generally viewed in the post-process and used for assessment:

1. Yield strain - The strain value at the yielding point.
2. Ultimate tensile strain - The strain value at the necking point.
3. Equivalent strain - The strain value calculated from von Mises stress. Note that in the explicit solver, the equivalent strain plotted at a given time step is usually a cumulated value [28].
4. Maximum principal strain - The strain vector (i.e., value and direction) for a given coordinate or a component of the strain tensor.

Note that if material models with multiple damage criteria are used in the simulation, the strain results may be interpreted based on other strain concepts (e.g., Green strain or Almansi strain). For some advanced damage models, additional parameters (e.g., Lode angle [171]) work along with the strain results.

3/4.7 Energy Results

The strain energy is usually viewed in an analysis. For structural design, the strain energy results can indicate the local stiffness or ability of a structure to sustain the load. For example, in a linear analysis (e.g., stiffness and vibration), a region with high strain energy results infers that the structure has a relatively lower stiffness but also indicates the subject high strain energy area is the predominant load paths. Therefore, in a structural optimization analysis, the strain energy results are always used as one of the key constraint factors. Not that the design philosophy in a crash-resistant design is to balance compliance and stiffness. This focus is different from the linear analysis, which is stiffness focused.

In addition to the strain energy, the following (but not limited to) energy indexes are also viewed in the nonlinear analysis:

- Internal energy
- Kinetic energy
- Contact or sliding energy
- Hourglass energy
- System damping energy
- Rigidwall energy

The internal energy not only includes the energies for 2D and 3D elements, but also 1D elements, e.g., spring, damper, and joint. Note that if element erosion or similar element deleting functions are used, there can be an energy imbalance.

3/4.8 Fracture Results

Nowadays, many FEA software can automatically calculate the stress intensity factor (K), T-stress

(T), J integral (J), and C* integral (C*) results and export the results to text files. However, other fracture parameters and related damage parameters, e.g., CTOD and CTOA, require “labor-intensive” postprocessing. That is mainly because the parameter either have different definitions (e.g., CTOD) or are under development (including user-defined element and/or materials).

The fracture results from FEA analyses are often used as input for subsequent structural integrity analyses. For example, K is used for Pairs’ law-based fatigue life calculation; K or J is used for calculating the fracture ratio (K_r) in the failure assessment diagram (FAD) [12,172–177]. Figure 3/4-3 schematically shows a FAD, where L_r is the load ratio, and the cut-off line refers to the local plastic collapse state. The fracture results located in the “green” region are safe and acceptable. Note that in the fitness-for-service design codes [12,13,63,114,178–181], the different levels (or options) of assessment adopt different fracture results in the fracture resistance analysis.

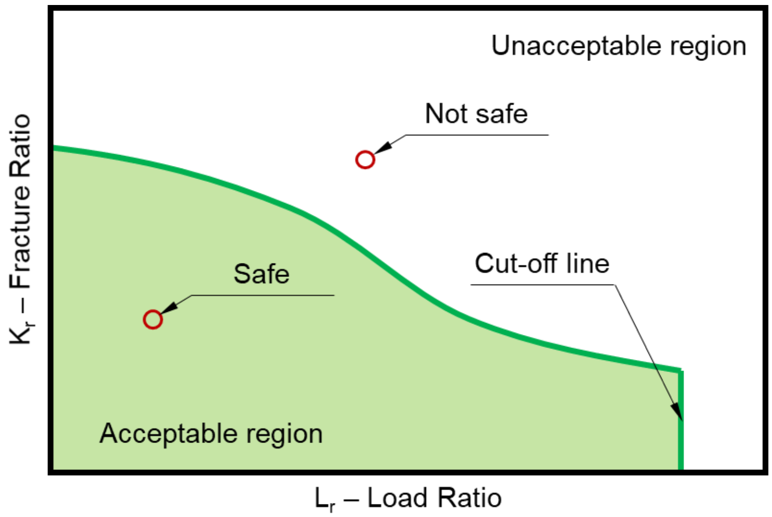


Figure 3/4-3 Schematic Failure Assessment Diagram (FAD) for Fracture Resistance Analysis

3/4.9 Fatigue Results

Several FEA software has embedded or optional add-ons for directly calculating fatigue life. These features may require either a third-party software (e.g., nCode) or manual post-processing for the fatigue life calculation. As above mentioned in sections 3/4.5 and 3/4.6, the stress or strain results are usually viewed before calculating the fatigue life. The calculation of fatigue results is relatively straightforward.

It should be noted that in the case of non-homogeneous materials, other types of stress indexes may be used for the fatigue life calculation, including:

- Absolute or signed maximum shear stress
- Absolute or signed von Mises stress
- Absolute or signed maximum principal stress

3/4.10 Vibration Results

3/4.10.1 Free Vibration Analysis Results

As discussed in Section 3/3.3.5.4, it is important that the mesh density is selected in accordance with the mode shapes that are evaluated. A course mesh density can result in accuracy issues when higher-order mode shapes are being evaluated.

The general observations regarding the sensibility of displacements and continuity of displacement should be considered as the first check of analysis results.

3/4.10.2 Frequency Response Vibration Analysis Results

There may be a significant error in Frequency Responses Vibration Analyses results. The FE prediction accuracy [88] in forced vibration analysis can be generalized as follows:

- Prediction of resonance frequencies for local ship structures to within 5-10% or less.
- Prediction of local response magnitudes to known input forces and machinery sources within a factor of 2-3, with results often being less than a factor of 2.

The above cautions should be taken into consideration when evaluating the results of this type of analysis.

3/5 CONCLUSIONS CHECK

This section deals with the final phase, conclusions and recommendations, of a finite element analysis (FEA). It is necessary to perform these checks to ensure that the loading, strength, and acceptance criteria are considered in arriving at the conclusions. This is a critical aspect of a finite element analysis since engineering decisions will typically be based on recommendations contained in this section. The following sections are grouped into five subsections dealing with various aspects of FEA conclusions.

3/5.1 FEA Results and Acceptance Criteria

A statement confirming that all analysis procedure quality assessment checks have been executed satisfactorily should be included.

Finite element analysis is an approximate solution technique, and, in spite of the careful effort, the results can only be approximations of the real solution. Therefore, the FEA results should always be validated using an alternative method/s. The alternative method includes comparison with experimental data, approximate analytical models, textbook and handbook cases, preceding numerical analyses of similar problems, numerical analysis of a related but simpler problem, and results for the same problem predicted by a different program (which could be based on a different numerical method). Many closed-form solutions of structures with simple geometry are available in handbooks and manuals, which could provide a suitable means for comparison. Numerical analysis using FEA of similar but simpler models could also be used for comparison. An example could be the use of a grillage model to check the results of a finite element model of the typical deck structure.

Despite the remarks made in the previous paragraph, the results from alternative solution methods should also be treated cautiously. Analytical models incorporate idealizations, mistakes may be made in the calculations, textbooks and handbooks may contain errors, numerical solutions are subject to errors in coding and in data preparation, and experiments may be improperly performed, and the results misinterpreted. Therefore, when the FEA results do not compare well with alternative methods, the possible reasons should be investigated.

The results should be presented so that they can be easily compared with the design/acceptance criteria. Finite element analysis results are identified based on node numbers and element numbers. These should be translated into the actual physical problem. For example, in a lattice mast, the members that do not meet the safety requirements should be highlighted on a figure of the model for easy identification.

When the FEA results do not meet the acceptance criteria, possible reasons should be explored and documented. In the case of large deviations, further justification regarding the validity of the FEA results should be provided.

The results should be assessed based on the knowledge of the physical problem. For analyses of high importance, an independent assessment should always be done by a qualified and experienced person.

3/5.2 Load Assessment

In case of discrepancies in the results, the loading applied to the model should be reviewed as part of the investigation into the source of the problem. The appropriateness of the types of loads, load cases, magnitudes, directions, load combinations, load factors, boundary conditions, etc., should be reviewed.

The loads applied to a finite element model are approximations of the actual loads. The analyst should provide a general description of the method used to approximate the actual loads. If the load distribution is simplified to more regular or uniform distribution, this should be justified to ensure that the simplified load distribution closely approximates the actual distribution in magnitude and direction. For example, if concentrated forces, at nodes, are used to approximate a pressure distribution, the calculations used in assigning the values of nodal forces should be explained. When concentrated forces are used to duplicate pressure, it is important that the load is applied such that the resultant acts through the center of pressure.

Details on load factors used in the analysis should also be provided. The information on whether the loads are based upon serviceability limit states or ultimate limit states should also be provided.

Finally, an assessment of the accuracy of the applied loads should be used in describing the results from the analysis.

3/5.3 Strength/Resistance Assessment

In design situations using traditional methods, the practice is to apply a nominal design load to the structure and compare the computed stress with some allowable stress. The latter is usually some fraction of the yield stress or the theoretical buckling stress.

In the modeling process, several assumptions are made, which may, or may not be, conservative. An assessment of conservatism, or otherwise, should be made particularly in regard to the underlying assumptions implicit in the design criteria that are being applied. Often design criteria have evolved with design methods based on hand calculation. Different design criteria may be appropriate if FEA is used to compute stresses. This factor should be included as part of the strength/resistance assessment.

In making an assessment of the strength/resistance of the structure based on the results of an FEA, appropriate allowances should also be made for factors that were not accounted for in the analysis. Some of these factors include geometric and material imperfections, misalignments, manufacturing tolerance, initial strains, and corrosion.

The design criteria being applied may implicitly include an allowance for some, or all, of these factors.

3/5.4 Accuracy Assessment

In assessing the accuracy of FEA results, factors to be considered include the level of detail and complexity modeled, type of behavior modeled, mesh refinements, etc. In deciding the level of detail, the analyst would necessarily have omitted some elements of the structure. The effect of these on the results should be assessed. The limitations of the element type/s used should also be assessed with respect to its capacity to model the required behavior.

For example, the element type used might model only the membrane actions when both membrane and bending behavior are significant.

The joints and connections between members might not be properly detailed in the model, making the model behave in a significantly different way. The effect of the mesh density used on the results should also be assessed. Simple parametric studies on smaller models may sometimes be necessary to assess the accuracy of the mesh used in the model.

Performing checks on the numerical accuracy of an FEA is difficult. Generally, reliance is placed on a combination of following good modeling practices and on parameters output by the FEA program. Common parameter outputs include the ratio of the largest and smallest stiffness found in the stiffness matrix and the so-called residuals.

Unfortunately, satisfactory values for these parameters are necessary, but not sufficient, conditions for satisfactory numerical performance.

The acceptability, or otherwise, of the ratio of the largest to smallest stiffness depends on the computer hardware and software, and it is suggested that the guidance provided by the warning and error messages issued by the FEA program are heeded.

The frequency response vibration analysis range requirements discussed in Section 3/2.8.3 and accuracy issues discussed in Section 3/4.10 should be taken into consideration.

3/5.5 Overall Assessment

All of the above-described factors should be used in conducting an overall assessment of the FEA. The results of this overall assessment should be included as part of the documentation. Deviations, if any, from the actual response should be justified.

Recommendations, if any, for future FEA should be clearly stated. If there is an anticipated continuation for the project at a later date, information on all computer files, documentation, etc. should be documented.

PART 4

BENCHMARK PROBLEMS FOR ASSESSING FEA SOFTWARE

4/1 INTRODUCTION

The assessment methodology presented in PART 2 includes a requirement that suitable FEA software be used. The determination of the suitability of a particular FEA code should involve, among other things, an assessment of its capability to analyze the types of problems that will be applied. This part describes the development and application of a series of standard benchmark test problems that can be used to assess the suitability of new, or significantly modified, FEA software for ship structure analysis.

As a means of qualifying FEA software, the benchmarks represent a category of tests between that of large-scale validation efforts and that of smaller-scale verification problems. The actual structural behavior of even the simplest component depends on such a large number of variables of varying complexity that isolating the response modeled by FEA codes is extremely difficult. As such, large scale validation of FEA software is typically very complex and expensive, often requiring comparison of FEA predictions with physical test results. Although such validation testing may be a requirement for certain critical structure applications, it is not a practical approach for assessing FEA software on a routine basis.

Most FEA software developers perform verification tests as part of their internal quality assurance procedures. For example, the verification test set for the ANSYS FEA program consists of over 5500 test cases at revision 5.1. Some software developers publish and / or make available a subset of the tests in the form of examples or verification manuals. Other developers include "textbook" verification examples in their marketing media. Verification problems of this sort are usually simple and small-scale in character and typically have closed-form theoretical solutions. They are generally designed to test a very specific aspect of the FEA code, such as the numerical performance of a certain type of element in a certain geometry, loading condition, and type of analysis. However, the verification problems rarely resemble "real life" engineering problems involving irregular geometries with large numbers of element types, in various shapes and sizes, combined with several load types and boundary conditions. Thus, while verification problems of the type described above are a necessary step in verifying and validating FEA software, they are not sufficient on their own.

The benchmark problems presented here are intended to represent the next step in ensuring that the candidate FEA software is appropriate for the FEA of linear elastic ship structure. The benchmarks are summarized in Figure 4/1-1 and cover a range of typical problems and requirements encountered in "real life" ship structure FEAs. The problems involve simple configurations of a number of representative ship structures but are detailed enough to retain the key characteristics of the structural assembly or detail. The problems typically require that several types of elements, materials, and loads be used in combination. An attempt has been made to design the benchmarks such that, collectively, all key features that determine the quality of FEA packages are addressed. The benchmark problems are described in 4/2 with complete details given in Appendix D.

The benchmarks are designed to exercise the FEA software rigorously without making the evaluation process overly demanding. The problem size has been limited to a maximum of 200 nodes to ensure that the process of benchmarking new and modified software is not onerous. The 200-node limit should also allow, in some cases, for the user to test demonstration or evaluation versions of FEA software. Such versions are usually based on the "full" versions of the FEA code, but typically have limits on the number of nodes and elements that can be modeled. These are usually available from the FEA software developer at a small nominal fee to allow testing and evaluation prior to making a larger financial commitment.

The benchmarks do not have closed-form theoretical solutions. Instead, the results from analyzing the benchmark problems using three well known FEA software programs are used to establish the reference benchmark results. The three programs used were ANSYS, MSC/ NASTRAN, and ALGOR and are described in 4/3. The presentation and discussion of the benchmark results are included in Appendix D.

Care has been taken to ensure that the test models for the benchmark problems are sufficiently detailed or refined that the results approach a converged solution. Element formulations, stress averaging / extrapolation algorithms, and other aspects of FEA software performance tend to be optimized for ideal configurations. Testing different FEA software of an ideal configuration (e.g., a rectangular plate with uniform rectangular elements) will tend to give virtually identical results. However, once the FEA model deviates from an ideal configuration, as is the case for the benchmarks, differences in the results manifest themselves. In these circumstances, the rate of convergence of results from different FEA programs may differ. Ensuring that the results obtained by the test models are near a converged solution should minimize any discrepancies that can be attributed to poor mesh design of the benchmark test models.

New or significantly modified, FEA software can be evaluated by exercising the software with the benchmark problems and comparing the results obtained with the reference benchmark results. The process by which this should be accomplished is presented in 4/4.

WARNING

The benchmark problems and associated FEA models presented in this document are intended for the express purpose of evaluating FEA software for ship structural analysis applications. While attempts have been made to ensure that the FEA models follow good modeling practices, they should not necessarily be regarded as appropriate for any purpose other than that for which they are intended.

Features		Benchmark Problem				
		BM-1 Reinforced Opening	BM-2 Stiffened Panel	BM-3 Isolation System	BM-4 Mast	BM-5 Bracket Detail
Analysis Types	2D	●				
	3D		●	●	●	●
	Static	●	●		●	●
	Modal		●	●	●	
Element Types	Mass			●	●	
	Spring			●		
	Truss / Spar				●	●
	Beam			●	●	
	Membrane	●				
	Shell		●			●
	Brick					
Load Types	Force	●			●	
	Pressure		●			
	Acceleration				●	
	Displacement					●
Boundary Conditions	Displacement	●	●	●	●	●
	Symmetry	●	●			
Results	Displacement	●	●		●	●
	Reactions				●	●
	Stress	●	●		●	●
	Frequency		●	●	●	

Figure 4/1-1 Summary of Ship Structure FEA Benchmark Problems

4/2 THE BENCHMARK PROBLEMS

The ship structure FEA benchmarks include the following problems:

1. Reinforced Deck Opening
2. Stiffened Panel
3. Vibration Isolation System
4. Mast
5. Bracket Detail

Figure 4/1-1 summarizes the main modeling and analysis features that the benchmarks are intended to test. The following sections provide a summary description of the benchmark test problems. Complete details of the benchmark problems are presented in Appendix D.

4/2.1 BM-l Reinforced Deck Opening

Openings and penetrations are among the most commonly encountered sources of high-stress levels in surface ship structures. In most cases, the openings are reinforced by coamings or insert plates to attenuate the resultant stress concentrations. FEA may be required to evaluate the stress levels and the effectiveness of the reinforcement technique. This benchmark tests the capability of

FEA packages to analyze this category of a ship structure problem and is shown in Figure 4/2-1. The benchmark tests the FEA program's capability to analyze a plane stress concentration problem using either 4-node or 8-node shell elements. However, it goes beyond the classical hole-in-a- plate problem by including two plate thicknesses for the deck and the reinforcement insert plate, and by including stiffeners in the plane of the deck.

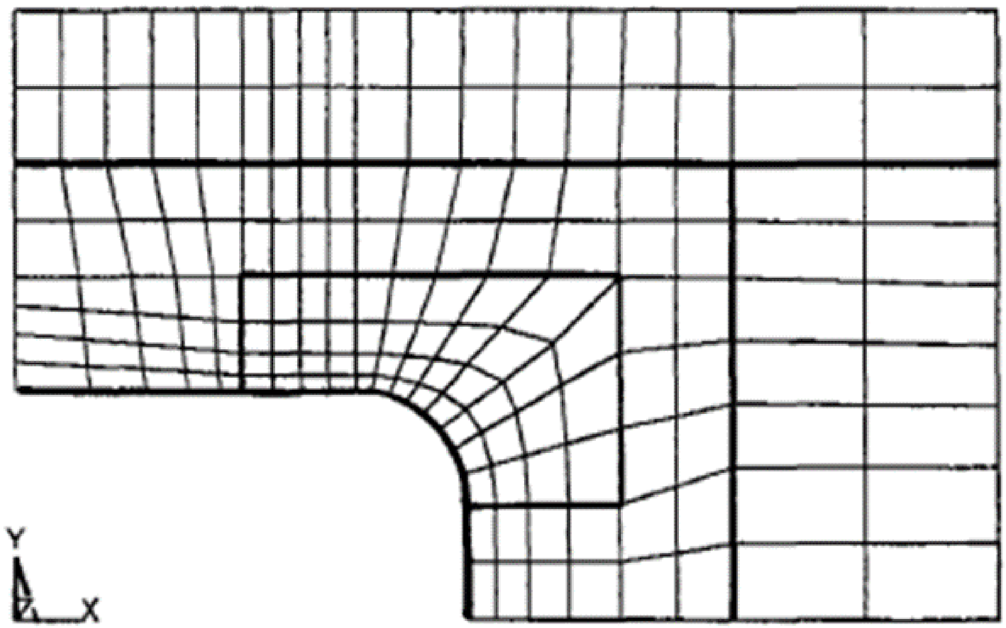


Figure 4/2-1 Benchmark Problem BM-1 Reinforced Deck Opening

4/2.2 BM-2 Stiffened Panel

Stiffened panels are the most common structural component in ships. This benchmark tests the capability of FEA packages to analyze this type of structure using the various plate and stiffener element modeling techniques. These include :

- 4-node shell elements for plate and in-plane beam elements for stiffeners
- 4-node shell elements for plate and off-set beam elements for stiffeners
- 4-node shell elements for plate and stiffeners
- 8-node shell elements for plate and stiffeners

Both static and modal analyses are conducted for each model. The static analysis involves surface pressure loading causing out-of-plane panel bending under symmetric boundary conditions (i.e., quarter model). The modal analysis tests the capability of the program for calculating natural frequencies and mode shapes under symmetric and antisymmetric boundary conditions.

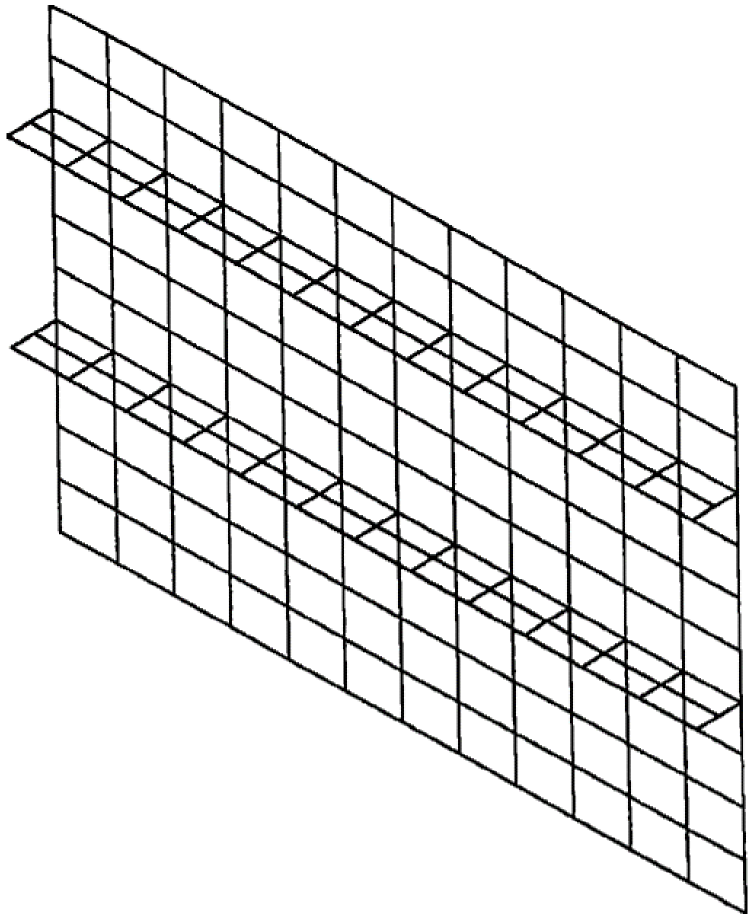
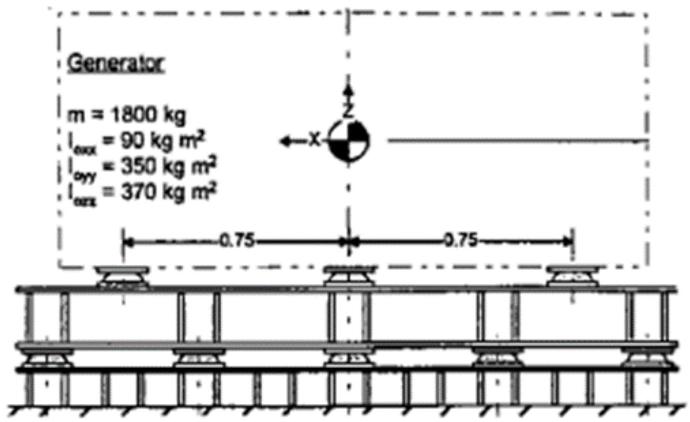


Figure 4/2-2 Benchmark Problem BM-2: Stiffened Panel

4/2.3 BM-3 Vibration isolation System

Vibration isolation systems are often required for ship equipment and machinery. FEA analyses may be used to optimize the isolation system and ensure that vibration and shock design criteria are achieved. This benchmark considers a 12 degree of freedom system consisting of a generator that is mounted and isolated on a raft structure, which is, in turn, isolated from the foundation structure. The problem is summarized in Figure 4/2-3. Some of the key testing features include of this benchmark include:

- Modal analysis;
- Point mass including rotational inertia terms (to the model generator)
- Spring elements with stiffness in three directions; and
- "Rigid" beam elements connecting generator mass and isolator springs to the raft.



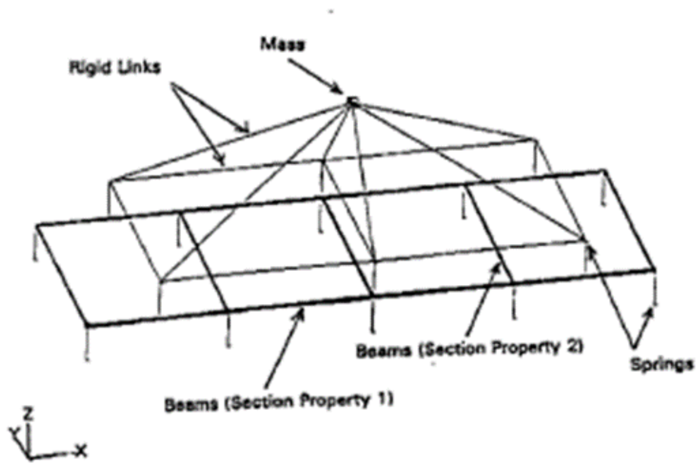


Figure 4/2-3 Benchmark Problem BM-3: Vibration Isolation System

4/2.4 BM-4 Mast Structure

Mast structures on ships must be designed to withstand environmental loads (wind and ship motions). Masts on naval ships usually have additional requirements for resisting shock and blast loading. The mast benchmark problem is summarized in Figure 4/2-4, and the key modeling and testing features include :

- Beam elements (with axial and bending stiffness) for main legs and polemast
- Axial line elements (spar, truss, rod) for braces
- Point mass elements for equipment "payloads"
- Inertial loading in three directions combined with nodal force loading
- Two materials (steel and aluminum)
- Modal analysis

While the benchmark problem is that of a lattice mast structure, it can be used to assess the FEA program's capabilities for modeling similar frame or truss-like structures such as booms and derricks, especially where beam and spar elements are used in combinations.

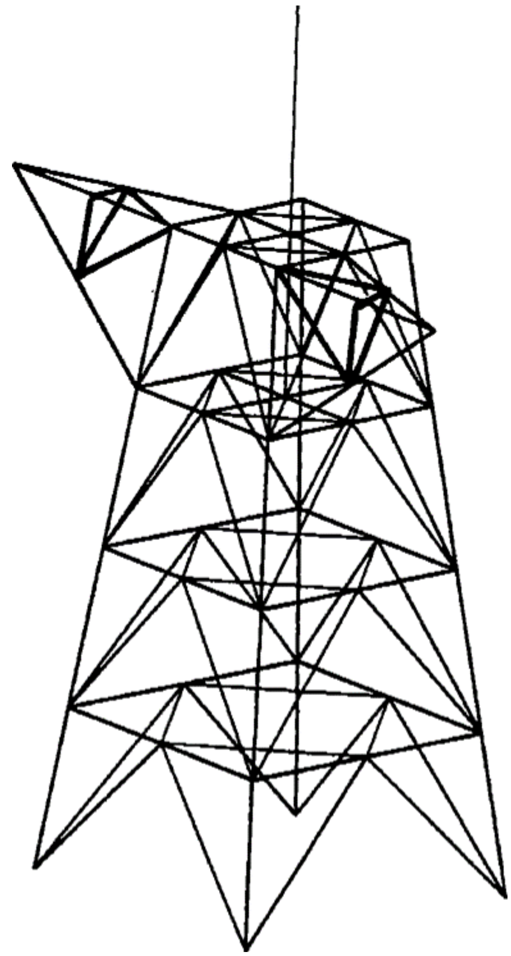


Figure 4/2-4 Benchmark Problem BM-4 : Mast Structure

4/2.5 BM-5 Bracket Connection Detail.

Welded connection details on ships are subject to fatigue loading. Poorly designed or constructed details can lead to premature fatigue failure. Finite element methods are frequently used to calculate fatigue stresses and to aid in the development of improved detail geometry and configurations. This benchmark problem is summarized in Figure 4/2-5. Some of the key modeling and testing features of this benchmark include:

- 3·0 geometry with shell elements of varying thicknesses;
- Axial line elements for bulkheads, deck, and flange of the bracket; Transition from coarse to fine mesh at the bracket weld;
- Prescribed nonzero nodal displacement boundary conditions.

The latter feature was included since, in many cases, the boundary conditions for a detail FEA are obtained from displacements and loads derived from a global FEA.

This particular bracket detail problem is complicated by the existence of a stress singularity at the end corner or toe of the bracket. In a linear elastic analysis, the stress at this point is theoretically infinite. Refining the finite element mesh gives

progressively higher stresses, which are meaningless. One method which is commonly used to get around this problem is to use the so-called "hot spot" stress. In calculating the hot spot stress no account is taken of the weld geometry, and in an idealized finite element representation (ignoring the weld) the stress is equal to the value at about one plate thickness from the corner (Chalmers, 1993).

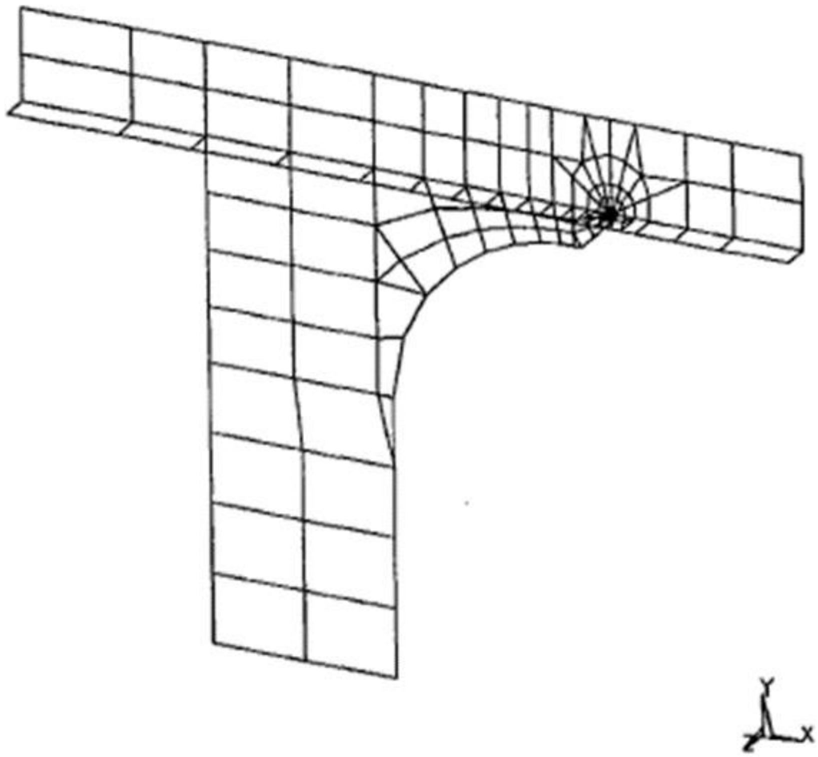


Figure 4/2-5 Benchmark Problem BM-5 : Bracket Detail

4/3 THE BENCHMARK TEST FEA PROGRAMS

As previously mentioned, the benchmark problems do not have readily obtainable theoretical solutions. Instead, the results from analyzing the benchmark problems using three well known FEA software programs are used to establish the reference benchmark results. The three programs used were ANSYS, MSC / NASTRAN, and ALGOR.

The ANSYS FEA program is developed and marketed by ANSYS Inc. of Houston, PA. ANSYS is a mature, general-purpose FEA program that has been commercially available on various computer platforms since 1970. It includes extensive analysis capabilities, a large, comprehensive library of elements, and extensive pre- and post-processing capabilities. The ANSYS Version 5.1 program was run on a DEC 3000 workstation for the benchmark test cases.

The MSC / NASTRAN FEA program is developed and marketed by The MacNeal- Schwendler Corporation, Los Angeles, CA. Traditionally it has been most widely used by the aerospace industry, having evolved from the National Aeronautics and Space Administration (NASA). MSC / NASTRAN is a very comprehensive and mature FEA program that has been commercially available for several decades. It is, to some extent, regarded, along with ANSYS, as the industry standard. MSC / NASTRAN for Windows 1.0 on an IBM 486 PC was used for the benchmarks.

The ALGOR FEA program is developed and marketed by ALGOR Inc., Pittsburgh, PA. It was one of the first FEA programs to be developed, especially for the personal computer, and has become one of the most popular FEA programs for PC applications. The program features a relatively wide range of modeling and analysis capabilities.

4/4 APPLICATION OF BENCHMARKS FOR ASSESSING FEA SOFTWARE

The intended application of the benchmarks is to provide a methodology for assessing FEA software. This assessment consists of modeling and analyzing the benchmark problem with the FEA software and comparing the results with those obtained by the reference FEA programs as presented in Appendix D. The data files for the benchmark problems in ANSYS, NASTRAN, and ALGOR formats may be obtained by contacting the Ship Structure Committee.

As was discovered in the benchmark results of the three reference FEA programs, there are liable to be differences between the results obtained by different FEA software packages. The differences may arise from a multitude of factors ranging from the numerical accuracy of the hardware and software platforms to different element formulations, solution algorithms, and results presentation techniques to actual errors or limitations in the FEA software. The question that arises is how much variation or deviation from the reference results is acceptable.

The authors suggest the following approach be used to judge the acceptability or otherwise of the benchmark results for any FEA software:

1. Result differences less than 2% with respect to the reference FEA software results for displacements, reaction forces, and lower mode natural frequencies are considered acceptable. The 2% limit is generally within what would normally be the required engineering accuracy for these types of problems.
2. Result differences between 2% and 5% are probably acceptable for beam and plate element stress results and higher mode natural frequencies. However, the user should endeavor to ensure that there are plausible explanations when differences get much past 2%. This may involve further testing of the problem by, for example, refining the FEA mesh or switching the analysis options to I from the defaults used by the FEA program.
3. Result differences greater than 5 % should be considered abnormal and require an explanation. If a reason cannot be found, the developer of the FEA software should be contacted and requested to investigate the difference. Where no explanation exists, the FEA software should probably be viewed as suspect for the particular type of analysis covered by the benchmark problem.

Particular attention should be paid to ensure that the proper loads and boundary conditions have been applied and that the stress contours, deformed shape or mode shapes (depending on what is applicable) are consistent with the reference results. The user should also be sure of the default analysis assumptions and solution techniques used by the software. These can be especially important for problems where transverse shear effects need to be considered, or when performing modal analyses. The user should also be aware of how the FEA software extrapolates and I or averages plate element stress results at nodes.

The benchmarks are necessary but by no means a complete method of validating an FEA program. The benchmarks primarily check that a particular FEA code will perform and produce results that are consistent with the three reference FEA codes. However, it is strongly recommended that users of new or significantly modified FEA software become fully aware of all features and limitations of that program for the particular applications involved. This should include testing the software on simplified versions of the main problems of interest in order to build confidence in the modeling approach, choice of elements, mesh densities, etc. as discussed in Part 3, Section 1.3.

PART 5
ADVANCED ANALYSIS SAMPLE APPLICATIONS

5/1 IMPACT AND PLASTICITY

5/1.1 Introduction

The FE model of a ship bow structure under frontal collision (see Figure 5/1-1) is described in this section. The objective of this study is to characterize the collision resistance of the bow structure and evaluate whether the ship structure has sufficient resistance to collision accidents. The nonlinear finite element analysis is conducted to assess the structural strength during the event. Note that the examples presented in this section were initially developed by researchers (e.g., Dr. Martin Storheim) from NTNU – the Norwegian University of Science of Technology and now adopted by DNVGL-RP-C208 [106].

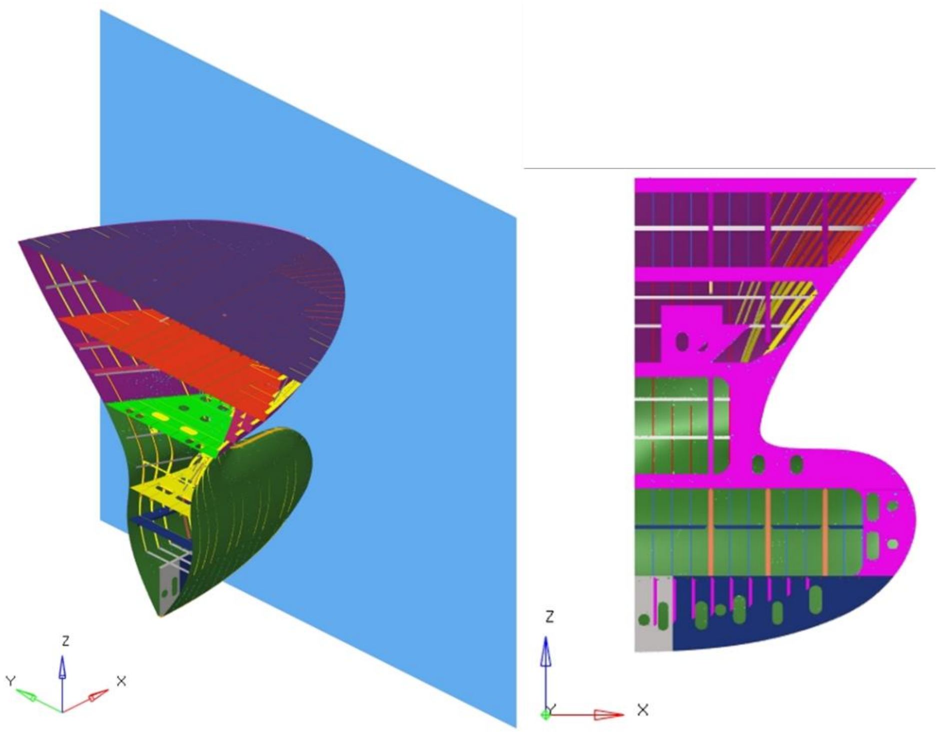


Figure 5/1-1 FE Models of Ship Bow Under Frontal Collision

5/1.2 Engineering Model

5/1.2.1 FEA Software

LS-DYNA® shared memory parallel (SMP) version R8.0.0 with double precision was employed for calculating the structure response. Altair® HyperMesh and HyperView are used for pre- and post-processing, respectively.

5/1.2.2 System Units

In this study, the unit system for this model is defined as follows:

1. Length millimeter [mm]
2. Mass metric tons [t]

3. Time second [s]
4. Force Newton [N]
5. Pressure megapascal [MPa] or [N/mm^2]
6. Stress megapascal [MPa] or [N/mm^2]
7. Strain no unit [%]
8. Velocity millimeter per second [mm/s]
9. Energy millijoule [mmJ]

5/1.2.3 Geometry assumptions

As shown in Figure 5/1-1, only bow structure is modeled and the rest of the ship structure is not included. Note that this is a partial ship structural analysis, and the geometry assumptions of stiffeners, supporting members, and openings follow the rules of DNVGL-CG-0127 [9].

5/1.2.4 Material properties

The bow structure is built of the S235 and S355 grade steel. The stress-strain relationship follows the Hollomon-type power-law relationship as follows:

$$\begin{cases} \sigma = \sigma_{yield} & if\ \varepsilon_p \leq \varepsilon_{p,yield-2} \\ \sigma = K\left(\varepsilon_{p,eff} + \varepsilon_p\right)^n & if\ \varepsilon_p > \varepsilon_{p,yield-2} \end{cases} \quad \text{Eq. (5.1)}$$

where n is the strain hardening exponent; K is the strength index or strength coefficient; ε and σ with subscripts are the strain and stress indexes. To account for the presence of the strain plateau, the work hardening is delayed until the plastic strain level reaches the plateau strain $\varepsilon_{p,yield-2}$. The effective plastic strain in the above equation is defined as

$$\varepsilon_{p,eff} = \varepsilon_{p,yield-1} - \varepsilon_{p,yield-2} = \left(\frac{\sigma_{yield}}{K}\right)^{1/n} - \varepsilon_{p,yield-2} \quad \text{Eq. (5.2)}$$

where $\varepsilon_{p,yield-1}$ is the strain at the initial yield point. Figure 5/1-2 schematically shows definitions of stress and strain indexes in the equation mentioned above. The strain hardening exponent n is defined as:

$$n = ln(1 + \varepsilon_{UTS}) \quad \text{Eq. (5.3)}$$

where ε_{UTS} is the ultimate tensile strain.

The engineering stress-strain data are converted to true stress-strain data by using the following equation.

$$\varepsilon_{true} = ln\left(1 + \varepsilon_{eng}\right) \quad \text{Eq. (5.4)}$$

$$\sigma_{true} = \sigma_{eng} exp(\varepsilon_{true}) \quad \text{Eq. (5.5)}$$

The parameters defining true stress-strain data are as follows:

1. Young's modulus E [MPa] 210,000
2. Poisson's ratio v [-] 0.3
3. Proportional stress σ_{prop} [MPa] 285.8 (S235), 384 (S355)
4. Yield strength σ_{yield} [MPa] 318.9 (S235), 428.4 (S355)
5. Yield strength – 2 $\sigma_{yield,2}$ [MPa] 328.6 (S235), 439.3 (S355)
6. Yield strain – 1 $\varepsilon_{yield-1}$ [%] 0. 4%

7. Yield strain – 2	$\varepsilon_{yield-2}$	[%]	2% (S235), 1.5% (S355)
8. Strength index	K	[MPa]	700 (S235), 900 (S355)
9. Ultimate tensile strength	ε_{UTS}	[%]	2% (S235), 1.5% (S355)
10. Strain hardening exponent	n	[-]	0.166

The density of steel is assumed to be 7.85×10^{-9} t/mm^3 in the study.

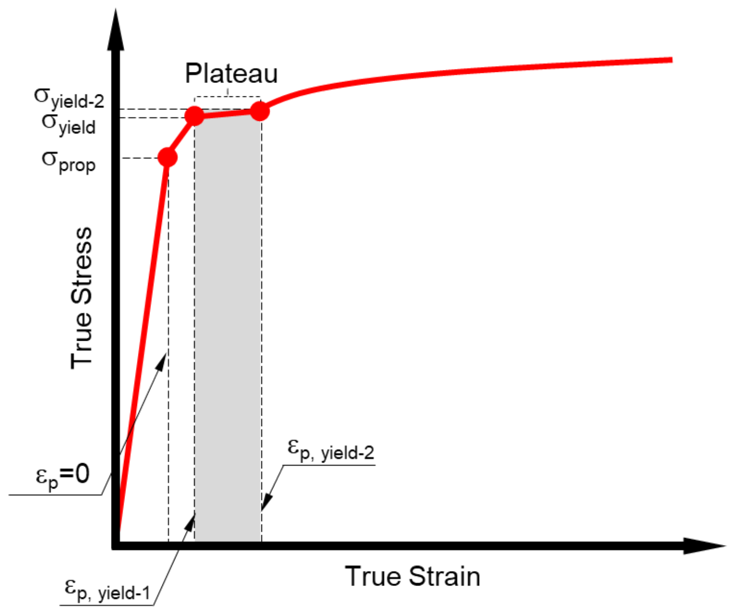


Figure 5/1-2 Schematic Stress and Strain Relationship With Yield Plateau and Strain Hardening Effects

The true stress and true strain curves of S235 and S355 for FE input are shown in Figure 5/1-3. Two curves are defined using the LS-DYNA command card – *DEFINE_CURVE _TITLE. The material card number 24 (*MAT_024 – *MAT_PIECEWISE_LINEAR_ PLASTICITY_TITLE) is employed associated with two defined stress and strain curves.

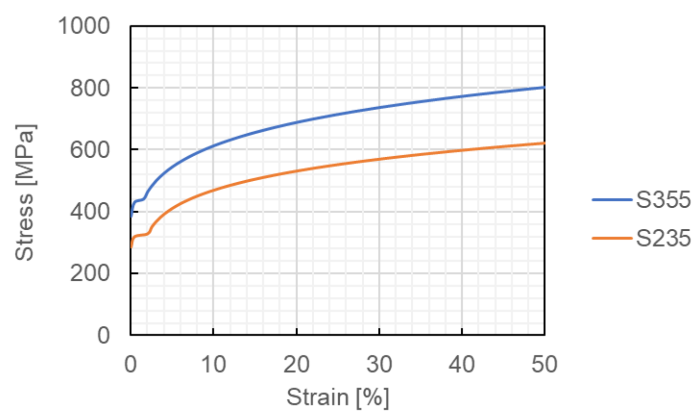


Figure 5/1-3 True Stress and True Strain Curves of S235 and S355 Grade Steel for LS-DYNA Input

5/1.2.5 Loading and Boundary Conditions

The bow structure is assumed to be fixed and impacted by a moving rigid-wall. Figure 5/1-4 schematically shows the boundary condition and applied load. All degrees of freedom of the bow structure boundaries are constrained. The rigid wall is moving toward the bow structure at a speed of 4 m/s along the x-direction, where the other five degrees of freedom are constrained. The initial velocity is defined by using command card *BOUNDARY_PRESCRIBED_MOTION_RIGID.

The penalty contact algorithm is used to define the contact among components. Two surface-to-surface contact pairs (*CONTACT_AUTOMATIC_SURFACE_TO_SUR-FACE_ID) are defined: (1) forecastle to rigid wall, and (2) bulb to a rigid wall. One single surface contact (*CONTACT_AUTOMATIC_SINGLE_SURFACE_ID) is defined for all bow structures. The friction effect of contact is not considered, i.e., static and dynamic friction coefficient (f_{static} and $f_{dynamic}$) = 0.

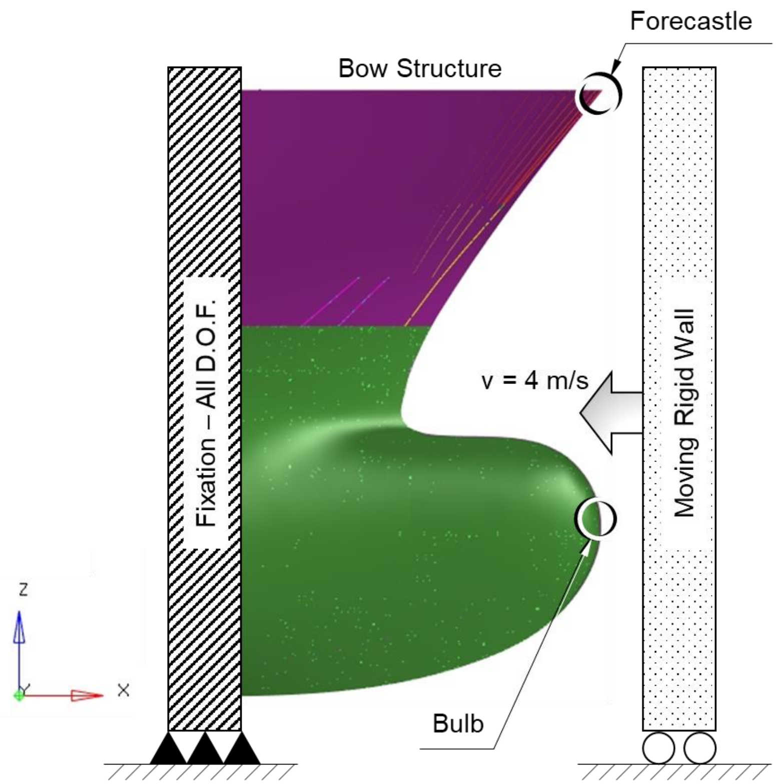


Figure 5/1-4 Schematic Illustration of Boundary Conditions and Applied Load

5/1.3 FE Models and Simulation Setups

5/1.3.1 Element

The bow structure is modeling by using 2D shell elements. The key parameters are summarized as follows:

- The Belytschko-Tsay reduced integration co-rotational formulation is adopted, i.e., ELFORM = 2.
- Five integration points through-thickness are defined to capture the variations of stress and strain gradients, i.e., NIP = 5.
- The shear correction factor is about 5/6, i.e., SHRF = 0.833.

5/1.3.2 Mesh design and quality control

The mesh was designed and tuned up by following DNVGL-CG-0127 [9], and the average element size is about 50 mm. The mesh of the FE model is revised based on the criteria shown in Table 5/1-1. Note the criteria are developed based on the description and discussion in PART 3. The mesh quality of the revised model meets all the criteria listed in Table 5/1-1.

Table 5/1-1 Crash problem with 50mm element size

#	Criterion	Ideal	Good	Warn	Fail	Worst
0	penalty value	0	0	1	2	10
1	min length	50	43.33	36.66	30	28.33
2	max length	50	62	73.2	90	112.9
3	aspect ratio	1	2	4	5	10
4	warpage	0	5	15	20	40
5	max angle quad	90	110	140	150	165
6	min angle quad	90	70	40	30	15
7	max angle tria	60	80	120	130	155
8	min angle tria	60	50	30	20	10
9	skew	0	10	50	60	75
10	jacobian	1	0.9	0.6	0.5	0.35
11	chordal dev	0	0.3	0.8	1	2
12	% of trias	0	3	4.5	6	9
13	taper	0	0.2	0.5	0.6	0.9

The bow structure FE model is constructed by using 336,261 nodes and 344,354 elements. The numbers of quad and tria elements are 335,146 and 9,208, respectively. It means there are 2.67% of tria elements in the model, which is less than the 6% “Fail” criterion in Table 5/1-1.

5/1.3.3 Include files

The FE models are decomposed into several key files:

- Bow structure
- Rigid wall
- Solution and output control
- System control (e.g., number of CPUs, size of memory, type of solver)

A master file is created to call all these key files using including file function – *INCLUDE.

5/1.4 Analysis Control and Solution Options

Several advanced features are also activated to control the simulation accuracy. For example:

- Hourglass control uses the stiffness-based method with Flanagan-Belytschko integration formulations, i.e., IHQ = 4 and QH = 0.03.
- Warping stiffness is captured by using Belytschko-Wong-Chiang formulations with full projection, i.e., BWC = 1, and PROJ = 1.
- Critical warpage angle of shell element is 20 degrees, i.e., WRPANG = 20.
- Mass scaling is applied to the critical element to reducing computational time, i.e., dt2ms = -3.0×10^{-6}.

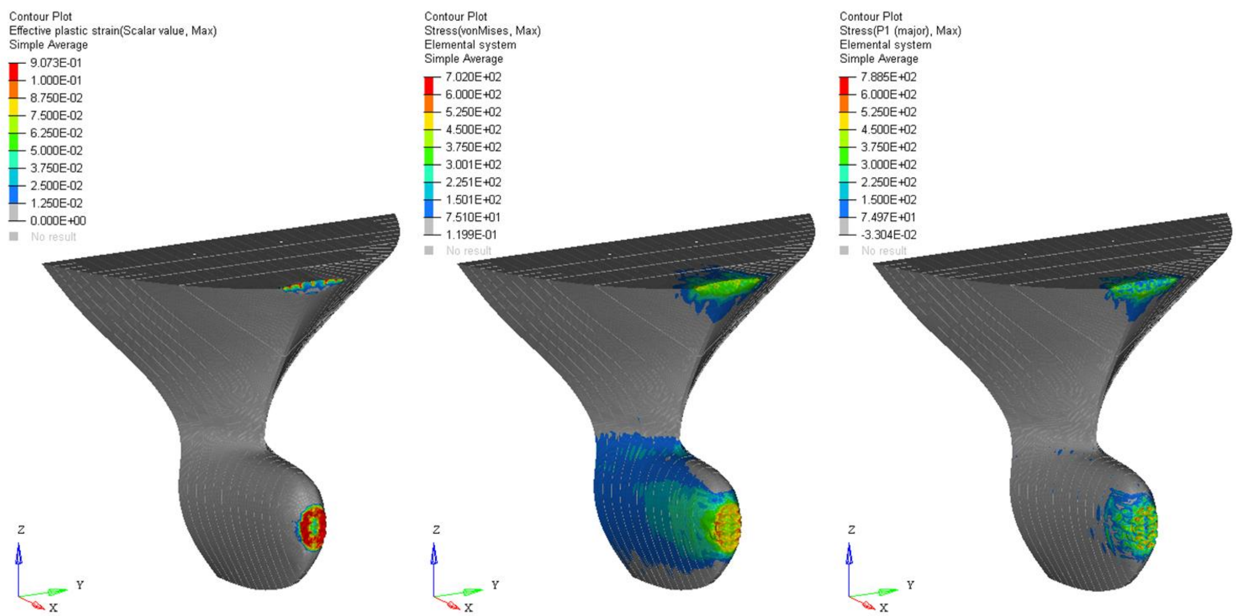
The following results are exported to ASCII text files at every 0.001s:

- Global data – *DATABASE_GLSTAT
- Material energy – *DATABASE_MATSUM
- Resultant interface forces – *DATABASE_RCFORC
- Sliding interface energy – *DATABASE_SLEOUT
- SPC fixation reaction force – *DATABASE_SPCFORC
- Boundary condition forces and energy – *DATABASE_BNDOUT
- Rigid body data – *DATABASE_RBDOUT

The binary results (*.d3plot files) of the entire model and contact interface are generated at every 0.1s, i.e., *DATABASE_BINARY_D3PLOT and *DATABASE_BIN-ARY_INTFOR.

5/1.5 Results

The results of von Mises stress, first principal stress, and effective plastic strain at the indentation of 2.0 m are shown in Figure 5/1-5. Figure 5/1-5 (a) shows that the bow structure has a local plastic deformation in the forecastle and bulb. It should be noted that the inertia effect is not considered in this study, which may induce different stress-strain distributions and failure mechanisms.



(a) Effective plastic strain (b) Von Mises stress (c) 1st principal stress

Figure 5/1-5 Results of Stress and Strain at the Indentation of 2.0 m

5/1.5.1 Impact Resistance

The impact force and indentation curves of the bow structure are shown in Figure 5/1-6. Note that the impact forces of bulb and forecastle to rigidwall are captured from the contact pairs. The design curve for a bow structure under the frontal impact is from standard DNVGL-RP-C204 [107] is also presented in Figure 5/1-6. It is seen that the resistance of the forecastle is similar to the design target at the indentation within 0.8m and less than the design target at the indentation between 0.8m and 2.0m. Compared with the forecastle, the bulb adsorbs the predominant impact energies and is the main structure to resist impact loads.

It should be noted that DNVGL-RP-C204 [107] does not consider the bow structure with a bulb. The current bow structure with both forecastle and bulb meets the design requirement of impact resistance.

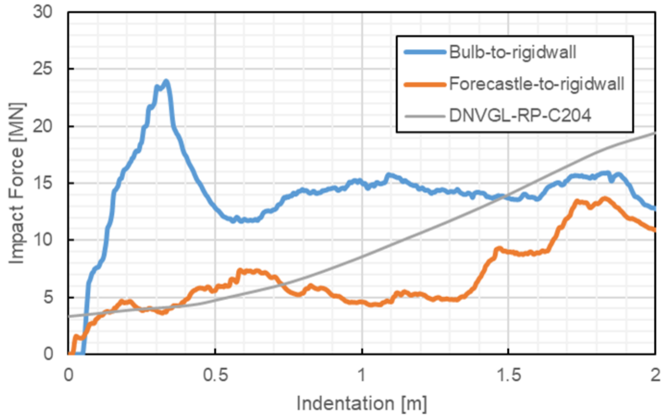


Figure 5/1-6 Force-Indentation Curves of the Bow Structures against the Design Curve from DNVGL-RP-C204

5/1.5.2 Failure mechanism

By assuming a flat rigid wall, the forecastle structure first contacts the rigid wall and then the bulb structure after time step of 0.01 s. The bulb tip flattens under the frontal impact shown in Figure 5/1-7. The predominant failure mechanisms are the local buckling of stringers and the plastic folding of structures. Figure 5/1-8 shows the local failure mechanisms of the bulb at two different time steps. The similar failure mechanisms are also observed in the forecastle.

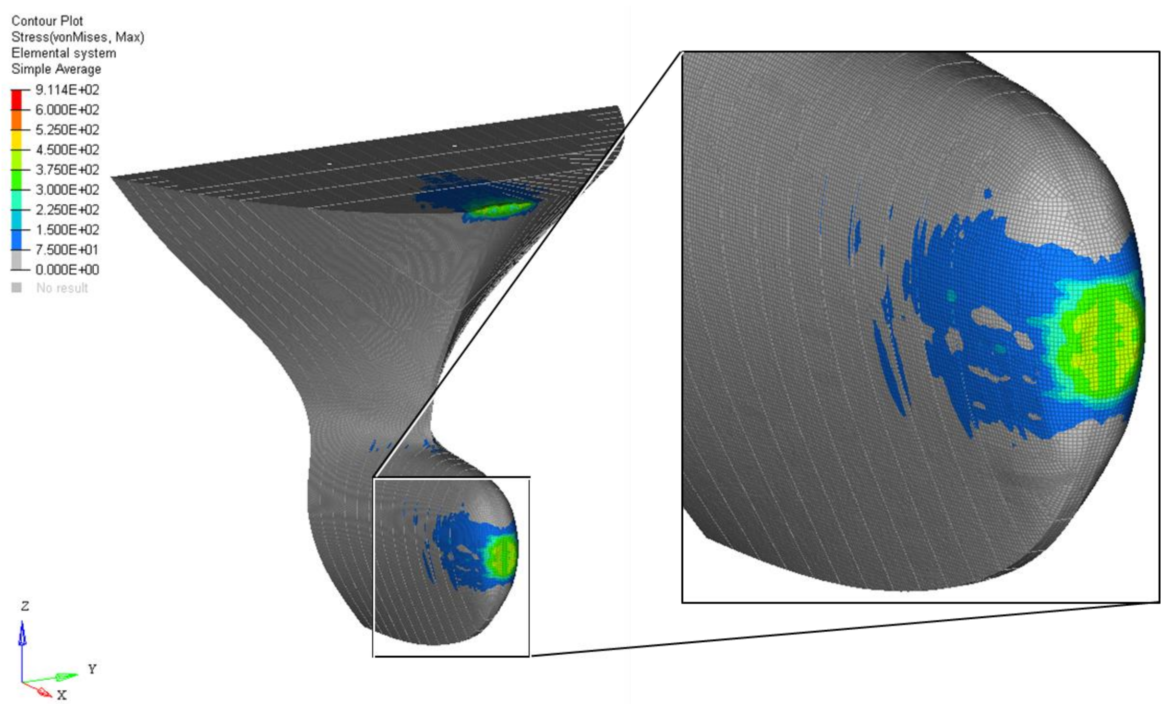
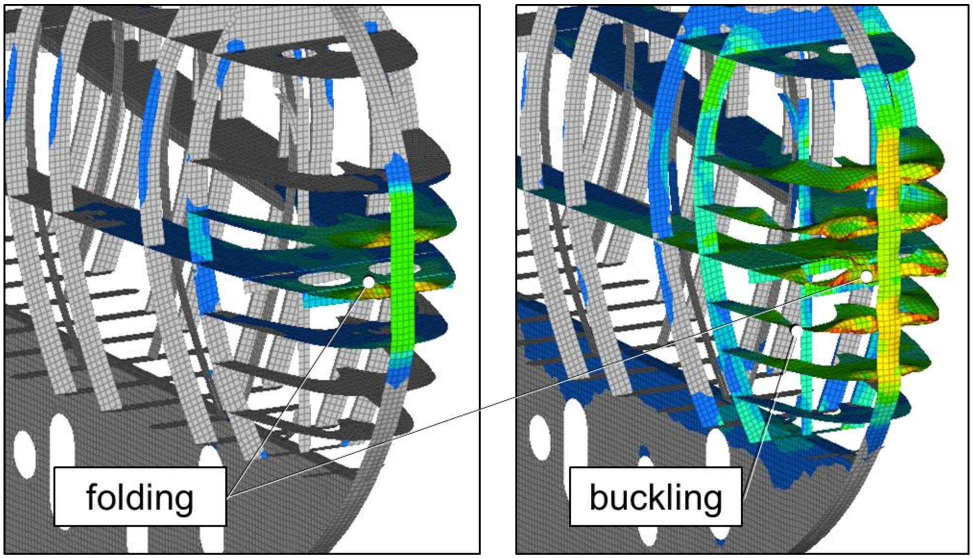


Figure 5/1-7 Flattening of the Bulb Tip under Frontal Impact of Rigid Wall



(a) t = 0.03 s (b) t = 0.08 s

Figure 5/1-8 Local Failure Mechanism of the Bulb Tip: Folding and Buckling

5/1.6 Discussions and Comments

The present FE model of the bow structure has been tuned up and can produce results with reasonable accuracies. In [40], a similar model is calculated by employing ABAQUS®. The results of the impact load indentations and stress/strain distributions from ABAQUS® are identical to those reported in this document.

This study was carried out based on simplified approaches of boundary conditions and material models. For example, the boundary condition of the present model is that a moving rigid wall impacts against a fixed bow structure. This setup does not consider the inertia effect of the ship structure, which can be different from the real collision cases.

In general, complete material behavior up to fracture should be modeled to capture the response of impact resistance and local failure mechanisms. That includes the stress-strain relation, strain rate effect, damage criteria, and, in special cases, thermal response. As discussed in PART 3, different types of damage criteria have been developed, e.g., strain-state dependence of fracture. The following sections discuss the impact of these factors on the simulation results.

Note that extra care is needed in preparing FE models if using advanced material models. For example, caution needs to be made concerning the material failure models that are inherent in finite element codes. Codes such as LS-DYNA have many different material response models, often including "failure." Code users often do not understand the question of what models to use for their problems, and how to obtain required parameters. The analyst needs to understand the assumptions and limitations built into these failure models. Before applying them for a practical analysis, their validity needs to be demonstrated through analysis of the results of experiments conducted with the same materials at the same rate of loading as in the analysis.

5/1.6.1 Work hardening effect

The work hardening controls where the strain localizes. Nowadays, a great number of plastic hardening models have been developed and implemented in the FEA software. Three material models with isotropic hardening are generally used in the ship structure design: (1) bilinear model per DNVGL-RP-C208 [106], (2) power-law model without a yield plateau, and (3) power-law model with a yield plateau (see Eq. 5.1). The hardening model is implemented based on the experimental data. It should be noted that since the stress-strain data from testing can include nonuniformity in results or test to test or variability, the yield strength of materials for ship structures is determined from the specified minimum value. That is typically at a fifth percentile of the probability distribution of testing data.

In the literature (e.g.,[154,155,157,158,182–186]), the effect of the work hardening has been systematically investigated in the damage extent in impact events. Paik [154,155] reported that the shape of stress-strain curves directly affects the simulation results, such as the local failure mechanism. Later, Hogstrom [157,182,183] carried out a parametric study on the work hardening effects associated with the damage model in impact events. It is found that not only the load-indentation relationship is directly dependent on the work hardening, but also damage or failure locations. Storheim [158,184–186] also investigated the effect of the work hardening variance in a full-scale ship collision scenario. Both upper- and low-bound stress-strain relationship was considered for a given strain hardening value. The upper bound strength is 1.15 times higher than the lower bound one. From the load indentation curves, the work hardening value has relatively low influences on the bow structure, but it has pronounced influences on the ship's side. The difference in impact resistance is mainly because of the governing failure mechanism. The governing deformation (e.g., buckling, and plastic folding) continuously develops and evenly spreads in the bow structure, whereas the governing deformation only happens when the failure propagates to stiffeners or vertical web frames in the ship side.

5/1.6.2 Strain rate effects

The strain rate is the velocity of deformation and defined in an explicit form – time derivate of strain:

$$\dot{\varepsilon} = \varepsilon / t \qquad \text{Eq. (5.6)}$$

Most of the steel materials show dependency on strain rate, which affects the yield strength, ultimate tensile strength, strain hardening, elongation limit, and fracture limit. In general, the yield strength, ultimate tensile strength, and strain hardening increase with the increase of strain rates, thus the flow stress increases to resist further deformation. Once the necking starts, the strain rate increases with the presence of necking localization. On the other hand, the elongation limit (also known as ductility) decreases as the increase of strain rate. The fracture limit does not show a clear correlation with the strain rate based on the experimental results. That can be because the fracture failure mechanism is complicated and sometimes also driven by other factors (e.g., level of triaxiality and Lode angles).

The strain rate effect is usually considered in the impact, explosion, or ballistic scenarios. Storheim and Amdahl [186] evaluated ship structures under different impact speed and pointed out two crucial points of strain rates in the ship collision:

1. The local instantaneous strain rate can be significant even at a low impact speed.
2. The strain rate is strongly mesh-size-dependent. For example, at the same impact speed – 4 m/s, the strain rate for an average mesh size of 100 mm can range from 5 – 15 s^{-1}, but the strain rate for a mesh size of 2 mm can be over 400 s^{-1}.

Note that the high strain rates often have a short time duration and usually show in the local necking zone.

Two material models are widely adopted in the FEA simulations: (1) Cowper-Symonds [187] and Johnson-Cook [188]. The Cowper-Symonds model defines the dynamic stress is proportional to the static stress with a scaling factor – known as dynamic hardening factor:

$$\sigma_{dynamic} = \left[1 + \left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}\right] \sigma_{static} \qquad \text{Eq. (5.7)}$$

where $\dot{\varepsilon}$ is the strain rate; *C* and *p* are parameters calibrated from experimental data. The standard DNVGL-RP-C208 [106] recommended calibrating the dynamic hardening factor to the maximum expected stress and strain. If there are no experimental data, parameters C = 4000 s^{-1} and p = 5 for the common offshore steel.

The Johnson-Cook model defines stress as a function of three factors: strain hardening, strain rate, and temperature.

$$\sigma = \left[K\left(\varepsilon_{p,eff} + \varepsilon_p\right)^n\right](1 + D\ln\varepsilon)\left(1 - T^{*^m}\right) \qquad \text{Eq. (5.8)}$$

where K and n are the same as those in Eq. 5.8; D and m are coefficients; T* is the temperature parameter relative to a reference temperature (e.g., room temperature). This model is usually adopted for ballistic simulations where the thermal load is coupling with mechanical loading.

5/1.6.3 Damage criteria

As mentioned above, the present study does not consider the material damage, and the stress or strain values are calculated over the elongation or fracture limit. The collision resistance of the ship structure can be overestimated without accounting for structural failure or damage. There are different damage criteria developed (see PART 3), e.g., stress-based, strain-based, and energy-based. The strain-based damage criteria are usually used to capture the structural failure in the impact event.

The maximum principal strain is usually used as the strain-based damage criteria, which assumes the predominant damage mechanism is caused by tensile failure. The procedure to calibrate the maximum principal strain is presented in [40,107]. In LS-DYNA®, the number 123rd material model – *MAT_123, which is similar to *MAT24 except for the additional maximum principal strain criteria. For two grade steels in this study, the calibrated damage criteria are as follows:

- Equivalent plastic strain 20.1% (S235) 19.6% (S355)
- Maximum principal strain 17.7% (S235) 17.4% (S355)

In the FEA software, the damage criteria usually collaborate with the element erosion (also known as deletion) function. For example, the shell element has five integration points through-thickness in this study, and the element will be deleted once all the integration points have failed.

5/1.6.4 Fracture and mesh dependence

The estimation of damage also shows strong mesh-size-dependency, similar to that of the strain rate. In [189,190], the authors carried out a series of numerical studies and discussed the fracture and mesh dependency. For example, Figure 5/1-9, reproduced from [189,190], shows a deformed plate section at the maximum indentation level in both testing and simulation. The figure shows the finer mesh model can accurately capture both the deformed shape and fracture path, but the coarser mesh model can only capture the fracture initiation point.

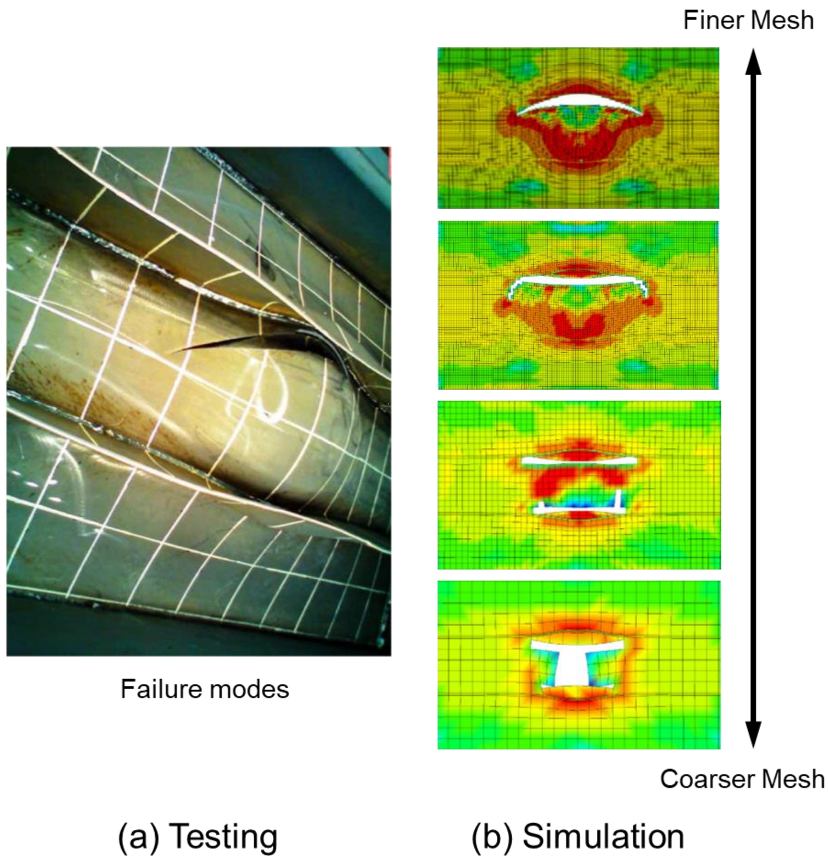


Figure 5/1-9 (a) Testing Results of Failure Mode; (b) Numerical Results of Failure Modes and von Mises Stress Distributions for Four Different Mesh Sizes (i.e., Length/Thickness = 1, 2, 5, and 10)

5/2 FRACTURE AND FATIGUE

5/2.1 Introduction

Several years ago, oil shipping companies started using double-hull tankers to replace single-hull tankers based on Annex I of MARPOL (Maritime Pollution) convention – “phase out single-hull tankers around the world by 2026” [191]. A client built a double bottom hull tank in the year 2000, and cracking zones have been observed throughout the double bottom after five-year services. This study investigates the cause of cracking in the double bottom using a fatigue analysis.

Figure 5/2-1 shows a crack is located at the toe of a welded connections of a double bottom floor stiffener. It should be noted that the cracking plane is atypical for longitudinal loading. In Figure 5/2-2, the cracking incidents observed in the double bottom are highlighted in the green shading. It is seen that the cracking spreads throughout the double bottom and is away from the bulkhead.

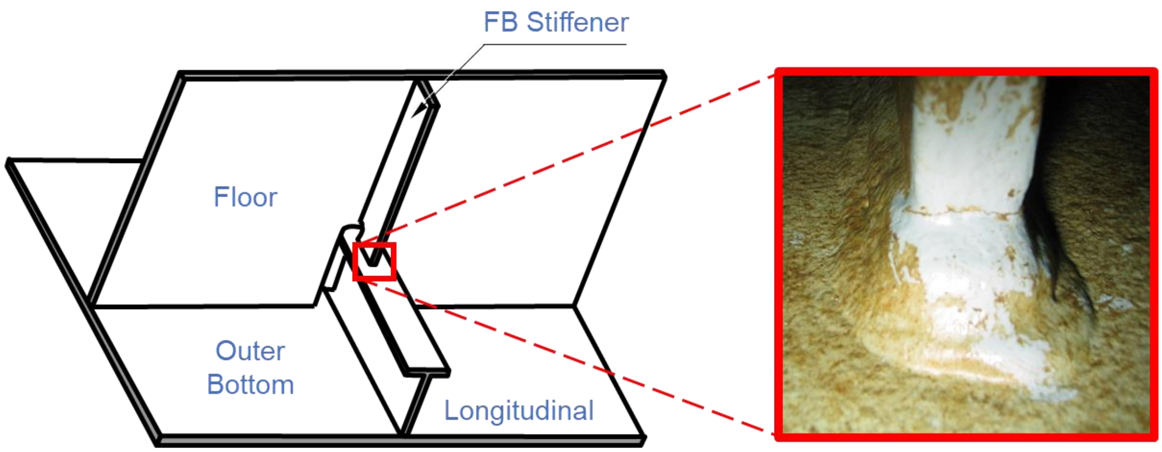


Figure 5/2-1 Cracking Occurring on a Horizontal Plane at Floor Stiffener to Longitudinal Connection

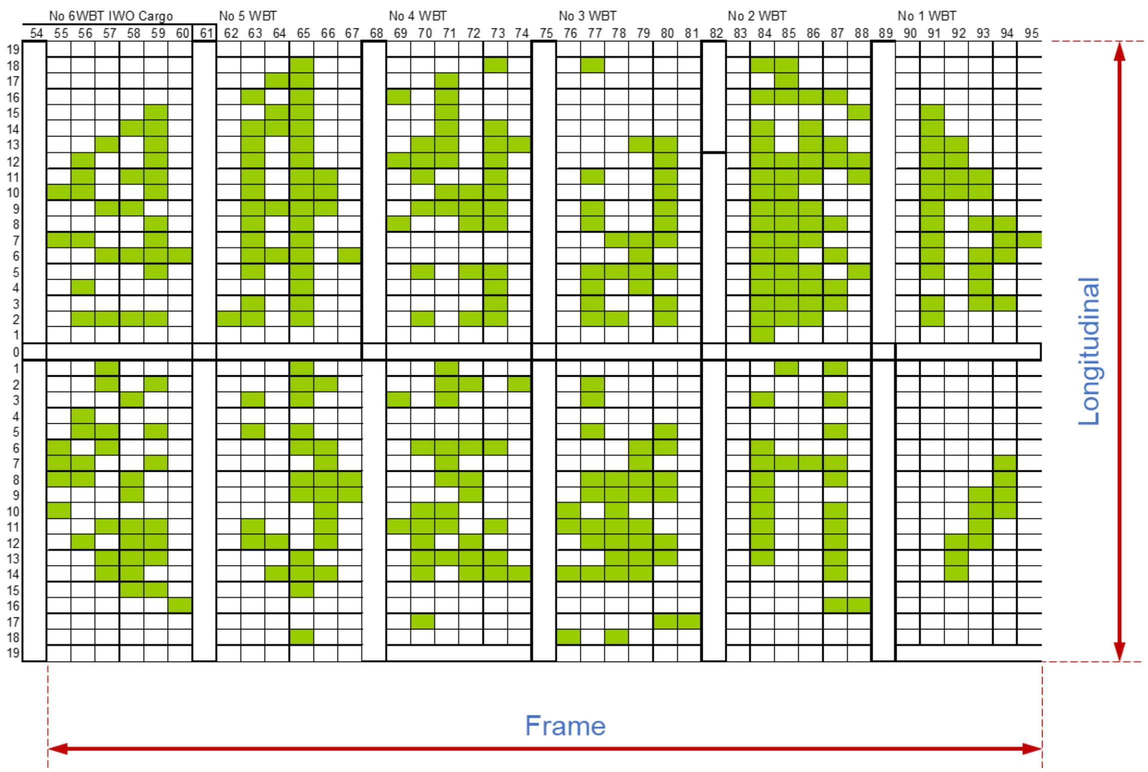


Figure 5/2-2 Locations of Cracking Incidents in the Double Bottom

The objective of this study is to understand the root cause of the cracking by carrying out numerical investigations on structural behaviors under two different loading scenarios. The investigations include two steps of analyses: static linear elastic FEA analysis and fatigue analysis, where the fatigue cyclic loadings are interpreted based upon the stress results from the Step-1.

5/2.2 Engineering model

5/2.2.1 FEA software

The general FEA software – ANSYS® Mechanical version 7.0 with double precision was employed for calculating the stress distributions. FlawCheck® [192] developed by BMT-Canada is used to carry out the fatigue analysis.

FlawCheck® is a comprehensive integrity assessment tool for metal structures designed to support the development and implementation of integrity assurance and maintenance programs. The current study uses FEA-based structural loading data to estimate the fatigue crack growth and to determine the durability life. In this study, the fatigue crack growth and fatigue life are evaluated by following the procedures described in BS 7910-2013 [12]. Figure 5/2-3 shows the FlawCheck's GUI (graphic user interface) and fatigue analysis module.

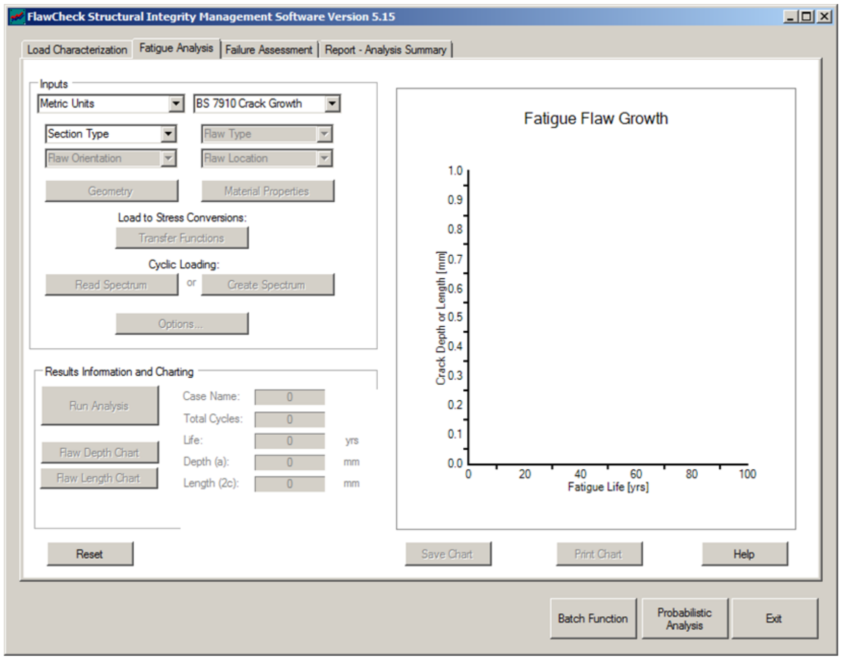


Figure 5/2-3 FlawCheck GUI® and Fatigue Analysis Module

5/2.2.2 System units

In this study, the metric unit system is used:

1.	Length	millimeter	[mm]
2.	Mass	metric tons	[t]
3.	Time	second	[s]
4.	Force	Newton	[N]
5.	Pressure	megapascal	[MPa] or [N/mm^2]
6.	Stress	megapascal	[MPa] or [N/mm^2]
7.	Strain	no unit	[%]
8.	Velocity	millimeter per second	[mm/s]
9.	Energy	millijoule	[mmJ]
10.	Fatigue life	year	[y]

5/2.2.3 Geometry assumptions

The geometric dimensions of this double bottom oil tanker are as follows:

1. Length B.P.: 245 m

2. Depth MLD: 20.4 m
3. Breadth MLD: 44.8 m

A typical hold of the vessel is modeled (see Figure 5/2-4), which includes rough scantlings, a longitudinal plane of symmetry, and a forward and aft bulkhead.

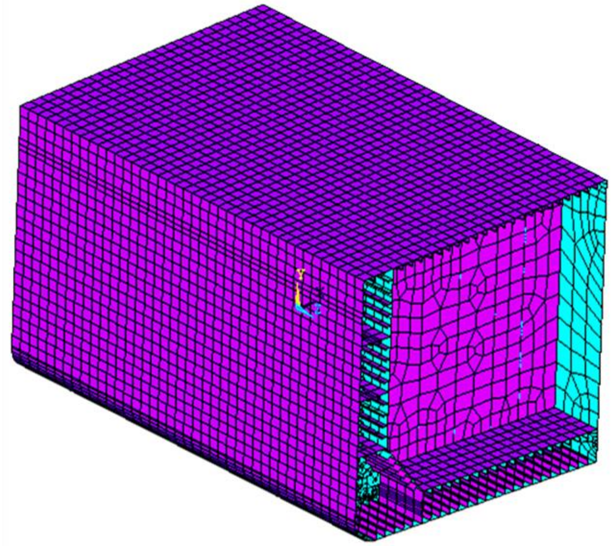


Figure 5/2-4 FE Model of a Typical Hold of a Vessel

Another level of the FE model with finer meshing (local submodel) is prepared to capture the variations of stress gradients in the hot spot. The loading and boundary conditions for the local model are interpreted from the results of the above-mentioned coarse meshing model. Figure 5/2-5 shows the finer meshing model and locations of hot spots.

Note that in the global analysis, only one hold model is used (see Figure 5/2-4), and Figure 5/2-5 shows the hierarchy structure the different levels of FE meshing.

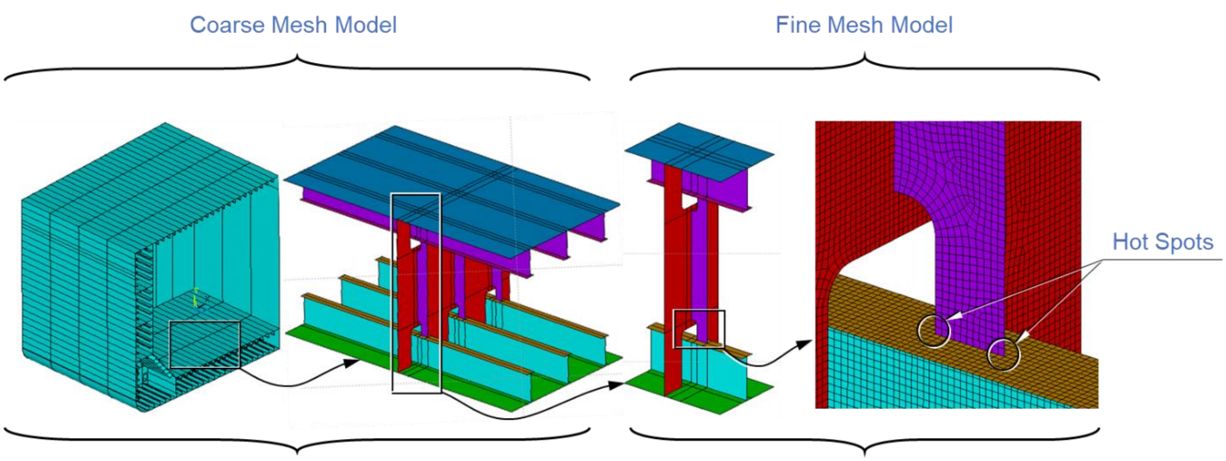


Figure 5/2-5 Local FE Model with Finer Meshing for Capturing Variations of Stress Gradients Close to Hot Spots

5/2.2.4 Material properties

For the linear elastic FE analysis, the material properties of ship grade steel are listed as follows:

1. Young's modulus E [MPa] 210,000
2. Poisson's ratio v [-] 0.3

A linear SN curve for a corrosive environment is used for the fatigue analysis, shown in Figure 5/2-6. Note that the failure locations are assumed in ballast tanks. The slope and intercept points of the SN curve are 3 and 11.705, respectively. The yield strength of the base metal is 360 MPa, and the limit of thickness effect is 25 mm.

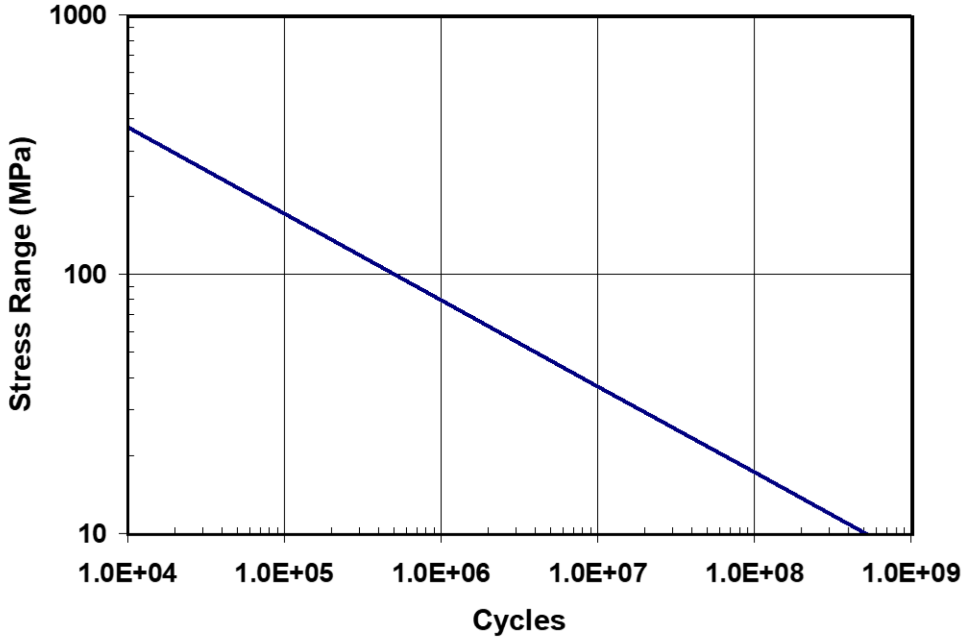


Figure 5/2-6 SN Curve of Ballast Steel for Corrosive Environment with Slope = 3 and Intercept Point = 11.705

5/2.2.5 Loading and boundary conditions

In the FEA analysis, three different load cases are considered in this study:

- Hull girder bending 2.1×10^7 N·mm
- Internal pressure – ballast see Figure 5/2-7 Illustrations of the pressure loads of (a) Ballast and (b) Cargo loaded cases
-
- External pressure – loaded see Figure 5/2-7 Illustrations of the pressure loads of (a) Ballast and (b) Cargo loaded cases
-

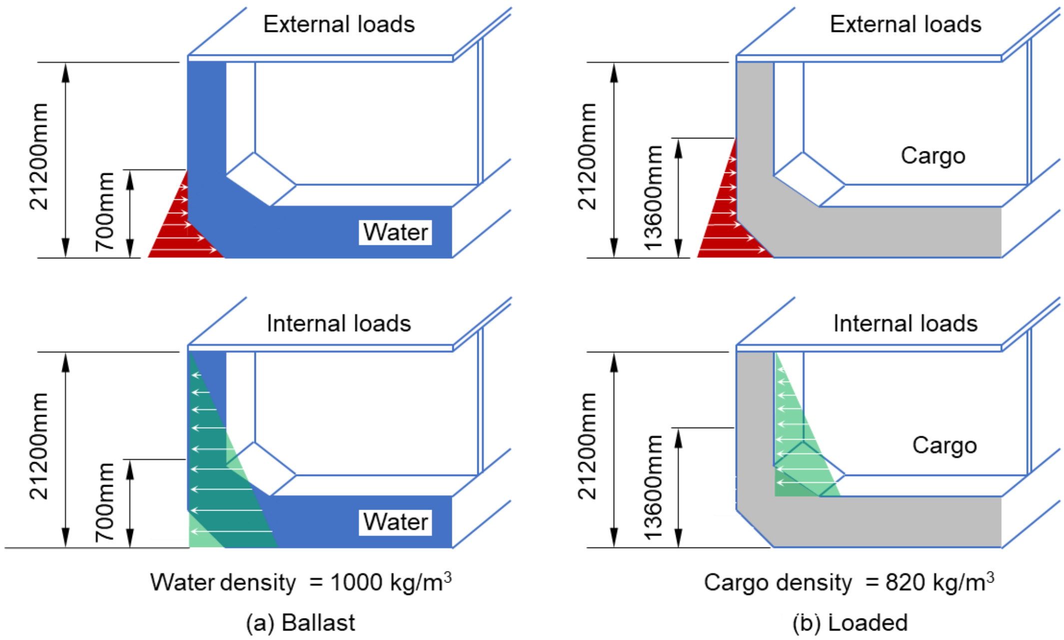


Figure 5/2-7 Illustrations of the pressure loads of (a) Ballast and (b) Cargo loaded cases

The loading and fixations of three load cases are schematically shown in Figure 5/2-8 Loading and Fixation Conditions of Three Load Cases. In Figure 5/2-7 (a), the yellow highlighted plane defines the longitudinal symmetry, and in Figure 5/2-7 (b) and (c), the green highlighted regions refer to fixation.

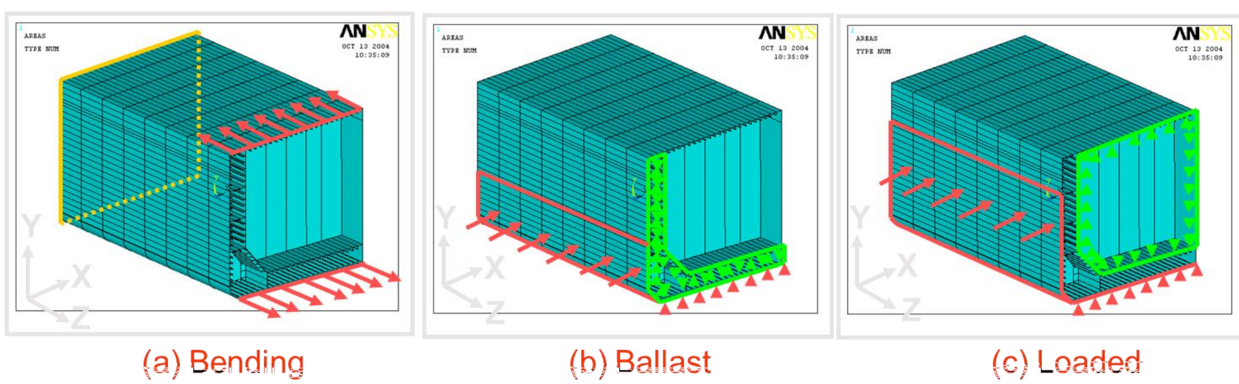


Figure 5/2-8 Loading and Fixation Conditions of Three Load Cases

In the analysis of the local submodel with finer meshing, the loading and fixation conditions for all the load cases are displacement-control, i.e., relative displacement, which are interpreted from results from the coarser meshing model. The concept is similar to the submodeling technique described in PART 3.

5/2.3 FE models and simulation setups

5/2.3.1 Element

The first-order shell elements (i.e., 3- and 4-nodes for tris and quads, respectively) are employed in both coarse and fine meshing models. The average element size for the coarse meshing model is about 200 mm, and that for the fine meshing model is about 12 mm that is consistent with the steel wall thickness.

5/2.3.2 Mesh design and quality control

The mesh was designed and tuned up by following DNVGL-CG-0127 [9], the mesh of global and local FE models is revised based on the criteria shown in Table 5/2-1 and Table 5/2-2, respectively. Note the criteria are developed based on the description and discussion in PART 3. The mesh quality of the revised model meets all the criteria listed in Table 5/2-1 and Table 5/2-2.

Table 5/2-1 Fatigue Problem with 200mm Element Size

#	Criterion	Ideal	Good	Warn	Fail	Worst
0	penalty value	0	0	1	2	10
1	min length	200	154	106.8	60	48.4
2	max length	200	260	316	400	514.4
3	aspect ratio	1	2	4	5	10
4	warpage	0	3.75	11.25	15	30
5	max angle quad	90	110	140	150	165
6	min angle quad	90	70	40	30	15
7	max angle tria	60	80	120	130	155
8	min angle tria	60	50	30	20	10
9	skew	0	10	50	60	75
10	jacobian	1	0.9	0.6	0.5	0.35
11	chordal dev	0	0.3	0.8	1	2
12	% of trias	0	1.5	2.25	3	4.5
13	taper	0	0.2	0.5	0.6	0.9

Table 5/2-2 Fatigue Problem with 25mm Element Size

#	Criterion	Ideal	Good	Warn	Fail	Worst
0	penalty value	0	0	1	2	10
1	min length	25	19.25	13.35	7.5	6.05
2	max length	25	32.5	39.5	50	64.3
3	aspect ratio	1	2	4	5	10
4	warpage	0	3.75	11.25	15	30
5	max angle quad	90	110	140	150	165
6	min angle quad	90	70	40	30	15
7	max angle tria	60	80	120	130	155
8	min angle tria	60	50	30	20	10
9	skew	0	10	50	60	75
10	jacobian	1	0.9	0.6	0.5	0.35
11	chordal dev	0	0.3	0.8	1	2
12	% of trias	0	1.5	2.25	3	4.5
13	taper	0	0.2	0.5	0.6	0.9

The FE models with coarse and fine meshing consist of about 55,000 and 85,000 nodes, respectively. The number of triangular elements is less than 3% of the total element numbers. Note that the mesh quality criteria for the fatigue analysis are different from those for the impact analysis presented in Example-1 (Table 5/1-1).

5/2.4 Analysis control and solution options

A special post-processing technique is adopted to estimate the stress levels in the point of interest to reduce the impact of the stress singularity at the notch tip. As above-mentioned, the shell element is used to construct the structure, and therefore the weld toe (see blue shading in Figure 5/2-9) is not practical to consider in the model. However, because of the presence of the geometric discontinuities – unreal notch, the estimated stress at the notch (σ_{notch}) can be significantly higher than the actual value (σ_g). Figure 5/2-9 schematically shows the technique to estimate the value of σ_g from stress values at t/2 and 3t/2, where t refers to the plate thickness.

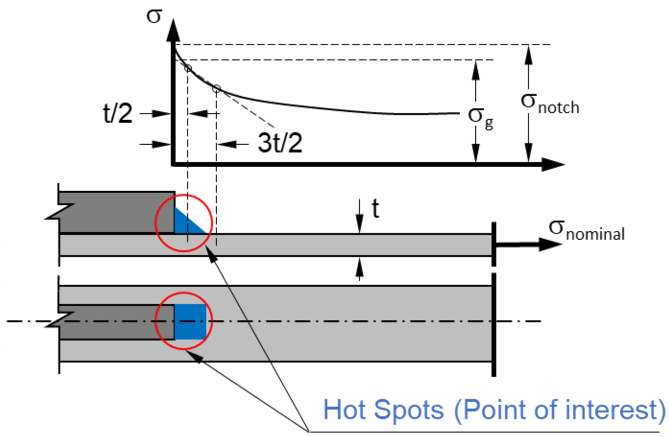


Figure 5/2-9 Hot Spot Stress Extrapolation in FE Models when Weld Toe (Blue-Color Shading) is not Modeled, Since the Use of Shell Elements

In the fatigue analysis, the stress range is calculated from results at hot spots. The fully reversed loading is assumed for the bending load case, and the stress range (σ_{range}) is two times of the first principal stress (σ_1) of hot spots:

$$\sigma_{range} = 2|\sigma_1|$$

For the pressure loading case, the stress ranges of upper and lower connections are the difference between the first principal stress (σ_1) of hot spots from fully loaded and ballast states:

$$\sigma_{range} = \sigma_1^{Ballast} - \sigma_1^{Fully-loaded}$$

where $\sigma_1^{Ballast} > 0 > \sigma_1^{Fully-loaded}$.

In a unit structure (i.e., a local submodel), there are two hot spots (see Figure 5/2-10), and there are four possible predominant load paths (A, B, C, and D). Note that the load path depends on the loading and boundary conditions. The values of the principal stress are extracted along the four load paths to calculate the stress (σ_g) at hot spots. The hot spots at both upper and lower connections are examined. Note that all the results are based on the nodal values.

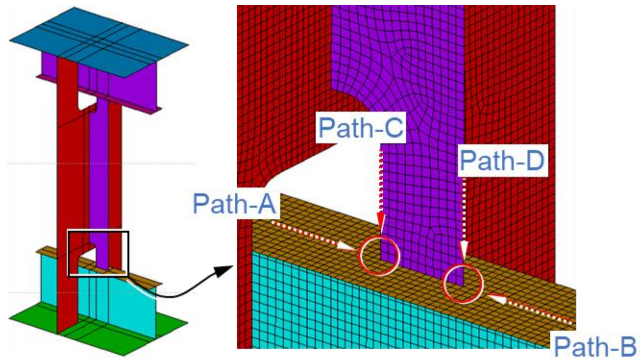


Figure 5/2-10 Possible Four Load Paths to Two Hot Spots at the Lower Connection

5/2.5 Results

5/2.5.1 Displacement results

The deformed shape and displacement results of the global model under longitudinal bending are shown in Figure 5/2-11. Note that the deformed shape in Figure 5/2-11 is exaggerated, aiming to qualitatively confirm the sense of loading and boundary conditions. The displacement results from the global model are used to model the structural response in the local sub-model. The same procedures are followed for the other two load cases.

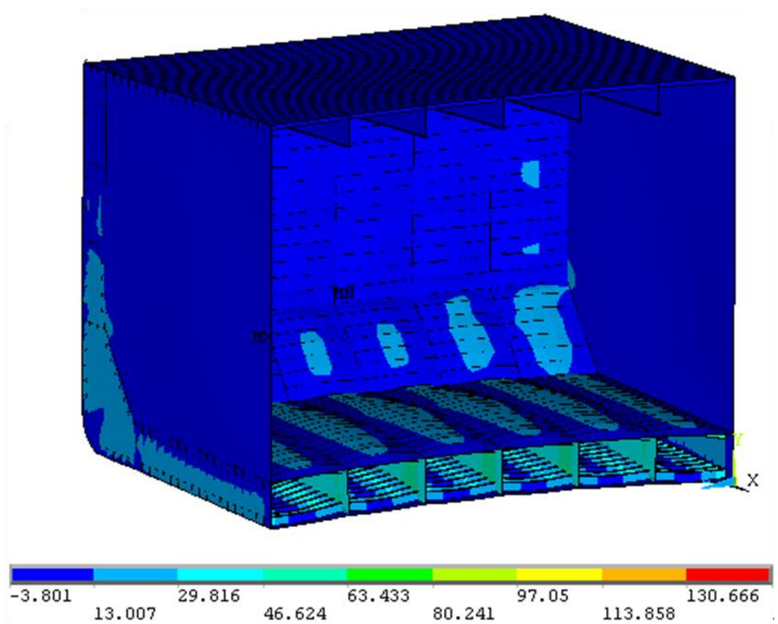


Figure 5/2-11 Displacement Results of Global Model Due to Longitudinal Bending (Load Case – 1)

5/2.5.2 Stress Results

In the analysis of the local sub-model, the distribution of the maximum principal stress due to the longitudinal bending is shown in Figure 5/2-12. It is clearly seen that the stress levels at two hot spots are significantly higher than the rest of the structure. The distributions of maximum principal stress due to ballast load and cargo load are shown in Figure 5/2-13 and Figure 5/2-14, respectively. In Figure 5/2-12, Figure 5/2-13, and Figure 5/2-14, the stress levels at point A are relatively higher than that at point B. The maximum principal stress in all three load cases is aligned with the stiffener axis. That indicates the fatigue failure mechanism is in Mode-1 (see Figure 3/2-17 of PART 3) and caused by cyclic tensile loading.

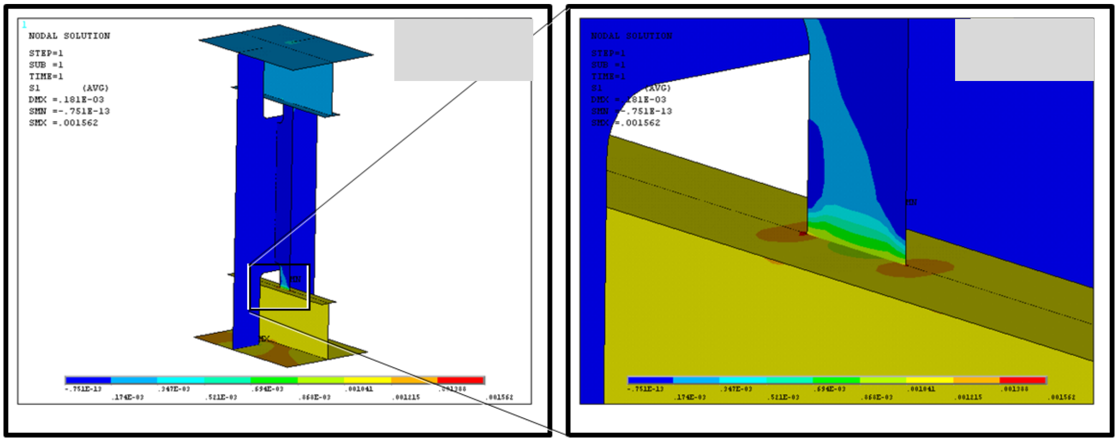


Figure 5/2-12 Maximum Principal Stress Due to Longitudinal Bending (Load Case – 1)

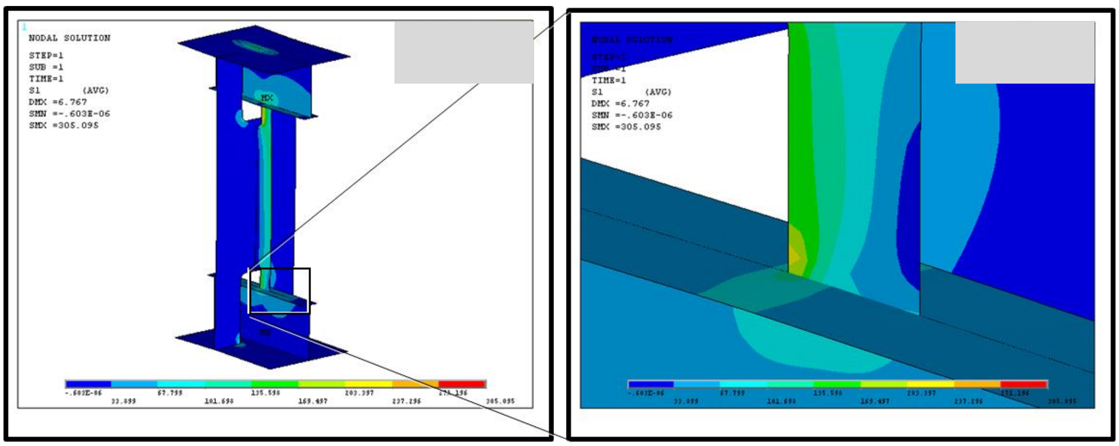


Figure 5/2-13 Maximum Principal Stress of Local Sub Model Due to Ballast Load Conditions (Load Case – 2)

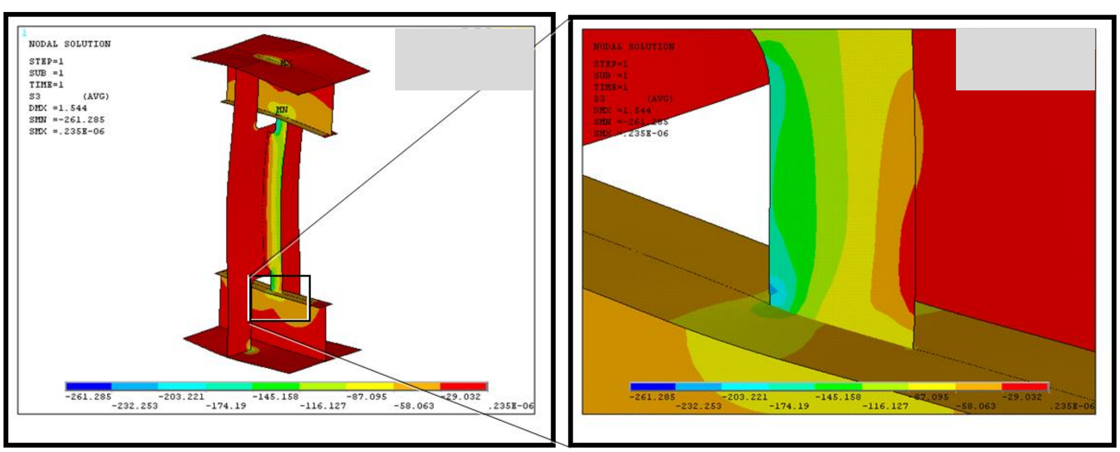


Figure 5/2-14 Maximum Principal Stress of Local Sub Model Due to Cargo Load Conditions (Load Case – 3)

The stress values of hot spots along four load paths are extracted based on the method above mentioned (see 5/2.4 and Figure 5/2-9). The results for three load cases are summarized in Figure 5/2-15.

At this point, all the results are preliminary for the subsequent fatigue analysis. The stress ranges (see green shading in Figure 5/2-15) are calculated from those hot spot stresses. Note that for the bending load case, it is a cyclic moment loading and the stress transfer functions are used instead of the stress range for the fatigue life calculation in FlawCheck®. The stress transfer function is defined as follows:

$$Transfer\ Function = Stress\ Range/Applied\ Moment$$

On the other hand, the stress transfer function for the cyclic membrane tension (i.e., load cases – 2 and – 3) equals the stress range, where the applied moment = 1.

For the load case – 1, the worst case is through load path A at lower connection, i.e., the maximum stress range = 2.95×10^{-3} MPa, and the stress transfer function = 1.4×10^{-10} MPa/(N·mm), see Figure 5/2-15. For the load cases – 2 and – 3, the worst case is through load path C at the upper connection, i.e., the maximum stress range = 635.6 MPa.

From the real-time measurement, the sample loading spectrum for the load case - 1 in a year is shown in Table 5/2-3. The screenshot of the loading spectrum setup in FlawCheck is shown in Figure 5/2-17. The loading spectrum for load cases – 2 and 3 (combined for fatigue analysis) is unit, which means the stress range occurs once per transit.

Load case – 1 (Bending)

Upper Connection

Path	Hot Spot Stress Value (MPa)	Stress Range (MPa)	Transfer Function (MPa/N-mm)
A	3.24E-04	6.49E-04	3.09E-11
B	3.27E-04	6.53E-04	3.11E-11
C	4.40E-04	8.79E-04	4.19E-11
D	3.31E-04	6.62E-04	3.15E-11

Lower Connection

Path	Hot Spot Stress Value (MPa)	Stress Range (MPa)	Transfer Function (MPa/N-mm)
A	1.47E-03	2.95E-03	1.40E-10
B	1.44E-03	2.87E-03	1.37E-10
C	1.15E-03	2.31E-03	1.10E-10
D	1.02E-03	2.04E-03	9.74E-11

Load case – 2, 3 (Ballast, Cargo)

Upper Connection

	Hot Spot Stress Value (MPa)		
Path	FULLY LOADED	BALLAST STATE	Stress Range (MPa)
A	-45.6	51.2	96.7
B	-56.0	62.5	118.5
C	-280.6	355.0	635.6
D	-137.3	204.1	341.3

Lower Connection

	Hot Spot Stress Value (MPa)		
Path	FULLY LOADED	BALLAST STATE	Stress Range (MPa)
A	-99.1	106.9	206.1
B	-57.2	56.1	113.3
C	-261.1	314.9	576.1
D	-78.8	81.9	160.7

Figure 5/2-15 The stress values of hot spots for four load paths from FEA and corresponding stress ranges at upper and lower connections

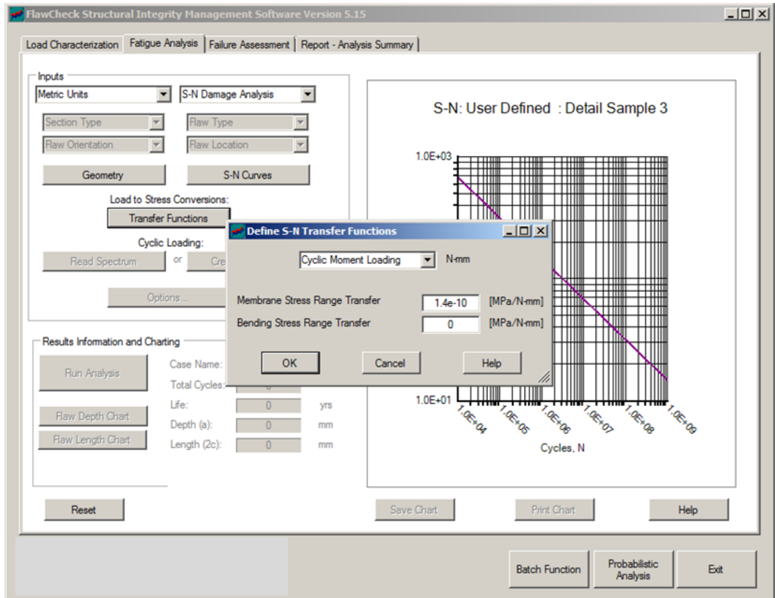


Figure 5/2-16 Defining SN Transfer Functions for Load Case – 1 (bending) in FlawCheck®

Table 5/2-3 Loading Spectrum Under Longitudinal Bending Moments in a Year

Bin No.	**Bin Size – ΔM [MN·m]**	**Number of occurrences**
1	150	2.0 × 106
2	200	1.0 × 106
3	250	5,000
4	300	500
5	350	5

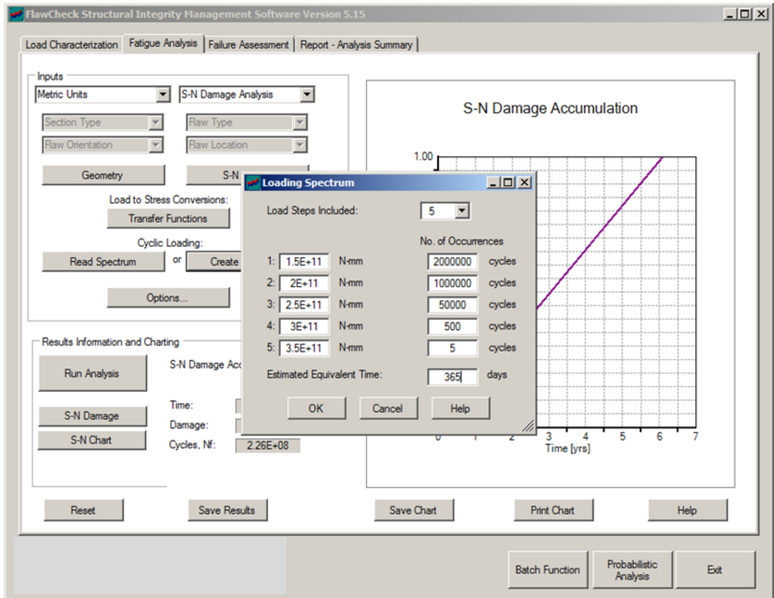


Figure 5/2-17 Setup of Loading Spectrum in FlawCheck®

5/2.5.3 Fatigue Results

The estimated fatigue life for the moment loading is approximately 11.8 years, which is equivalent to 3.6×10^{-7} cycles. It should be noted that this result can vary with a different applied load spectrum. The SN accumulated damage is shown in Figure 5/2-18 in terms of the number of years.

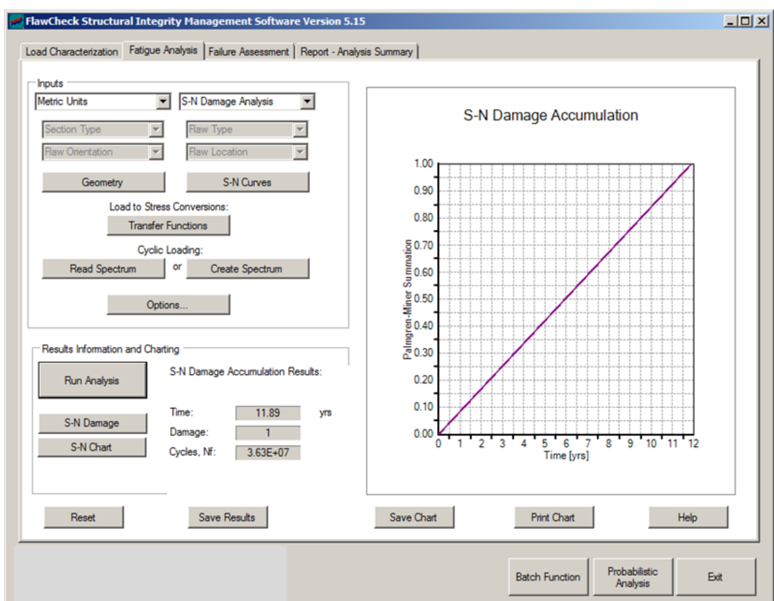


Figure 5/2-18 Fatigue Life Under the Moment Loading, where the Accumulated Damage Plots against Time

For the ballast and cargo pressure loading, the estimated fatigue life is about 1,975 cycles, which is equivalent to 5.41 years of service. Note that the result may vary with the refinement of local geometry, e.g., including weld toes.

The fatigue results from FEA and fatigue life analysis are similar to those from real-life observed observation. The ballast pressure and full cargo pressure are the major cause of short service life. It is recommended to revise the design close to the hot spots to reduce the stress level or stress concentration.

5/2.6 Discussions and Comments

The fatigue analysis method presented in the above section is based on the method of SN damage analysis. As discussed in PART 3, the fracture-mechanics-based fatigue analysis is another method that is usually adopted in an integrity analysis (including a fatigue life analysis). In this section, the fatigue life is evaluated by using the fracture-mechanics-based method. The procedures also follow the rules of BS 7910 [12].

It is assumed that there is a pre-existing flaw with 30 microns (i.e., 0.03 mm) and will propagate along the most critical load path C at the upper connection, as shown in Figure . The duration of this flaw to grow through the plate (either through-thickness or through-width direction) is estimated. The width and length of the interested plate (i.e., purple in Figure) are 200 mm and 12 mm, respectively.

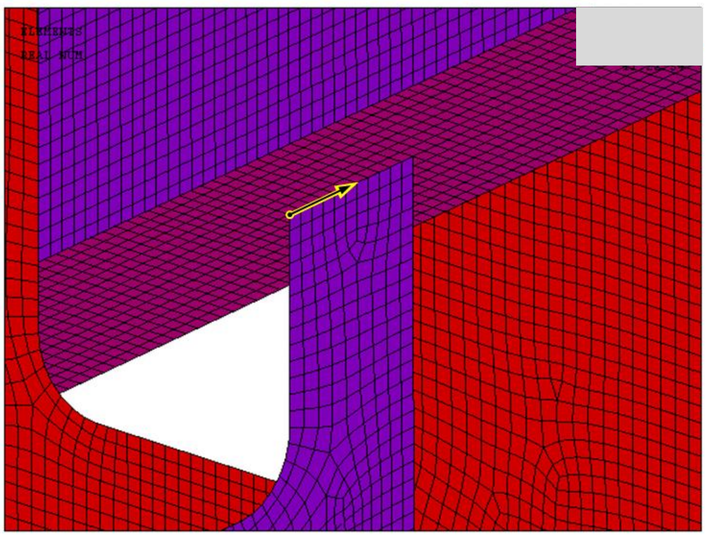


Figure 5/2-19 Direction of Crack Growth along the Load Path C at the Upper Connection

The material properties of ballast steel are as follows:

• Yield strength	[MPa]	360
• Ultimate tensile strength	[MPa]	400
• Paris' law constant – C	[-]	2.3×10^{-12}
• Paris' law constant – m	[-]	3

Note that the crack growth constants of Paris' law are recommended by BS 7910.

Using the same stress range and loading spectrum from the ballast and cargo pressure loading, the estimated fatigue life is about 2,436 cycles. That means it takes about 6.67 years to grow through the plate thickness. Figure 5/2-20 shows that the crack growth depth is about 90.09 mm along the plate width direction. This result is similar to that from the SN based fatigue life estimation. Note that this result will vary with a refinement of the initial flaw in terms of size and orientation.

Note that the effects of residual stress and mean stress need to be addressed in fatigue crack growth analysis. A crack may initiate in an area of high residual tensile stress but grow into an area of lesser stress, where the rate of crack growth will be slowed. On the other hand, as a crack grows across a ship, the stress levels increase as less structure is effective, and the rate of crack growth increases. If the finite element code does not adjust for the changes in the stress fields with crack growth, the analyst must do so by reinitiating the analysis.

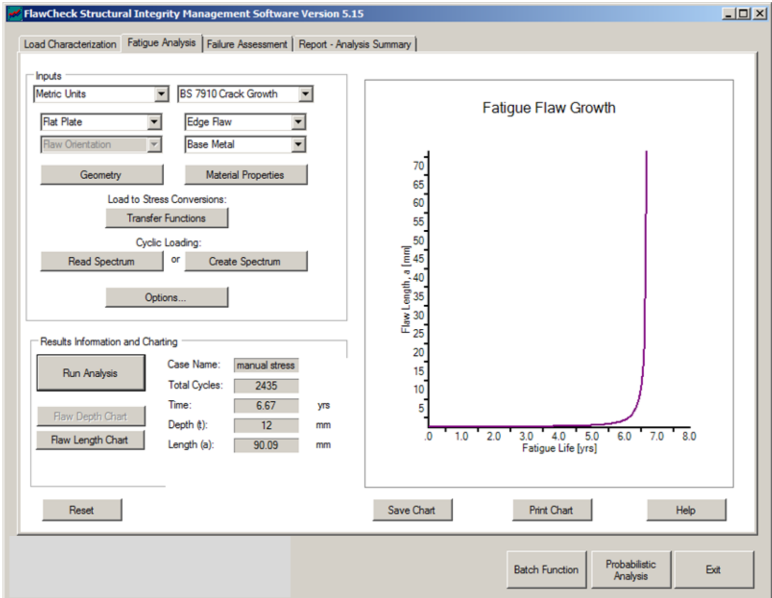


Figure 5/2-20 Fatigue Life Estimation Using the Fracture-Mechanics-Based Fatigue Analysis Method, Where the Flaw Length Plots Against Time

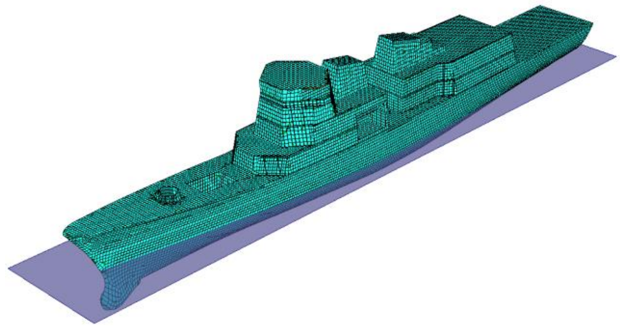
5/3 WHOLE SHIP ANALYSIS

5/3.1 Sample Application Description

An example is given to show how to perform a whole ship hull girder strength analysis. The sample application notional frigate has a length between perpendiculars L=135m, beam B=17m, depth D=9.4m, and displacement of 4,000 tons. This analysis uses MAESTRO 11.7.0, a finite element analysis-based toolset tailored to the design of ships and other floating platform structures by naval architects.

5/3.2 Finite Element Model Development

The notional frigate model was created using the NAPA-Steel initial design structural modeler, as shown in Figure 5/3-1 (a). The whole ship finite element model has over 61,000 nodes and 125,000 elements, as shown in Figure 5/3-1 (b). This model is an example of a finer mesh model, where the stiffeners are modeled explicitly as offset beams at their exact locations, the web plate of the frames and girders are modeled as shell elements, and the flange plate of the frames and girders is modeled as flat bar elements. One shell element is modeled between stiffeners, and three elements are modeled between frames. Figure 5/3-2 illustrates the finite element mesh density of the frigate model.



(a) (b)

Figure 5/3-1 A Notional Frigate

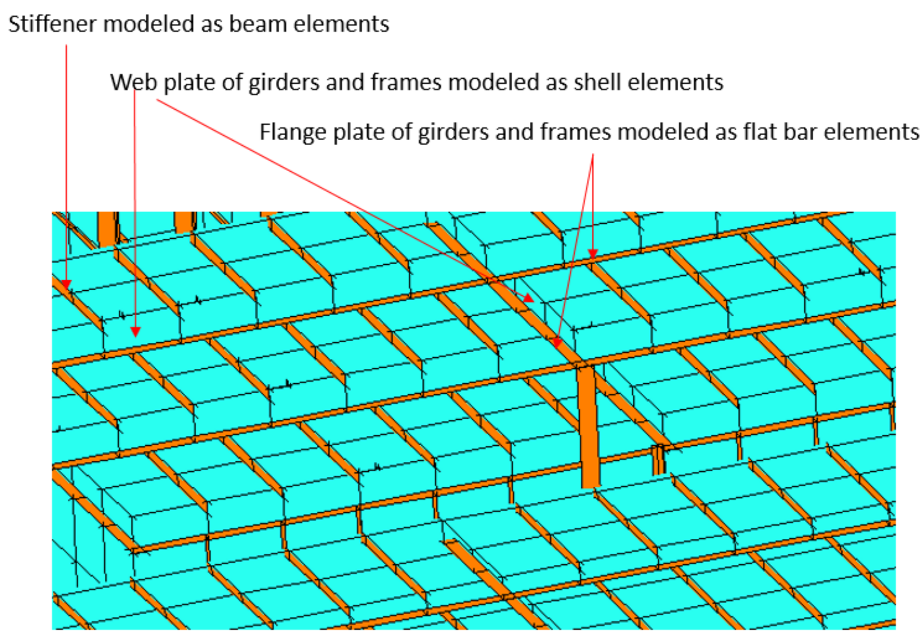


Figure 5/3-2 A Notional Frigate Meshing Approach

The full hull and superstructure are modeled. The model would typically be developed based on the geometry of a 3D hull model, general arrangements drawings, and scantling drawings at typical frames. This full-ship FE model and its mesh density are typical for use in designing hull, deck, and bulkhead scantlings for the final design. A simpler coarse mesh model could have been used in early-stage design to establish initial structural topology and scantlings.

5/3.3 FE Model Mass Properties and Hydrostatic Loading

The tank boundary elements and the hull wettable elements can be defined interactively using element grouping features. The frigate tank boundary definitions and wettable surface elements are shown in Figure 5/3-3 and Figure 5/3-4, respectively.

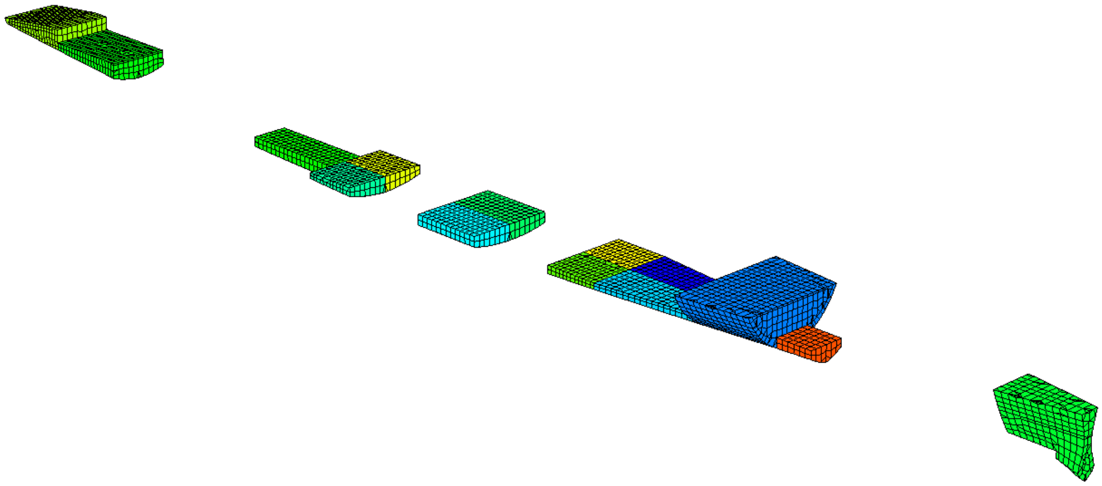


Figure 5/3-3 Liquid Tank Definition of the Notional Frigate

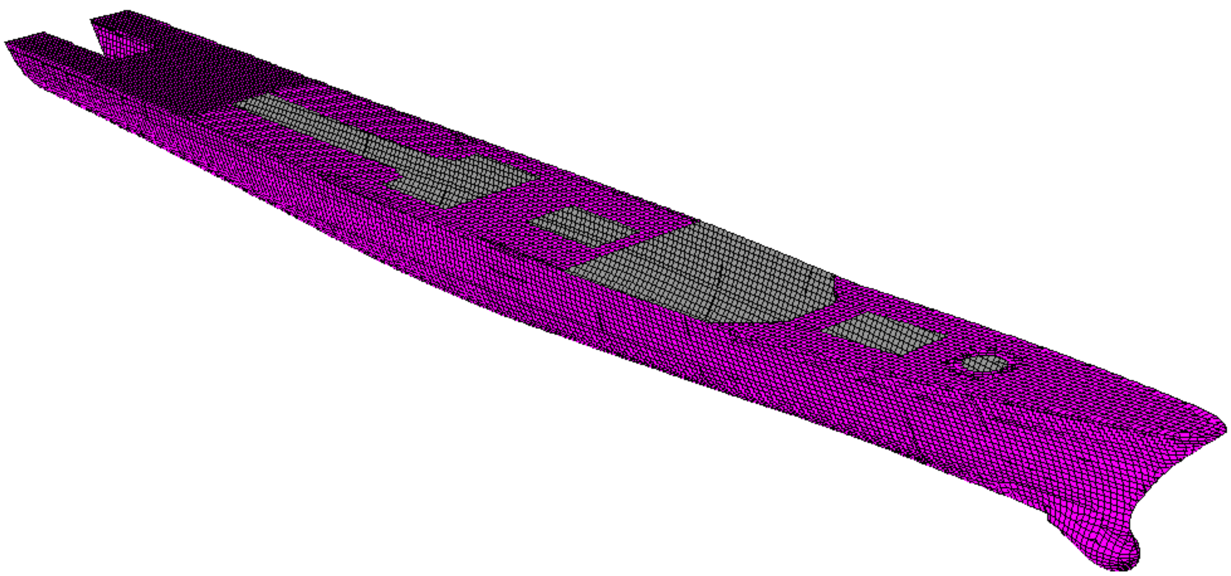
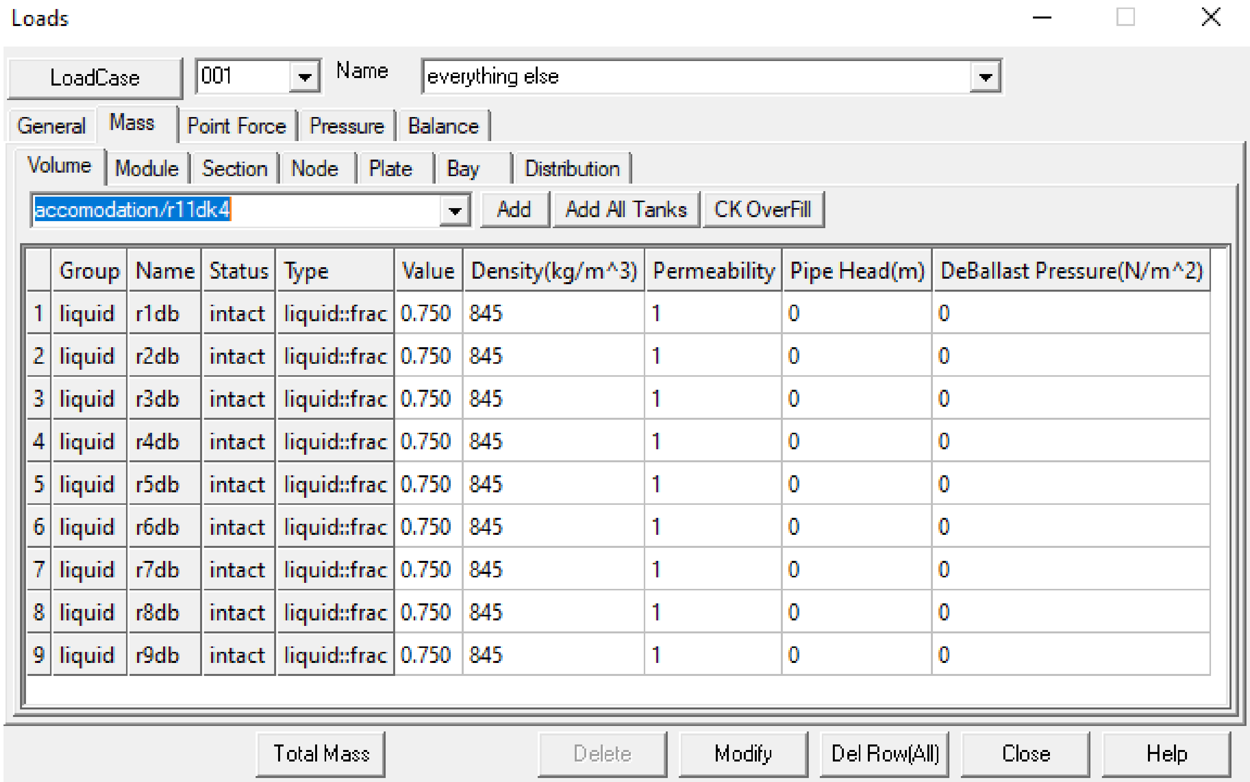


Figure 5/3-4 Frigate "Wettable" Elements

In addition, the weight distribution, tank loads, and floating conditions defined in the initial design modeler software hydrostatic module are also translated into the sample application FEA. Figure 5/3-5 and Figure 5/3-6 show how tank loads and lightship weight was entered in the sample application FEA software.



	Group	Name	Status	Type	Value	Density(kg/m^3)	Permeability	Pipe Head(m)	DeBallast Pressure(N/m^2)
1	liquid	r1db	intact	liquid::frac	0.750	845	1	0	0
2	liquid	r2db	intact	liquid::frac	0.750	845	1	0	0
3	liquid	r3db	intact	liquid::frac	0.750	845	1	0	0
4	liquid	r4db	intact	liquid::frac	0.750	845	1	0	0
5	liquid	r5db	intact	liquid::frac	0.750	845	1	0	0
6	liquid	r6db	intact	liquid::frac	0.750	845	1	0	0
7	liquid	r7db	intact	liquid::frac	0.750	845	1	0	0
8	liquid	r8db	intact	liquid::frac	0.750	845	1	0	0
9	liquid	r9db	intact	liquid::frac	0.750	845	1	0	0

Figure 5/3-5 Frigate Full Load Tank Definition

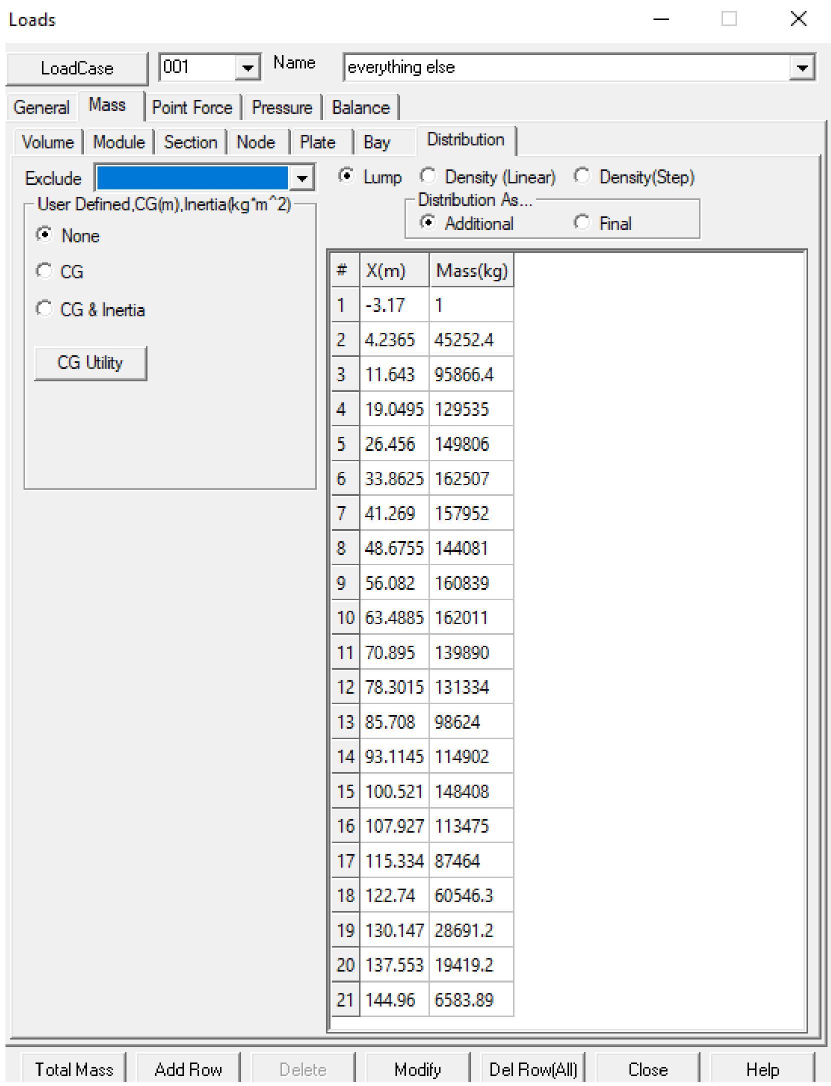


Figure 5/3-6 Frigate Lightship Weight Distribution

5/3.4 Boundary Conditions

Three traditional constraints are placed in the model as described in 3/2.7.5.1 and shown in Figure 5/3-7. For the still water condition, the total reaction force in the vertical direction is 1.6 tons, which is less than 1% of the ship's gross weight. Therefore, the constraints are considered satisfactory.

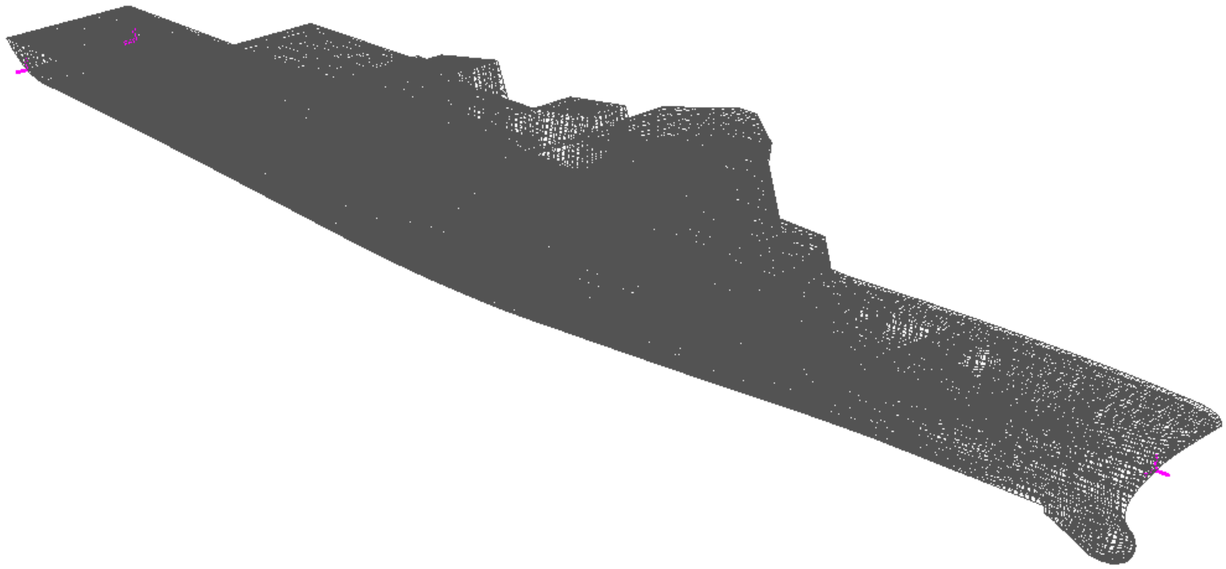


Figure 5/3-7 Typical Full-ship Boundary Constraints

5/3.5 **Hull Girder Design Waves**

Hull girder sagging and hogging design loads are generated using four different methods,

1. Quasi-static trochoidal waves
2. SPECTRA universal RAO
3. Frequency domain extreme loads
4. Time-domain extreme loads

Using the sample application FEA software internal hydrostatic balance feature, the prescribed trochoidal wave profiles are automatically adjusted to have proper sinkage, trim, and heel of the model into a quasi-static equilibrium between the defined weight and the buoyancy provided by the wettable elements. Figure 5/3-8 shows the wave pressure distribution of the frigate subjected to quasi-static sagging and hogging trochoidal waves.

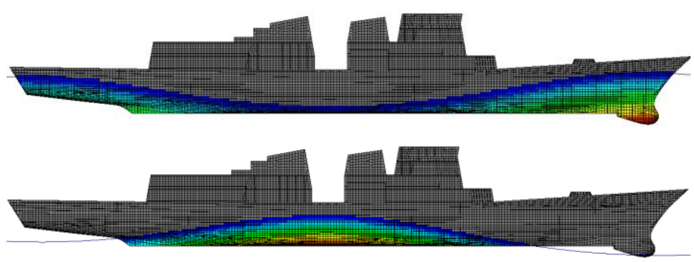


Figure 5/3-8 Quasi-static Pressure Under Sagging and Hogging Waves

The second method is to use SPECTRA to generate a hull girder lifetime extreme bending moment distribution envelope. Alternatively, one can create a design wave bending moment using equations provided by class society rule books. To simulate the bending moment distribution on a 3D finite element model, the sample application FEA software automatically places a large set of vertical nodal forces on the model such that the resulting hull girder longitudinal bending moment distribution matches the prescribed bending moment distribution. The third method uses the sample application FEA software hydrodynamic kernel to generate extreme design waves. This approach is based on ABS's DLA procedure. The frigate model has 19,266 wetted finite elements. The wetted elements can be further reduced to 8,202 evaluation patches, as shown in Figure 5/3-9. Those patches are then used for hydrodynamic loads calculations.

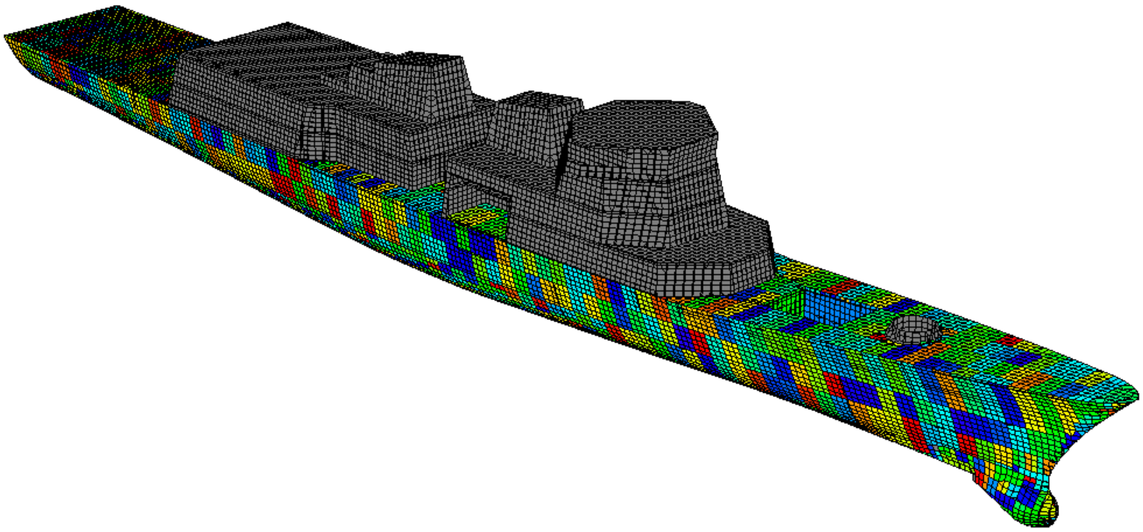


Figure 5/3-9 Frigate Model Evaluation Panels

For the frequency domain analysis, it is assumed the ship has a forward speed of 0, 5, 15, and 25 knots, 7 different headings, and 30 wave frequencies. A unit wave response database is generated. The database includes wave-induced accelerations, panel pressures, nodal forces due to hydrostatic restoring correction, and the hull girder loads. A design wave is generated based on a desired Dominant Load Parameter (DLP), a sea state scatter diagram, an operating profile, and the return period. The dynamic design wave, which has a perfect equilibrium (because the equations of motions are formulated on the FEA mesh rather than a hydrodynamic mesh), combined with the static loads, becomes a DLP static analysis load case. Figure 5/3-10shows an example of generating the extreme vertical bending moment design waves using the general Atlantic scatter diagram, combatant operating profile, Pierson Moskowitz wave spectrum, and 20 years return period. Figure 5/3-11 shows the hydrodynamic pressure and vertical bending moment distribution of the corresponding linear design waves.

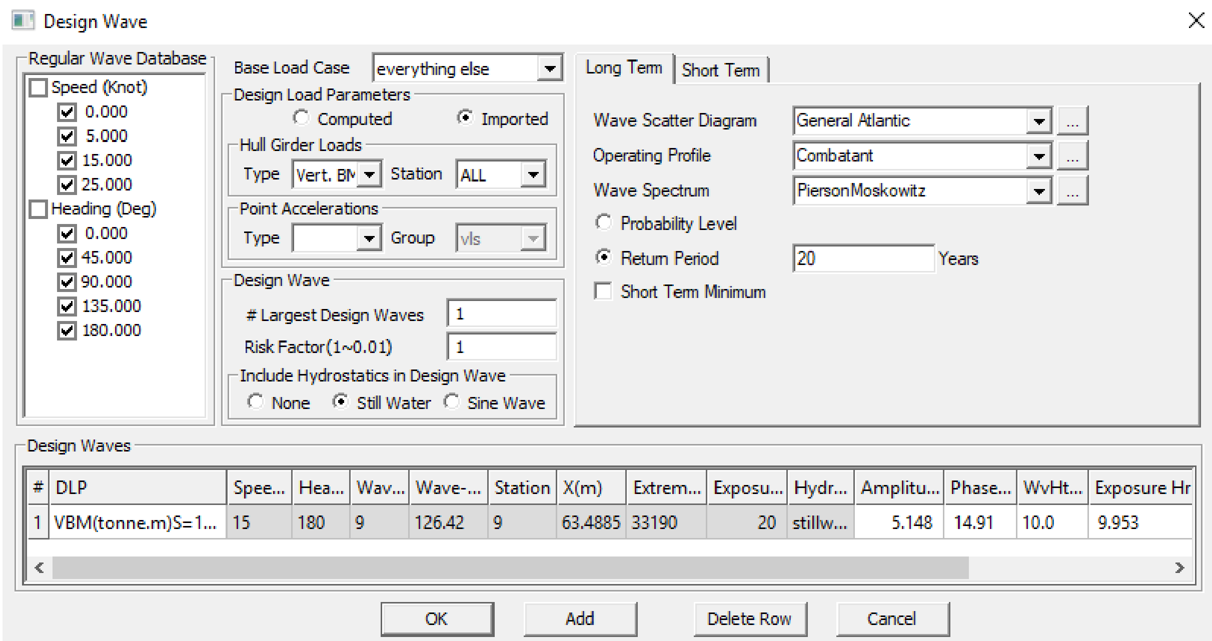
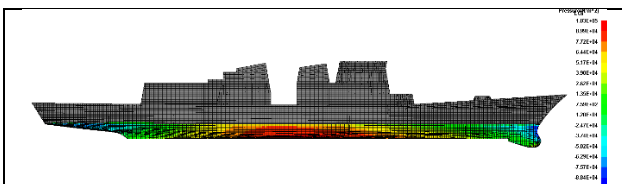
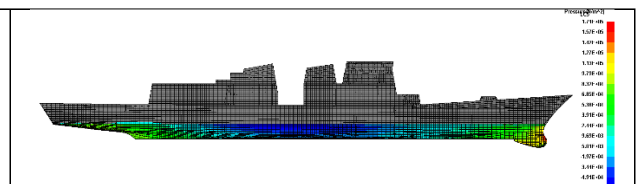


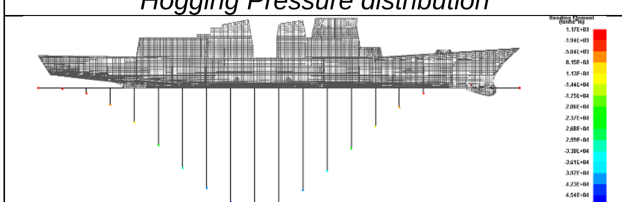
Figure 5/3-10 Example of Generating an Extreme Vertical Bending Moment Design Wave



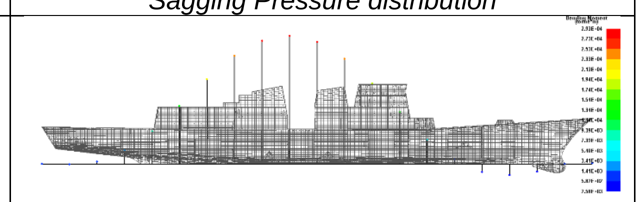
Hogging Pressure distribution



Sagging Pressure distribution



Hogging bending moment distribution



Sagging bending moment distribution

Figure 5/3-11 Hydrodynamic Pressure and Vertical Bending Moment of the Linear Design Waves

The fourth method is to use the sample application FEA software weakly nonlinear time-domain simulation to generate a hull girder extreme bending moment. From the frequency domain load analysis results, the expected most probable extreme hull girder vertical bending moment occurs when the ship sails head sea at 15 knots in sea state 8. A 20-minutes weakly nonlinear time-domain simulation at this condition was conducted. The maximum hull girder sagging and hogging moments were extracted from the time history. A load scale factor was determined using the ratio of the peak time-domain hull girder loads and the frequency domain hull girder loads. The panel pressures, point forces, and accelerations at the peak response time-step of each time-domain simulation were then scaled such that the magnitude of the DLP was equal to that predicted from the frequency domain analysis. If the peak of the time domain run exceeded the frequency domain extreme value, the peak of the time domain run was to be used for that particular DLP. Figure 5/3-12 shows an example of setting up a weakly-nonlinear time-domain simulation in a sea state. Figure 5/3-13 shows an example of vertical bending moment time history at mid-ship. Figure 5/3-14 shows the ship speed, heading, sea condition, and DLP VBM values where the time-domain simulation peaks occurred, and Figure 5.3-15 shows a snapshot of the maximum bending moment in the nonlinear simulation time history and the corresponding hydrodynamic pressure.

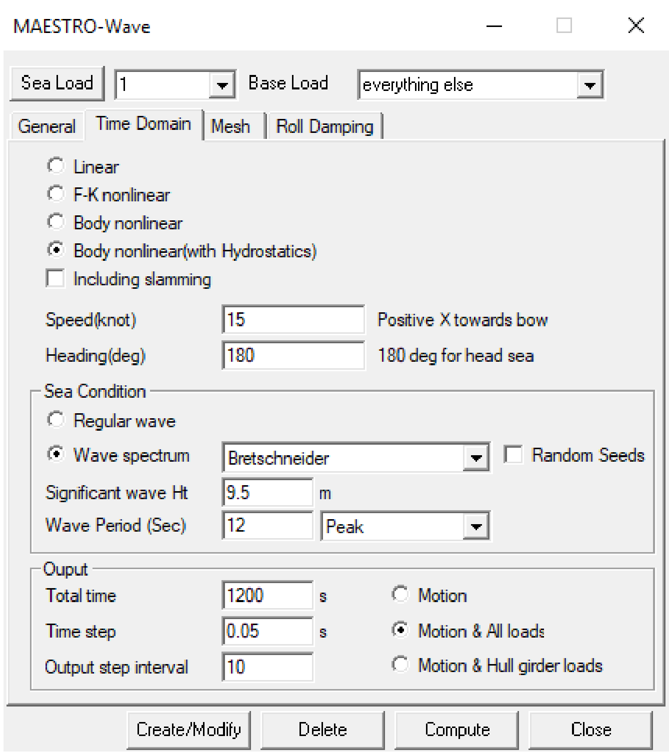


Figure 5/3-12 Example of Setting Up a Weakly-Nonlinear Time Domain Simulation

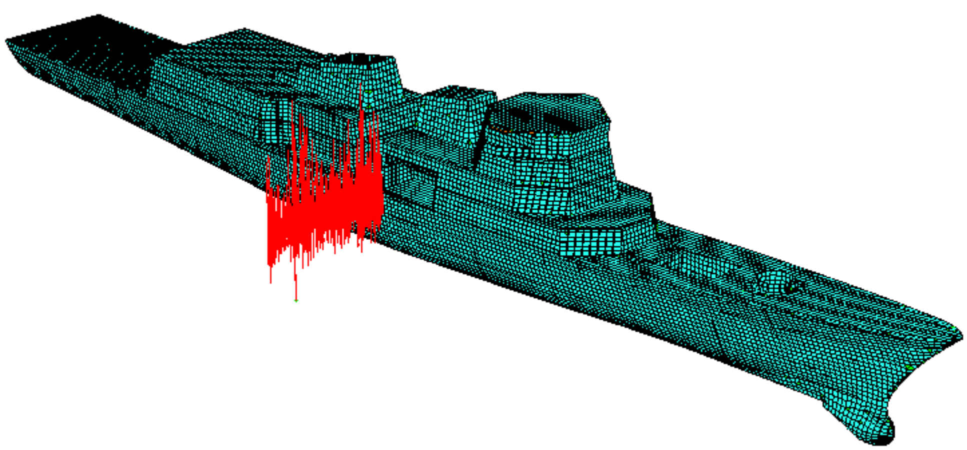


Figure 5/3-13 Example of Vertical Bending Moment Time History at Mid-ship

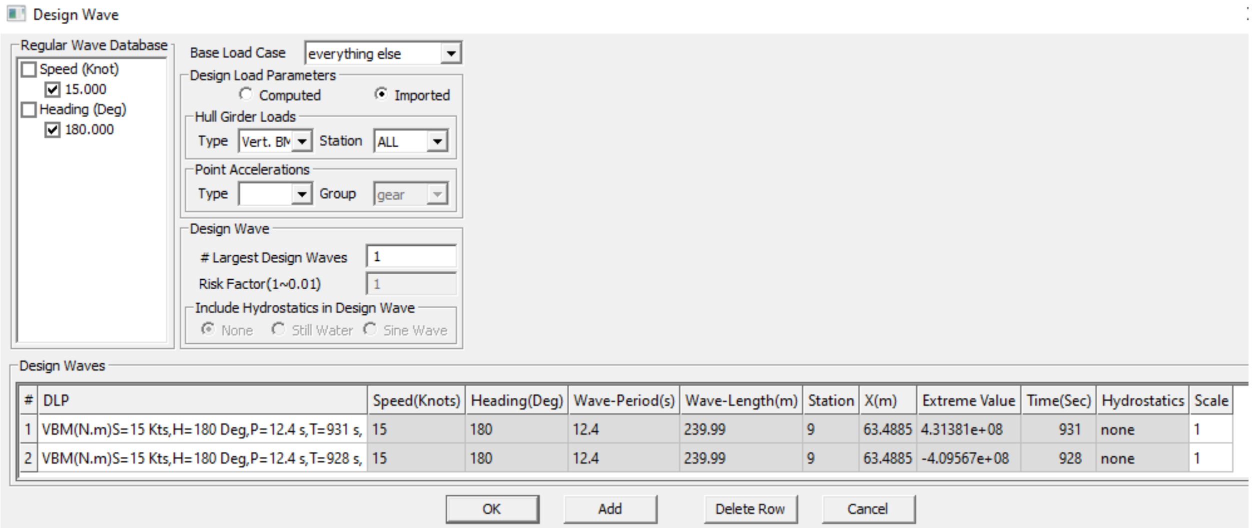


Figure 5/3-14 Maximum Vertical Bending Moments in this Time Domain Simulation

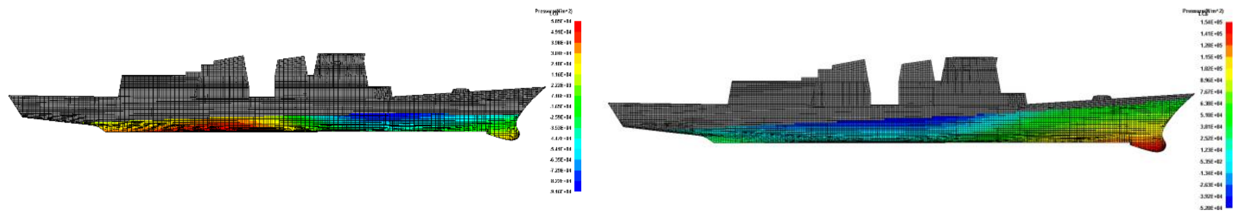


Figure 5/3-15 Pressure Distribution when the Maximum Vertical Bending Moments Occur

The extreme sagging and hogging moments computed by the four methods are listed in Table 5/3-1. For SPECTRA, a 20-year design life, the combatant operating profile, Pierson-Moskowitz wave spectrum and General Atlantic sea scatter diagram were used to generate the dynamic hull girder loads

Table 5/3-1 Design Bending Moments

	Sagging(ton*m)	**Hogging(ton*m)**
Quasi-Static Trochoidal Design Wave	22,614	28,856
SPECTRA	37,455	50,850
Frequency Domain	22,618	41,455
Time Domain	43,138	40,957

5/3.6 Limit State or Failure Criteria

To assess the strength of whole ship primary structures, the ABS High-Speed Naval Craft (HSNC) 2017 [86] limit states have been integrated into the sample application FEA software limit state evaluation and optimization frameworks, and are used in this example. The acceptance criteria include not only stress limits but also buckling rules and other limit states. Figure 5/3-16 shows the limit state evaluation panels automatically defined in the sample application FEA software for the entire ship's structure. Each colored group of elements represents a single evaluation patch, which is evaluated against each of the selected limit states for all load cases defined.

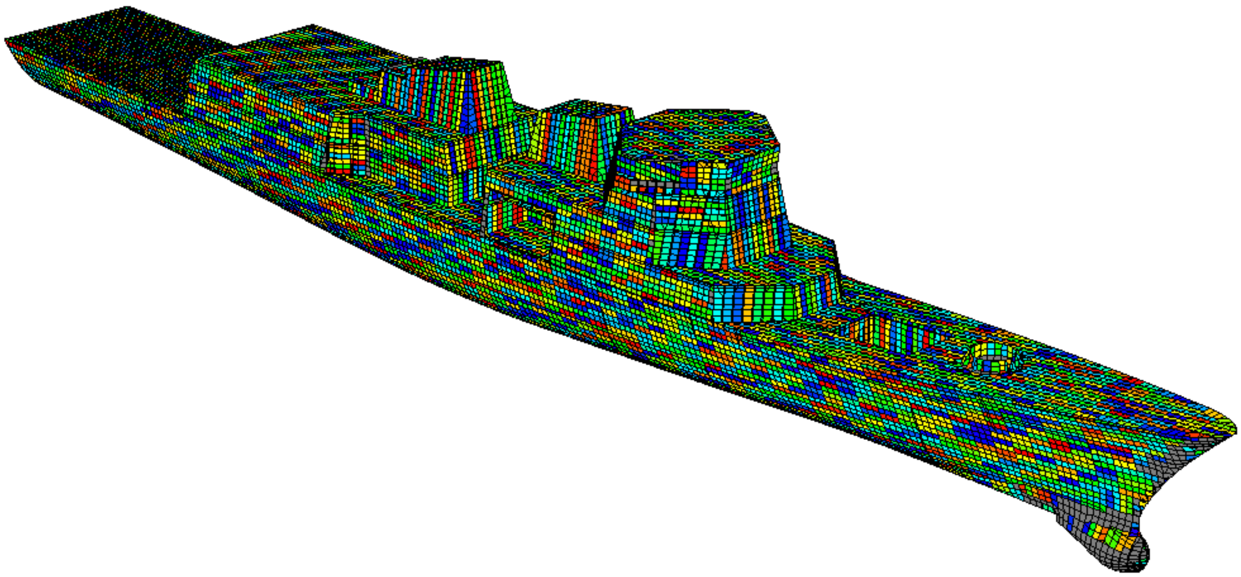


Figure 5/3-16 Frigate Evaluation Panels

Figure 5/3-17 presents the buckling, and ultimate strength limit states evaluated in the sample application FEA software during the optimization process or limit state evaluations outside of optimization. The HSNC 2017 buckling and ultimate strength limit states reference the criteria in 5-1-5/5.3.1 and 5-1-5/5.3.2 of the guide ABS Rules for Building and Classing Steel Vessels [65] .

3 Plate Panels

For rectangular plate panels between stiffeners, buckling is acceptable, provided that the ultimate strength given in 3/3.5 of the structure satisfies the specified criteria. Offshore practice demonstrates that only an ultimate strength check is required for plate panels. A buckling check of plate panels is necessary when establishing the attached plating width for stiffened panels. If the plating does not buckle, the full width is to be used. Otherwise, the effective width is to be applied if the plating buckles but does not fail.

3.1 Buckling State Limit

For the Buckling State Limit of plates subjected to in-plane and lateral pressure loads, the following strength criterion is to be satisfied:

$$\left(\frac{\sigma_{x\max}}{\eta\sigma_{Cx}}\right)^2+\left(\frac{\sigma_{y\max}}{\eta\sigma_{Cy}}\right)^2+\left(\frac{\tau}{\eta\tau_C}\right)^2\le 1$$

3.3 Ultimate Strength under Combined In-plane Stresses

The ultimate strength for a plate between stiffeners subjected to combined in-plane stresses is to satisfy the following equation:

$$\left(\frac{\sigma_{x\max}}{\eta\sigma_{Ux}}\right)^2-\varphi\left(\frac{\sigma_{x\max}}{\eta\sigma_{Ux}}\right)\left(\frac{\sigma_{y\max}}{\eta\sigma_{Uy}}\right)+\left(\frac{\sigma_{y\max}}{\eta\sigma_{Uy}}\right)^2+\left(\frac{\tau}{\eta\tau_U}\right)^2\le 1$$

3.5 Uniform Lateral Pressure

In addition to the buckling/ultimate strength criteria in 3/3.1 through 3/3.3, the ultimate strength of a panel between stiffeners subjected to uniform lateral pressure alone or combined with in-plane stresses is to also satisfy the following equation:

$$q_u\le \eta 4.0\sigma_0\left(\frac{t}{s}\right)^2\left(1+\frac{1}{\alpha^2}\right)\sqrt{1-\left(\frac{\sigma_e}{\sigma_0}\right)^2}$$

5.1 Beam-Column Buckling State Limit

The beam-column buckling state limit may be determined as follows:

$$\frac{\sigma_a}{\eta\sigma_{CA}(A_e/A)}+\frac{C_m\sigma_b}{\eta\sigma_0[1-\sigma_a/(\eta\sigma_{E(C)})}\le 1$$

5.3 Flexural-Torsional Buckling State Limit

In general, the flexural-torsional buckling state limit of stiffeners or longitudinals is to satisfy the ultimate state limit given below:

$$\frac{\sigma_a}{\eta\sigma_{CT}}\le 1$$

5.5 Local Buckling of Web, Flange and Face Plate

The local buckling of stiffeners is to be assessed if the proportions of stiffeners specified in Subsection 3/9 are not satisfied.

Figure 5/3-17 ABS HSNC Buckling and Ultimate Strength Limit States

5/3.7 Finite Element Analysis and Results

A finite element and limit state analysis of the frigate's structural design was conducted using the eight load cases created using the methods discussed in Section 5/3.5. The results for the extreme vertical sagging bending moment (time-domain weakly nonlinear simulation under full load, sea state 8, 15 knots, head sea, Sagging VBM = 43,138 ton*m) are shown here. The colored elements shown in Figure 5/3-18 indicate those elements that do not meet the maximum allowable stress limits as specified in the HSNC Rules. It should be noted that, for the most part, except hatch corners, the stresses are within the ABS HSNC limits. That is not true of the structure's evaluation against the HSNC buckling limits. Figure 5/3-19 shows those elements that fail to meet, in this load case alone, the HSNC's various buckling and ultimate limit state criteria as outlined in Section 5/3.6 (Figure 5/3-17).

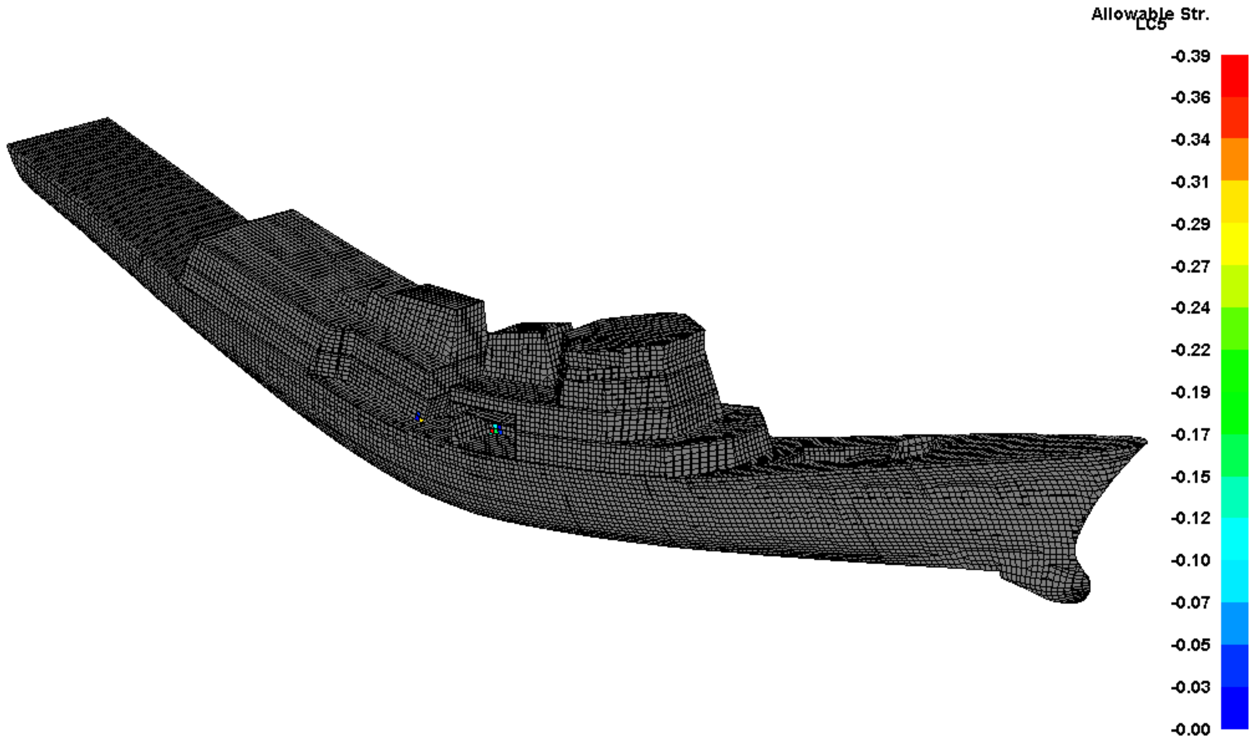


Figure 5/3-18 Allowable Stress Under Time Domain Sagging Condition

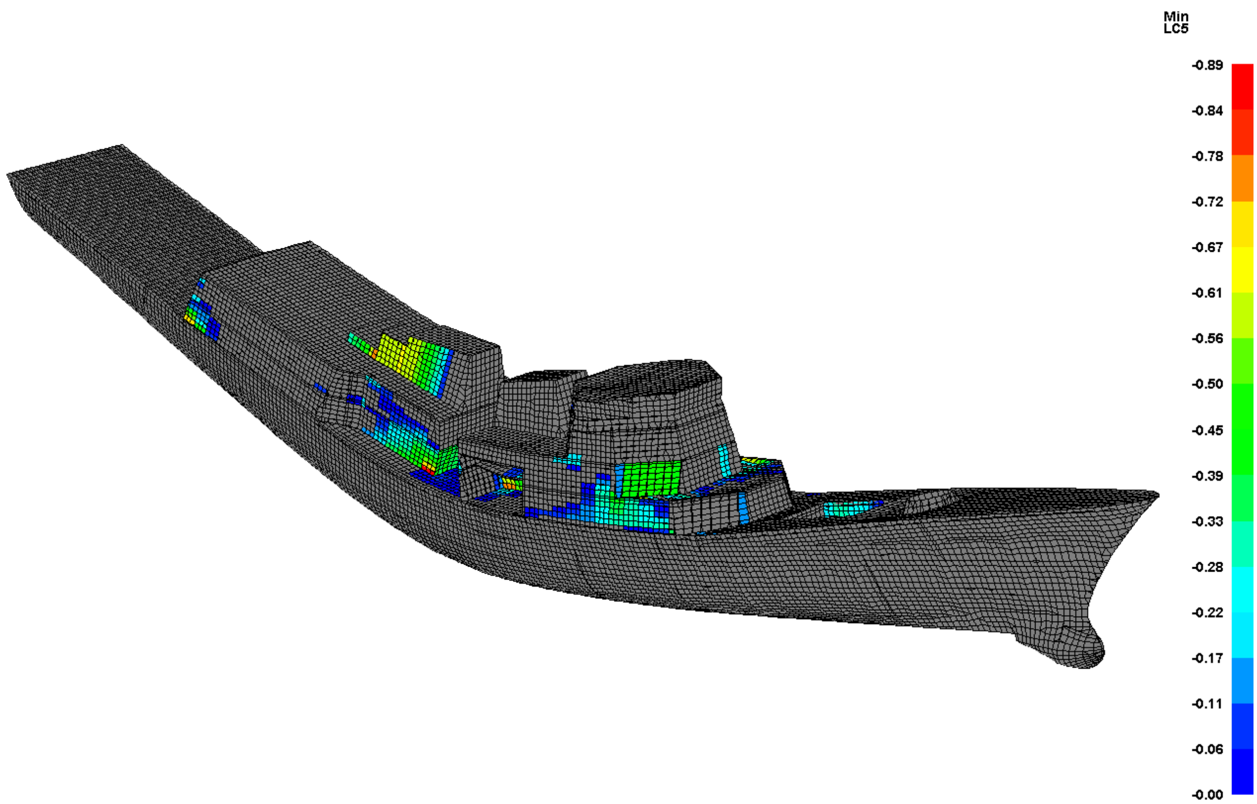


Figure 5/3-19 All Limit States Under Time Domain Sagging Condition

5/3.8 Sample Showing the Significance of Equilibrium for Whole Ship Analysis

For a floating structure, it is important to obtain equilibrium before performing a finite element analysis. Imbalance in a model causes errors in the results. To illustrate this, a design exercise is given in this section.

A frigate model is given in section 5/3.1. The hydrostatic model was balanced using the hydrostatic software in the design stage prior to a finite element analysis. The floating condition of a draft, trim, and heel was given from the hydrostatic software. The balanced floating condition had a draft of 3.96 m, with no heel and -0.382 degree trim. In regular finite element analysis, this floating condition is used to load the finite element model. Figure 5/3-20 and Figure 5/3-21 show the weight and buoyancy distribution of NAPA (the hydrostatic software) and MAESTRO (the sample FEA software), respectively. While the weight distribution of these software has a good agreement, the buoyancy distribution of the sample FEA software and the hydrostatic software does not have good agreement. Such a discrepancy may be caused by several possible reasons: A) Rudders and propellers are usually not modeled in the finite element model for hull girder strength analysis. B) The integration schemes are different. The buoyancy calculation in the hydrostatic software is volume-based, using continuous curves and/or surfaces. For finite element analysis packages, the buoyancy calculation is pressure based, as a result of integrating the hydrostatic pressure over the faceted shell elements in the finite element model.

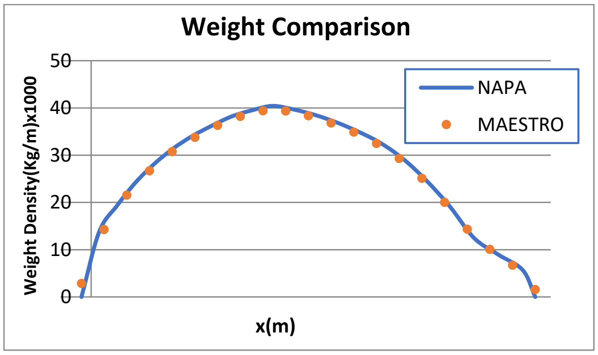


Figure 5/3-20 Weight Distribution Comparison

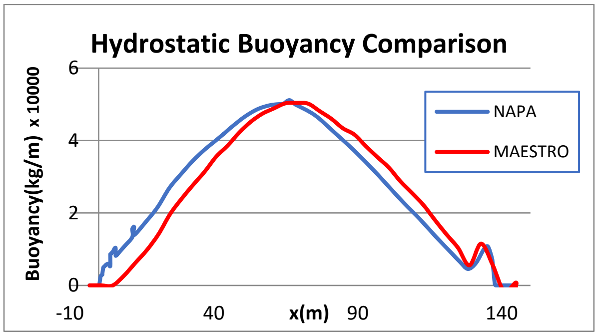
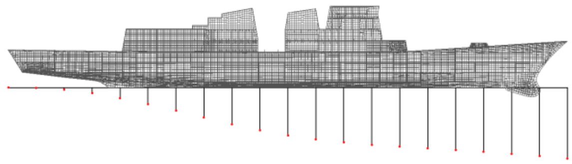
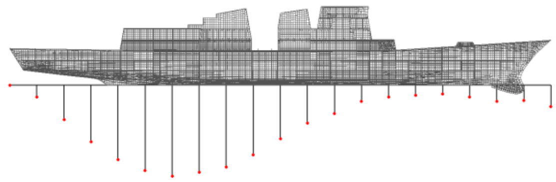


Figure 5/3-21 Buoyancy Distribution Comparison

To solve the above finite element model, three nodal constraints were placed near the longitudinal neutral axis of the model to prevent the rigid body motion, with two located at the stern and one at the bow. If the model is properly balanced, the restraining forces will be negligible. In order to check the balance, the sample application FEA software computes and plots hull girder responses before the restraints are applied. Figure 5/3-22 shows the vertical bending moment, and vertical shear force before the restraints are applied. Since the curves are not closed, they reveal that the model is not well balanced. Figure 5/3-23 shows the resulting bending moment distribution, which includes the reaction forces due to the constraints. Figure 5/3-24 shows the deflection and stress distribution. The lack of buoyancy at the stern and the resulting non-negligible restraint forces cause an incorrect change of sign in the bending moment and in the curvature of the deflected hull.



(a) Bending moment



(b)Shear force

Figure 5/3-22 Bending Moment and Shear Force Distribution before Applying Restraining Forces

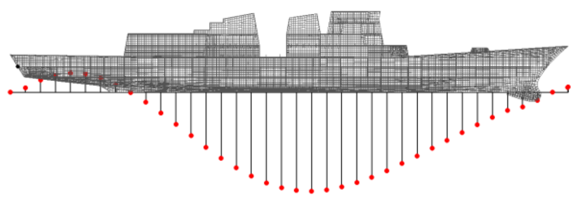


Figure 5/3-23 Bending Moment Distribution after Applying Restraining Forces

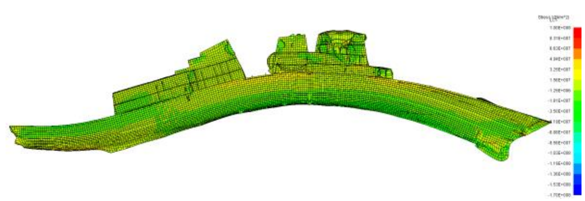


Figure 5/3-24 Deflection and Stress Distribution

If the analyst is using a general-purpose FEA program, then they must work to resolve as much of the force and moment imbalance as possible prior to evaluating any FEA results. Either the vessel draft and trim may need to be modified in the FEA model, or the applied loads may need to be reviewed for accuracy. The results shown in Figure 5/3-24 would be questionable.

Finite element tools specifically developed for floating structures usually provide a “hydrostatic balancing kernel” by which an imbalanced finite element model can be automatically rebalanced by adjusting draft, heel, and trim. After this hydrostatic balancing, the finite element model has a draft of 4.07m and a trim of 0.437 degrees, which are small changes comparing to the original hydrostatic software’s floating condition (0.397m draft, and -0.382 degree trim). Once the model is balanced, the buoyancy discrepancy between the hydrostatic software and the FEA software is corrected, as shown in Figure 5/3-25. Likewise, the distributions of vertical bending moment and shear force are correctly closed at both ends, as shown in Figure 5/3-26, and the corresponding deflection and stress distributions are as expected, as shown in Figure 5/3-27.

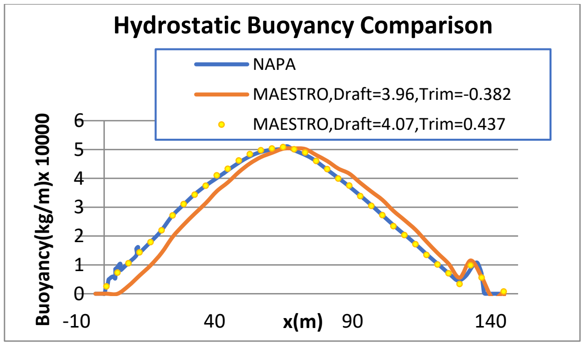
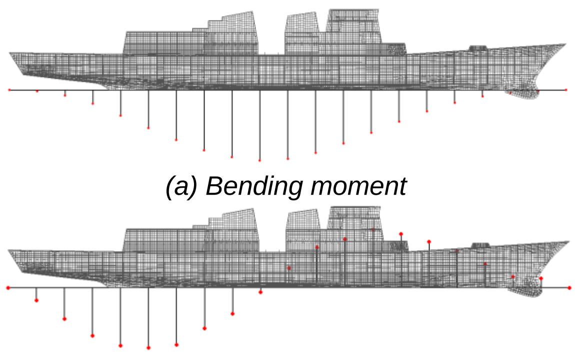


Figure 5/3-25 Buoyancy Distribution Comparison with MAESTRO Auto-Balancing



(a) Bending moment

(b) Shear force

Figure 5/3-26 Bending Moment and Shear Force Distribution after Hydrostatic Balance

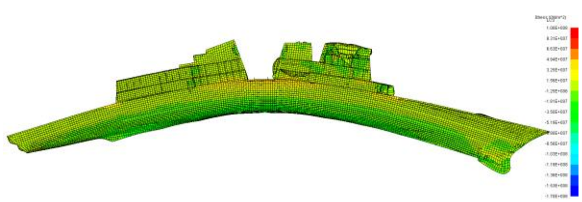
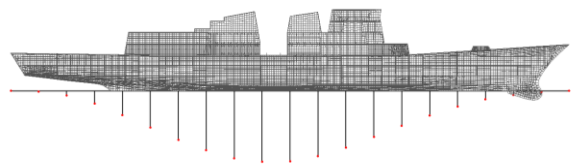


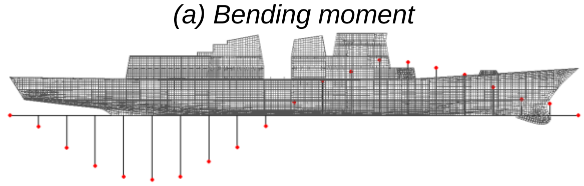
Figure 5/3-27 Stress Distribution after Hydrostatic Balance

Hydrostatic imbalance can also result when the loads are derived from a linear seakeeping analysis, where the mean water surface plane is prescribed. There are two main reasons for this as follows: (1) hydrostatic balancing can only achieve equilibrium in heave, heel and trim, but not in surge, sway and yaw, and(2) a hydrostatic rebalance would cause a change of the mean water surface plane, which would require re-running the linear seakeeping analysis.

The “inertia relief” method is commonly used to resolve imbalance forces and moments. Figure 5/3-28 shows the distributions of vertical bending moment and shear force after using “inertia relief,” and Figure 5/3-29 shows the corresponding deflection and stress distribution.



(a) Bending moment



(b) Shear force

Figure 5/3-28 Bending Moment and Shear Force Distribution after Inertia Relief

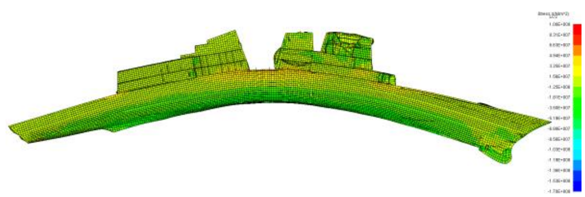


Figure 5/3-29 Stress Distribution after Inertia Relief

The hull girder responses may be different depending on whether the finite element model is loaded using inertia relief, hydrostatic balancing within the finite element model, or hydrostatic balance outside the finite element model. Figure 5/3-30 shows the longitudinal bending moment distribution of (a) the model is balanced in the original hydrostatic model (b) the model is balanced using inertia relief, and (c) the model is hydrostatically balanced using the finite element geometry and weight distribution.

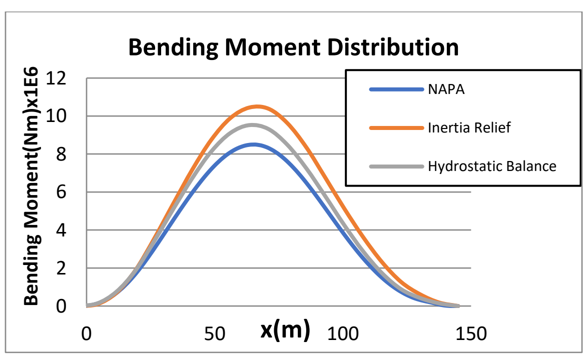


Figure 5/3-30 Comparison of Bending Moment Distribution

5/4 FREQUENCY RESPONSE VIBRATION ANALYSIS

5/4.1 Sample Application Description

The following sample application is an example of how to perform a frequency response vibration analysis (i.e., a forced vibration analysis). The sample application is a modified version of a report issued to a commercial customer. Some vessel identifying information has been removed.

5/4.2 Introduction

This report validates the main engine and gear box foundation design vibration characteristic for a vessel with a conventional low-speed diesel engine propulsion system. This report is an example of how to perform a forced vibration analysis. Note that this report is a modified version of a report issued to a commercial customer; some vessel identifying information has been removed.

5/4.3 Method

5/4.3.1 General

This report evaluates the subject foundation based on [14] ABS Guidance Notes on Ship on Ship Vibration criteria for local structure. This analysis uses ANSYS 19.0, an industry-leading finite element analysis software.

The analysis model was created based on the project foundation drawing and the available project ship structure drawings. The analysis model extends from main engine room bulkhead to main engine room bulkhead and includes the tank top, floors, girders, and shell plate. It also includes the main engine room stanchions. The edges of the model are simply supported. Images of this model are shown in Figure 5/4-1 to Figure 5/4-4.

The model is created using shell elements except for the main engines and gearboxes. The main engine and gearbox are modeled using brick elements. The main engine and gearbox models approximate the correct equipment masses, centers of gravity, and connections with the foundation.

The model boundary is the engine room bulkheads and the tops of the stanchions. These boundaries have a zero-displacement boundary condition.

The foundation’s lowest natural frequencies are first determined using a modal analysis. A frequency response (forced vibration) analysis is then performed to determine the subject foundation responses to the excitation forces.

The modal superposition method is used for the frequency response analysis. The frequency response analysis calculates the main engine induced maximum vibrations and propeller-induced maximum vibrations separately. The separate vibrations values are added together where applicable. This is conservative where added together because the maximum motions likely do not occur at the same location in the same direction. The total vibration results are then compared against the criteria in [14].

The analysis is done for one case with just the port side power train operating, and for three cases with both power trains operating. The three cases with both power trains operating have the starboard main engine 0 degrees, 90 degrees, and 180 degrees out of phase of the port engine. This was done to test the impact of out of phase main engine excitation. The propeller excitation is kept in sync for all these cases.

The Main Engine Technical Information and Propeller Blade Design Report describe the main engine and propeller excitation forces, respectively. The excitation frequencies range from 0 to 25.9 Hz. There are three cyclical loads of concern, and they are as follows:

- The main engine is rated for up to 15 Hz (900 rpm).
- The gearbox ratio is 4.64:1, and the propeller has four blades. The blade rate, therefore, is 12.9 Hz (776 rpm).

- The prop has a v-strut in front of it, so a frequency of twice the blade rate or 25.9 Hz may also be an important forcing frequency. Note also that this cyclical load may be occurring simultaneously with the other two.

Reference [88] recommends that a range of frequencies +/- 5 to 10% be evaluated surrounding each excitation frequency and that the worst-case frequency be used. Therefore for simplicity, the frequencies were evaluated at 5 Hz increments surrounding the target excitation frequencies. In practice, this works out to taking the highest response from three frequencies for each of the cyclical loads. As an example, in the case of the main engine cyclical load, the three frequencies that are relevant are the 10 Hz, 15 Hz, and 20 Hz results.

Further, the vibration is assumed to be linear, and so, therefore, the maximum individual response values for each cyclical load can be summed together to get a conservative maximum total response. The maximum total response in velocity, therefore, is taken as the summation of three maximum individual responses in velocity: the main engine response velocity, the propeller blade rate analysis response velocity, and the propeller twice the blade rate response velocity. The result is compared against the ABS criteria maximum allowable velocity.

The maximum total velocity is determined without consideration for where on the model, the maximum individual velocities are occurring. This is only because the results were much less than the allowable criteria. Had the results been closer to the limiting criteria, then the locations of these maximums would have been taken into consideration.

5/4.3.2 Model Size Variation

This report also includes additional results using a smaller model to show the impact of the model size. The smaller model only includes the equipment and foundation structure above the tank top on the port side. Images of this model are shown in Figure 5/4-5 to Figure 5/4-8.

5/4.4 Given and Assumed Parameters

5/4.4.1 Equipment

Equipment List:

Main Engine: Make/Model - Mass

Gear Box: Make/Model – Mass

5/4.4.2 Vibration Criteria

ABS Guidance Notes on Ship on Ship Vibration Section 7 Acceptance Criteria, Part 5 Vibration Limits for Local Structures [14] says the following:

- For each peak response component (in either vertical, transverse, or longitudinal direction), from 1 Hz to 5 Hz, the displacement is recommended below 1.0 mm, and the damage is probable above 2.0 mm
- For each peak response component (in either vertical, transverse, or longitudinal direction), from 5 Hz and above, the velocity is recommended below 30 mm/sec, and the damage is probable above 60 mm/sec.

The peak excitation forces occur above 5 Hz. Therefore it is required that the maximum structural velocity is less than 30 mm/sec.

ABS Guidance Notes on Ship on Ship Vibration Section 7 Acceptance Criteria, Part 7 Vibration Limits for Machinery [14] provides the Table 5/4-1 machinery vibration limits.

Table 5/4-1 ABS Machinery Vibration Limits [14]

Propulsion Machinery	*Limits (rms)*
Thrust Bearing and Bull Gear Hub	5 mm/s
Other Propulsion Machinery Components	13 mm/s
Stern Tube and Line Shaft Bearing	7 mm/s
Diesel Engine at Bearing	13 mm/s
Slow & Medium Speed Diesel Engine on Engine Top (over 1000 HP)	18 mm/s
High Speed Diesel Engine on Engine Top (less 1000 HP)	13 mm/s

Therefore it is required that the maximum diesel engine velocity is less than 13 mm/sec, and the maximum thrust bearing and bull gear hub (gearbox) is less than 5 mm/sec.

The peak displacements are also reported for reference.

5/4.4.3 Excitation Forces

5/4.4.3.1 Main Engine Excitation Forces

The Main Engine Technical Information reports describe the main engine excitation forces as follows:

> AS A DESIGN FIGURE THE DYNAMIC FORCE PRODUCED AT BOTH ENDS OF THE ENGINE, AT 900 RPM ENGINE SPEED, WILL NOT EXCEED 500 LBS, AT A FREQUENCY OF 15 HZ.

Therefore a rotating force of 500 lbs is applied about the longitudinal axis of the engine. This force is applied to the forward and aft face of the main engine.

This force will peak at 15 Hz, as stated.

5/4.4.3.2 Propeller Excitation Forces

Propeller Blade Design Report determines the propeller excitation force amplitude is 1.8 kN or 405 lbs. Therefore, a dynamic force of 405 lbs is applied in the direction of the shaft centerline. This force is applied to the aft face of the gearbox in this analysis.

The blade rate, therefore, is 12.9 Hz.

The prop has a v-strut in front of it, so a frequency of twice the blade rate may also be an important forcing frequency; 25.9 Hz.

5/4.5 Results

5/4.5.1 Bulkhead to Bulkhead Model

The modal analysis determined the 12 lowest modal frequencies as listed in Table 5/4-2 below for the bulkhead to bulkhead model. The lowest natural frequencies are at the high end of the highest excitation frequencies. Therefore it is expected, and the frequency response results show that the most significant vibration motions occur as a result of the high-end excitation frequencies.

Table 5/4-2 Bulkhead to Bulkhead Modal Frequencies

Mode No.	Nat. Freq [Hz]	Description
1	24.9	Aft centerline stanchion surging near boundary condition.
2	25.2	Middle centerline stanchion surging near boundary condition.
3	27.9	Small outboard port side floor unconnected to bottom shell (modeling error).
4	28.0	Small outboard starboard side floor unconnected to bottom shell (modeling error).

Mode No.	Nat. Freq [Hz]	Description
5	28.7	Starboard outboard aft stanchion surging between its end connections.
6	28.7	Port outboard aft stanchion surging between its end connections.
7	32.5	Shell and tank top heaving below main engines.
8	33.2	Main engines swaying side to side in tandem.
9	34.2	Middle centerline stanchion surging between boundary conditions.
10	34.6	Aft centerline stanchion surging between boundary conditions.
11	35.0	Port outboard aft stanchion swaying between its end connections.
12	35.1	Starboard outboard aft stanchion swaying between its end connections.

Table 5/4-3 and Table 5/4-4 show the bulkhead to bulkhead model maximum vibration-induced displacements & velocities based on the frequency response analysis. The data is provided for a range of forcing frequencies near the predominant 15 Hz and 25 Hz forcing frequencies.

Table 5/4-3 shows the bulkhead to bulkhead model results for the cases with both power trains operating. Table 5/4-4 shows the results with only the port power train operating. Table 5/4-3 cases with both power trains operating results in more severe motions. The most severe motions within the cases with both power trains operating depend on the forcing frequency.

The main engine maximum individual velocity response is 0.408 mm/s. The blade rate maximum individual response is 0.099 mm/s. The twice blade rate maximum individual response is 0.562 mm/s. The maximum total velocity response is, therefore, 1.069 mm/s. This is significantly less than the least allowable maximum velocity of 5 mm/sec in [14]. The foundation, therefore, will not have vibration issues.

Table 5/4-3 Bulkhead to Bulkhead Model Vibration Results: Port and Starboard Power Trains in Sync

	Stbd Phase Lag of 0°		Stbd Phase Lag of 90°		Stbd Phase Lag of 180°			
	Main Engine P/S		Main Engine P/S		Main Engine P/S		Propeller P/S	
	Displacement	Velocity	Displacement	Velocity	Displacement	Velocity	Displacement	Velocity
[Hz]	[mm]	[mm/s]	[mm]	[mm/s]	[mm]	[mm/s]	[mm]	[mm/s]
5	0.002	0.070	0.002	0.054	0.002	0.074	0.001	0.022
10	0.002	0.148	0.002	0.113	0.002	0.155	0.001	0.045
15	0.003	0.251	0.002	0.184	0.002	0.256	0.001	0.070
20	0.003	0.408	0.002	0.280	0.002	0.405	0.001	0.099
21							0.001	0.107
22							0.001	0.195
23							0.002	0.308
24							0.003	0.431
25							0.004	0.562
26							0.003	0.477
27							0.002	0.383
28							0.002	0.281
29							0.002	0.308
30							0.002	0.395

Table 5/4-4 Bulkhead to Bulkhead Model Vibration Results: Port Power Train Only

	Main Engine P						Propeller P	
	Displacement	Velocity					Displacement	Velocity
[Hz]	[mm]	[mm/s]					[mm]	[mm/s]
5	0.002	0.071					0.001	0.002
10	0.002	0.149					0.001	0.050
15	0.003	0.247					0.001	0.077
20	0.003	0.385					0.001	0.106
21							0.001	0.112
22							0.001	0.119
23							0.001	0.160
24							0.001	0.224
25							0.002	0.292
26							0.002	0.247
27							0.001	0.199
28							0.001	0.180
29							0.001	0.214
30							0.001	0.251

5/4.5.2 Tank Top Model

The tank top port side model main engine maximum individual velocity response is 0.206 mm/s. This is only 53% of the same result for the bulkhead to bulkhead model results with just the port side forces. The blade rate maximum individual response is 0.093 mm/s. This is only 88% of the same result for the bulkhead to bulkhead model results with just the port side forces. The twice blade rate maximum individual response is 0.144 mm/s. This is only 49% of the same result for the bulkhead to bulkhead model results with just the port side forces. In summary, the maximum individual responses are all significantly less for the tank top model. The extra effort to develop the larger bulkhead to bulkhead model was, therefore, worthwhile in order to avoid reporting a non-conservative result.

Table 5/4-5 Tank Top Model Vibration Results

	Main Engine P		Propeller P	
	Displacement	Velocity	Displacement	Velocity
[Hz]	[mm]	[mm/s]	[mm]	[mm/s]
5	0.001	0.047	0.001	0.023
10	0.002	0.096	0.001	0.046
15	0.002	0.148	0.001	0.069
20	0.002	0.206	0.001	0.093
25			0.001	0.118
30			0.001	0.144

5/4.6 Conclusions

The calculated vibrations are acceptable. They do not exceed the allowable limits defined in [14].

5/4.7 Appendix A: GEOMETRY PLOTS

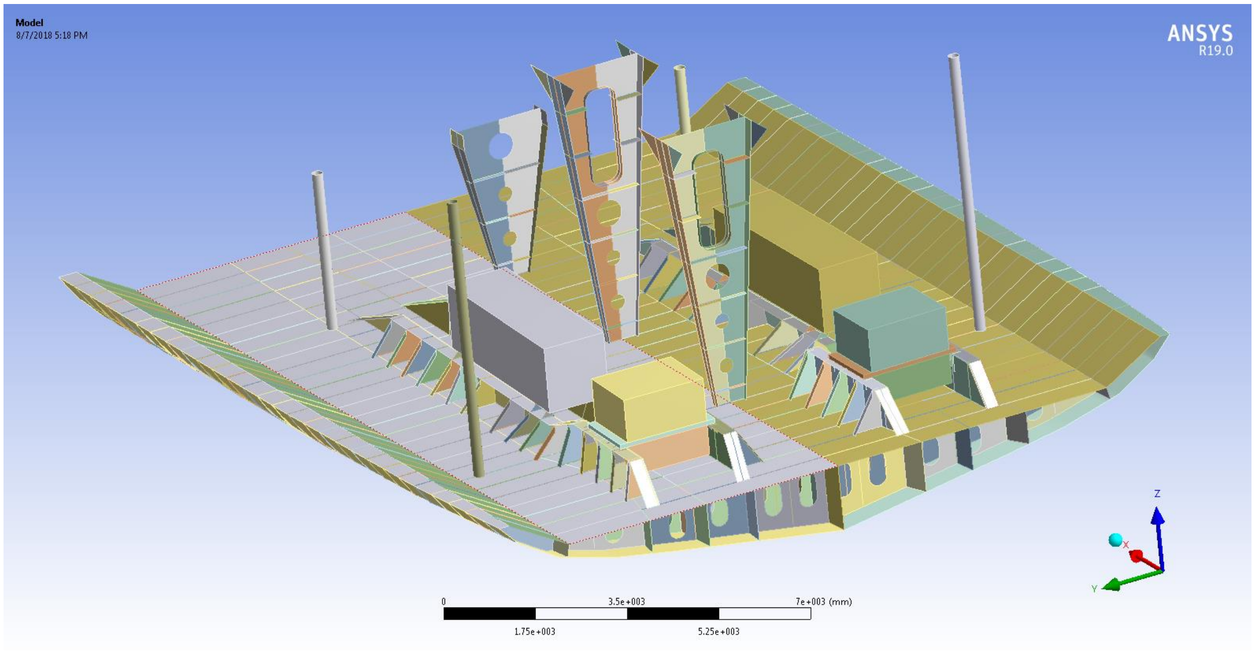


Figure 5/4-1 Bulkhead to Bulkhead Model Image 1

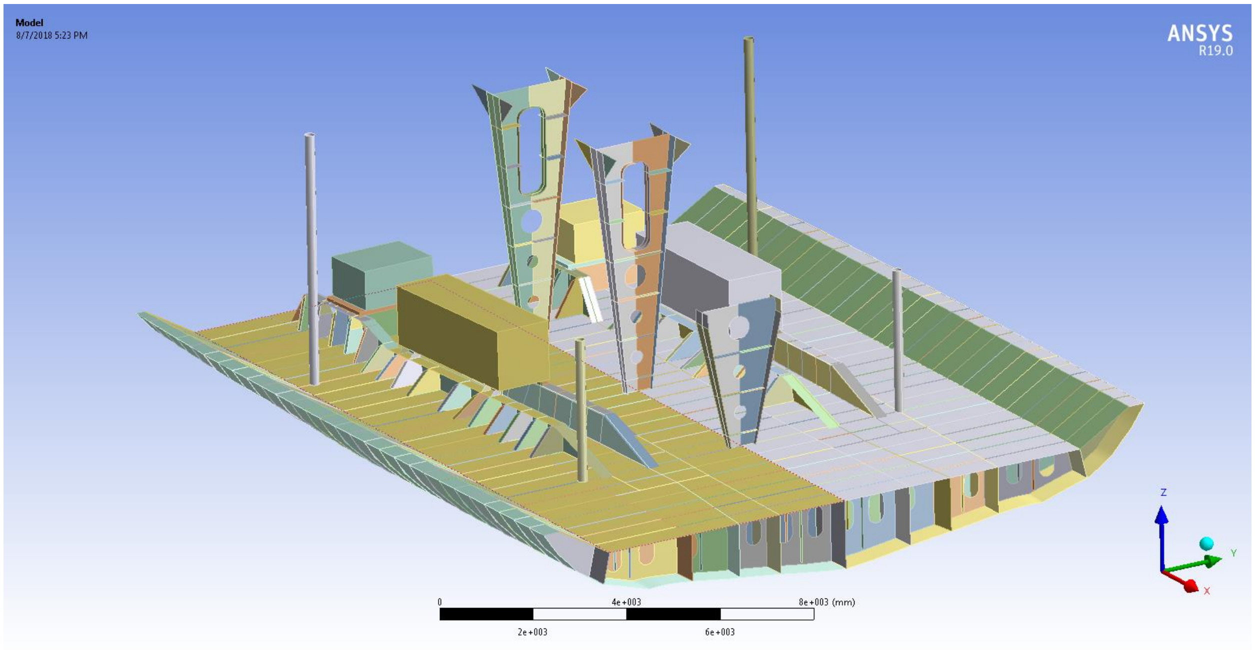


Figure 5/4-2 Bulkhead to Bulkhead Model Image 2

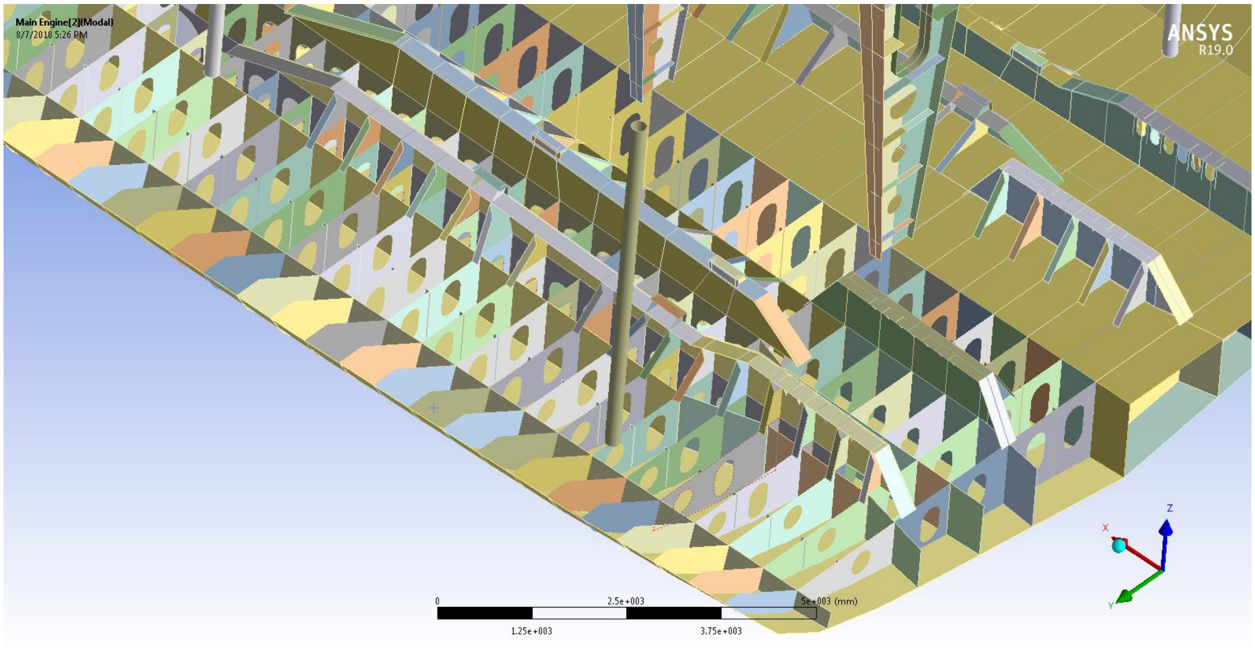


Figure 5/4-3 Bulkhead to Bulkhead Model Image 3

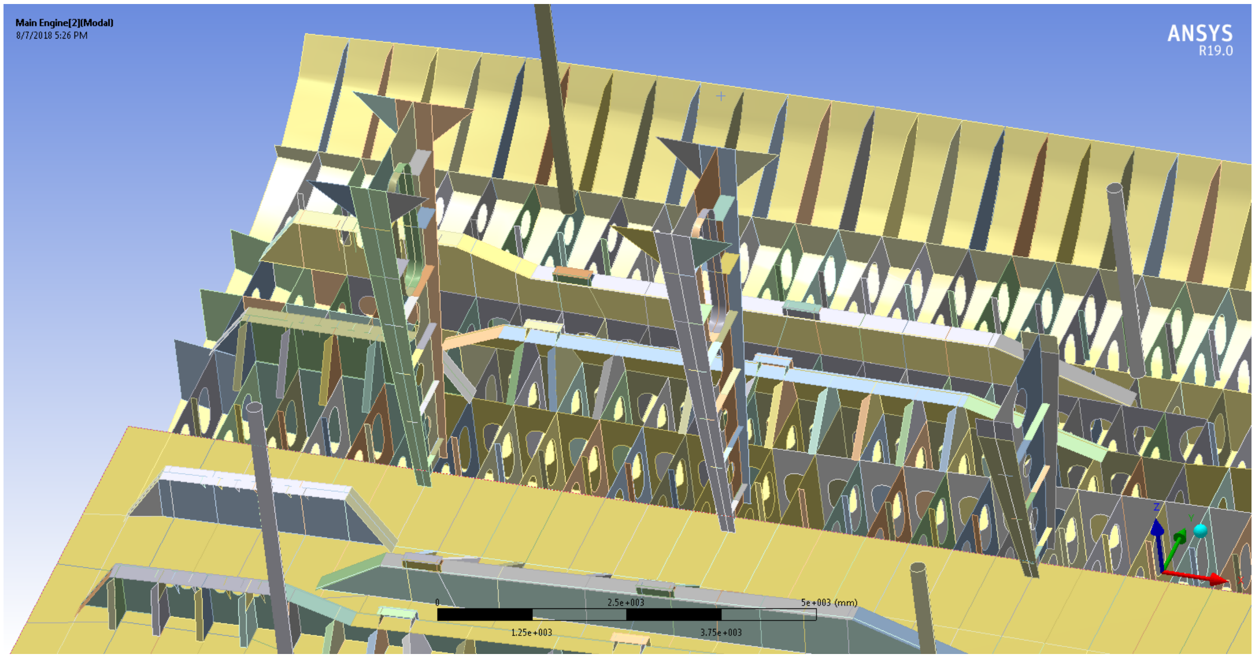


Figure 5/4-4 Bulkhead to Bulkhead Model Image 4

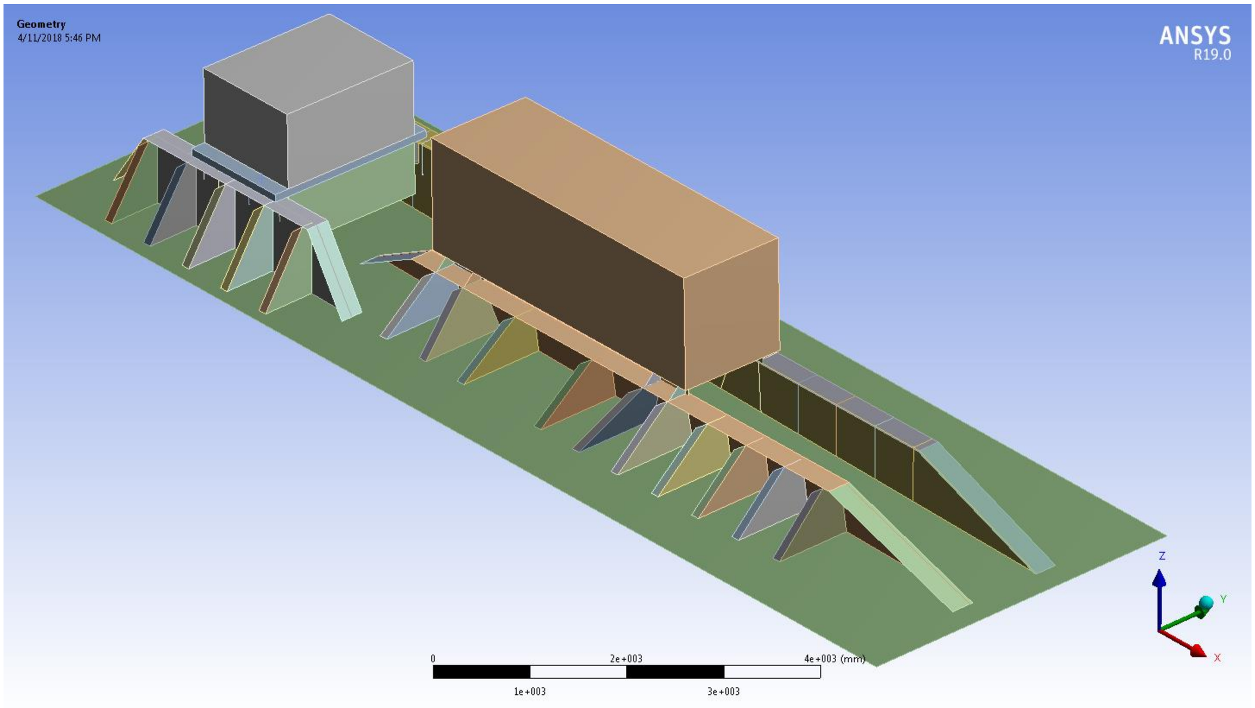


Figure 5/4-5 Tank Top Model Image 1

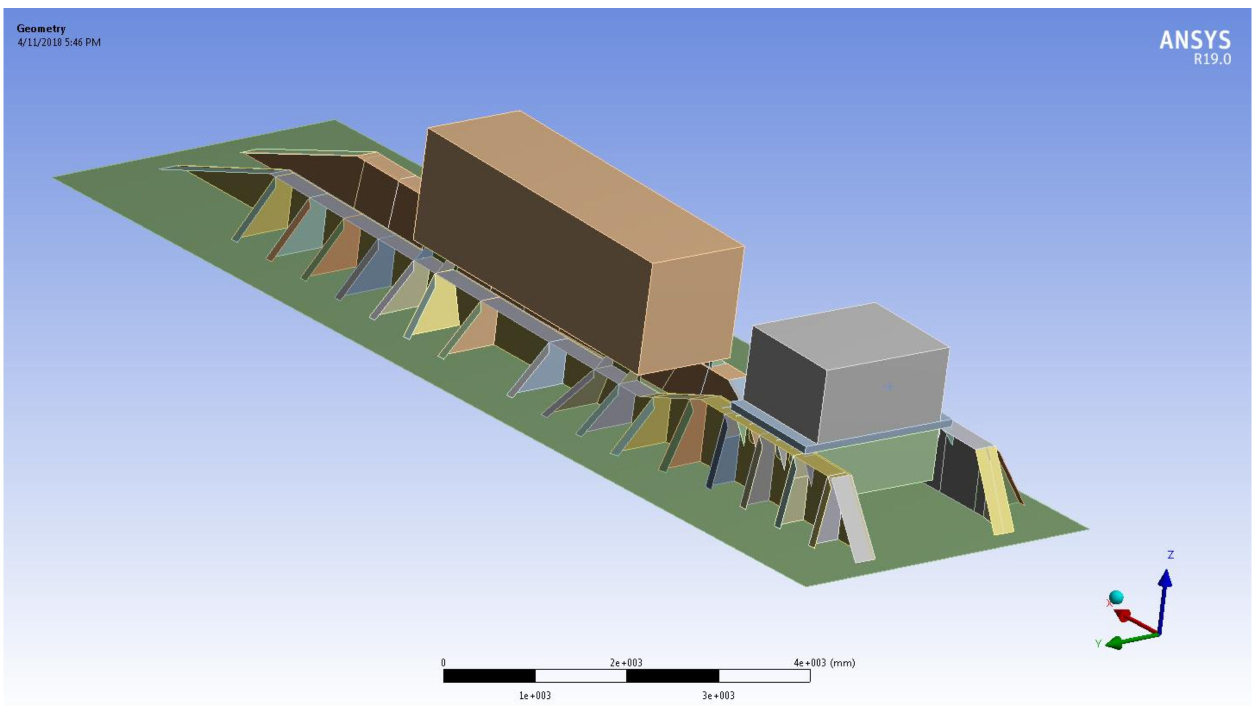


Figure 5/4-6 Tank Top Model Image 2

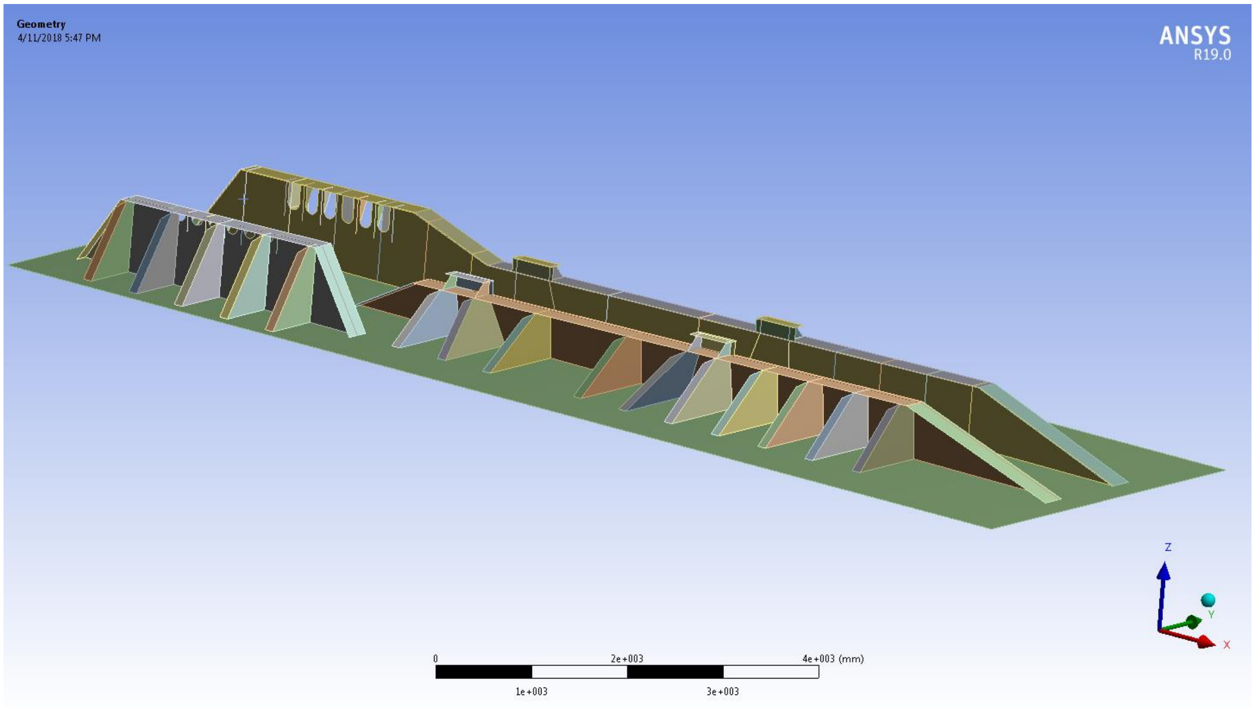


Figure 5/4-7 Tank Top Model Image 3

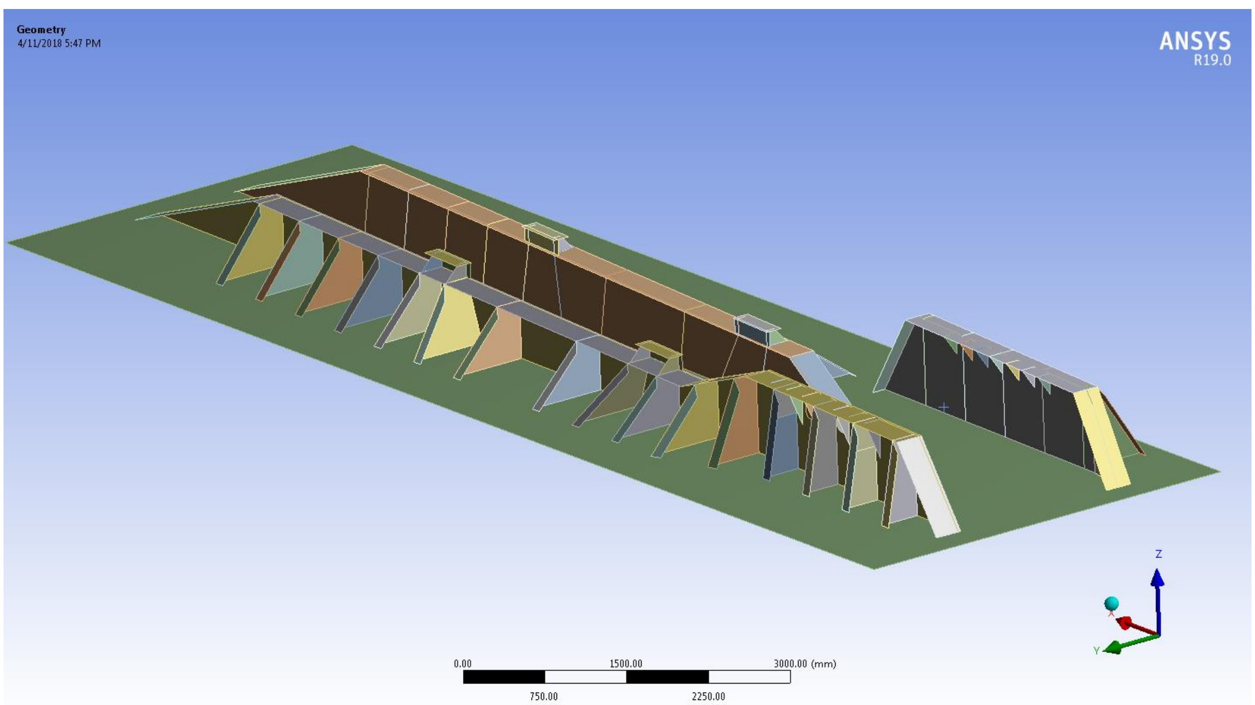


Figure 5/4-8 Tank Top Model Image 4

5/4.8 Appendix B: LOADING AND BOUNDARY CONDITION PLOTS

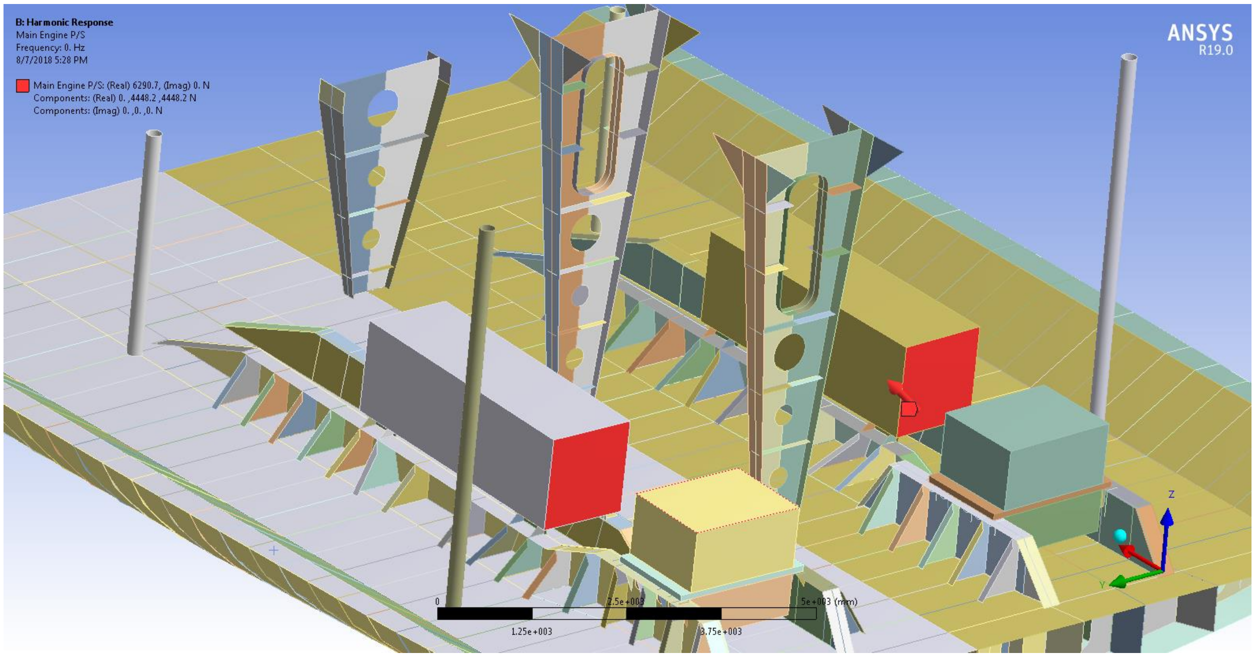


Figure 5/4-9 Main Engine Forcing Loading Condition Image 1

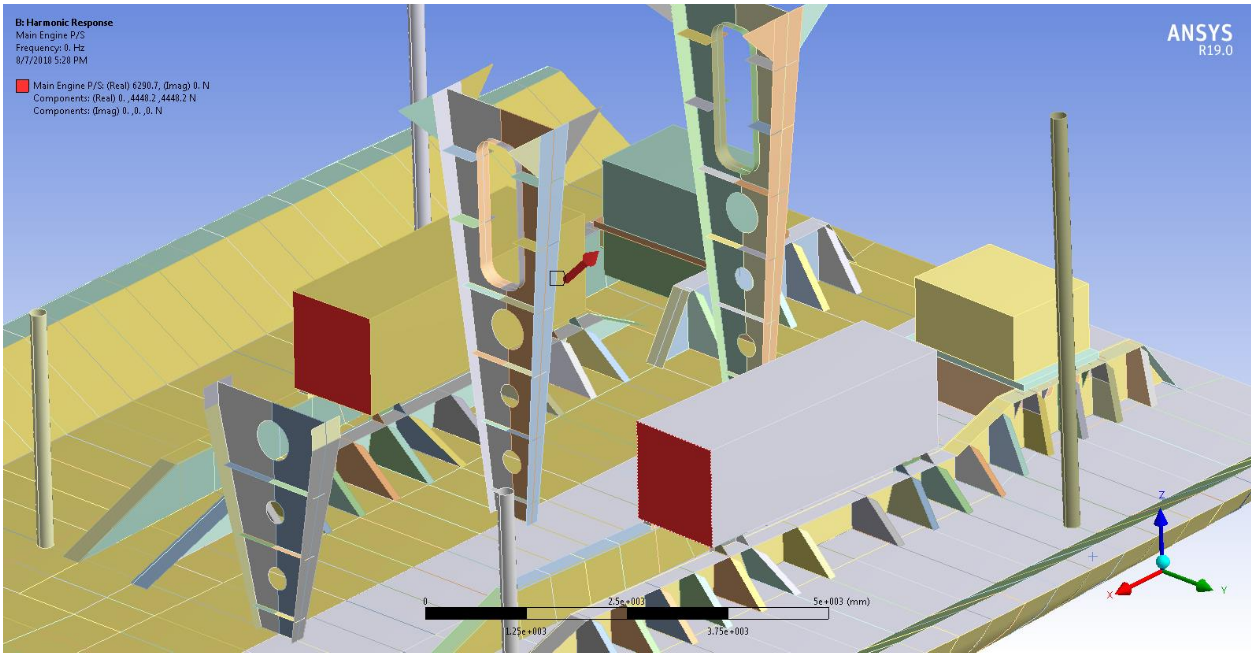


Figure 5/4-10 Main Engine Forcing Loading Condition Image 2

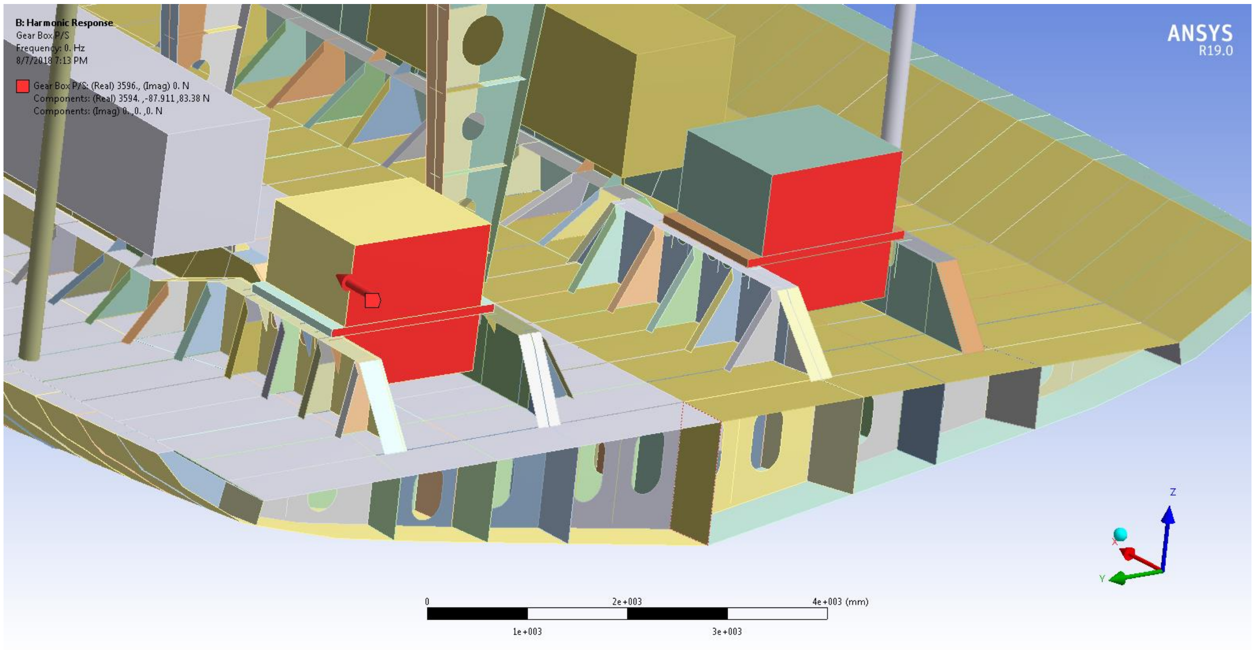


Figure 5/4-11 Gear Box Forcing Loading Condition Image

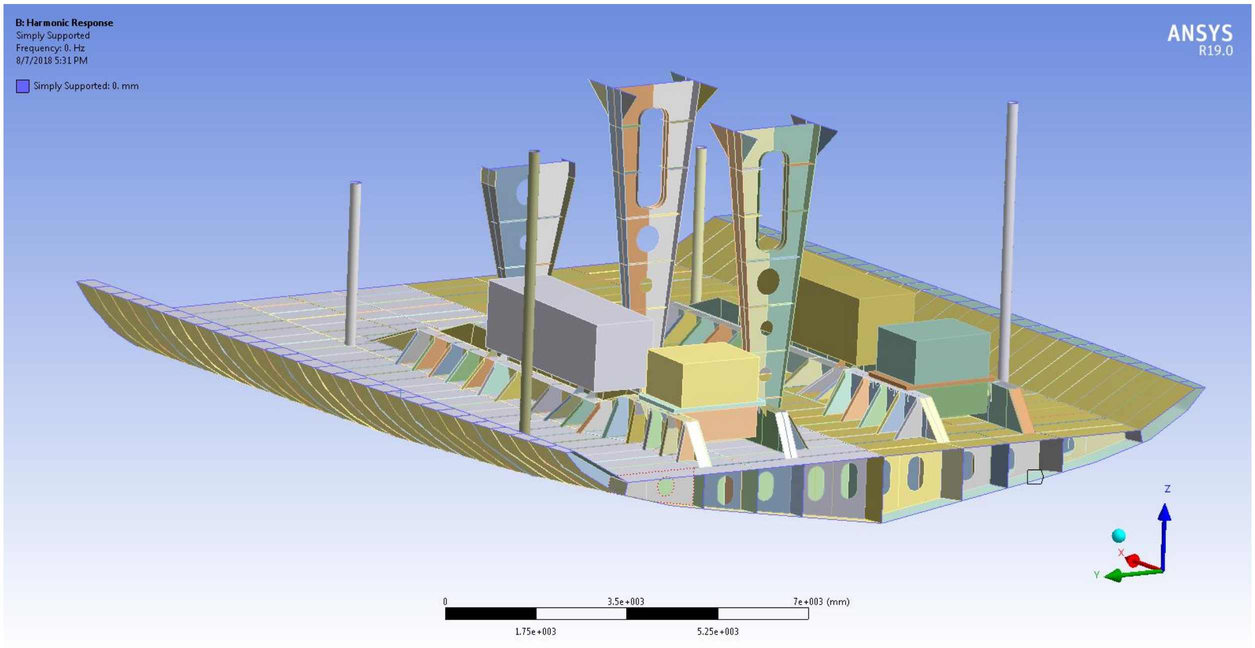


Figure 5/4-12 Boundary Conditions Image

5/4.9 Appendix C: DISPLACEMENT PLOTS

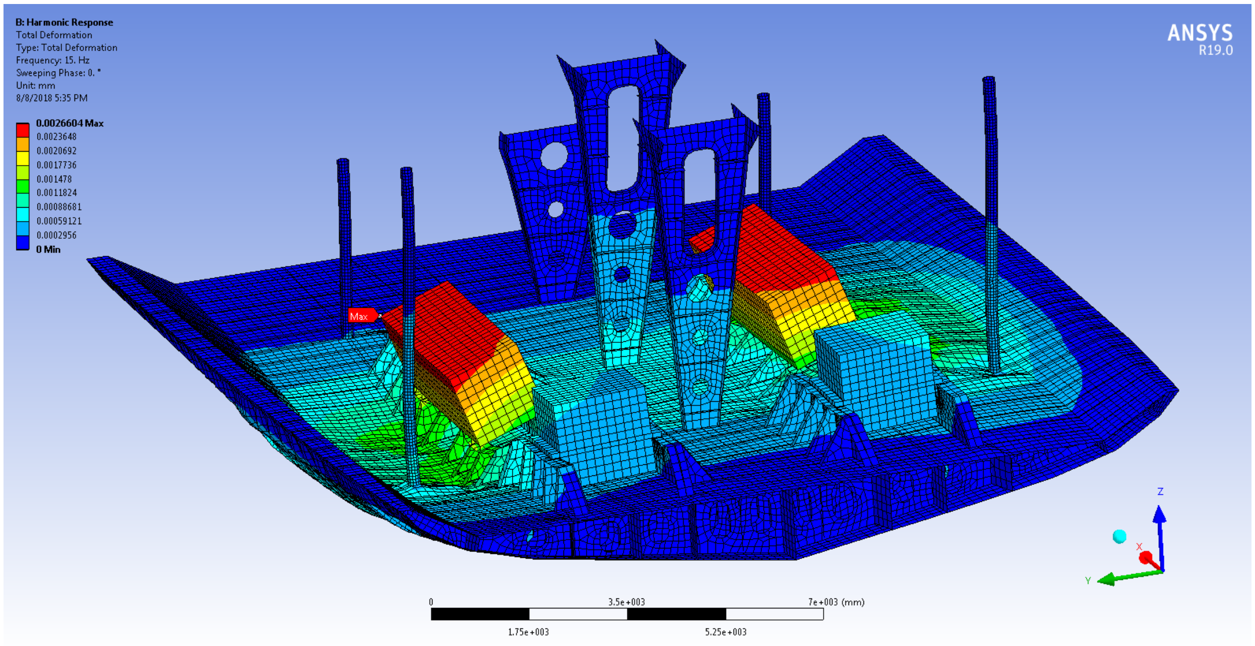


Figure 5/4-13 Main Engine Maximum Displacement at 15 Hz

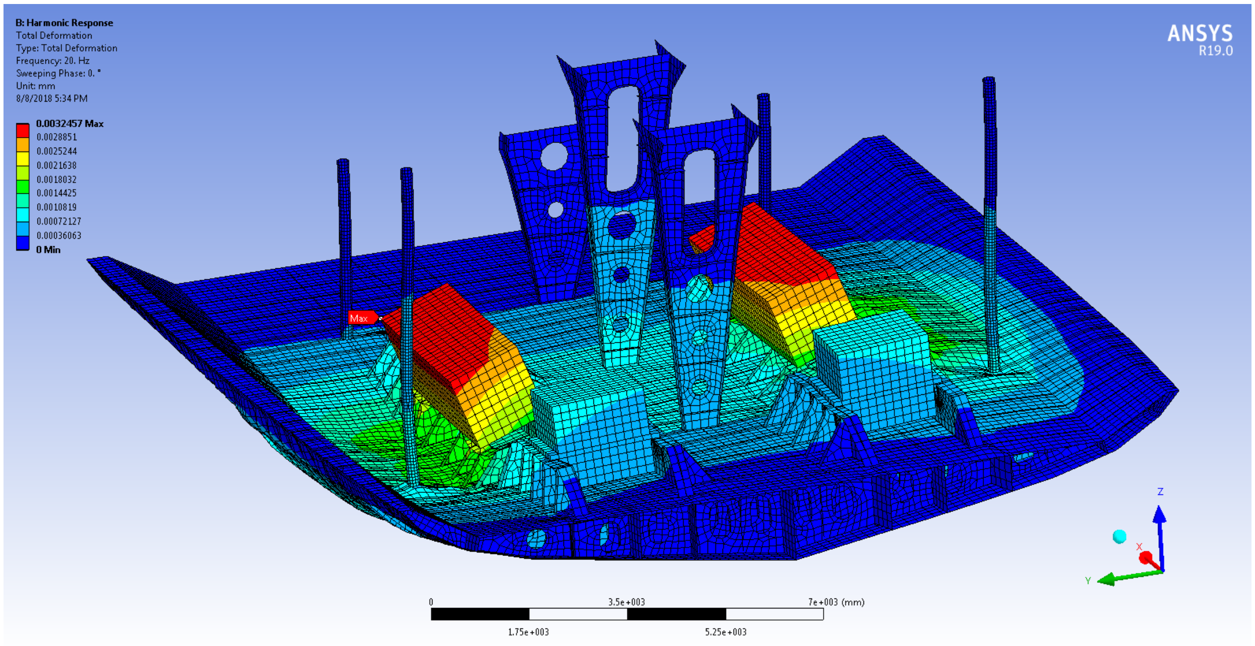


Figure 5/4-14 Main Engine Maximum Displacement at 20 Hz

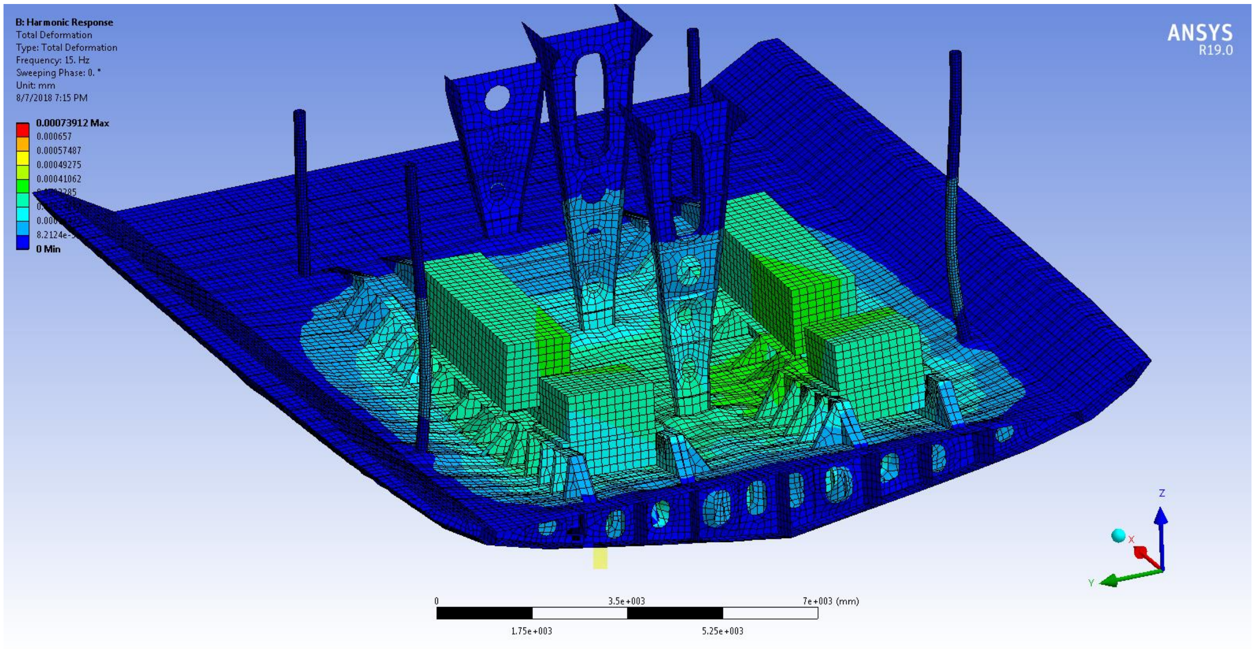


Figure 5/4-15 Gear Box Maximum Displacement at 15 Hz

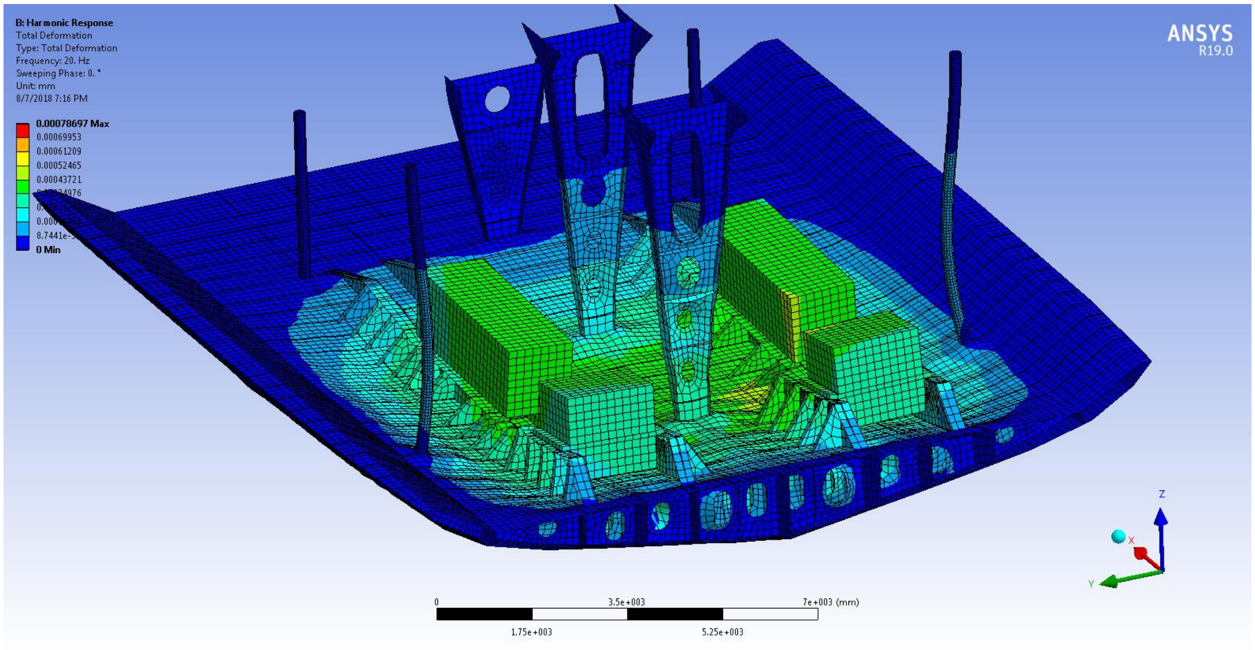


Figure 5/4-16 Gear Box Maximum Displacement at 20 Hz

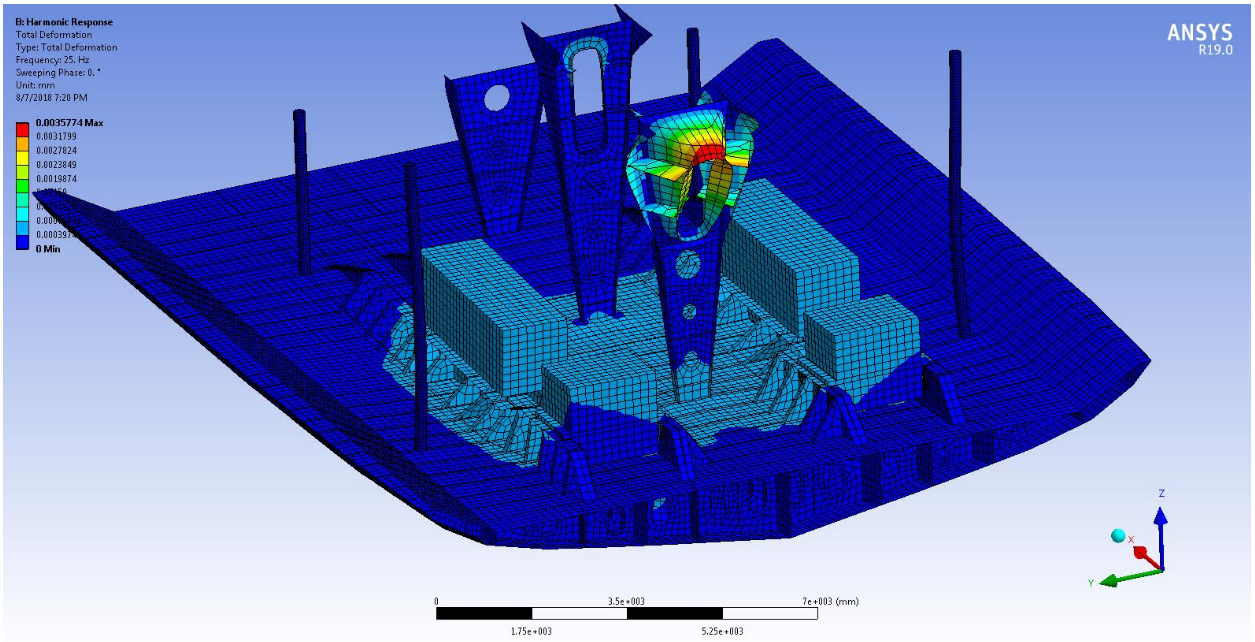


Figure 5/4-17 Gear Box Maximum Displacement at 25 Hz

5/4.10 Appendix D: VELOCITY PLOTS

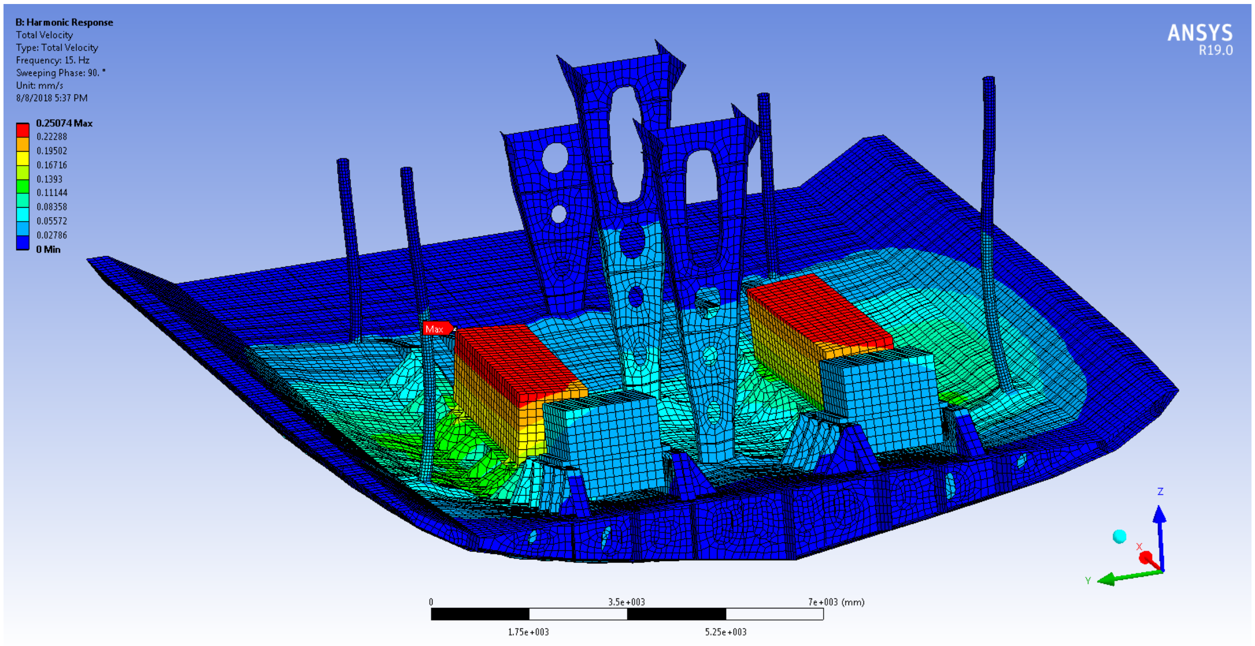


Figure 5/4-18 Main Engine Maximum Velocity at 15 Hz

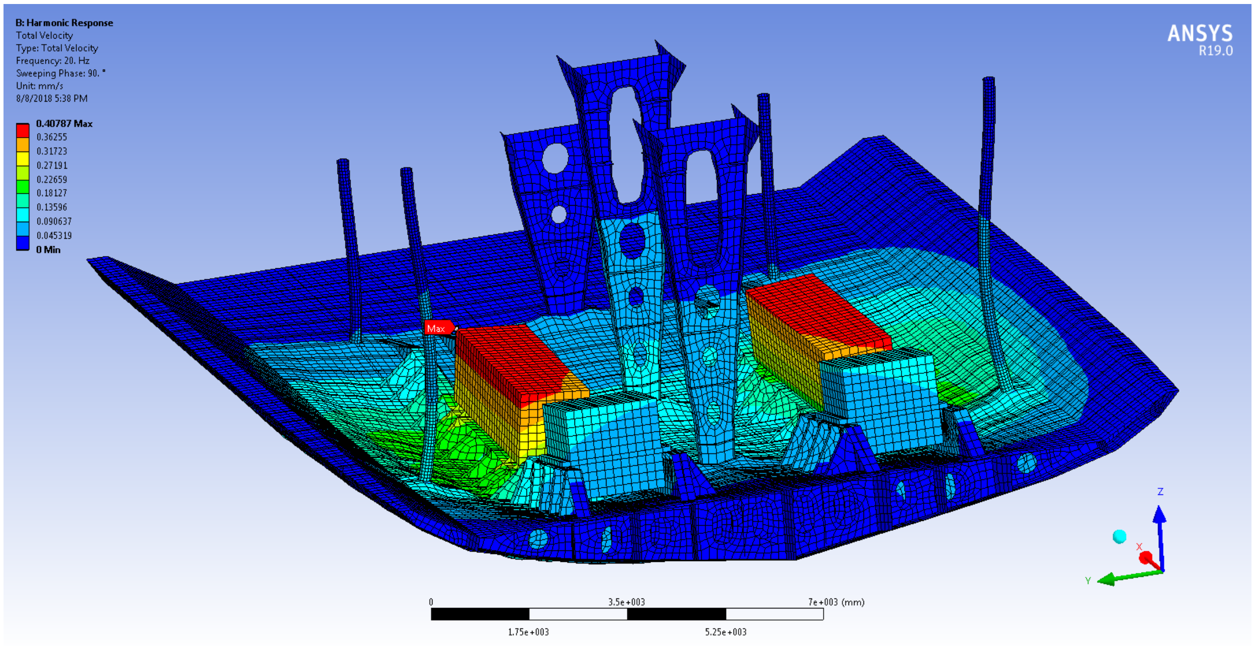


Figure 5/4-19 Main Engine Maximum Velocity at 20 Hz

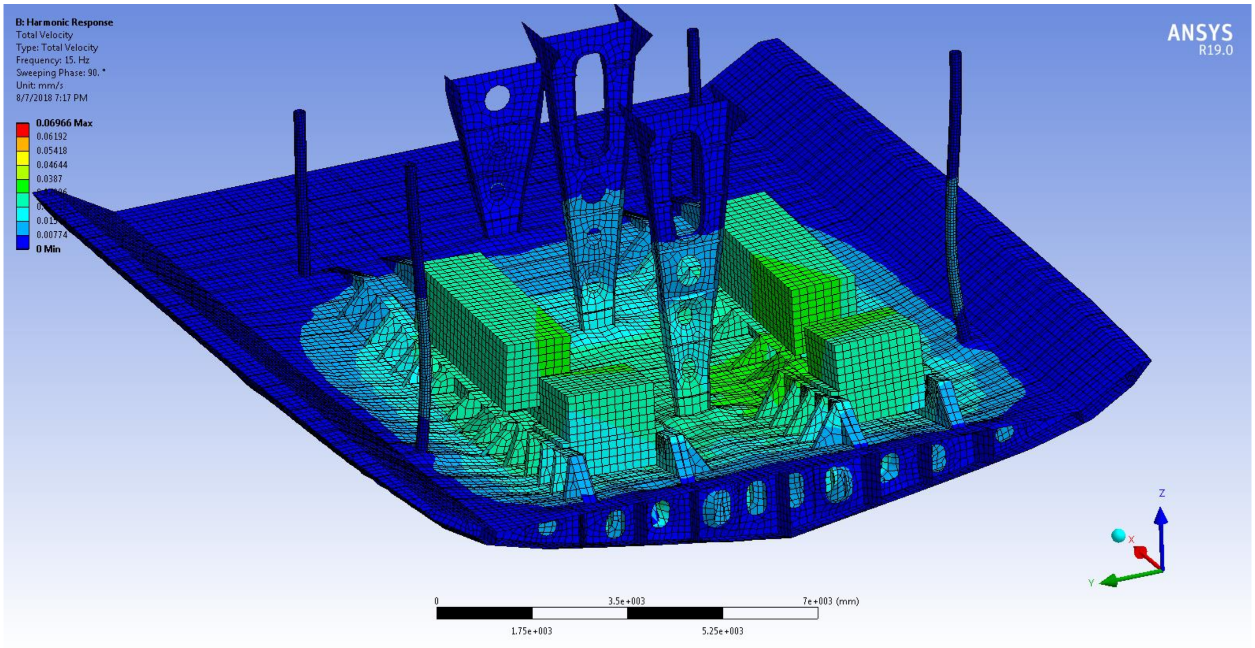


Figure 5/4-20 Gear Box Maximum Velocity at 15 Hz

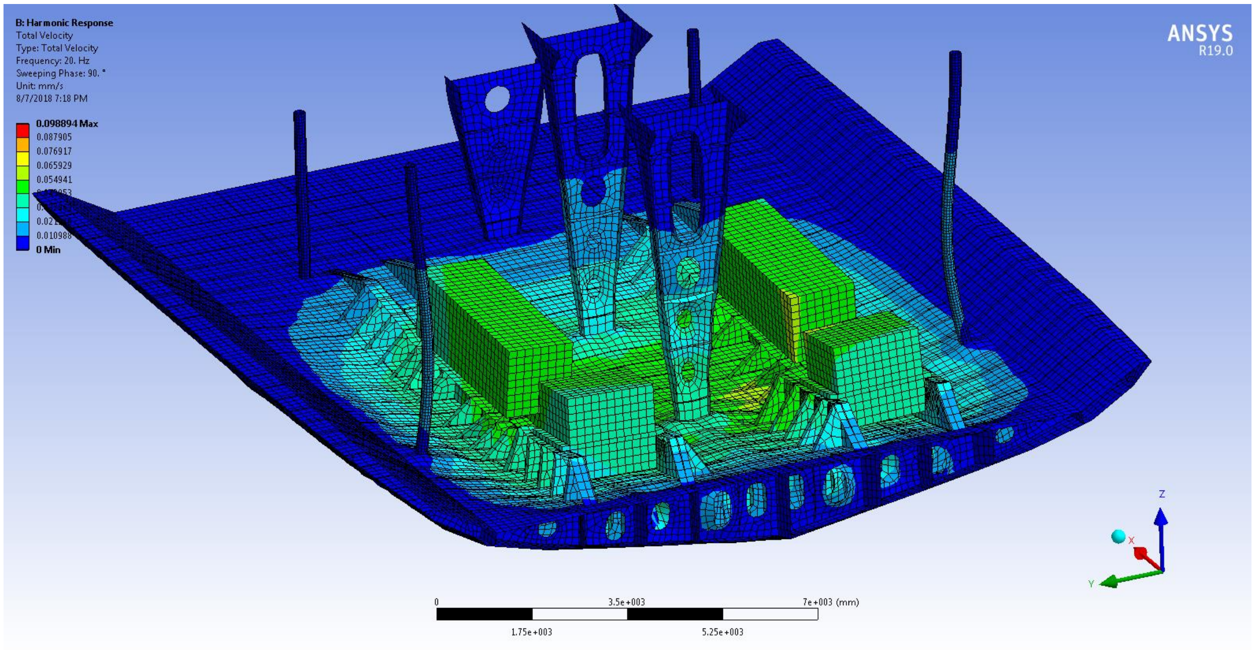


Figure 5/4-21 Gear Box Maximum Velocity at 20 Hz

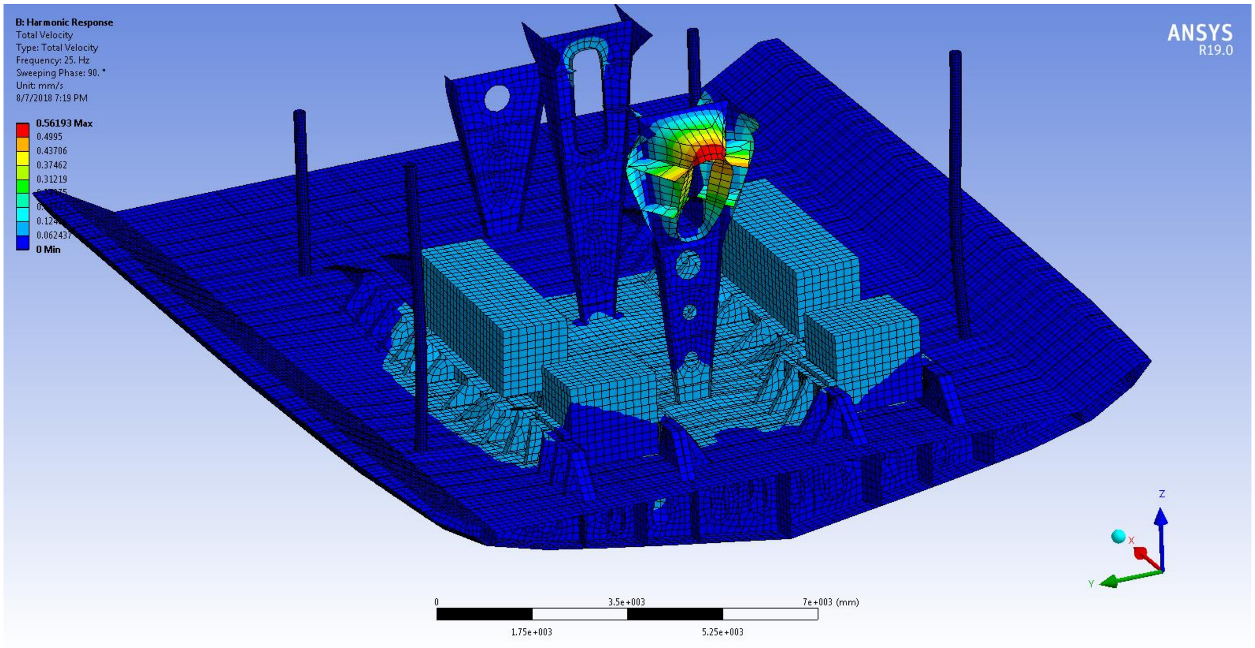


Figure 5/4-22 Gear Box Maximum Velocity at 25 Hz

PART 6
CONCLUSIONS AND RECOMMENDATIONS

There has been much development in the theory and application of FEA to ship structures since the 1996 SSC-387 was published. The relative quantity of references in this updated version versus the original version attests to the level of development. The work presented in this report provides an updated guide to those that are faced with the problem of evaluating the FEA work performed by other parties. As an aid to the evaluation process, a comprehensive, systematic, updated assessment methodology is presented in this report. It is designed to be flexible in terms of the level of skill expected of the evaluator, and in terms of the size and complexity of the FEA that the methodology can be applied to.

The methodology is structured on three levels. The first level is essentially an overview checklist of features of an FEA that needs to be evaluated. A more detailed checklist, based on the first level, is presented in the second level of the methodology. The third level provides guidance in narrative and illustrative form and is structured to match the first and second level checklists. Further guidance is provided through a series of illustrative examples which show the influence of varying finite element modeling practice on FEA results. These are intended to help the evaluator in assessing the levels of accuracy that might be attained in the FEA that is being evaluated.

The benchmarking information from the SSC-387 guide has been retained here, but it is typically no longer necessary for well-established FEA software that has been used repeatedly already on ship structure type problems. However, new FEA software or FEA software that has not been used on ship structure type problems still requires benchmarking. The benchmark problems of the type presented in this report can be regarded as a further level of qualification for such software.

These benchmark problems are intended to test the ability of software to provide accurate solutions for structural assemblies typical of ship structures. Unlike the typical verification problem used by software houses, benchmark problems consider non-ideal configurations, multiple element types, several load cases, etc.

FEA codes are large and complex and hence can never be guaranteed to be free of errors. However, it is suggested that FEA software that has been thoroughly tested by the vendor at the verification example level, will, by successfully yielding solutions for the benchmark problems, provide another level of assurance that the software is fit for performing ship structure FEA.

Several recommendations are presented below for consideration:

1. The scope could be broadened to include composite materials and the application of the many varied types of contacts available in modern FEA software.

2. The benchmarks presented in this report might be considered as a starting point for building a library of benchmark problems that include advanced analysis problem benchmarks. These problems could also include high-quality and well-documented experiments on ship structure assemblies.

3. There may be a benefit in a new study comparing the tradeoffs of using solid elements versus using shell elements. There has been a significant advancement in computing power, and FEA software capabilities since the marine industry generally adopted the use of shell elements for ship structure problems. The automotive and aerospace industry is tending to use more solid element models because less analyst intervention is required. Such a new study would determine using current technology which types of ship structure FEA problems are more efficiently done using solid elements versus shell elements.

Appendix A
Evaluation Forms for Assessment of Models and Results

	Process	
No.	Level-1	Level-2
1	○ Preliminary Checks (Backgrounds and objectives)	○ Documentation Requirements
		○ Job Specification Requirements
		○ Finite Element Analysis Software Requirements
		○ Personnel Qualification Requirements
2	○ Engineering Model Checks (Inputs and assumptions)	○ Analysis Type
		○ Analysis Geometry
		○ Material Properties
		○ Loads and Boundary Conditions
		○ Impact and Plasticity
		○ Fatigue and Fracture Analysis
		○ Whole Ship
		○ Frequency Response Vibration Analysis
3	○ Finite element model checks (Pre-processing)	○ CAD Importing
		○ Element Types
		○ Mesh Design
		○ Substructure and Submodelling
		○ Loads and Boundary Conditions
		○ Analysis Controls and Solution Options
4	○ Finite element results checks (Post-processing)	○ General Solution Checks
		○ Post Processing Methods
		○ Displacement Results
		○ Force Results
		○ Stress Results
		○ Strain Results
		○ Energy Results
		○ Fracture Results
		○ Fatigue Results
		○ Vibration Results
5	○ Conclusions Checks (Evaluations and Recommendations)	○ FEA results and Acceptance Criteria
		○ Load Assessment
		○ Strength / Resistance Assessment
		○ Accuracy Assessment
		○ Overall Assessment

Figure A/0-1 Overall Evaluation Methodology Checklist

FINITE ELEMENT ANALYSIS ASSESSMENT		I PRELIMINARY CHECKS
Project No.	I Project Title :	
Company Name:		IDate:
Analyst:	I Checker:	

A/1.1 Documentation Requirements

In order to perform a comprehensive assessment of an FEA, certain essential information must be provided in the documentation submitted.

Finite Element Analysis Assessment Check	Refer to Guideline Section	Result	Comments
1.1.1 Has the following information been provided in the FEA documentation?	3/1.1		
a. Scope and objective of analysis.			
b. Timeline for delivery of analysis			
c. Reference codes, manuals, and/or standards required.			
d. Physical problem documentation references			
e. FEA software used.			
f. Contractor qualifications.			
g. Analysis type(s).			
h. Description of physical problem.			
i. Description of engineering model.			
j. Description of the estimated range of stress and displacements expected from the FEA analysis.			
k. Description of the FEA model.			
l. Plots of full FEA model and local details.			
m. System of units.			
n. Coordinate axis systems.			
o. Material properties.			
p. Stiffness and mass properties.			
q. Loads and boundary conditions.			
r. Element type(s).			
s. Meshing idealizations/ assumptions/ representations/ simplifications.			
t. Meshing criteria for 2D and 3D elements.			
u. FE loads and boundary conditions.			
v. Solution options and procedures.			
w. Results.			

x. Comparison of results with acceptance criteria.		
y. Accuracy assessment.		
z. Conclusions and recommendations for amendments.		

Based on the above checks, answer Question 1.1, and enter the result in Figure A/0-1.	**Result**
1.1 Is the level of documentation sufficient to perform an assessment of the FEA?	
Comments	

A/1.2 Job Specification Requirements

Perform these checks to ensure that the analysis addresses the objectives, scopes, requirements, and intent of the job specification (e.g., the contract document, work specification, statement of work, etc.).

Finite Element Assessment Check	Refer To Guideline Section	Result	Comments
1.2.1 Is the job specification identified and referenced in the analysis documentation?	3/1.2		
1.2.2 Are the objectives and scopes of the analysis clearly stated, and are they consistent with those of the job specification?	3/1.2		
1.2.3 Are the analysis requirements clearly stated, and are they consistent with those of the job specification?	3/1.2		
1.2.4 If certain requirements of the job specification have not been addressed (such as certain load cases), has adequate justification been given?	3/1.2		
1.2.5 Are the design / acceptance criteria clearly stated, and are they consistent with those of the job specification?	3/1.2		
1.2.6 Is there reasonable justification for using FEA for this problem?	3/1.2		
1.2.7 Has the advantage been taken of any previous experimental, analytical, or numerical works that are relevant to this problem? If so, are these previous works cited in the subject analysis report?	3/1.2		

Based on the above checks, answer Question 1.2, and enter the result in Figure A/0-1.	**Result**
1.2 Does the analysis address the job specification requirements?	
Comments	

A/1.3 Finite Element Analysis Software Requirements

The FEA software should meet certain minimum standards to be considered acceptable for ship structural analysis applications.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
1.3.1 Is the FEA software appropriate for the application or on a list of approved programs for ship structural analysis applications in the organization?	3/1.3		
If the answer to Check 1.3.1 is "Y," you may skip Checks 1.3.2 and 1.3.3.			
1.3.2 Are the capabilities and limitations of the FEA software used to perform the required analysis stated in the analysis documentation?	3/1.3		
1.3.3 Is evidence of this capability documented and available for review (e.g., verification manual, results of ship structure FEA benchmark tests, previously approved FEA of similar problems)?	3/1.3		
1.3.4 Does the vendor of the FEA software have a quality system to ensure that appropriate standards are maintained in software development and maintenance.	3/1.3		
1.3.5 Other concerns: • Does the FEA solver have both single and double precisions? • Does the FEA solver include implicit and/or explicit solvers? • Does the FEA solver have a HPC (high performance computing) version? Parallel processing: SMP (Shared memory parallel) version? DMP (distributed memory parallel) version? • Does the GPU (graphics processing units) hardware precision match the solver's precision? Note that the double-precision GPU is recommended for those CPU and GPU based simulations • Does RAM have an error checking and correcting (ECC) function?	3/1.3		

Based on the above checks, answer Question 1.3, and enter the result in Figure A/0-1.	**Result**
1.3 Is the FEA software qualified to perform the required analysis?	
Comments	

NOTE: Part 4 of this report presents benchmark problems for the purpose of assessing the quality and suitability of FEA software for performing ship structural analysis. On its own, the successful performance of the candidate FEA software in exercising the benchmark problems is not sufficient evidence of the quality and suitability of the software. The assessor should, in addition, be able to answer the other questions in the table above affirmatively.

A/1.4 Personnel Qualification Requirements

The personnel should possess certain minimum qualifications for performing ship structure FEA. In addition, the personnel should adhere to a Quality Assurance (QA) system to ensure that proper management, administrative, and checking procedures have been applied in the analysis.

Finite Element Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
1.4.1 Do the personnel have adequate academic training and experience qualifications to perform finite element analysis?	3/1.4		
1.4.2 Do the personnel have adequate engineering experience qualifications for performing ship structural design or analysis?	3/1.4		
1.4.3 Do the personnel have adequate professional certification qualifications?	3/1.4		
1.4.4 Does the personnel have a working system of Quality Assurance (QA) procedures and checks that are satisfactory for the requirement?	3/1.4		
1.4.5 Do the personnel have adequate experience with the FEA software used for the analysis?	3/1.4		

Based on the above checks, answer Question 1.4, and enter the result in Figure A/0-1.	**Result**
1.4 Is the personnel adequately qualified for performing ship structure FEA?	
Comments	

FINITE ELEMENT ANALYSIS ASSESSMENT	ENGINEERING MODEL CHECKS
Project No.	I Project Title :
Company Name:	IDate:
Analyst:	I Checker:

A/2 Engineering Model Checks

A/2.1 Analysis Type

Perform these checks to ensure that the assumptions used in developing the engineering model or idealization of the physical problem are adequate

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
2.1.1 Does the engineering model employ enough dimensions and freedoms to describe the structural behavior (e.g., 1-D, 2-D, 3-D, or a combination of these)?	3/2.1		
2.1.2 Does the engineering model address the appropriate scale of response for the problem (e.g., global, intermediate, or local response)?	3/2.1		
2.1.3 Is the type of analysis appropriate for the type of response and loading of interest (e.g., linear, static, modal vibration, frequency response vibration, buckling, nonlinear, quasi-static, dynamic, crash, impact, fracture, fatigue, crack, creep, damage, residual fields, pre-loaded, DoE (design of experiments), optimization – topology / tomography / etc.)?	3/2.1		
2.1.4 Does the engineering model address all the required results parameters (e.g., critical stress / strain / displacement / force values, frequency, permanent deformation, penetration depth, buckling load, contact force, strain energy, deformation energy, crack driving force(s) – J / CTOD / CTOA, stress intensity factor – K_I / K_{II} / K_{III}, weight, stiffness, strength, velocity, acceleration, etc.)?	3/2.1		
2.1.5 Are all assumptions affecting the choice of engineering model and analysis type justified (watch for unusual assumptions)?	3/2.1		
2.1.6 Is the level of detail, accuracy, or conservatism of the engineering model appropriate for the criticality of the analysis and type of problem?	3/2.1		
2.1.7 Does the analysis employ a consistent set of units?	3/2.1		
2.1.8 Does the analysis employ a consistent global coordinate axis system, and if applicable, a consistent local coordinate axis system?	3/2.1		

Based on the above checks, answer Question 2.1, and enter the result in Figure A/0-1. **Result**

2.1 Are the assumptions of the type of analysis and engineering model acceptable?	
Comments	

A/2.2 Analysis Geometry

A/2.2.1 Analysis Geometry General

Perform the following checks to ensure that correct procedures have been followed for defining the geometric properties of the structure.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
2.2.1.1 Does the extent of the model geometry capture the main structural actions, load paths, vibrations (if applicable), failure mode(s), critical value(s), and response parameters of interest?	3/2.2		
2.2.1.2 If the analysis is vibration analysis, is the model boundary conditions far enough away from the vibration response zone(s), e.g. potential maximum displacement/force location, that the model is not overly stiff?	3/2.2		
2.2.1.3 Are correct assumptions used to reduce the extent of model geometry (e.g., symmetry, boundary conditions at changes in stiffness)?	3/2.2		
2.2.1.4 Will the unmodelled structure (i.e., outside the boundaries of the engineering model) have an acceptably small influence on the results?	3/2.2		
2.2.1.5 Are there the impacts of geometric simplifications (such as omitting local details, cut-outs, holes, stiffeners, openings, etc.) on the accuracy of the analysis acceptable?	3/2.2		
2.2.1.6 For local detail models, have the aims of Saint-Venant's principle been satisfied?	3/2.2		
2.2.1.7 Do the dimensions defining the engineering model geometry correspond to the dimensions of the structure adequately?	3/2.2		
2.2.1.8 For buckling analysis, does the geometry adequately account for discontinuities and imperfections affecting buckling capacity?	3/2.2		
2.2.1.9 Has corrosion been considered over the applicable design life when selecting the material thickness? Is this decision in accordance with any selected evaluation criteria guidance on corrosion?	3/2.2		
2.2.1.10 Is the criteria for the use of material thickness in the model (i.e. net scantlings or gross scantlings) provided?	3/2.2		

Based on the above checks, answer Question 2.2.1 and enter the result in Figure A/0-1. **Result**

2.2.1 Are the geometry assumptions in the engineering model acceptable?	

Comments

A/2.2.2 Mass and Added Mass

Perform the following checks to ensure that correct procedures have been followed for defining the mass and added mass properties of the structure.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
2.2.2.1 Are all components that have a significant effect on the mass of the structure accounted for in the engineering model?	3/2.2.2		
2.2.2.2 Has the material density properties been appropriately defined (see also Check 2.3.3)?	3/2.2.2		
2.2.2.3 Has the added mass of entrained water been adequately accounted for with structure partially or totally submerged?	3/2.2.2		
2.2.2.4 Are lumped mass representations of structural mass and / or equipment correctly consolidated and located?	3/2.2.2		
2.2.2.5 If rotational inertia is expected to be important, are mass moments of inertia properties correctly defined for masses?	3/2.2.2		
2.2.2.6 If element erosion is to be considered, are the criteria for mass reduction provided?	3/2.2.2		
2.2.2.7 If the nodal mass element approach is adopted, does its location match the physical observation without creating additional loads?	3/2.2.2		
2.2.2.8 Are the values of the mass properties considered supported by acceptable calculations and / or references?	3/2.2.2		
2.2.2.9 If relevant, has fluid-structure interaction been accounted for? Has the added mass been included in the model?	3/2.2.2		
2.2.2.10 Are the units for the stiffness and mass properties consistent with the system of units for other parts of the analysis?	3/2.2.2		

Based on the above checks answer, Question 2.2.2, and enter the result in Figure A/0-1.	**Result**
2.2.2 Are the assumptions and data defining the mass and added mass properties acceptable?	
Comments	

A/2.2.3 Shock Analysis Mass Modeling Reduction

In shock analyses, it is sometimes desirable or necessary to reduce the size of the problem by reducing the number of dynamic degrees of freedom (DOF). Perform these checks to ensure that the correct procedures have been followed for selecting dynamic degrees of freedom.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
2.2.3.1 Are dynamic DOF defined along enough directions to model the anticipated dynamic response behavior of the structure?	3/2.2.3		
2.2.3.2 Are the number of dynamic DOF at least three times the highest mode required (e.g., if 30 modes are required, at least 90 DOF are needed)?	3/2.2.3		
2.2.3.3 Are the dynamic DOF located where the highest modal displacements are anticipated?	3/2.2.3		
2.2.3.4 Are the dynamic DOF located where the highest mass-to-stiffness ratios occur for the structure?	3/2.2.3		
2.2.3.5 Are dynamic DOF located at points where forces are to be applied for dynamic response analyses?	3/2.2.3		
2.2.3.6 Are the number of dynamic DOF such that at least 90% of the structural mass is accounted for in the reduced model in each direction?	3/2.2.3		

Based on the above checks, answer Question 2.2.3 and enter the result in Figure A/0-1.	**Result**
2.2.3 Are the assumptions and data defining shock analysis mass modeling acceptable?	
Comments	

A/2.3 Material Properties

Perform the following checks to ensure that correct procedures have been followed for defining the material properties of the structure.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
2.3.1 Are all materials of structure crucial to the problem included in the engineering model? (Note: it is preferable to number each material separately as well as associated components in the assembly of FE models)	3/2.3		
2.3.2 Are the assumed behaviors valid for each material (eg. linear elastic, nonlinear elastic, elasto-plastic, isotropic, anisotropic, orthotropic, hardening, fatigue, fracture, creep, failure, damage, pre-loaded, relaxation, rate- or temperature dependency, composites)?	3/2.3		
2.3.3 Are the required material parameters defined for the type of analysis? For example: • Density • Thermal conductivity (if applicable) • Electric resist (if applicable) • Young's modulus (moduli, if temperature dependent) • Poisson's ratio • Stress-strain curves or models (if temperature and/or strain rate dependent) • Failure criteria (e.g., element erosion, accumulated strain) • Damage criteria (e.g., Weibull stress, Lematre model, and Gurson model)	3/2.3		
2.3.4 For cellular type materials (i.e., metallic and non-metallics), are the required material parameters defined for the type of analysis? For example: • Matrix materials All listed material parameters listed in 2.3.3 • Composite materials All listed material parameters listed in 2.3.3 • Cellular material (e.g., honeycomb, open- or close-foams, wood, sandwich structure, glass- or carbon-fiber-reinforced composite), as well as associated parameters, defined the material (e.g., density ratio, strength ratio, representative cells, interface strength)	3/2.3		
2.3.5 Are orthotropic and / or layered properties defined correctly for non-isotropic materials such as metallic foam?	3/2.3		
2.3.6 Are orthotropic properties defined correctly where material orthotropy is used to simulate structural performance?	3/2.3		

2.3.7 Are the correct material models are adopted for non-isotropic materials?	3/2.3		
2.3.8 If strain rate effects are expected to be significant for this problem, are they accounted for in the material property data?	3/2.3		
2.3.9 If temperature effects are expected to be significant for this problem, are they accounted for in the material property data?	3/2.3		
2.3.10 If electrical and/or magnetic effects are expected to be significant for this problem, are they accounted for in the material property data?	3/2.3		
2.3.11 Are the values of the material property data traceable to an acceptable source or reference (e.g., handbook, mill certificate, coupon tests)?	3/2.3		
2.3.12 Are the units for the material property data consistent with the system of units adopted for other parts of the analysis?	3/2.3		
2.3.13 For the analysis requiring plasticity correction (e.g., low cycle and high cycle fatigue, or yielding state analysis), is the Neuber correction adopted and appropriately defined?	3/2.3		
2.3.14 If the user-defined material model is adopted in FE simulation, has the material model been verified against testing data as well as validated in terms of reliabilities and accuracies? If possible, the user-defined material should be enclosed in the submitted report (e.g., source code or encrypted format).	3/2.3		

Based on the above checks, answer Question 2.3, and enter the result in Figure A/0-1.	**Result**
2.3 Are the assumptions and data defining the material properties acceptable?	
Comments	

A/2.4 Loads and Boundary Conditions

Perform the following checks to ensure that correct procedures have been followed for defining the loads and boundary conditions of the problem.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
2.4.1 Are all required loadings/load cases accounted for and has sufficient justification been provided for omitting certain load effects?	3/2.3.5		
2.4.2 Are the loading assumptions stated clearly, and are they justified?	3/2.3.5		
2.4.3 Has an assessment been made of the accuracy and / or conservatism of the loads?	3/2.3.5		
2.4.4 Are the procedures for combining loads/load cases (e.g., superposition) described adequately, and are they justified?	3/2.3.5		
2.4.5 Have the boundary conditions assumptions been stated clearly, and are they justified?	3/2.3.5		
2.4.6 Do the boundary conditions adequately reflect the anticipated structural behavior?	3/2.3.5		
2.4.7 Has an assessment been made of the accuracy of the boundary conditions, and if they provide a lower or upper bound solution?	3/2.3.5		
2.4.8 Is the information on loads and boundary conditions provided? For example: • Loading history and loading sequence (force-based or displacement-based or other-types) • Whether inertial relief is needed	3/2.3.5		
2.4.9 If nonlinear loads or boundary conditions (e.g., follower forces or contacts) are considered, is the loading information converted adequately to represent the physical problem.	3/2.3.5		
2.4.10 If the analysis includes pre-tension (e.g., bolt preload tension) and/or pre-loaded (e.g., residual stress and strain), are the input values as well as the corresponding directions adequate and sufficiently representing the physical observations.	3/2.3.5		
2.4.11 Are the units for the data of loads and boundary conditions (e.g., forced displacement) consistent with the system of units adopted for other parts of the analysis?	3/2.3.5		

2.4.12 If the engineering problem involves contact(s), which type(s) of contact(s) should be adopted? For example: • self-contact • point-to-point contact • point-to-surface contact • surface-to-surface contact	3/2.3.5		
2.4.13 If required, are contact model(s) defined and applied appropriately? Are the related parameters provided and documented? Note that for the frictional contact, the dynamic and static frictional coefficients should be distinguished.	3/2.3.5		
2.4.14 If required, have the load and boundary conditions as well as the mesh designing of the contact point(s) and/or surface(s) been revised to be compatible with the adopted contact type(s) and model(s)?	3/2.3.5		
2.4.15 If required, have the normal direction(s) and slave and/or master been reviewed for the contact pair(s)?	3/2.3.5		
2.4.16 If a moving rigid barrier(s) is/are used, is the coordinate defining loads or boundary conditions consistent with the input data, e.g., local or global coordinate?	3/2.3.5		

Based on the above checks, answer Question 2.4, and enter the result in Figure A/0-1.	**Result**
2.4 Are the assumptions and data defining loads and boundary conditions reasonable?	
Comments	

A/2.5 Impact and Plasticity

Perform the following checks to ensure that correct procedures have been followed for an impact and plasticity problem.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
2.5.1 Is the study an impact analysis?	3/2.5		
2.5.2 Is the study a static or quasi-static analysis involving plastic deformation?	3/2.3		
2.5.3 For the impact analysis, do the material properties depend on the strain rate?	3/2.3.3		
2.5.4 Is it clear which level of impact analysis the study focuses on? • Local cross-section • Component or sub-structure • Total system	3/2.5		

2.5.5 Does the force- or displacement-controlled loading input provided? For example,
• Quasi-static analysis
  ○ Maximum displacement (displacement control)
  ○ Maximum force (force control)
• Impact analysis
  ○ Velocity and time curve
  ○ Displacement and time curve
	3/3.5		

2.5.6 Is the geometric imperfection considered in the model?	3/3.5		
2.5.7 For a perfect geometric shape, is the deformation trigger considered in the model?	3/3.5		

Based on the above checks, answer Question 2.5, and enter the result in Figure A/0-1.	**Result**
2.5 Are the assumptions and data for this impact and plasticity analysis acceptable?	
Comments	

A/2.6 Fatigue and Fracture Analysis

Perform the following checks to ensure that correct procedures have been followed for fatigue and fracture analysis problems.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
2.6.1 Is the study a fatigue analysis? For example: • Stress life fatigue analysis • Strain life fatigue analysis • Fracture mechanics-based fatigue analysis • Vibration fatigue analysis	3/2.6.1		

2.6.2 Is the study a fracture analysis? For example:
• Linear elastic fracture mechanics
• Elastic-plastic fracture mechanics
  ○ Small-scale yielding
  ○ Large-scale yielding
• Elastic-plastic fracture mechanics with multiple constraint parameters
• Damage included (crack propagation)
	3/2.6.3		

2.6.3 Are the material properties provided, including conditions such as the ambient temperature or corrosive environment? • General material properties • Fatigue analysis • Stress-cycle (SN) or strain-cycle (EN) curve • Fracture analysis	3/2.3		
2.6.4 Are the loading histories provided?	3/2.6.2.1		
2.6.5 Are the geometric detail profiles provided for the fatigue analysis?	3/3.3.5.3		
2.6.6 Are the geometric detail profiles provided for the fracture analysis?	3/3.3.5.2		
2.6.7 Is the crack explicitly included in the model?	3/2.6.2.1		
2.6.8 Is the crack extension considered in the fatigue life estimation?	3/2.6.2.1		
2.6.9 Is the initial crack or flaw size correct or calibrated using NDT (nondestructive testing) tools?	3/2.6.2.1		

2.6.10 If the submodel technique is used for the fatigue analysis, does the constraint equation and mesh refinement defined appropriately?	3/2.6.2.1		
2.6.11 For the fracture mechanics-based fatigue analysis, is the accumulative damage model used, i.e., Paris's law?	3/2.6.2.1		
2.6.12 For low cycle fatigue analysis, is the Neuber correction method used?	3/2.3.5		
2.6.13 For a fracture analysis, is it clear which parameter(s) will be considered? For example: • Stress intensity factor – K • J-integral – J • Crack-tip opening displacement – CTOD	3/2.6.4.2		

Based on the above checks, answer Question 2.6, and enter the result in Figure A/0-1.	**Result**
2.6 Are the assumptions and data for this fatigue and fracture analysis acceptable?	
Comments	

A/2.7 Whole Ship

Perform these checks to ensure that the correct procedures have been followed for a whole ship analysis.

If the analysis is not a whole ship analysis, you may proceed directly to Part A/2.8.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
2.7.1 Is the scope of the whole-ship analysis is clear? Specifically, is it clear which criterion the model is evaluated against, and how many sub-sections of the model will be used for the local stress analysis?	3/2.7		
2.7.2 If buckling is of interest, is an appropriate buckling criterion used for the given vessel type and risk profile?	3/2.7		
2.7.3 Are the hydrostatic loads applied to all surfaces correctly?	3/2.7		
2.7.4 Is a hull girder design wave applied if applicable?	3/2.7		
2.7.5 Does the model mass match the loading condition being evaluated?	3/2.7		
2.7.6 Is the model is hydrostatically balanced, or have the residual forces been minimized and an appropriate rigid body displacement restraints been applied?	3/2.7		
2.7.7 Does the mesh sizing match the requirements of the applied evaluation criteria?	3/3.3.1		
2.7.8 Are the beam and shell element properties appropriate for the given mesh design?	3/3.3.2		
2.7.9 Is the deflected hull girder shape qualitatively correct?	3/2.7		

Based on the above checks, answer Question 2.7, and enter the result in Figure A/0-1.	**Result**
2.7 Are the assumptions and data defining the whole ship analysis acceptable?	
Comments	

A/2.8 Frequency Response Vibration Analysis

Perform these checks to ensure that the correct procedures have been followed for a frequency response vibration analysis (i.e., a forced vibration analysis).

If the analysis is not a frequency response vibration analysis, you may proceed directly to Part A/3.1.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
2.8.1 Are all relevant cyclical excitation forces included in the analysis? If the analysis is a main machinery analysis, have the (1) the main engine excitation, (2) propeller alternative thrust excitation, and (3) propeller hull pressure excitation been considered?	3/2.7		
2.8.2 Are the excitation force magnitudes based on acceptable references?	3/2.7		
2.8.3 Have the directions of the excitation forces provided by the equipment manufacturer been considered?	3/2.7		
2.8.4 Do propeller hull pressure excitation forces have due consideration for cavitation?	3/2.7		
2.8.5 Is the model extent selected with due consideration for all the aforementioned natural and excitation frequencies, so that all relevant resonant conditions can be captured in the model?	3/2.7		
2.8.6 Have the hull-girder natural frequencies, main machinery/shafting system longitudinal vibration natural frequencies, superstructure fore-and-aft vibration natural frequencies all been considered?	3/2.7		
2.8.7 Does the model include all relevant masses? This includes the mass of all lightship weight, all loading condition weight such as cargo and tank liquids, and added mass where appropriate.	3/2.7		
2.8.8 Has the inclusion of damping been considered appropriately?	3/2.7		
2.8.9 Has a free vibration (modal) analysis been performed to identify both the hull-girder natural frequencies and the local natural frequencies that may impact the results? Are these natural frequencies compared against the excitation frequencies?	3/2.7		
2.8.10 If there is more than one cyclic load and these loads occur at different frequencies simultaneously, are two analyses run and the results combined during post-processing?	3/2.7		
2.8.11 If there is more than one cyclical load and these loads occur at the same frequencies simultaneously, are any phase angle shifts applied correctly? Alternatively, are the worst-case phase angles used?	3/2.7		
2.8.12 Are ranges of frequencies surrounding the excitation frequencies evaluated as required by Section 3/2.8.3?	3/2.7		

Based on the above checks, answer Question 2.8, and enter the result in Figure A/0-1.	**Result**
2.8 Are the assumptions and data defining vibration analysis acceptable?	
Comments	

FINITE ELEMENT ANALYSIS ASSESSMENT	FINITE ELEMENT MODEL CHECKS
Project No. I Project Title :	
Company Name:	IDate:
Analyst:	I Checker:

A/3 Finite Element Model Checks

A/3.1 CAD Importing

Perform the following checks to ensure that correct procedures have been followed for importing CAD. Relevant, material, loading, and boundary condition checks need to be completed in addition to general geometry considerations listed here.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
3.1.1 Is the CAD model compatible with the FEA software?	3/3.1		
3.1.2 Does the CAD model include all the geometric information for FE modeling?	3/3.1		
3.1.3 Does the CAD model need to clean up for meshing?	3/3.1		
3.1.4 Does the CAD model need to perform geometric defeaturing?	3/3.1		
3.1.5 Does the CAD model interfere or overlap with other components in the assembly? If not, do the gap distance between surfaces recorded?	3/3.1		

Based on the above checks, answer Question 3.1, and enter the result in Figure A/0-1.	**Result**
3.1 Are the assumptions and data for CAD importing acceptable?	
Comments	

A/3.2 Element Types

Perform these checks to ensure that the correct types of elements have been used to model the problem.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
3.2.1 Are all the different types of elements used in the FEA model identified and referenced in the analysis documentation?	3/3.2		
3.2.2 Are the element types available in the FEA software used appropriately to ship structural analysis?	3/3.2		
3.2.3 Do the element types support the kind of analysis, geometry, materials, and loads that are of importance for this problem?	3/3.2		
3.2.4 If required, do the selected beam element types include capabilities to model transverse shear and / or torsional flexibility behavior?	3/3.2		
3.2.5 If required, do the selected beam element types include capabilities to model tapered, off-set, or unsymmetric section properties?	3/3.2		
3.2.6 If required, do the selected beam element types include capabilities for nodal DOF end releases (e.g., to model partially pinned joints)?	3/3.2		
3.2.7 If required, do the selected plate element types include capabilities to model out-of-plane loads and bending behavior?	3/3.2		
3.2.8 If required, do the selected plate element types include capabilities to model transverse shear behavior (i.e., thick plate behavior)?	3/3.2		
3.2.9 If the model is 2-D, are the selected element types (or options) correct for plane stress or plane strain (whichever case applies)?	3/3.2		
3.2.10 If required, can the selected element types model curved surfaces or boundaries to an acceptable level of accuracy?	3/3.2		
3.2.11 If the model includes multiple element types, do the connected or transient elements have formulations that allow them to be connected and produced correct results? Check the software literature if there are questions about this.	3/3.2		
3.2.12 In a linear fracture mechanics analysis, does the set of 2-D or 3-D elements in the vicinity of the crack tip have 25% offset towards the tip and coincided/collapsed nodes, which can represent the stress singularity (i.e., 1/sqrt(r))?	3/3.2		

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
3.2.13 In a nonlinear fracture mechanics analysis, is the high-order element type used to capture the crack-tip blunting?	3/3.2		
3.2.14 If required for automatic mesh refinement, does the element type is compatible with user-defined mesh splitting or mesh refinement algorithm?	3/3.2		
3.2.15 In an analysis with element erosion or structural damage (e.g., flaw coalescence and crack propagation), is the element type compatible with material models? Have the element parameters been appropriately defined?	3/3.2		
3.2.16 In certain special analyses, hybrid or user-defined types of elements (note: user compiling needed) need to be adopted. Does the definition of elements meet the analysis requirements and can the analysis be performed accurately?	3/3.2		
3.2.17 Have the limits of each element type in the model been documented?	3/3.2		
3.2.18 For shell elements, are the number of integration points in-plane and through-thickness sufficient to capture tension/compression and bending behaviors? Have the locations of these integration points been recorded?	3/3.2		
3.2.19 For solid elements, are the number of integration points sufficient to capture tension/compression and bending behaviors? Have the locations of these integration points been recorded?	3/3.2		
3.2.20 For lower-order elements, are the numbers of the DOF of each node (especially the rotational DOF) appropriately defined to keep the calculation accuracy?	3/3.2		,
3.2.21 For 2-D shell elements, if the thickness offset feature is activated, has the offset value been verified and documented?	3/3.2		
3.2.22 If the automatic mesh refinement feature is activated, is the algorithm compatible with the current element type without inducing element inconsistency?	3/3.2		
3.2.23 If the element erosion feature (associated with material model) is activated, is the calculation of critical values verified without inducing inaccuracy?	3/3.2		
3.2.24 If a special element, e.g., a cohesive element, is adopted, are the related variables appropriately defined based on the FEA codes? Note: there are differences among FEA codes when dealing with special elements.	3/3.2		
3.2.25 If a user-defined element is adopted, is the element verified and documented? It is recommended to enclose the source code or encrypted file in the report.	3/3.2		

Based on the above checks, answer Question 3. 2 and enter the result in Figure A/0-1.	**Result**
3.2 Are the types of elements used in the FEA model acceptable?	
Comments	

A/3.2.1 Stiffness and Thickness Properties

Perform the following checks to ensure that correct procedures have been followed for defining the stiffness and thickness properties of the structure.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
3.2.1.1 Are all components that have a significant effect on the stiffness of the structure accounted for in the engineering model (under-considered load cases)?	3/3.2.1		
3.2.1.2 Are the assumed stiffness behaviors valid for each structural component (e.g., linear, membrane, bending, shear, and torsion)?	3/3.2.1		
3.2.1.3 Are the required stiffness parameters defined for each component and each contact pair, e.g.: • Truss members – A • Beams, bars – A, I_{yy}, I_{zz}, others • Plates, shells – t (uniform or varying, or offset) • Springs – K (axial or rotational) • Contact springs – K, friction coefficient	3/3.2.1		
3.2.1.4 Do the section properties of stiffeners (where modeled with beams) include correct allowances for the effective plate widths?	3/3.2.1.1		
3.2.1.5 If torsion flexibility is expected to be important, are torsion flexibility parameters correctly defined for beam sections?	3/3.2.1.1		
3.2.1.6 If shear flexibility is expected to be important, are shear flexibility parameters correctly defined for beam and/or plate elements?	3/3.2.1.1		
3.2.1.7 Are the values of the stiffness and thickness properties data supported by acceptable calculations and / or references?	3/3.2.1.1		
3.2.1.8 Are the units for the stiffness and thickness properties data consistent with the system of units for other parts of the analysis?	3/3.2.1.1		
3.2.1.9 Are the orientations of the beam element axes correct for the defined section properties?	3/3.2.1.1		3.2
3.2.1.10 Are the cross-section of the beam elements correct for the defined section properties?	3/3.2.1.1		3.2

Based on the above checks, answer Question 3.2.1, and enter the result in Figure A/0-1.	**Result**
3.2.1 Are the assumptions and data defining stiffness and thickness properties acceptable?	

Comments

A/3.3 Mesh Design

As a finite element method is essentially a piece-wise approximation technique, the accuracy is dependent mainly on the mesh design and corresponding quality. Perform the following checks to ensure that the finite element mesh is acceptable.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
3.3.1 Does the mesh design adequately reflect the geometry of the problem (e.g., overall geometry, stiffener locations, and details)?	3/3.3		
3.3.2 Does the mesh design adequately reflect the anticipated structural response (e.g., stress/strain gradients, stress/strain concentrations, deflections, mode shapes, failure modes, load paths)?	3/3.3		
3.3.3 Is the mesh free of unintentional gaps or cracks, overlapping, or missing elements?	3/3.3		
3.3.4 Is the mesh free of duplicate elements?	3/3.3		
3.3.5 Is proper node continuity maintained between adjacent elements (also continuity between beam and plate elements in stiffened panels)?	3/3.3		
3.3.6 Does the analysis documentation state or show that there are no "illegal" elements in the model (i.e., no element errors or warnings)?	3/3.3		
3.3.7 Are element aspect ratios acceptable (e.g., < 5 for shell), particularly near and in the areas of interest?	3/3.3		
3.3.8 Are element taper angles acceptable (e.g., < 0.6 for shell), particularly near and in the areas of interest?	3/3.3		
3.3.9 Are element skew angles acceptable (e.g., < 60 for shell), particularly near and in the areas of interest?	3/3.3		
3.3.10 If flat shell elements are used to model curved surfaces, are the curve angles < 10° for stresses, or < 15° for displacement results?	3/3.3		
3.3.11 If flat shell elements are used for double or tapered curve surfaces, is warping avoided or appropriately controlled (e.g., small curve angles, use of triangles, or < 20 for quad shell)?	3/3.3		
3.3.12 Does the analysis documentation state or show that there is an upper limit of the percentage of triangle shell and/or tetra solid elements in the model?	3/3.3		

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
3.3.13 Are differences in rotational DOF I moment continuity for different element types accounted for (e.g., beam joining solid)?	3/3.3		
3.3.14 Are the element shapes in the areas of interest acceptable for the types element used and degree of accuracy required?	3/3.3		
3.3.15 Are mesh transitions from coarse regions to areas of refinement acceptably gradual?	3/3.3		
3.3.15 Are mesh transitions from coarse regions to areas of refinement having no disconnected nodes/elements?	3/3.3		
3.3.16 Are nodes and elements correctly located for applying loads, support and boundary constraints, connections to other parts, and pairs of contacts?	3/3.3		
3.3.17 Are the outward normal directions for plate/shell elements of a surface in the same direction?	3/3.3		
3.3.18 If the analysis is a vibration analysis, is the mesh size sufficient to model the relevant vibration modes.	3/3.3.5.4		

Based on the above checks, answer Question 3.3, and enter the result in Figure A/0-1.	**Result**
3.3 Is the design of the finite element mesh acceptable?	
Comments	

A/3.4 Substructures and Submodeling

Substructuring or submodeling techniques may be employed to reduce the size of the problem for computing and / or to take advantage of repetitive geometry in the structure. Perform the following checks to ensure that the acceptable procedures have been followed.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
3.4.1 Is the overall substructure or submodeling scheme or procedure adequately described in the analysis documentation?	3/3.3.8		
3.4.2 Are all individual substructure models, global models, and refined submodels identified and described in the analysis documentation?	3/3.3.8		
3.4.3 Are the master nodes located correctly, and are the freedoms compatible for linking the substructures?	3/3.3.8		
3.4.4 Are the master nodes located correctly for the application of loads and boundary conditions upon assembly of the overall model?	3/3.3.8		
3.4.5 Are loads and boundary conditions applied at the substructure level consistent with those of the overall model?	3/3.3.8		
3.4.6 Does the boundary of the refined submodel match the boundary of coarse elements/nodes in the global model in the region of interest?	3/3.3.8		
3.4.7 If the technique of super-element is used, does the boundary as well as the node numbers of submodel match the super-elements at the region of interest, especially in the use of including FEA files (e.g., Nastran *.pch files)?	3/3.3.8		
3.4.8 Is the boundary for the submodel at a region of relatively low-stress gradient or sufficiently far away from the area of primary interest?	3/3.3.8		
3.4.9 Does the refined submodel correctly employ forces and/or displacements from the coarse model as boundary conditions?	3/3.3.8		
3.4.10 Does the submodel include all other loads applied to the global model (e.g., surface pressure and acceleration loads)?	3/3.3.8		
3.4.11 Have stiffness differences between the course global mesh and refined submodel mesh been adequately accounted for?	3/3.3.8		
3.4.12 Are the super-element(s) created correctly and verified by trial runs? (Note: it is recommended to carry out a preliminary modal analysis, and it is commonly seen a certain degree of difference in terms of eigenvalues.)	3/3.3.8		

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
3.4.13 Is the gravity center of the submodel or substructure reviewed?	3/3.3.8		
3.4.14 Does the model size (in terms of the total DOF of global models, including submodel or substructure or super element) meet the hardware capacity?	3/3.3.8		

Based on the above checks, answer Question 3.4, and enter the result in Figure A/0-1.	**Result**
3.4 Are the substructuring or submodeling procedures acceptable?	
Comments	

A/3.5 Loads and Boundary Conditions

Perform the following checks to ensure that correct procedures have been followed for defining the loads and boundary conditions of the finite element model.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
3.5.1 Are point load forces applied at the correct node locations on the structure, and are they defined with the correct units, magnitude, and direction?	3/3.5		
3.5.2 Are distributed loads applied at the correct locations on the structure, and are they defined with the correct units, magnitude, and direction?	3/3.5		
3.5.3 Are surface pressure loads applied at the correct locations on the structure, and are they defined with the correct units, magnitude, and direction?	3/3.5		
3.5.4 Are translational accelerations defined with the correct units, and do they have the correct magnitude and direction?	3/3.5		
3.5.5 Are rotational accelerations defined with the correct units, magnitude, and direction and about the correct center of rotation?	3/3.5		
3.5.6 Are prescribed displacements applied at the correct locations on the structure and are they defined with the correct units, magnitude, and direction.	3/3.5		
3.5.7 Are the displacement boundary conditions applied at the correct node locations?	3/3.5		

Based on the above checks, answer Question 3.5, and enter the result in Figure A/0-1.	**Result**
3.5 Are the loads and boundary conditions applied correctly?	
Comments	

A/3.6 Analysis Controls and Solution Options

Perform the following checks to ensure that correct solution options, techniques, or procedures have been used for the finite element model.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
3.6.1 Have any special solution options and procedures been used, and, if so, have they been documented?	3/3.6		
3.6.2 If non-standard options been invoked, have they been documented, and the reasons for their use been explained?	3/3.6		
3.6.3 If the problem is a dynamic analysis, is the method for eigenvalue and mode extraction appropriate?	3/3.6		

Based on the above checks, answer Question 3.6, and enter the result in Figure A/0-1.	**Result**
3.6 Are the solution options and procedures followed for the FEA acceptable?	
Comments	

FINITE ELEMENT ANALYSIS ASSESSMENT	FINITE ELEMENT RESULTS CHECKS
Project No. I Project Title :	
Company Name:	IDate:
Analyst: I Checker:	

A/4 Finite Element Results Checks

A/4.1 General Solution Checks

Perform these checks to expose any gross errors. Most program output values of gross parameters associated with the solution process. These parameters typically include summed applied loads and reactions, total mass, the position of the center of gravity, time steps, and other related parameters.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
4.1.1 Are all error and warning messages issued by the software reviewed and understood?	3/4.1		
4.1.2 Is the magnitude of the mass of the finite element model approximately as expected?	3/4.1		
4.1.3 Is the location of the center of gravity of the model, as calculated by the program, reasonable?	3/4.1		
4.1.4 Are the applied forces in equilibrium with the applied reactions?	3/4.1		
4.1.5 Are the reaction loads and moments at any supports added to prevent rigid body motion minimized?	3/4.1		
4.1.6 Does the solution converged within a number of time steps?	3/4.1		

Based on the above checks, answer Question 4.1, and enter the result in Figure A/0-1.	**Result**
4.1 Are the general solution parameters acceptable?	
Comments	

A/4.2 Post Processing Methods

Perform these checks to ensure that the methods and their limitations, used by the program to post-process the results are understood.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
4.2.1 Are the methods for reducing the analysis results described (e.g., calculation of safety factors and other parameters calculated by manipulating raw output)?	3/4.2		
4.2.2 Are the methods for "correcting" FE results described (e.g., correction factors and smoothing factors)?	3/4.2		
4.2.3 Are any path plotting of data and similar data collection tools appropriately applied to collect data on the correct surface or mid thickness location of the element?	3/4.2		

Based on the above checks, answer Question 4.2, and enter the result in Figure A/0-1.	**Result**
4.2 Is the methodology used for post-processing the results satisfactory?	
Comments	

A/4.3 Displacement Results

Perform these checks to ensure that the displacement results are consistent with expectations.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
4.3.1 Are the displacement results described and discussed?	3/4.3		
4.3.2 Are plots of the deformed structure (or mode shape) presented?	3/4.3		
4.3.3 Are the directions of displacements consistent with the geometry, loading, and boundary conditions?	3/4.3		
4.3.4 Do the magnitudes and sense/direction of displacements make sense?	3/4.3		
4.3.5 Is the deformed shape (or mode shape) smooth and continuous in the area of interest?	3/4.3		
4.3.6 Are unintentional slits or cuts (indicating elements not connected where they should be) absent?	3/4.3		

Based on the above checks, answer Question 4.3, and enter the result in Figure A/0-1.	**Result**
4.3 Are displacement results consistent with expectations?	
Comments	

A/4.4 Force Results

Perform these checks to ensure that the force results are consistent with expectations.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
4.4.1 Are the force results available for review? For example, the results are exported into text files.	3/4.4		
4.4.2 Do the force magnitudes and directions make sense? Are they explainable?.	3/4.4		
4.4.3 Do the forces and moments balance? For example: • Global-level • Local-level • Contact forces • Following forces	3/4.4		
4.4.4 For strain-life fatigue analysis, are the force levels suitable for using Neuber correction method?	3/4.4		
4.4.5 For the fracture analysis, are the force results using for validating fracture toughness J-integral?	3/4.4		

Based on the above checks, answer Question 4.4, and enter the result in Figure A/0-1.	**Result**
4.4 Are force results consistent with expectations?	
Comments	

A/4.5 Stress Results

Perform these checks to ensure that the stress results are consistent with expectations.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
4.5.1 Are the stress results described and discussed? Is any stress index used (i.e. equivalent von Mises)?	3/4.5		
4.5.2 Are stress contour plots presented? In the stress plots, are the stress parameters or components defined using appropriate units and scale/fringe levels?	3/4.5		
4.5.3 Is the method of smoothing stress results, or averaging stress results described (e.g., element stresses vs. nodal average stresses)?	3/4.5		
4.5.4 Are the units of stress parameters consistent?	3/4.5		
4.5.5 Are the magnitudes of stresses consistent with intuition?	3/4.5		
4.5.6 In cases where there are adjacent plate elements with different thicknesses, does the method for averaging stresses account for the differences?	3/4.5		
4.5.7 Are the stress contours smooth and continuous, particularly in the region of primary interest?	3/4.5		
4.5.8 Are the stress contours at boundaries consistent with the boundary conditions applied (e.g., stress contours perpendicular to the boundary if there is a symmetry boundary)?	3/4.5		
4.5.9 Are stresses local to the applied loads reasonable?	3/4.5		
4.5.10 Do stress levels at the boundaries (applied loads and fixtures) make an engineering sense by comparing it with the area of interest? Note that the area of interest should be sufficiently remote from boundaries (applied loads and fixtures).	3/4.5		
4.5.11 Are there areas in which stresses are above yield (which would invalidate linear elastic analysis)? Are these areas small enough to be ignored?	3/4.5		

Based on the above checks, answer Question 4.5, and enter the result in Figure A/0-1.	**Result**
4.5 Are stress results consistent with expectations?	

Comments

A/4.6 Strain Results

Perform these checks to ensure that the strain results are consistent with expectations.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
4.6.1 Are the strain results described and discussed?	3/4.6		
4.6.2 Are strain contour plots presented? In the strain plots, are the strain parameters or components defined using appropriate units and scales/fringe levels?	3/4.6		
4.6.3 Is the method of smoothing strain results, or averaging strain results described (e.g., element strains vs. average nodal strain)?	3/4.6		
4.6.4 Are the units of strain parameters consistent?	3/4.6		
4.6.5 Are the magnitudes and directions of strains consistent with intuition?	3/4.6		
4.6.6 In cases where there are adjacent plate elements with different thicknesses, does the method for averaging strain account for the differences?	3/4.6		
4.6.7 Are the strain contours smooth and continuous, particularly in the region of primary interest?	3/4.6		
4.6.8 Are the strain contours at boundaries consistent with the boundary conditions applied (e.g., strain contours perpendicular to the boundary if there is a symmetry boundary)?	3/4.6		
4.6.9 Are strains local to the applied loads reasonable?	3/4.6		
4.6.10 Are there areas in which strains are over yield (which would invalidate linear elastic analysis)?	3/4.6		

Based on the above checks, answer Question 4.6, and enter the result in Figure A/0-1.	**Result**
4.6 Are strain results consistent with expectations?	

Comments

A/4.7 Energy Results

Perform these checks to ensure that the energy results are consistent with expectations.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
4.7.1 Are the energy-related parameters exported into text files? For example: • Strain energy • Internal energy • Kinetic energy • Contact energy • Hourglass energy • Damping energy	3/4.7		
4.7.2 Are the internal and external energy balanced?	3/4.7		
4.7.3 Are the amount of hourglass and/or damping energy within the limit?	3/4.7		
4.7.4 For the fracture analysis, are the energy-related fracture toughness parameter employed?	3/4.7		

Based on the above checks, answer Question 4.7, and enter the result in Figure A/0-1.	**Result**
4.7 Are energy results consistent with expectations?	
Comments	

A/4.8 Fracture Results

Perform these checks to ensure that the fracture results are consistent with expectations.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
4.8.1 Are the fracture mechanics parameters exported? For example: • J (J-integral) • K (stress intensity factor) • CTOD (crack-tip opening displacement) • CTOA (crack-tip opening angle) • C* (creep fracture mechanics parameter)	3/4.8		
4.8.2 Are the results used for a fatigue analysis? Are the results adequate for evaluating the fatigue crack growth rate?	3/4.8		
4.8.3 Are the results used for an integrity analysis? If yes, which design code does the analysis follows? Is the analysis procedure documented?	3/4.8		
4.8.4 Are the results adequate for evaluating parameters in integrity analysis, e.g., the FAD method?	3/4.8		

Based on the above checks, answer Question 4.8, and enter the result in Figure A/0-1.	**Result**
4.8 Are fracture results consistent with expectations?	
Comments	

A/4.9 Fatigue Results

Perform these checks to ensure that the fatigue results are consistent with expectations.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
4.9.1 Are the fatigue life calculation performed manually, in software or explicitly in the FEA?	3/4.9		
4.9.2 Which index is used for fatigue life estimation? For example: • Absolute or signed maximum shear stress • Absolute or signed von Mises stress • Absolute or signed maximum principal stress	3/4.9		
4.9.3 What is the safety factor (SF)? For example: • SF = 2	3/4.9		
4.9.4 Does the estimated fatigue life agree with that from testing?	3/4.9		
4.9.5 Does the explicit crack growth direction or stress direction make sense?	3/4.9		
4.9.6 Is the crack tip plasticity reasonably localized?	3/4.9		
4.9.7 Are the node release criteria respected?	3/4.9		
4.9.8 Are the load shedding and boundary conditions of sub-models behaving appropriately?	3/4.9		

Based on the above checks, answer Question 4.9, and enter the result in Figure A/0-1.	**Result**
4.9 Are fatigue results consistent with expectations?	
Comments	

A/4.10 Vibration Results

Perform these checks to ensure that the vibration analysis results are consistent with expectations.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
4.10.1 Are the frequencies expressed in the correct units?	3/4.10		
4.10.2 Are the magnitudes of natural frequencies consistent with the type of structure and the modal number?	3/4.10		
4.10.3 Are the mode shapes smooth?	3/4.10		
4.10.4 Are results being reported for a range of frequencies near the excitation frequency?	3/2.8.3 3/4.10.2		
4.10.5 Do the results and criteria being used consider that the local response magnitudes may be off by a factor of 2 to 3?	3/4.10.2		

Based on the above checks, answer Question 4.10, and enter the result in Figure A/0-1.	**Result**
4.10 Are vibration results consistent with expectations?	
Comments	

FINITE ELEMENT ANALYSIS ASSESSMENT	CONCLUSIONS CHECKS
Project No. I Project Title :	
Company Name: IDate:	
Analyst: I Checker:	

A/5 Conclusion Checks

A/5.1 FEA Results and Acceptance Criteria

Perform these checks to ensure that the results are in a form suitable for comparison with specified acceptance criteria.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
5.1.1 Are the results presented in units and summarized in a manner that allows comparisons with acceptance criteria, or alternative solutions or data?	3/5.1		
5.1.2 Are satisfactory explanations provided where the results do not meet acceptance criteria, or where they differ significantly from other comparable solutions or data?	3/5.1		

Based on the above checks, answer Question 5.1, and enter the result in Figure A/0-1.	**Result**
5.1 Are the results presented in sufficient detail to allow comparison with acceptance criteria?	
Comments	

A/5.2 Load Assessment

Perform these checks and evaluations to ensure that the loads applied in the FEA, and their accuracy, are understood.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
5.2.1 Has an assessment been made of the accuracy or degree of conservatism of the loads used in the FE model with respect to the following aspects:	3/5.2		
a. types of loads/load cases that were included and excluded			
b. basis or theory used to derive loads (e.g., linear strip theory for sea motion loads, base acceleration vs. DRS for shock, drag coefficients for wind loads, etc.)			
c. magnitudes of loads			
d. loading directions included / excluded			
e. load combinations			
f. load factors			
g. boundary conditions			

Based on the above checks, answer Question 5.2, and enter the result in Figure A/0-1.	**Result**
5.2 Are the accuracy and conservatism, or otherwise, of the applied loading modeled understood?	
Comments	

A/5.3 Strength / Resistance Assessment

Perform these checks and evaluations to ensure that an adequate assessment of the capability of the structure has been made.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
5.3.1 Has an assessment been made of the accuracy or degree of conservatism of the strength or resistance of the modeled structure with respect to the following aspects:	3/5.3		
a. failure theory, failure criteria, allowable stresses, safety factors, etc.			
b. section properties			
c. material properties			
d. allowances for imperfection, misalignment, manufacturing tolerances			
e. allowances for corrosion			

Based on the above checks, answer Question 5.3, and enter the result in Figure A/0-1.	**Result**
5.3 Has an adequate assessment been made of the capability of the structure?	
Comments	

A/5.4 Accuracy Assessment

The checks listed below are intended to ensure that an attempt has been made to assess the accuracy of the FEA.

Finite Element Analysis Assessment Check	Refer To Guideline Section	Result	Comments
5.4.1 Has an assessment been made of the scale of the FE model and its level of detail and complexity?	3/5.4		
5.4.2 Have the types of behavior modeled and not modeled (e.g., membrane only instead of membrane plus bending) been assessed?	3/5.4		
5.4.3 Has the influence of mesh refinement on accuracy been considered?	3/5.4		
5.4.4 Has a comparison with other results (e.g., other solutions, and experiments) been made?	3/5.4		
5.4.5 Based on the above, has an overall assessment of the accuracy of the relevant results been made?	3/5.4		

Based on the above checks, answer Question 5.4, and enter the result in Figure A/0-1.	**Result**
5.4 Has an adequate assessment of the accuracy of the analysis been made?	
Comments	

A/5.5 Overall Assessment

The checks listed below are to ensure that the overall conclusions and recommendations resulting from the FEA have been presented and are generally satisfactory.

Finite Element Analysis Assessment Check	**Refer To Guideline Section**	**Result**	**Comments**
5.5.1 Are conclusions from the FEA provided, and are they consistent with the material presented?	3/5.5		
5.5.2 If appropriate, has a way ahead or potential solutions been presented?	3/5.5		
5.5.3 Based on consideration of all previous checks, is the overall assessment that the FEA is acceptable?	3/5.5		

Based on the above checks, answer Question 5.5, and enter the result in Figure A/0-1.	**Result**
5.5 Is the finite element analysis generally assessed satisfactorily?	
Comments	

Appendix B
Example Application of Assessment Methodology

B1.0 INTRODUCTION

The purpose of this Appendix is to illustrate the application of the FEA assessment methodology, and the guidelines presented in Parts 2 and 3 of this document.

An example finite element analysis (FEA) of a web frame from an Arctic-going tanker design subject to ice loads is used for this purpose. The approach used to illustrate the assessment methodology and guidelines includes :

- a sample report of the Arctic tanker web frame FEA annotated with references to relevant sections of the FEA assessment methodology and guidelines; and
- completed checklists as required by the assessment methodology.

The annotated report and the completed checklists are presented in Annexes 8-1 and 8-4, respectively.

B2.0 EXAMPLE FINITE ELEMENT ANALYSIS

The example FEA is adapted from an analysis for an actual design[1] of an icebreaking tanker. The tanker is double-hulled. Transverse strength is provided by a series of closely spaced web frames, and the longitudinal load transfer is achieved through several longitudinal stringers. The design requirements are based on current Canadian rules.

The primary interest tor this analysis is the behavior of a typical web frame in response to ice loads. Other loads are ignored as negligible compared with the ice loads. The analysis was performed to ensure that the side structure that directly resists the ice loads responds in the manner expected by the designers and that the structure is as optimized as possible.

This example illustrates several aspects of finite element modeling common in ship structures, including:

- behavior of stiffened plate structures
- openings in structures
- discontinuities often found in ship structures
- integrated nature of typical ship structures
- use of most types of elements commonly used in the FEA of ship structures.

For reasons explained in Annex B-1, it was necessary to make modifications to the original analysis, particularly in regard to the level of ice load, to make it suitable for the purposes of the present work.

[1] The design was undertaken by Canarctic Shipping Co. Ltd., Ottawa, Ontario, Canada under

contract to the Transportation Development Centre, Montreal, Quebec, Canada

B3.0 ANNOTATED REPORT

Annex B-1 presents a sample report of the Arctic tanker web frame FEA that has been prepared by a contractor ("BB Engineering") and has been subjected to the assessment methodology. For illustrative purposes, the report has been annotated with short descriptions identifying the relevant part of the assessment methodology presented in Parts 2 and 3 of this document. Except for the annotations, the report is meant to be typical of the documentation that an evaluator of FEA might receive.

B4.0 CHECKLISTS

A sample of completed FEA evaluation checklists for the report in Annex B-1 is presented in Annex B-4.

Acknowledgment

The finite element analysis described in the following pages is adapted from an analysis performed by MIL Systems Engineering, Ottawa, Ontario, for Canarctic Shipping Ltd., Ottawa, Ontario, under a contract awarded by the Transportation Development Centre, Montreal, Quebec.

Warning

This example is presented solely for the purpose of illustrating the assessment methodology described in Part 2. As such, it is not necessarily complete in all details, particularly in regard to parameters such as number of loading types, design criteria, and number of structural responses considered. Furthermore, this example should not be construed as representative of the requirements for finite element analysis of other marine structures.

Annex B-1
Finite Element Analysis of Arctic Tanker Web Frame

BB Engineering Ltd.
13-1300 Finite Drive
Ottawa, Ontario
XXX XXX

FINITE ELEMENT ANALYSIS OF 50000 DWT TANKER SINGLE MIDBODY WEB FRAME

1.0 INTRODUCTION

AA Shipping Company Limited has developed a design for a 50000 DWT Arctic tanker. The focus of the work has been to design cost-optimized midbody and bow structures.

The BB Engineering Co Ltd. (BBE) has been tasked to undertake a finite element analysis (FEA) of a typical midbody web (diaphragm) frame. The purpose of the FEA reported in this report is to assess the response of the midship structure to ice loads.

Section 2 of this report provides a summary of the requirements for the analysis, and data on the software and the resources applied to the problem. The engineering model is described in Section 3. This section includes a discussion of the subject structure and the assumptions made in developing the engineering model. Section 4 describes the finite element model, and Section 5 presents the results of the analysis.

2.0 PRELIMINARY INFORMATION

2.1 Job Specification

The job specification calls for a static, linear elastic, FEA of a web frame from the midbody section of the 50000 DWT tanker at a design ice load of 4435 kN.

The finite element model is based on the drawings provided in Arctic Tanker Structural Evaluation - Midship Sections, Bow Sections, and Repair Drawings (Ref. 2).

The acceptance criteria for the analysis are as follows:

1. maximum stress not to exceed the material yield stress except as noted in item 2.
2. very localized stresses in excess of yield stress are considered acceptable

2.2 Rationale for using Finite Element Method

The structure under investigation is too complex to be analyzed by hand calculation particularly in regions of high-stress concentrations.

FEA Software
Para. 1.3.1

2.3 FEA Software

ANSYS finite element software (Version 5.1), developed and supported by ANSYS Inc. of Houston, PA, was used for the finite element work performed and presented here. ANSYS is a well-established FEA package that has a proven track record in analyzing structures of the type under consideration. BBE currently has maintenance and technical support contract with ANSYS, Inc.

The software updates and error reports received from ANSYS are reviewed by all BBE staff involved in FEA and filed along with other ANSYS documents. ANSYS's shell and beam elements have been validated by BBE for use in ship structural analysis. ANSYS has been evaluated against benchmarks designed to test the capability of the software to perform ship structural FEA.

2.4 Contractor and Analyst Qualifications

Information on qualifications of the contractor, the analysts, and the supervisor, to perform the required FEA is provided in Annex B-2 of this document.

Contractor I Personnel Qualification

Para. 1.4

3.0 ENGINEERING MODEL

3.1 Analysis Type and Assumptions

Since the stresses are limited to the yield stress, the material behavior is assumed to be linear. Similarly, because large deflections are not expected geometric behavior is assumed to be linear as well.

Analysis Type & Assumptions
Para. 2.

Justification for using FEA

Para. 1.2.6

The load is assumed to be static, and interest is centered on the strength of the frame. Hence, the dynamic behavior of the frame is not within the scope of this analysis. Instability behavior is also not considered in this analysis. However, it should be considered as part of the design process.

The overall strength of the frame is the primary focus of this analysis, and therefore the analysis is not optimized to examine stress concentrations at structural discontinuities such as those that will exist around openings for example. Again, these should be addressed as part of the normal design process.

3.2 Global Geometry of 50000 DWT Tanker

Geometry Assumptions Para. 2.2

The 50000 DWT tanker has a waterline length of 242 meters, a breadth of 34.6 meters and a depth of 18.1 meters. The vessel has seven cargo tanks. In the cargo tank region of the vessel the distance between transverse bulkheads is 19.2 meters. Each cargo tank has approximate dimensions of 18 m x 30.6 m x 14.6m.

The vessel is double-hulled. The distance between the inner and outer hulls is 2000 mm. The bottom structure wraps around the turn of the bilge and connects to the side shell structure at a point

4.0 meters above baseline. The side shell structure connects with the deck structure at a point of 15.0 meters above the baseline. Therefore, the side shell structure vertically spans a distance of

11.0 meters. The structure is transversely framed by web frames (diaphragms) spaced at 1000 mm intervals. Longitudinal framing is provided by several stringers spanning between bulkheads.

The midship section is shown in Figure[1] 3.1

3.3 Frame Selected

Extent of Model Para. 2.2.1

The ice load for the 50000 DWT tanker is approximately 22 meters in length and, therefore, if centrally positioned, spans across a pair of bulkheads. The ice load applied to side structure is resisted by the transverse frames (each acting essentially as a ring), the deck structure, the bottom structure and the transverse bulkheads.

Any transverse loads applied to the side structure are distributed vertically to the bottom and deck structures by transverse frames,

· and longitudinally to bulkheads through stringers.

The most severe loading case for a web frame is from ice load

1 *Figures are presented at the end of this document*

applied to the frame midway between bulkheads and centrally disposed with respect to the frame. The characteristics of the load are discussed in Section 3. 7.

3.4 Extent of Model

The structure of the vessel, between transverse bulkheads, is a series of ring frames comprising inner and outer hull plating with a stiffened plate diaphragm connecting them. These frames are connected by all longitudinally oriented structure (framing members and plating).

Extent of Model
Para. 2.2.1

An alternative method to account for the influence of the surrounding structure would be to model adjacent web frames and stringers approximately.

It is sufficient to model a single transverse ring frame if the correct boundary conditions are applied as discussed in Section 3.6. Due to the symmetry (structure and load) that exists along the vessel centerline it is also sufficient to model one half of the ring frame.

This ring frame extends from the bottom of the ship at centerline around to the vessel centerline at the deck. The width of the model needs to be the frame spacing (1000 mm) and will include the inner and outer shell plating and the stiffened plate diaphragm.

Figure 3.1 illustrates the midbody frame that was analyzed. Figure

3.2 shows the outer dimensions for the frame.

3.5 Material Properties

Figure 3.2 indicates that the vessel material in the outer shell plating is Grade EH50 and that the inner shell and framing components are Grades DH36 and EH36. Table 3.1 lists the relevant material properties as taken from Reference 3 for these steel grades.

Material Properties
Para. 2.3

The Young's Modulus was taken as 208,700 MPa for all steel types. Parameters such as initial imperfections and residual strains were not included in the analysis, and no allowance is made for corrosion. These assumptions are consistent with the design criteria.

TABLE 3.1: Steel Mechanical Properties

Property	**Steel Grade**	
	EH50	**DH36,EH36**
Yield Stress (min.) (MPa)	500	355
Tensile Stress (MPa)	610-770	490-620
Elongation %	16	21
Young's Modulus (MPa)	208700	208700
Poisson's Ratio	0.3	0.3

3.6 Interaction **with Adjacent Structure**

Influence of unmade/led structure Para. 2.2.3

The midbody web frame is part of an integrated structural system comprising the inner and outer shells, the transverse frames and longitudinal girders. However, for the reasons discussed above, it is reasonable to isolate a single web frame for analysis provided that the interaction with adjacent structure is accounted for.

The primary interaction with adjacent structure (for the load pattern of interest to this analysis) is through load transfer via longitudinal structure. A reasonable approximation for this configuration is to account for the support provided by the longitudinal structure by using springs representing the stiffness of this structure.

With reference to Figure 3.1, springs are required at the following locations:

1. Centerline of Main Deck to account for the deck centerline longitudinal girder (vertically);
2. On Main Deck to account for the inboard side girder (vertically);
3. On Main Deck to account for the outboard side girder (vertical and horizontal components);
4. On side shell to account for the upper stringer (horizontal);

5. On side shell to account for the lower stringer at the top of the turn of the bilge (horizontal);
6. Bottom structure to account for the girders (3 locations - vertically);
7. Centerline of bottom structure to account for the centerline girder (vertically); and
8. Bottom structure to account for the bottom shell longitudinals (vertically).

Spring constants for the above items have been calculated as the inverse of the deflection at the midspan of the longitudinal member being evaluated (list above) due to a unit point load placed at each of the points of intersection with a midbody web frame along its length. The ends of the longitudinal member(s) have been conservatively assumed as pinned. If a fixed end condition had been assumed, the stiffness of the longitudinal structure would have been overestimated resulting in a greater load transfer from the midbody web frame than would be the case in reality.

Spring constants calculated and used in the FE model are listed in Section 84.4 Beam Section Properties.

3.7 Loads

The ice load[2] is a function of vessel displacement, power of the vessel, the region of the ship, and the Arctic Class. Taking account of the various factors associated with ship parameters the total load applied to the web frame is 4435 kN. This is applied as a uniform pressure of 1 meter width (which equals the web frame spacing) and 2.85 meter height. This translates to a pressure of

1.556 MPa. As required by the standard the pressure patch is positioned such that 10% of its height is above the waterline.

Loads

Para. 2.6

Para. 3.4

Influence of Extent of Model Para. 2.2.1

The load applied is illustrated in Figure 3.3.

[2] *The ice loads are adapted from Ref. 1. The structural design philosophy of this standard is based on plastic design. Hence design loads calculated from this standard will, for a well-designed structure, result in extensive yielding. For the purposes of this example FEA, which assumes linear elastic behavior, the load applied has been arbitrarily halved to ensure the structure remains elastic.*

3.8 Boundary Conditions

Symmetry is assumed about a vertical plane through the longitudinal axis of the ship. Therefore, symmetry boundary conditions are applied to all nodes along the outer (longitudinal) edges of the plates. This provides translational restraint along the longitudinal axis of the vessel, and rotational restraint about the other two axes.

Boundary Conditions Para. 2.6

Para. 3.4

Symmetrical boundary conditions are applied to the bottom structure and the deck structure intersecting the vertical plane through the longitudinal axis of the ship. In addition, the bottom shell plating along the centerline is fixed in the vertical translation to avoid rigid body motion

4.0 FINITE ELEMENT MODEL

4.1 General Information

SI units were used throughout the finite element model. Therefore, the units of length, area, moment of inertia, Young's Modulus, and pressure were mm, mm^2 , mm^4 , MPa, and MPa, respectively.

Units

Para. 2.1.7

The global coordinate system for the problem is as follows:

Global axes system Para. 2.1.8

Global X-axis :	athwartshi
Global Y-axis :	p vertical
Global 2 axis :	parallel to ship CL

4.2 Element **Selection**

The elastic shell element (SHELL63) of ANSYS was selected and used for modeling the web frame, and stiffeners from the bottom stringer of the side shell structure at the top of the turn of the bilge to the start of the sloped section on the outboard edge of the main deck. The stiffeners in other areas were modeled using 3-D elastic beam elements (BEAM44) of ANSYS. The stiffness of longitudinal girders was modeled using linear spring elements (COMBIN14).

Element Types Para. 3.1

The SHELL63 element is well suited for modeling linear behavior of flat or warped, thin to moderately thick, shell structures.

The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the element x, y, and z axes. The deformation shape is linear in the two in-plane directions. The out-of-plane motion is predicted using a mixed interpolation of tensorial components. The element is defined by four corner nodes, four thicknesses, and the orthotropic material properties (if required). A triangular-shaped element may be formed by defining the same node numbers for the third and fourth nodes. Pressure load may be applied as surface loads on the element.

The stiffeners in the deck and bottom structure of the mid-body section have been modeled using 3-D elastic offset beam elements (BEAM44). BEAM44 is a uniaxial element with tension, compression, torsion, and bending capabilities. This element also has six degrees of freedom per node. The stiffeners in the side structure diaphragms were modeled using shell elements (SHELL63).

To simulate the overall stiffness of the rest of the structure, as discussed in Section 2.4, the connection points of the frame to other structures were modeled with linear springs (COMBIN14) elements. COMBIN14 elements are uniaxial tension-compression elements with up to three degrees of freedom at each node: translations in the nodal x, y, and z directions. Two sets of elements, one for springs in the horizontal direction and the other for springs in the vertical direction, were defined.

4.3 Mesh Design

Mesh Design

Para. 3.2

The response of the side shell structure is of primary interest, particularly in the vicinity of the loading. Therefore, the frame structure has been modeled with a fine mesh of shell elements in the following areas:

1. side shell structure between the turn of the bilge and the side shell upper stringer; and
2. outer edge of the deck structure between the side shell upper stringer and the deck angled outboard girder.

The remainder of the frame has been modeled using a coarse mesh of shell and beam elements. This ensures that the stiffness of this part of the structure is reasonably modeled in an economical manner.

The mesh, consisting of beam and shell elements, used for the frame analysis is shown in Figure 4. 1. The mesh design is consistent with the results expected from the finite element model, that is, a fine mesh is provided in the regions where a high-stress gradient is expected with a coarse mesh provided elsewhere. The mesh is the densest around openings, which are sources of stress concentrations. Since the primary interest is in establishing overall adequacy of the structure, the mesh density adopted is designed to yield stresses that are accurate for this purpose. Based on preliminary analyses the mesh around these openings should allow the prediction of peak stresses with an accuracy of roughly ± 5%.

The finite element model contains 3758 elements, 3578 nodes, and 18131 total active degrees of freedom.

4.4 Finite Element Attributes and Spring Constants

Stiffness and Mass Properties

Para. 2.4

The attributes of the elements used in the model are listed in Table 4.1 . The spring constants calculated based on the stiffness properties of the adjacent structure are listed in Table 4.2.

To avoid ill-conditioning in the stiffness matrix, ANSYS prints a warning if the ratio of largest to smallest stiffness value is greater than 1.0e08. The largest stiffness in the stiffness matrix being 4. 179e + 11, the smallest stiffness allowed is 4179 N/mm. Therefore, springs with stiffness less than 4179 N/mm were not used. Because of their relatively low stiffness values, these springs will have a negligible effect on the overall behavior of the web frame.

TABLE 4. 1: Finite Element Attributes

Item **No.**	Description	Element **Type** & **No.**	**Met. Type &No.**	**Real Cons. No**	**Thickness or Area mm/mmi**	Izz **x10"** mm"	Iyy **x10^{3}** mm"	TKZT1 mm	TKYT1 mm
1	Diaphragms / Web Plating	She1143	EH36	101	16				
2	Floors - Web Plating	"	"	102	26				
3	Deck Transverses - Web 1500x12	"	"	103	12				
4	Deck Plating	Shell43	EH36	104	14				
5	Outer Shell Plating	"	EH50	105	36				
6	Bottom Shell Plating	"	AH36	106	29				
7	Deck Transverses - Flange	Shell43	EH36	107	19				
8	Inner Deck Plating	"	..	108	14				
9	Inner Shell Plating	"	"	109	16.5				
10	Inner Shell Plating - Bilge	"	"	110	17				
11	Tank Top Plating	"	"	111	13				
12	Transverse Stiffeners - Diaphragms	Shell43	EH36	112	16				
13	Stringers	"	"	113	16				
14	Transverse Stiffeners - Tank Top	Beam44	AH36	114	5700	38.58	190.0	10	142.5
15	Girders - Tank Top	Shell43	AH36	115	15				
16	Deck Transverse Stiffeners	Beam44	EH36	116	1575	2.95	14.47	5.25	75
17	Side Girders	Shell43	EH36	117	14				
18	Deck Plating (with openings)	Shell43	EH36	118	9.34				
19	Beam Elements for stiffeners at	Beam44	EH36	119	6576	92.56	140.3	8	205.5
20	Beam Elements for the bilge and	Beam44	EH36	120	6576	92.e6	140.e4	8	205.5
21	Vertical Springs - to account for	Combin14	-	see Table 4.2 for spring stiffness					
22	Horizontal Springs - to account for	Combin14	-	see Table 4.2 for spring stiffness					

TABLE 4.2 Spring Stiffness Calculated Based on Stiffness of Adjacent Structure

Description	**Spring Direction**	**Element Type**	**Real Constant No.**	**Spring** Stiffness **N/mm**
Deck Centerline Girder	Vertical	5	121	231
Inboard Side Girder	Vertical	5	122	3785
Outboard Side Girder	Vertical	5	123	3012
Outboard Side Girder	Horizontal	6	124	56
Upper & Centre Stringer	Horizontal	6	125	7151
Lower Stringers	Horizontal	6	126	7151
Bottom Girder - Outboard	Vertical	5	127	6508
Bottom Girders	Vertical	5	128	5913
Bottom Centre Line Girder	Vertical	5	129	3631

4.5 FE Model Loads and Boundary Conditions

Loads and Boundary Conditions

Para. 2.6

Para. 3.4

General information on the applied load is provided in Section 3. 7. The design ice load was applied as a pressure of 1.556 MPa.

The finite element model boundary conditions are as explained in Section 3.8. Referring to the global coordinate system described in Section 4.1,all nodes with Z - co-ordinate of +500or -500mm have symmetry boundary conditions along the Z-axis. This provides translation restraints in the Z-axis and rotational restraints in the X and Y axes. All nodes along the bottom center

line have symmetry boundary conditions along the X - axes, i.e., translations restrained in the X and rotations restrained in the Y & Z axes. The nodes along the bottom centerline for the bottom shell plating were also restrained in the Y direction. For the top centerline, all nodes have symmetry boundary conditions along the X-axis.

4.6 FE Model Checks

Before the finite element model was run, the following prerun checks were performed on the FE model :

Finite Element Model Checks

Para. 3.0

- consistent units
  coordinate system
- element attributes and real
  constants boundary conditions and
  loads

The following prerun checks were conducted using the graphical user interface provided by ANSYS. ANSYS provides a listing of requested information for specifically selected entities. Also, symbols can be turned on/off to view various aspects, such as boundary conditions, loads, element connectivity, etc., of the model.

- nodal coordinates of extremities of model
- free edge plots to check for structural discontinuities element shape; aspect ratio, taper, skew, orientation shrink plots and element edge plots to check element connectivity
- checks for property assignment to elements - using color coding based on element type, material type, physical property type, etc.
- element plot showing element coordinate system to check for element orientation
- true scale 3D plot of beam elements to ensure correct beam size, orientation, and offsets
- boundary conditions - using model plots with boundary condition symbols
- pressure load magnitude and direction (using arrows)

The following prerun checks are built into ANSYS and are performed during the data checking process. Warning or error messages are issued when the model fails to pass the check. The output from such a data check run was reviewed for warning and/or error messages.

- nodes not connected to structure
- elements not connected to structure
- missing material properties
- missing physical properties

- element aspect
- ratio element
- warping element
- skewness

4.7 FE Solution Option and Procedures

The following solution options and procedures used were:

- New Analysis
- Static Analysis
- No Stress Stiffening
- Small Deflections
- Store all results for all load steps
- Print all output to a listing file

Solution Options and Procedures

Para. 3.5

5.0 ANALYSIS RESULTS

5.1 General Solution Checks

The following post-run checks were performed:

comparison with simple hand calculations to ensure that the results are reasonable (these calculations are included as Annex 8-3)

equilibrium between the applied load and the reactions
inspection of the displaced shape of the structure to ensure that there were no discontinuities in the model

inspection of stress contours to ensure the adequacy of the mesh used

All error and warning messages output by the program were investigated and resolved.

The total applied load in the X direction is 4434.9 kN. No forces are applied in the Y and Z directions. The summed reactions in the X, Y, and Z directions are 4434.9 kN, 0 kN, and 0 kN, respectively.

General Solution Checks

Para. 4.1

5.2 Post Processing Methods

Post-processing Methods

The ANSYS graphical post-processor was extensively used to review stress and displacement results. Listings were reviewed to obtain specific magnitudes for various quantities. In all of the stress contour plots, nodal averaging was used. For the shell element used in the model, the nodal values are calculated by extrapolating from the values at the integration points.

Para. 4.2

Para. 4.3

Para. 4.4

5.3 Structural Response

The deflected shape of the structure is shown in Figure 5.1, where the displacements are scaled up by a factor of 20. The maximum vertical displacement at the top centerline of the vessel is 1 24 mm. The maximum horizontal displacement is 51.08 mm and occurred on the inner shell in the vicinity of the load application.

The out of plane displacement, which was relatively small at 1.96 mm, occurred in the diaphragm between the side shell and the opening, also in the area of load application.· This displacement occurred between two stiffeners indicating a possible location for shear buckling. This possibility should be checked using classical methods.

FEA Results and Acceptance Criteria Para. *5.1*

The von Mises stress plot for the area of interest is shown in Figure 5.2. The contours are arranged, such that color orange indicated stresses past yield (355 MPa) in all areas except the outer shell. Dark red shading is used to indicate stresses past yield (500 MPa) in the outer shell. It is clear from the figure that at the applied load the overall structure remains elastic, except for a small area around the openings where the stresses are past-yield. The maximum stress recorded here is 573 MPa.

Figure 5.3 shows contours of bending stress, Sy. The outer shell is in compression with a maximum compressive stress of 307 MPa. The inner shell has maximum tensile stress of 330 MPa. High bending stresses, past yield stress, were again observed around openings. Clearly the bending stresses in the outer and the inner shells are below the yield stress.

A contour plot of shear stresses in the diaphragm is shown in Figure 5.4. The maximum and minimum stresses recorded were 188 and 164 MPa, respectively. The yield stress in shear being 205 MPa, the structure remains elastic at the applied load. Figure

5.5 contains an enlarged view of shear stresses around the opening. which is directly under the load. The stress concentrations around the opening are clearly visible in the figure.

The smoothness of the contours suggests that the mesh density is probably adequate for the purposes of this study.

6.0 CONCLUSIONS

Overall Assessment Para. 5.5

The midbody framing section of the 50000 DWT tanker as designed and analyzed meets the acceptance criteria. At the applied load, the structure remains predominantly elastic except in a very localized region around openings. The tendency towards an out-of-plane displacement in the diaphragm, between two stiffeners in the area of an opening, could result in instability at higher loads. This needs further investigation.

7.0 REFERENCES

1. PROPOSED EQUIVALENT STANDARDS FOR THE CONSTRUCTION OF ARCTIC CLASS SHIPS; Arctic Ship Safety, (AMNB) Canadian Coast Guard - Northern; Dated March 1993.

2. Arctic Tanker Structural Requirement Evaluation MIDSHIP SECTIONS, BOW SECTIONS AND REPAIR DRAWINGS; AA Shipping Company Limited; Dated June 1994.

3. LLOYD'S REGISTER - RULES FOR THE MANUFACTURE, TESTING, AND CERTIFICATION OF MATERIALS; Dated January 1993

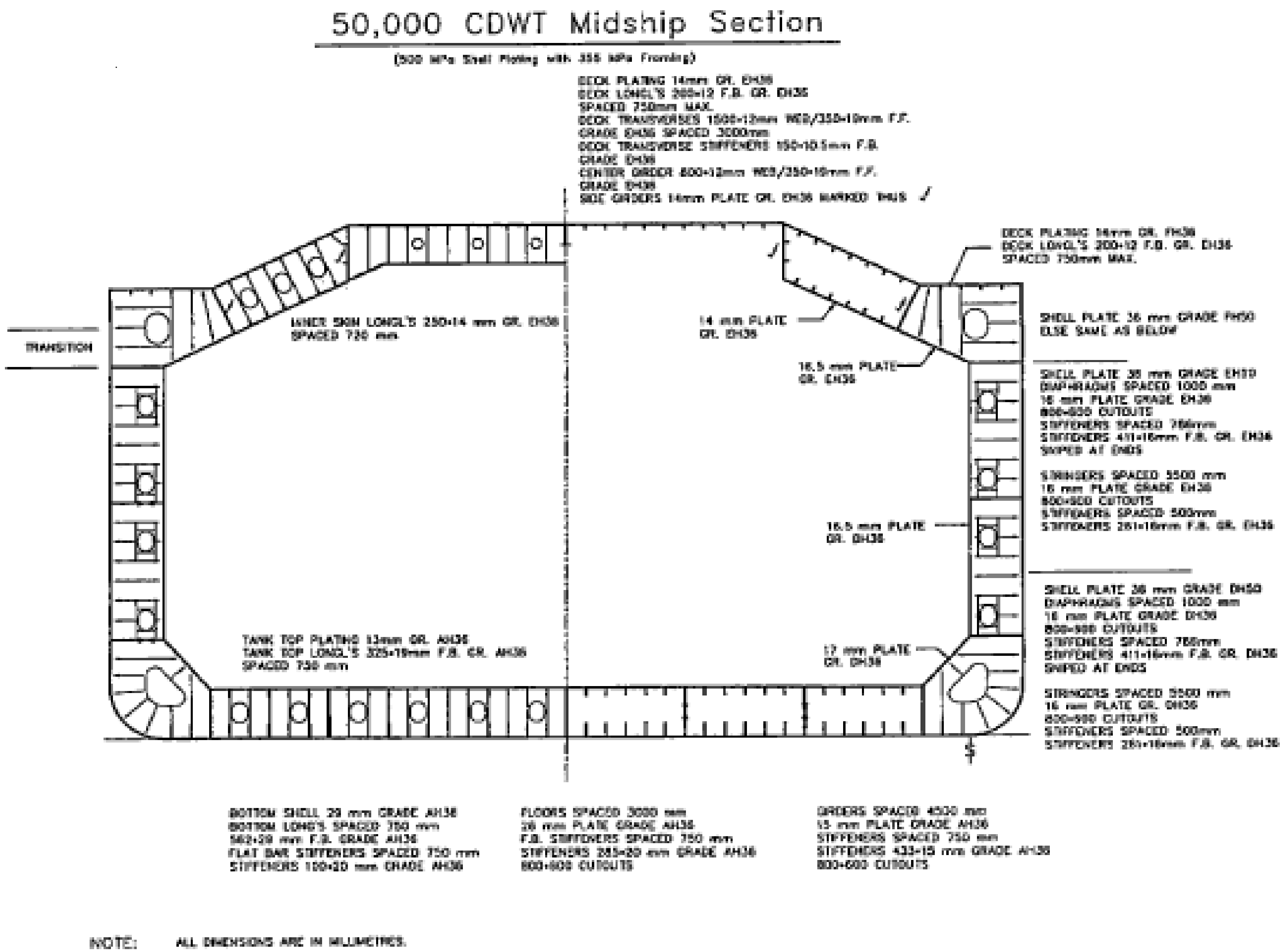


FIGURE 3.1 Midbody Web Frame

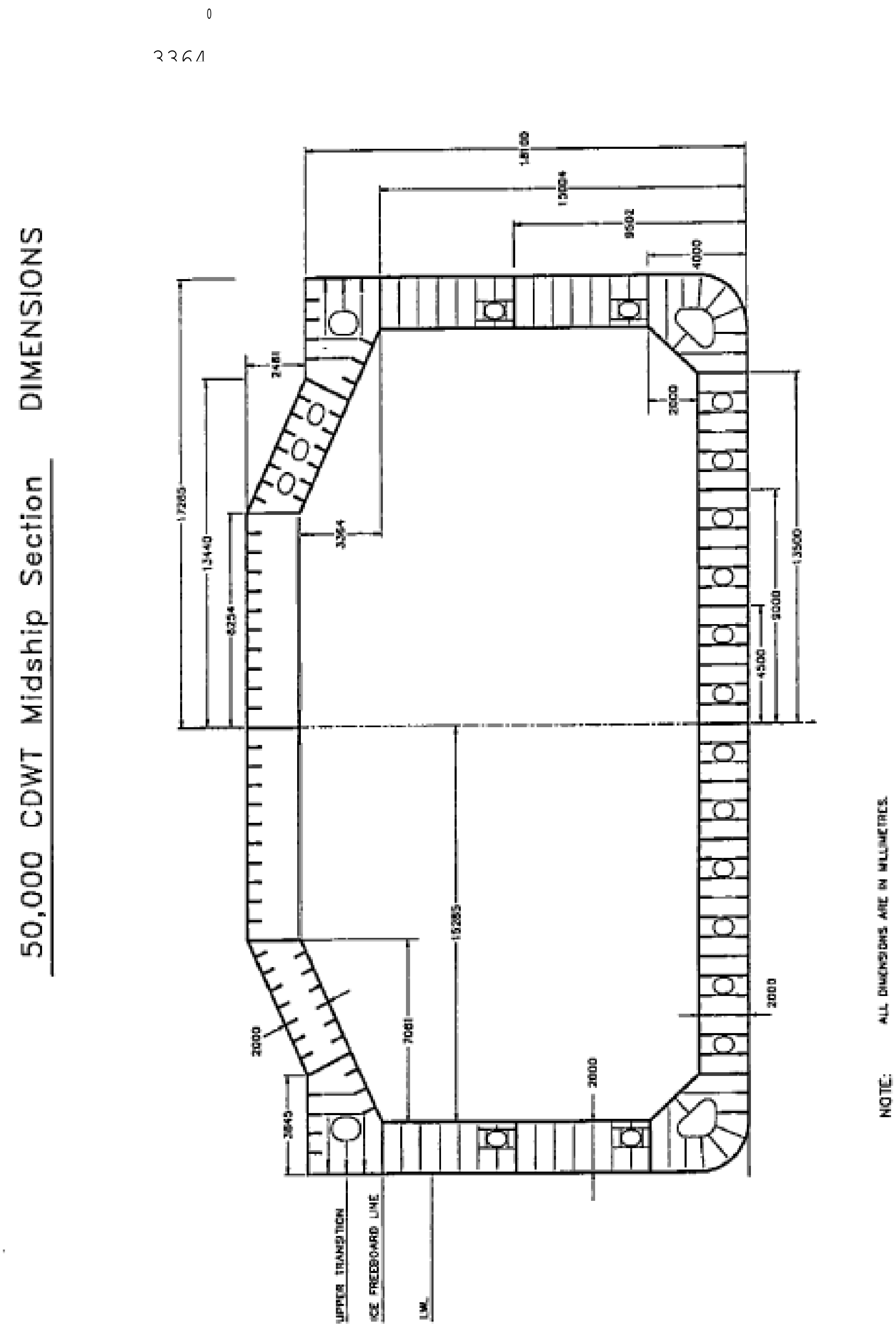


FIGURE 3.2 Outer Dimensions of Web Frame

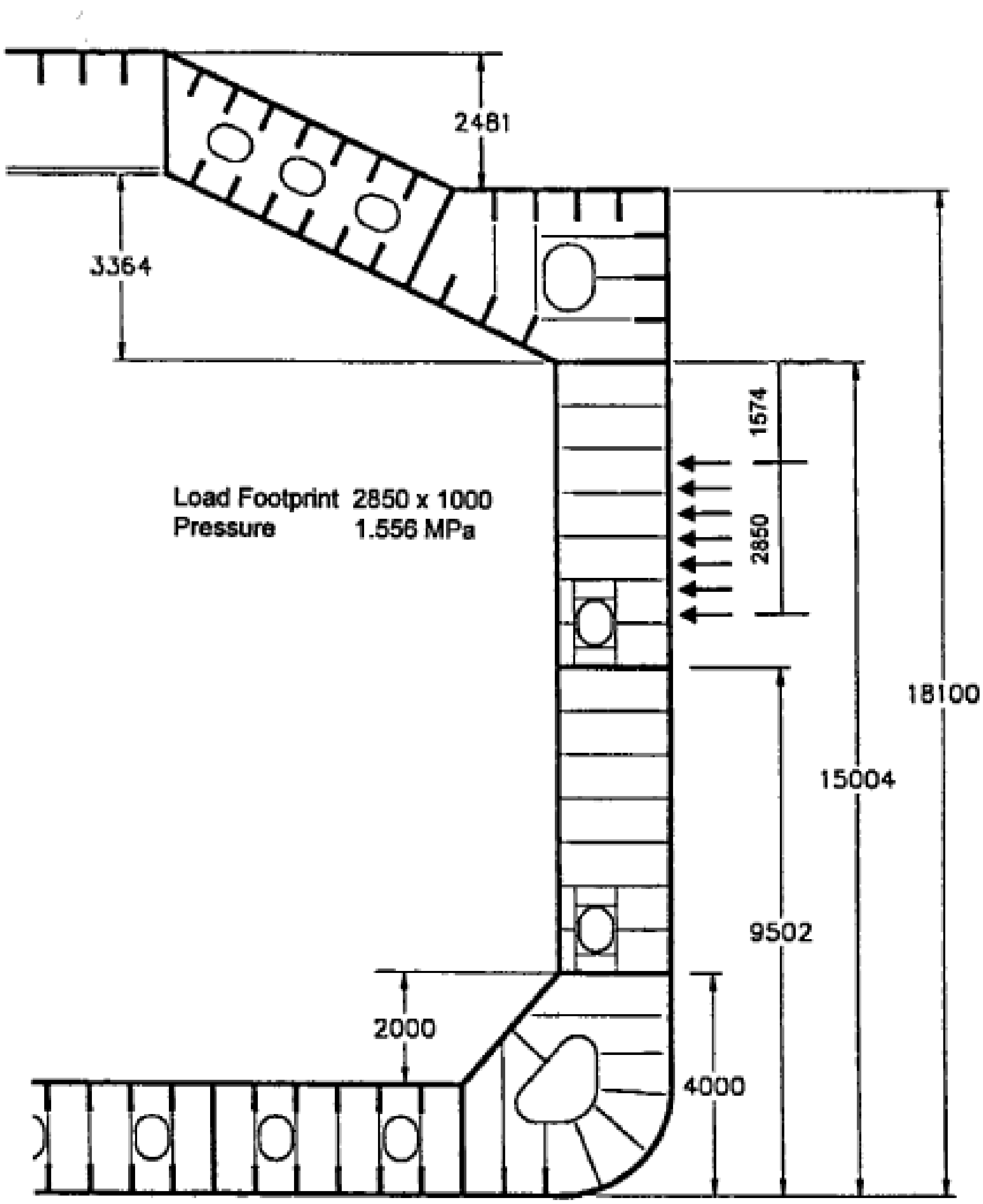


FIGURE 3.3 Characteristics of Load

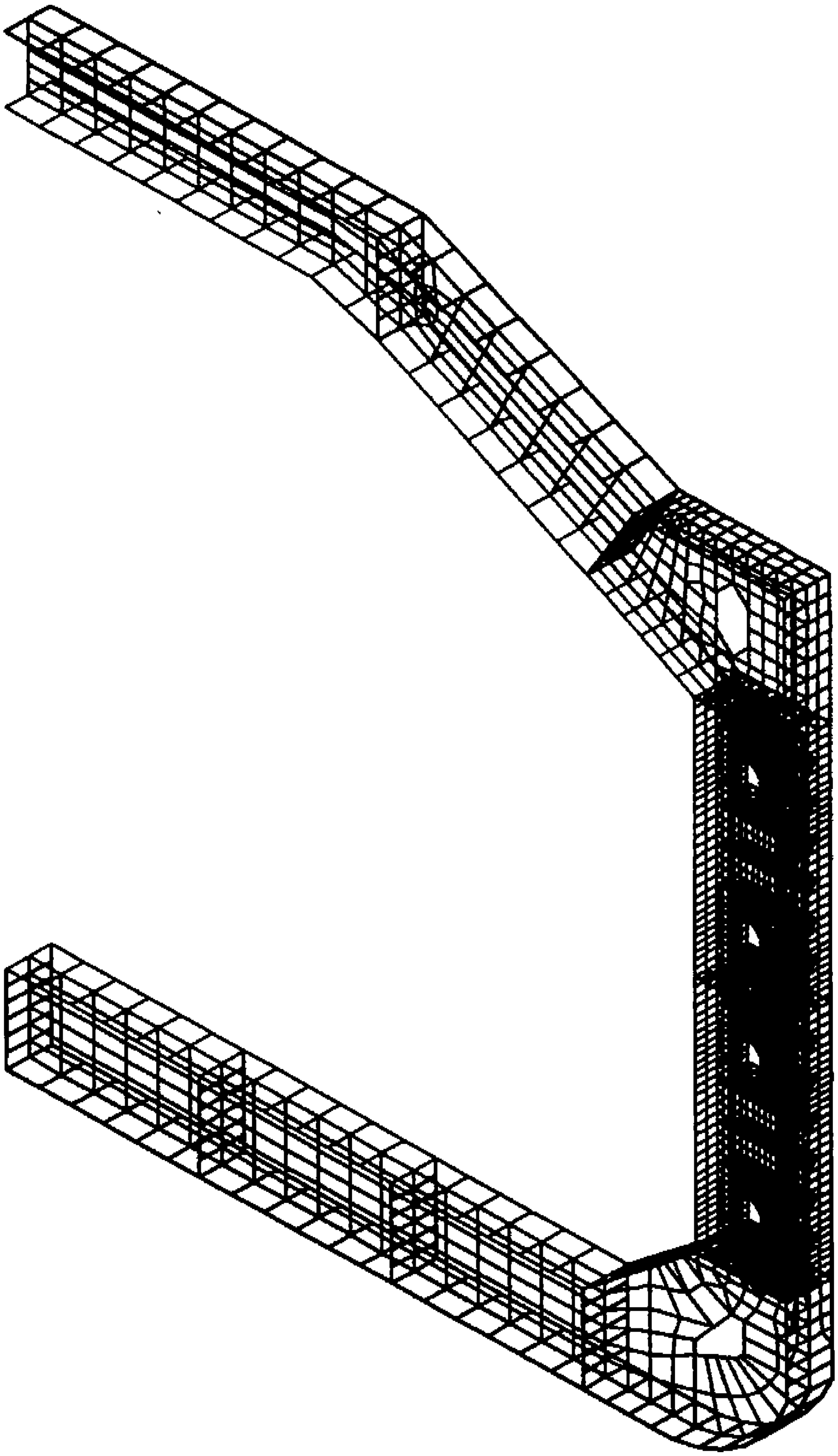


FIGURE 4.1 Finite Element Model of Web Frame

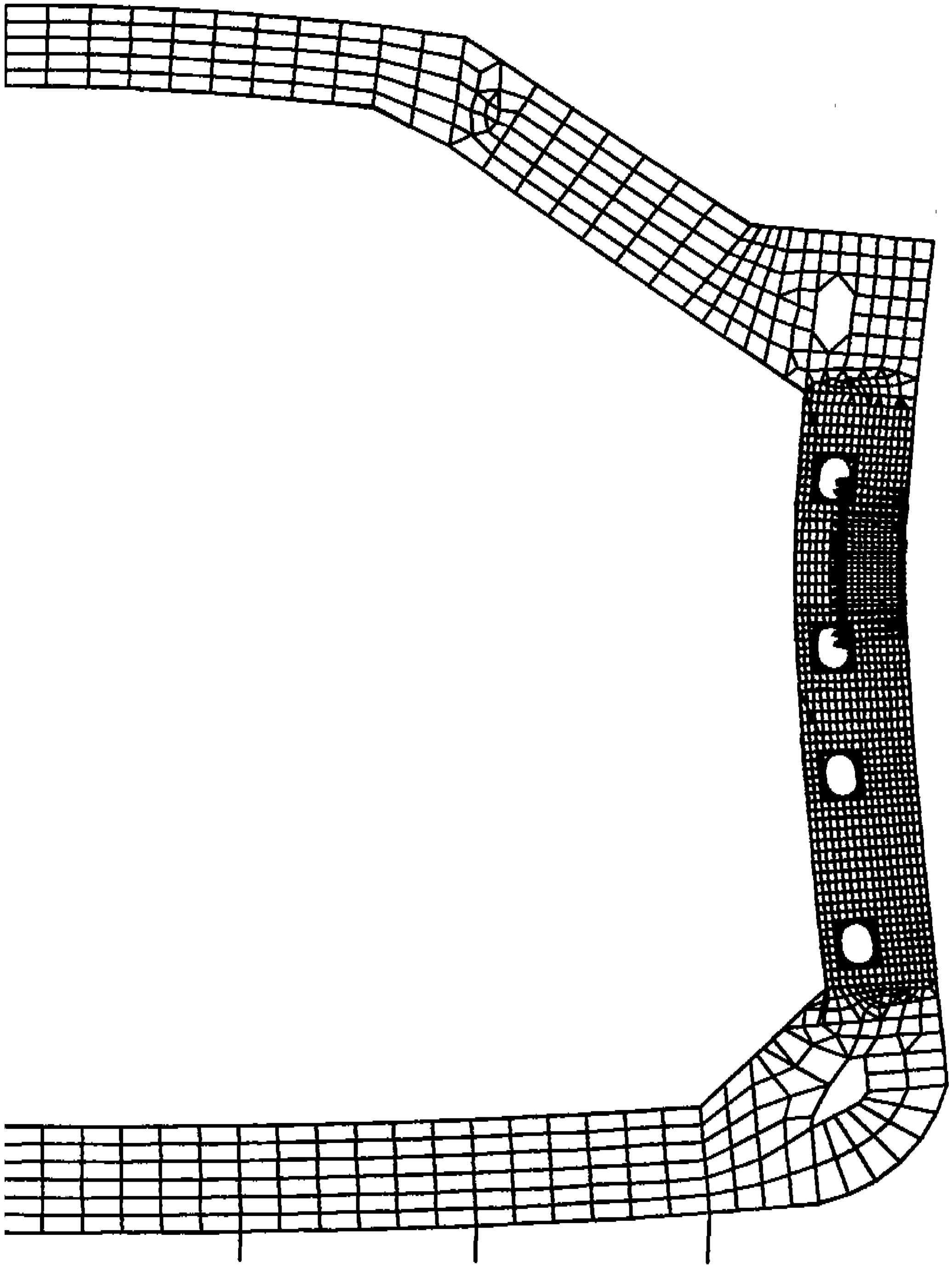


FIGURE 5.1 Deflected Shape of Web Frame

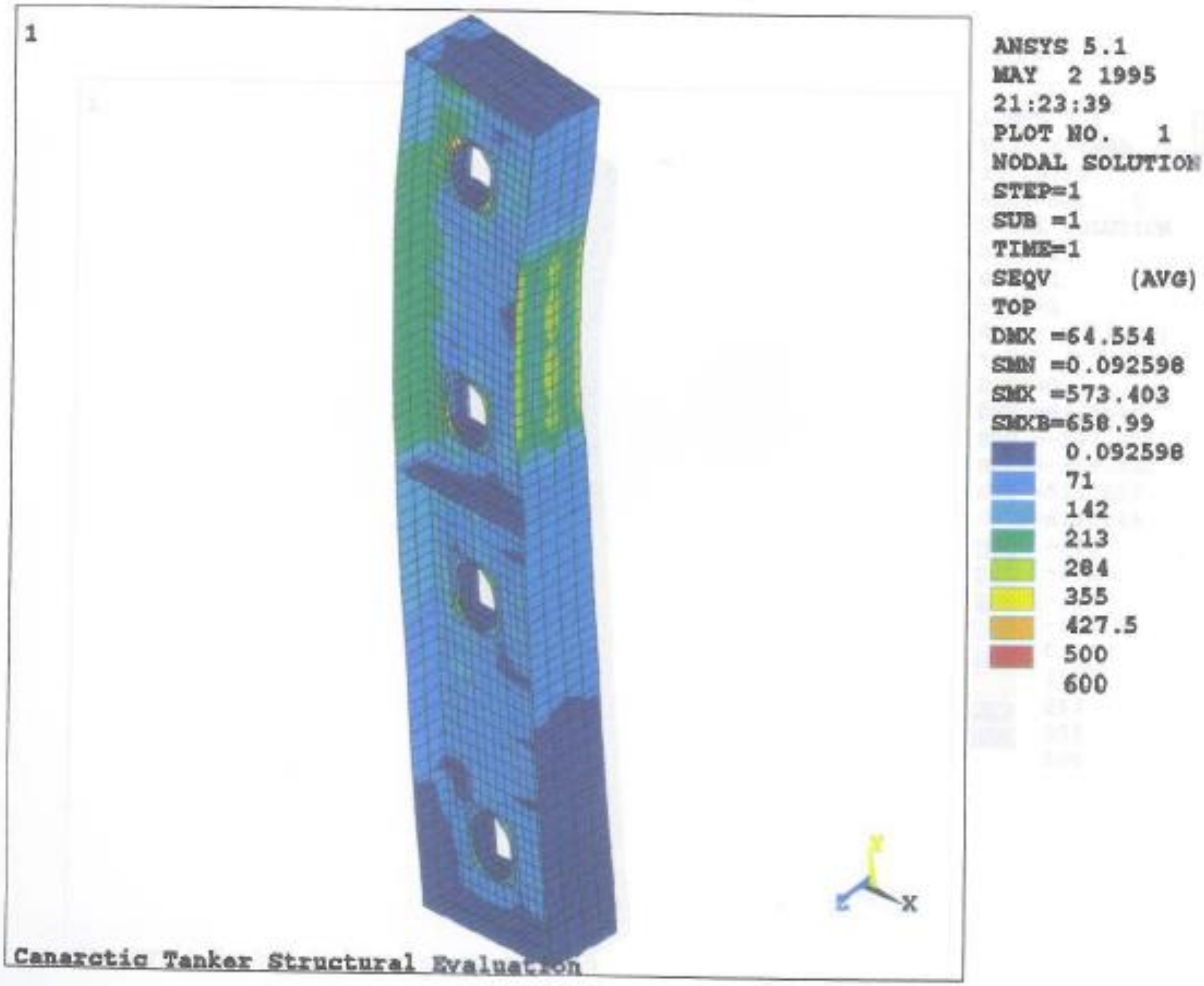


FIGURE 5.2 Von Mises Stress Plot

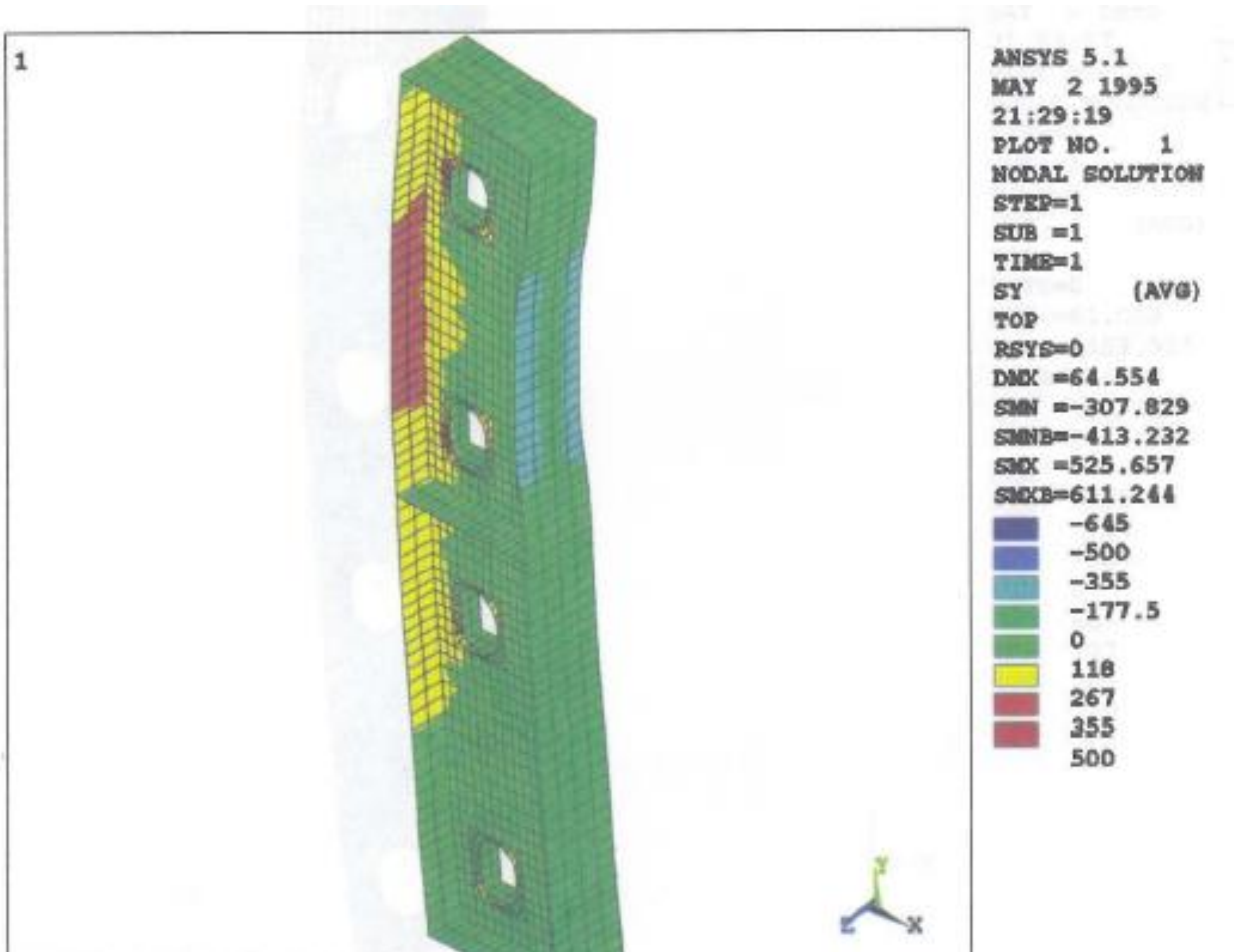


FIGURE 5.3 Bending Stresses

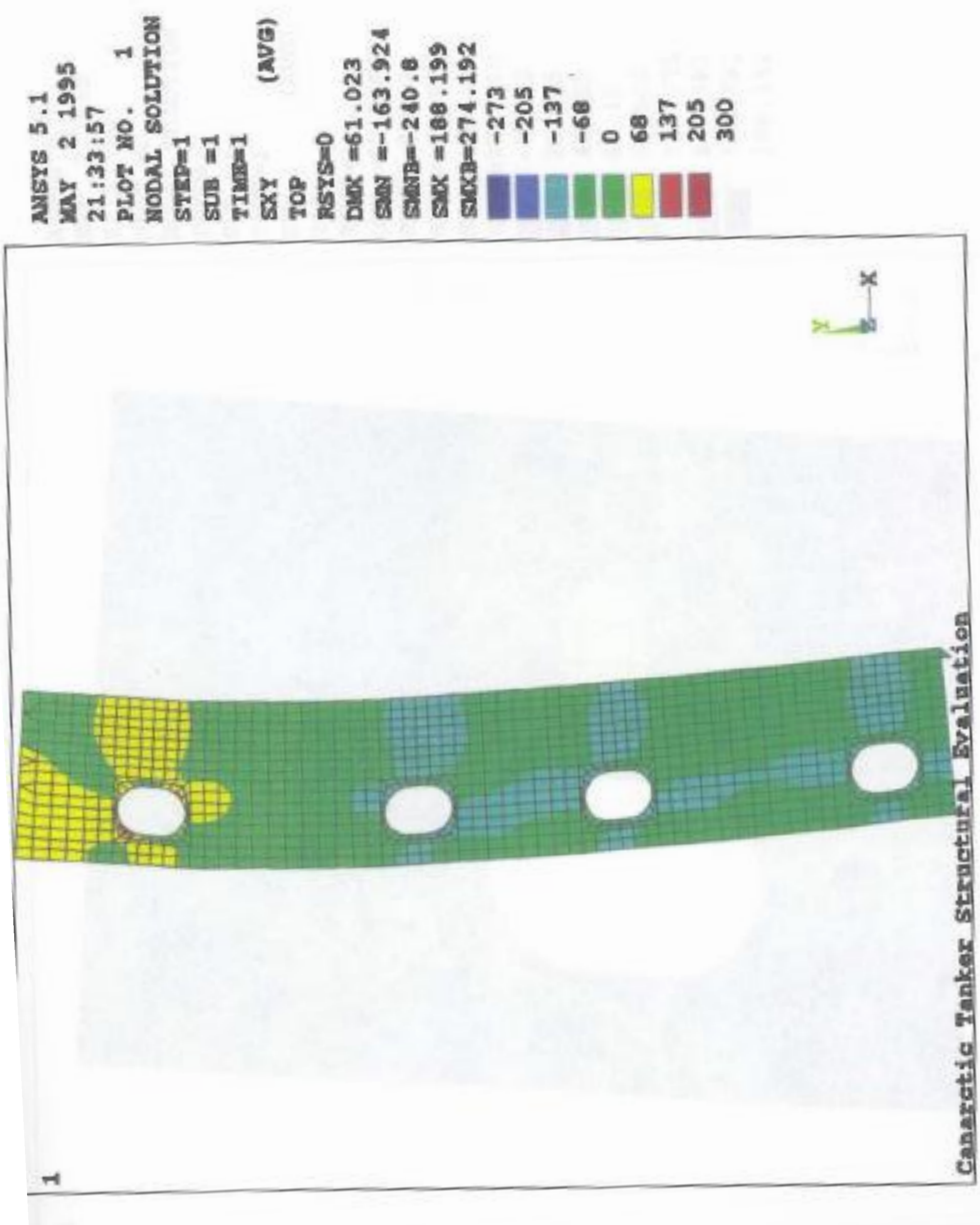


FIGURE 5.4 Shear Stresses in Diaphragm

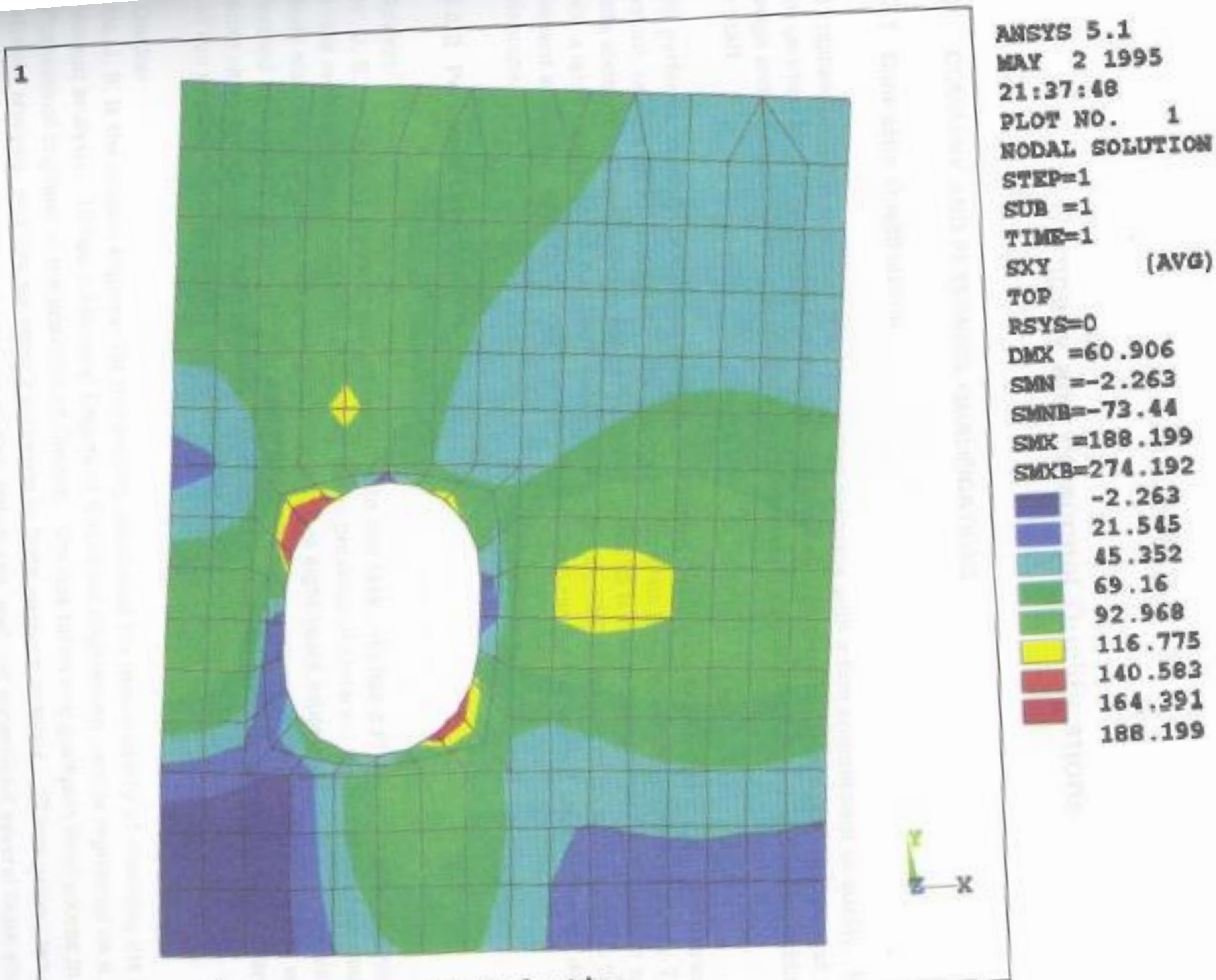


FIGURE 5.5 Shear Stresses Around Opening

Annex B-2
Company and Personnel Qualifications

B-2 COMPANY AND PERSONNEL QUALIFICATIONS

B-2.1 Contractor Qualifications

BB Engineering (BBE) is an ISO 9001 compliant company with a firm commitment to quality. It is also certified by the Association of Professional Engineers of Ontario. BBE's primary business is Ship Design and Analysis. It has several qualified professional structural engineers and naval architects on its staff.

BBE performs all its finite element analysis on either a Dec Station 5000, running on Ultrix operating system, or on a 60 MHz, 486 PC. For the current analysis the Dec Station 5000 was used. The finite element software used is called "ANSYS". ANSYS is a well-established finite element software with a large user base. It has been successfully used by BBE in several of its ship structure finite element analyses. ANSYS provides all the required features for the current task and hence deemed adequate.

B-2.2 Personnel Qualifications

Analyst

Mr. J. S. is the finite element analyst assigned to this task. He has a Ph.D. in Structural Engineering, and is registered as a Professional Engineer in the province of Ontario. He has taken two courses in finite element analysis at the graduate level and has eight years experience in using finite element method as an analysis tool. JS has a total of five years experience in using ANSYS, out of which three years are ship structure specific. Information on specific finite element analysis problems that JS has worked on in the past is available on request.

Checker

Ms. J. B. is the project engineer for this project and holds the responsibility of checking the finite element analysis. JB has a Masters' Degree in Structural Engineering and is registered as a Professional Engineer in the province of Ontario. She has taken one graduate level course in finite element analysis and has six years experience in finite element analysis. JB has gained ten years experience in the design and analysis of ship structures and has supervised several finite element analysis projects. JB has three years experience in using ANSYS. Information on projects that JB has worked in the past is available on request.

Annex B-3
FEA Results Verification

B-3 FEA RESULTS VERIFICATION

Accuracy Assessment Para. 5.4

The FEA results were compared with hand calculations. Two analyses have been performed as follows:

1. An elastic beam analysis of the frame with a span of 11000 millimeters, ends fixed, openings ignored, subjected to a uniformly distributed load of length 2850 millimeters equal to 3.112 MN/m (9.373*0.8*0.5*0.83), for a total load of 8869 kN.

   The structure has a bending stress of 550 MPa at the top support in the inner hull plating. Shear stresses in the portion of structure above the load are 195 MPa.

   This structure reached first yield (in bending) at a load of approximately 5700 kN.

2. An elastic frame analysis of the structure was FE modeled, except that the inner shell and bottom structure was analyzed with a flange width equal to 40 times the plate thickness and the frame was assumed to be fixed on centerline at the deck and at the bottom. In this analysis side sway of the frame was ignored. The bending moments calculated were within a few percents of those found in the first analysis.

   By comparison, the FEA predicts first yield of the inner hull plating at the top of the 11000 mm portion of the side shell framing at a load of approximately 4835 kN. This comparison suggests that the FEA results are broadly consistent with the results from the approximate simplified analyses.

Annex B-4

Sample Completed
Assessment Methodology Forms

EVALUATION OF FINITE ELEMENT MODELS AND RESULTS

Project#: *XXXX*

Project Title: *Finite Element Analysis of Arctic Tanker Web Frame*

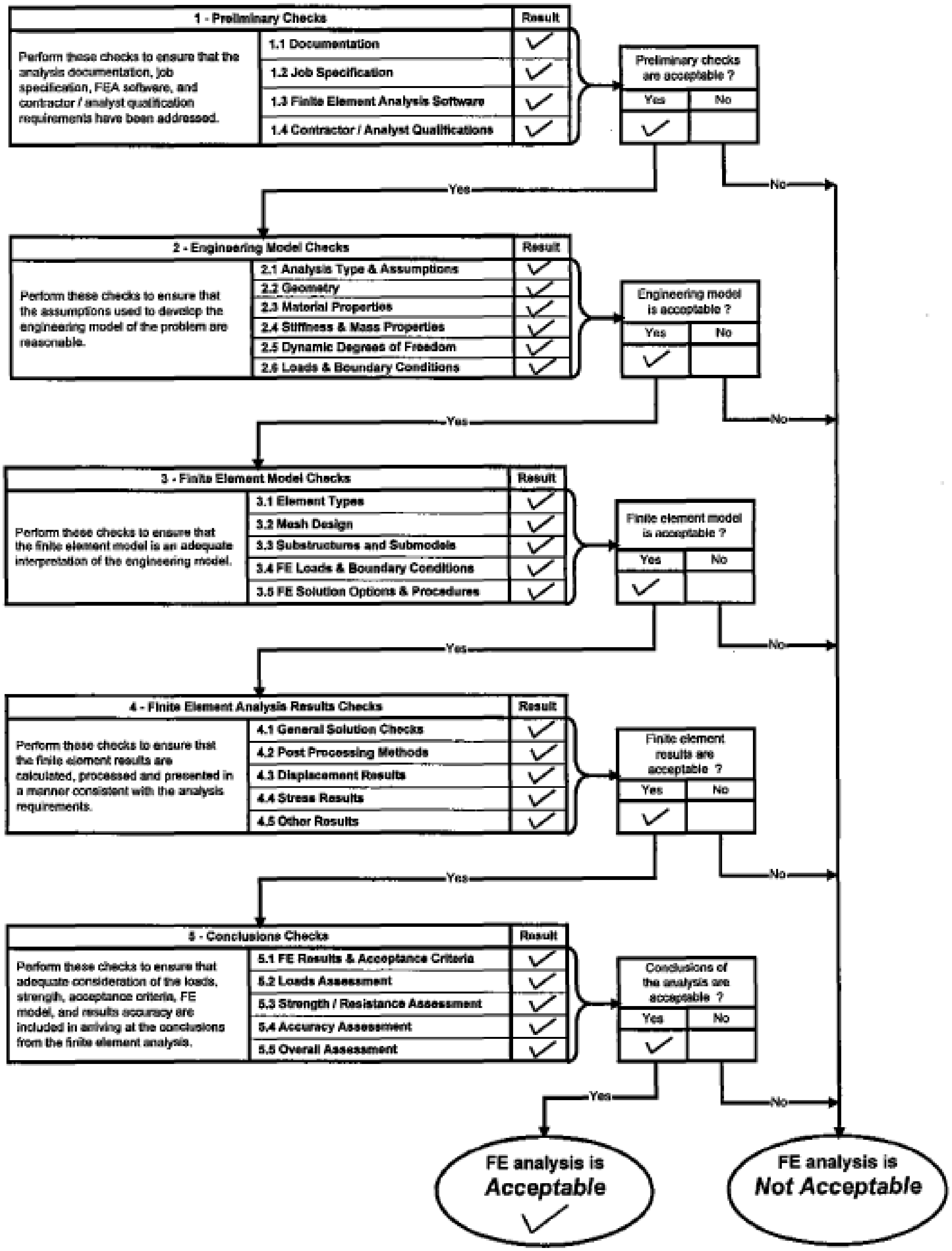
Project Description: *Linear. static analysis of web frame tp ensure the adequacy of frame ice load*

Contractor: *BB Engineering ltd.*

Result of Evaluation: *Generally satisfactory. Final approval subject to the supply of data on some details of the model*

Evaluator: *John Doe*

Date: *May* 1995


1 - Preliminary Checks
Result
Perform these checks to ensure that the analysis documentation, job specification, FEA software, and contractor / analyst qualification requirements have been addressed.
1.1 Documentation
1.2 Job Specification
1.3 Finite Element Analysis Software
1.4 Contractor / Analyst Qualifications
Preliminary checks are acceptable ?
Yes
No
Yes
No
2 - Engineering Model Checks
Result
Perform these checks to ensure that the assumptions used to develop the engineering model of the problem are reasonable.
2.1 Analysis Type & Assumptions
2.2 Geometry
2.3 Material Properties
2.4 Stiffness & Mass Properties
2.5 Dynamic Degrees of Freedom
2.6 Loads & Boundary Conditions
Engineering model is acceptable ?
Yes
No
Yes
No
3 - Finite Element Model Checks
Result
Perform these checks to ensure that the finite element model is an adequate interpretation of the engineering model.
3.1 Element Types
3.2 Mesh Design
3.3 Substructures and Submodels
3.4 FE Loads & Boundary Conditions
3.5 FE Solution Options & Procedures
Finite element model is acceptable ?
Yes
No
Yes
No
4 - Finite Element Analysis Results Checks
Result
Perform these checks to ensure that the finite element results are calculated, processed and presented in a manner consistent with the analysis requirements.
4.1 General Solution Checks
4.2 Post Processing Methods
4.3 Displacement Results
4.4 Stress Results
4.5 Other Results
Finite element results are acceptable ?
Yes
No
Yes
No
5 - Conclusions Checks
Result
Perform these checks to ensure that adequate consideration of the loads, strength, acceptance criteria, FE model, and results accuracy are included in arriving at the conclusions from the finite element analysis.
5.1 FE Results & Acceptance Criteria
5.2 Loads Assessment
5.3 Strength / Resistance Assessment
5.4 Accuracy Assessment
5.5 Overall Assessment
Conclusions of the analysis are acceptable ?
Yes
No
Yes
No
FE analysis is *Acceptable*
FE analysis is *Not Acceptable*

FINITE ELEMENT **ANALYSIS ASSESSMENT** PRELIMINARY CHECKS	
Project No. *xxxx*	I Project Title : *FEA of Arctic Tanker Web Frame*
Company Name: *BB Engineering Ltd*	I Date: *May 1995*
Analyst: *JS*	IChecker: ***JS***

Documentation Requirements

Finite **Element Analysis Assessment Check**	**Refer to Guideline Section**	**Result**	**Comments**
1.1.1 Has the following information been provided in the FEA documentation?	3.1.1		
a) Objectives and scope of the analysis.		✓	
bl Analysis requirements and acceptance criteria.		✓	
cl FEA software used.		✓	
d) Description of physical problem.		✓	
e) Description of engineering model.		✓	
f) Type of analysis.		✓	
g) System of units.		✓	
h) Coordinate axis systems.		✓	
i) Description of FEA model.		✓	
j) Plots of full FEA model and local details.		✓	*Some detail missing**
k) Element types and degrees of freedom per node.		✓	
I) Material properties.		✓	
m) Element properties (stiffness & mass properties).		✓	
n) FE loads and boundary conditions.		✓	
o) Description and presentation of the FEA results.		✓	
pl Assessment of accuracy of the FEA results.		✓	
q) Conclusions of the analysis.		✓	
r) List of references.		✓	

Based on the above checks answer Question 1. 1 and enter result in Figure 1.O.	***Result***
1.1 Is the level of documentation sufficient to perform an assessment of the FEA?	☐

Comments

**Request additional detail on stiffener/web connection*

Job Specification Requirements

Finite **Element Assessment Check**	**Refer To Guideline Section**	**Result**	**Comments**
1.2.1 Is the job specification identified and referenced in the analysis documentation?	3-1.2	✓	
1.2.2 Are the objectives and scope of the analysis clearly stated and are they consistent with those of the job specification?	3-1.2	✓	
1.2.3 Are the analysis requirements clearly stated and are they consistent with those of the job specification?	3-1.2	✓	
1.2.4 If certain requirements of the job specification have not been addressed (such as certain load cases), has adequate justification .been given?	3-1.2	*N/A*	
1.2.5 Are the design / acceptance criteria clearly stated and are they consistent with those -of the job specification?	3-1.2	✓	
1.2.6 Is there reasonable justification for using FEA for this problem?	3-1.2	✓	
1.2.7 Has advantage been taken of any previous experimental, analytical, or numerical works that are relevant to this problem?	3-1.2	*N/A*	

Based on the above checks answer Question 1.2 and enter result in Figure 1.0.	***Result***
1.2 Does the analysis address the job specification requirements?	✓
Comments	

Finite Element Analysis Software Requirements

Finite Element **Analysis Assessment Check**	**Refer To Guideline Section**	**Result**	**Comments**
1.3.1 Is the FEA software on the list of approved programs for ship structural analysis applications?	**3-1.3**	✓	
If the answer to Check 1.3.1 is "Y", you may skip Checks 1.3.2 and 1.3.3.			
1.3.2 Are the capabilities and limitations of the FEA software used to perform the required analysis stated in the analysis documentation?	**3-1.4**	✓	
1.3.3 Is evidence of this capability documented and available for review (eg. verification manual, results of ship structure FEA benchmark tests, previous approved FEA of similar problems)?	3-1.3	✓	
1.3.4 Does the vendor of the FEA software have a quality system to ensure that appropriate standards are maintained in software development and maintenance.		✓	

Based on the above checks answer Question 1.3 and enter result in Figure 1.0.	***Result***
1.3 Is the FEA software qualified to perform the required analysis?	✓
Comments	

1.4 Contractor / Personnel Qualification Requirements

Finite Element Assessment Check	Refer To Guideline Section	Result	Comments
1.4.1 Do the contractor personnel have adequate academic training and experience qualifications to perform finite element analysis?	**3-1.5**	✓	
1.4.2 Do the contractor personnel have adequate engineering experience qualifications for performing ship structural design or analysis?	3-1.5	✓	
1.4.3 Do the contractor and contractor personnel have adequate professional certification qualifications?	3-1.5	✓	
1.4.4 Does the contractor have a working system of Quality Assurance (QA) procedures and checks that are satisfactory for the requirement?	3-1.5	*X*	*Not documented but using well established software*
1.4.5 Do the contractor personnel have adequate experience with the FEA software used for the analysis?	3-1.5	✓	

Based on the above checks answer Question 1.4 and enter result in Figure 1.0.

		Result
1.4	Is the contractor adequately qualified for performing ship structure FEA?	✓

Comments

FINITE ELEMENT ANALYSIS ASSESSMENT	ENGINEERING MODEL CHECKS
Project No. *XXXX*	Project Title : *FEA of Arctic Tanker Web Frame*
Company Name: *BB Engineering Ltd*	Date: *May 1995*
Analyst: ***JS***	Checker: ***JB***

2.1 Analysis Type and Assumptions

Finite Element **Analysis Assessment Check**	**Refer To Guideline Section**	**Result**	**Comments**
2.1.1 Does the engineering model employ enough dimensions and freedoms to describe the structural behavior (eg. 1-0, 2-D, or 3-0)?	3-2.1	✓	
2.1.2 Does the engineering model address the appropriate scale of response for the problem (eg. global, intermediate, or local response)?	3-2.1	✓	
2.1.3 Is the type of analysis appropriate for the type of response and loading of interest (eg. linear, static, dynamic, buckling analysis)?	3-2.1	✓	
2.1.4 Does the engineering model address all the required results parameters (eg. stress, displacement, frequency, buckling load)?	3-2.1	✓	
2.1.5 Are all assumptions affecting the choice of engineering model and analysis type justified (watch for non-standard assumptions)?	3-2.1	✓	
2.1.6 Is the level of detail, accuracy or conservatism of the engineering model appropriate for the criticality of the analysis and type of problem?	3-2.1		*Appears marginal - may require more data on results to complete evaluation*
2.1.7 Does the analysis employ a consistent set of units?	3-2.1	✓	
2.1.8 Does the analysis employ a consistent global coordinate axis system?	3-2.1	✓	

Based on the above checks answer Question 2. 1 and enter result in Figure 1.0.

		Result
2.1	Are the assumptions of the type of analysis and engineering model acceptable?	✓

Comments

See above

Appendix C
Examples of Variations in FEA Modeling Practices and Results

INTRODUCTION

The purpose of this Appendix is to illustrate the effect of varying certain FEA modeling parameters on the results using typical ship structure example problems.

Three typical ship structure examples are used. The first example, presented in Section C1, concerns the modeling of stiffened panels. Four different approaches for modeling stiffened panels are considered and the results presented. In the second example, presented in Section C2, the modeling of stress concentrations arising from openings in a deck structure is considered. In the third example, presented in Section C3, variations in the approach to modeling a truss type mast structure are illustrated. A brief introduction is provided for each problem, followed by a pictorial overview of the FEA model and results. A brief discussion of the results is provided at the end of each example.

It is not the intention of this Appendix to endorse any particular modeling method. Rather, it represents an effort to illustrate various modeling practices and present the variations in results. This should provide some insight into the consequences of adopting a particular modeling approach. The choice of the appropriate method, for a given problem, depends on the purpose and objectives of the FEA.

In all cases the ANSYS program was used. The following element types were used:

- four-node membrane shell elements
- four-node shell elements with bending capabilities
- eight-node shell elements with bending capabilities
- two-node 3-D beam elements
- two-node 3-D truss elements
- mass elements

In certain cases, converged solutions are referred to. These solutions result from very fine mesh models which are known to have converged (by comparison with less fine mesh models).

C1.0 Stiffened Panel

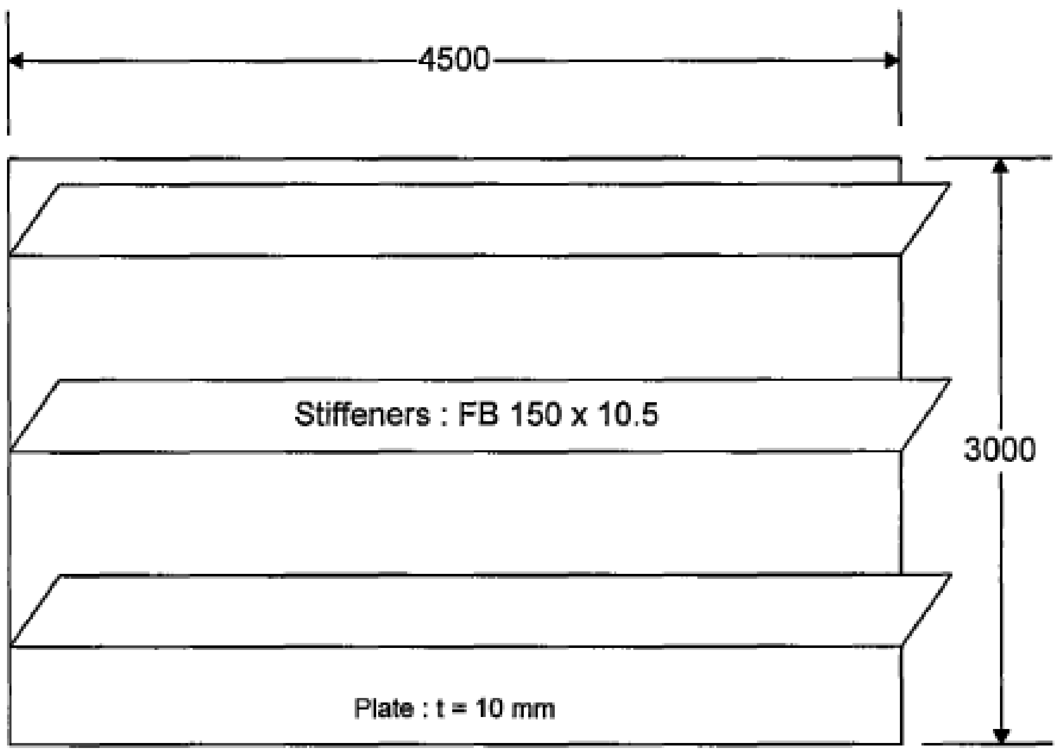
The majority of the structural weight in conventional ship structures is stiffened panels that comprise the shell, decks, bulkheads, and superstructure. The panels are stiffened with structural sections that are usually spaced in a regular fashion. The appropriate modeling approach for stiffened panels depends on both the scale of the response (i.e., local or global response) and the main structural actions of interest. Two main structural actions typically modeled are 1) bending action due to loading normal to the panel surface, and 2) membrane action due to loading in the plane of the panel. The first part of this section deals with bending action and hence focusses on stiffened plate subjected to transverse loading. Membrane action in a stiffened plate as a result of in-plane loads is briefly examined in the second part.

FEA Example No. 1	**Title :** Stiffened Panel - Transverse Loading

Problem Description:

There are various techniques available for modelling stiffened panels. The choice of a particular technique depends on the purpose of the analysis. Using a simple stiffened panel structure, the differences in the accuracy of stress and deflection results for some of these techniques are examined.

Engineering Model :



Material Properties :	**Geometric Properties :**	**Loading :**
$E = 207 \times 10^3$ MPa $v = 0.3$	Plate t = 10 mm Stiffeners 150 x 10.5 FB	$P_z = 15000$ Pa

Modelling Features : Four modelling approaches are considered:

1. Modelling stiffeners with off-set beams (beam properties defined at beam centroid which is rigidly off-set from plane of plate);
2. Modelling stiffeners with in-plane beams (beam properties includes an effective width of plating and are defined at beam centroid which is in the plane of the plate);
3. Explicit modelling of stiffeners using shell elements; and
4. Modelling the plate with orthotropic material properties (in-plane loads / membrane action only)

FEA Example No. 1	Title : Stiffened Panel - Transverse Loading

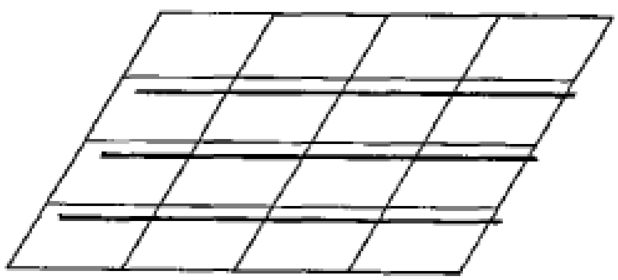
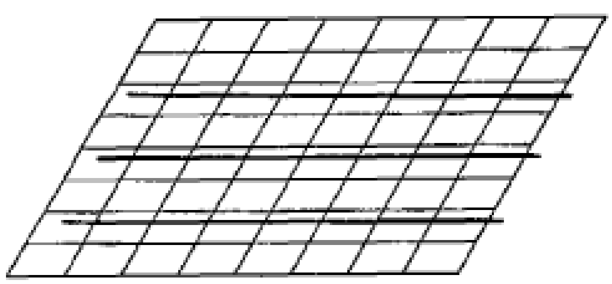
Finite Element Models :

A total of 12 FE models, grouped into four sets, were studied. Each set contained three models representing the three modelling techniques. The mesh and element types are as follows :

Set 1	4x4 element mesh; 4 noded elements
Set 2	8x8 element mesh; 4 noded elements
Set 3	16x16 element mesh; 4 noded elements
Set 4	16x16 element mesh; 8 noded elements

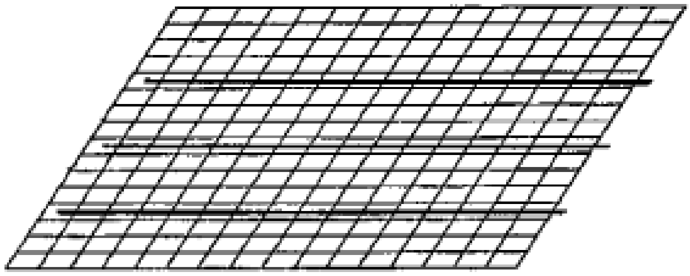
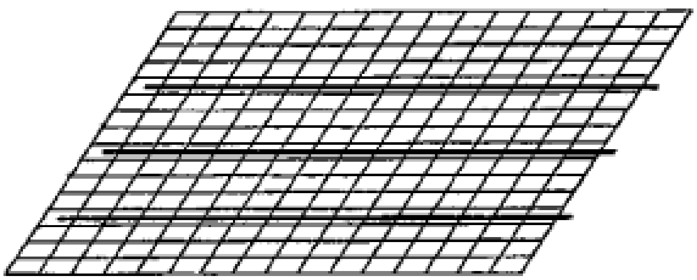
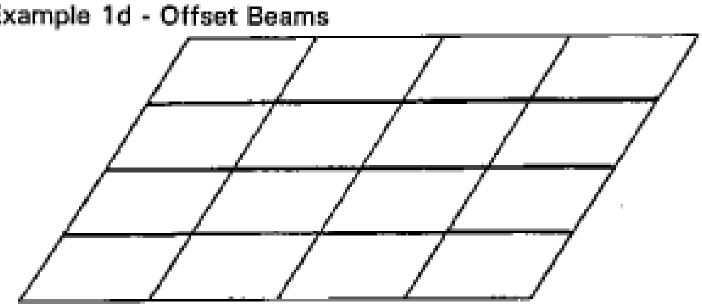
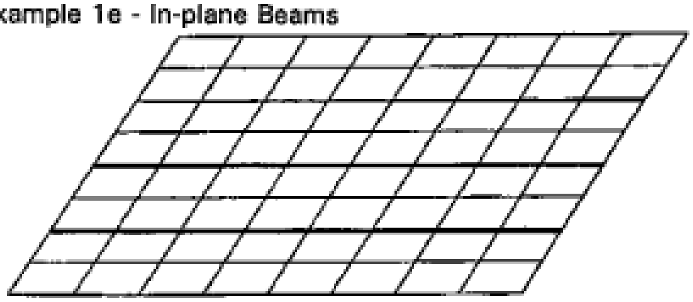
All models are fully fixed along the four edges. A uniform transverse pressure load of 15 kN/m^2 is applied.

For the in-plane beam models the effective width of plating was assumed to be 40t, where t is the thickness of the plate. The inertia properties of the beam were calculated based on stiffener and an effective width of plating. However, for the area, the area of the stiffener alone was input.

Model	Nodes	Elements	Degrees of freedom
 Example 1a - Offset Beams	25	28	150
 Example 1b - Offset Beams	81	88	486

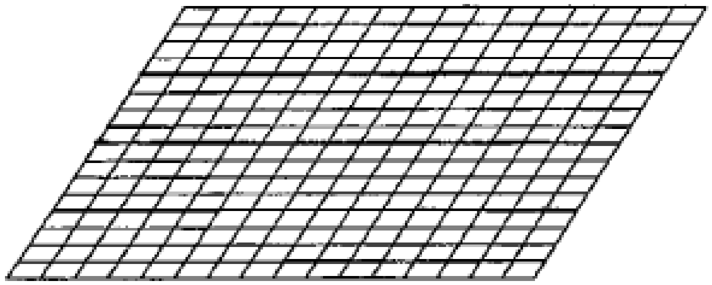
FEA Example No. 1	Title : Stiffened Panel - Transverse Loading

Finite Element Models :

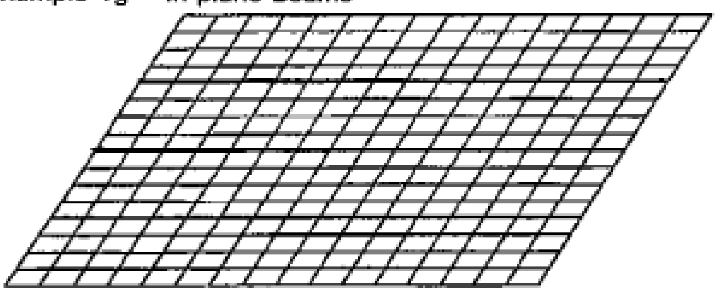
Model	Nodes	Elements	Degrees of freedom
 Example 1c - Offset Beams	289	304	1734
 Example 1d - Offset Beams	833	352	4230
 Example 1e - In-plane Beams	25	28	150
 Example 1f - In-plane Beams	81	88	486

FEA Example No. 1	Title : Stiffened Panel - Transverse Loading

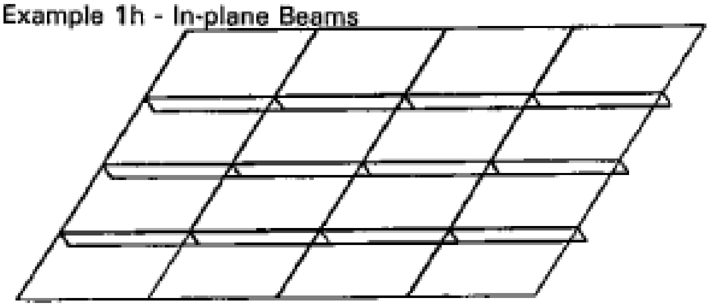
Finite Element Models :



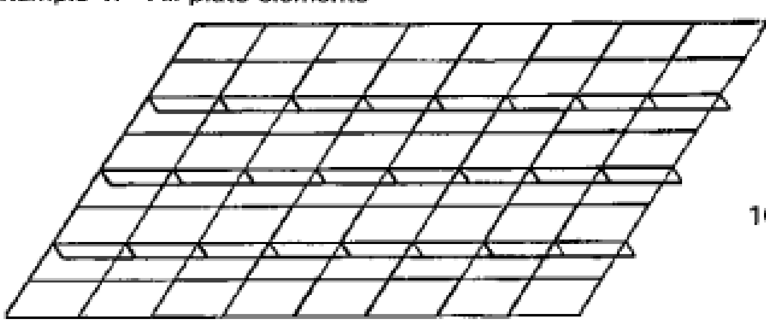
Example 1g - In-plane Beams



Example 1h - In-plane Beams



Example 1i - All plate elements

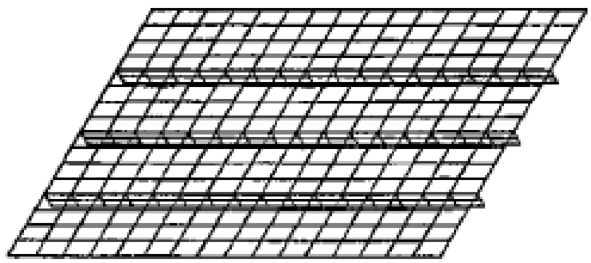
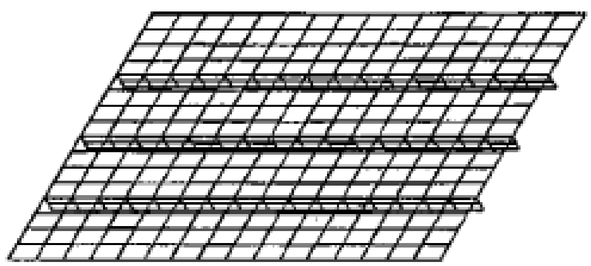


Example 1j - All plate elements

Model	Nodes	Elements	Degrees of freedom
Example 1g	289	304	1734
Example 1h	833	352	4230
Example 1i	40	28	240
Example 1j	108	88	648

FEA Example No. 1	**Title :** Stiffened Panel - Transverse Loading

Finite Element Models :

	Nodes	Elements	Degrees of freedom
 Example 1k - All plate elements	391	352	2346
 Example 1l - All plate elements	1133	352	5886

FEA Example No. 1	Title : Stiffened Panel - Transverse Loading

DISCUSSION OF RESULTS

Key results are summarized in Table C1.1. The maximum vertical deflection is at the centre of the panel (see Figure C1.1). The peak stresses reported in the table are at the ends of the central stiffener (at supports) . The three mode shapes associated with the three frequencies are shown in Figure C1.2. Figure C1.3 shows the longitudinal stress contours for the plate and the stiffeners.

Figure C1.4 summarizes the deflection results for all twelve models. From Figure C1.4 it is evident that the deflection solution starts to converge for an 8x8 mesh. Figure C1.4 also shows the stress results in the stiffener. Some general observations for the three modelling types are :

<u>In-Plane Beams:</u> Despite the approximation of 40t as the effective width of plating this method seems to provide the most economical solution for deflection prediction. The same is true even for stress prediction.

<u>Offset Beams:</u> Deflection decreases with mesh refinement contrary to the expectation that displacement-based FEA model becomes more flexible with more elements. This is probably due to the presence of a spurious moment generated at the ends of the stiffener as a result of two axial forces (in the plate and in the beam) being offset. However, with mesh refinement this effect tends to diminish resulting in reasonable predictions of deflections.

<u>All Plate Elements:</u> In this case the performance approaches that of the in-plane beam models with an 8x8 mesh.

All three techniques predict natural frequencies and mode shapes fairly well.

In modelling stiffeners as in-plane beams, the greatest uncertainty is the choice for the effective breadth of plating. The most important parameter which determines effective breadth of plating is the ratio of actual flange width to the length between points of zero bending moment. The effective breadth of plating can be estimated from charts (see, for example, Hughes[1]). Another important aspect to note with this technique is that the effective breadth thus used is only effective at the location of maximum bending moment. However, for design purposes the stresses at the section of maximum bending moment is of most importance.

In conclusion, the approach recommended will depend on the nature of the analysis. If the plate-stiffener combination is subjected to transverse loading, modelling stiffeners with in-plane beams provides the most economical approach in terms of overall stiffness, and stresses in the stiffener at the location of maximum bending moment. When more detailed stress information is required then the explicit modelling of the stiffener with plate elements appears most appropriate. The use of the offset beam is attractive since there is no approximation required for effective breadth. With a reasonable mesh density (at least 3 elements between stiffeners) this technique should provide reasonable prediction of the overall stiffness of the structure.

TABLE C1.1 Stiffened Panel FEA - Results

Modelling of stiffener	Offset beams	In-plane beams	Plate elements
SET 1 : 4 x 4 Mesh	**1a**	**1e**	**1i**
Max. Vertical Deflection (mm)	9.51	5.95	4.48
Max. stress in plate (MPa)	32.87	45.20	16.11
Max. bending stress in stiffener at ends (MPa)	-379.90	-246.40	-98.31
	289.30	45.20	5.59
First three natural frequencies (Hz)	24.94	30.89	30.02
	29.12	34.00	33.93
	38.34	43.54	35.24
SET 2 : 8 x 8 Mesh	**1b**	**1f**	**1j**
Max. Vertical Deflection (mm)	7.70	6.86	6.64
Max. stress in plate (MPa)	33.87	47.69	24.12
Max. bending stress in stiffener at ends (MPa)	-339.20	-259.95	-175.58
	181.80	47.69	15.81
First three natural frequencies (Hz)	28.11	29.71	30.50
	31.89	32.40	33.93
	43.33	43.96	45.60
SET 3 : 16 x 16 Mesh	**1c**	**1g**	**1k**
Max. Vertical Deflection (mm)	6.90	6.69	6.80
Max. stress in plate (MPa)	38.96	48.22	33.15
Max. bending stress in stiffener at ends (MPa)	-307.50	-262.88	-226.17
	112.98	48.22	26.02
First three natural frequencies (Hz)	29.59	29.87	29.84
	33.31	32.60	33.51
	45.29	44.64	45.55
SET 4 : 16 x 16 Mesh (8 node)	**1d**	**1h**	**1l**
Max. Vertical Deflection (mm)	6.70	6.65	6.88
Max. stress in plate (MPa)	47.26	48.47	50.55
Max. bending stress in stiffener at ends (MPa)	-289.67	-264.25	-287.29
	75.37	48.47	41.42
First three natural frequencies (Hz)	30.02	29.94	29.58
	33.73	32.70	33.35
	45.95	44.93	45.53

FEA Example No. 1	Title : Stiffened Panel - Transverse Loading

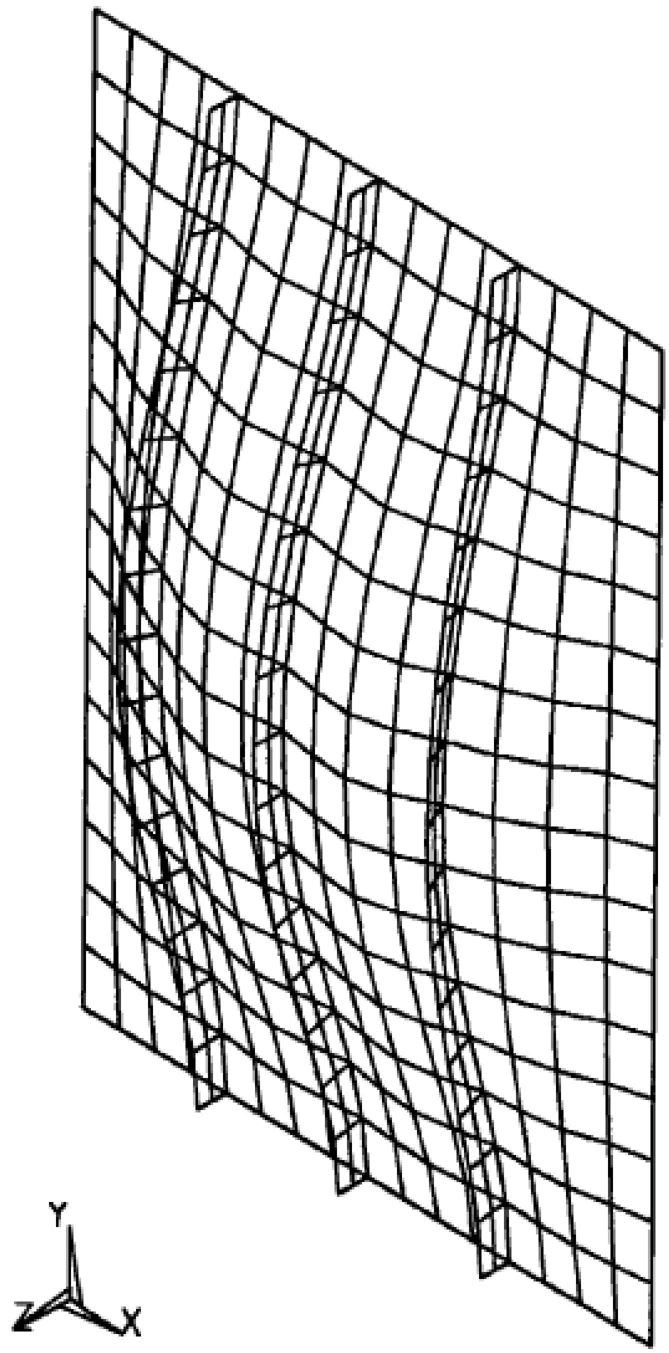
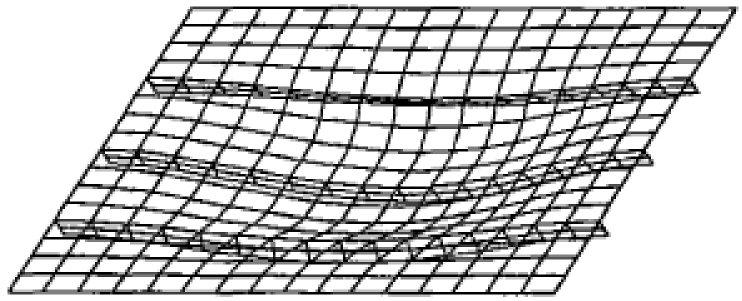
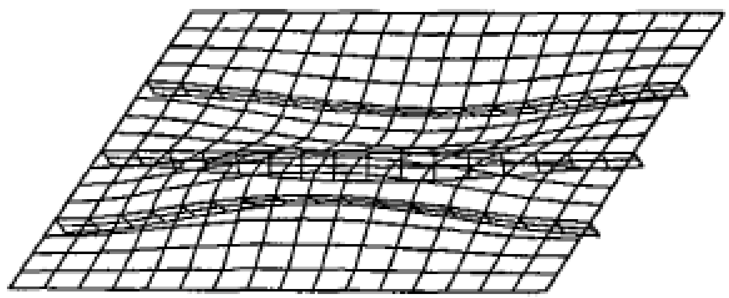


Figure C1.1 Deflected Shape

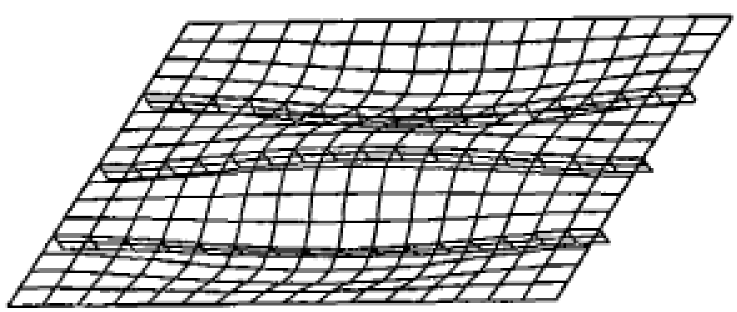
FEA Example No. 1	Title : Stiffened Panel - Transverse Loading



Mode 1



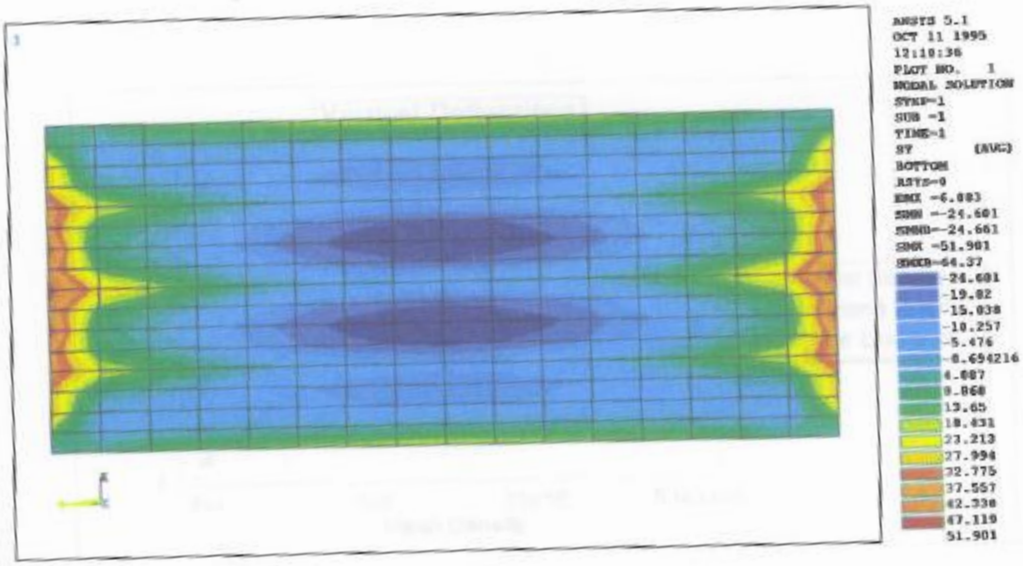
Mode 2



Mode 3

Figure C1.2 Mode Shapes

FEA Example No. 1	Title : Stiffened Panel - Transverse Loading



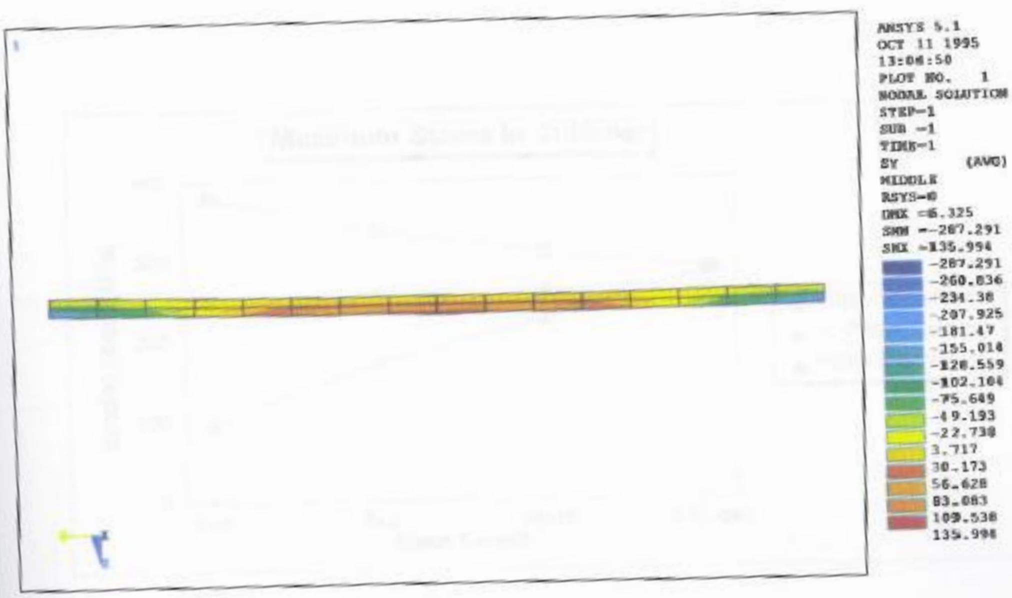
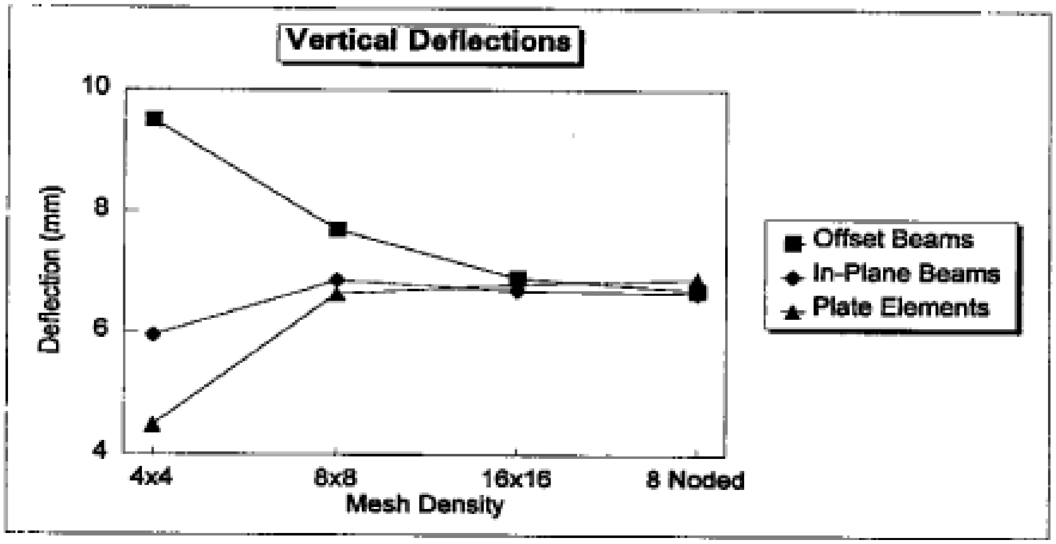


Figure C1.3 Longitudinal Stress (σ_y)

FEA Example No. 1	Title : Stiffened Panel - Transverse Loading



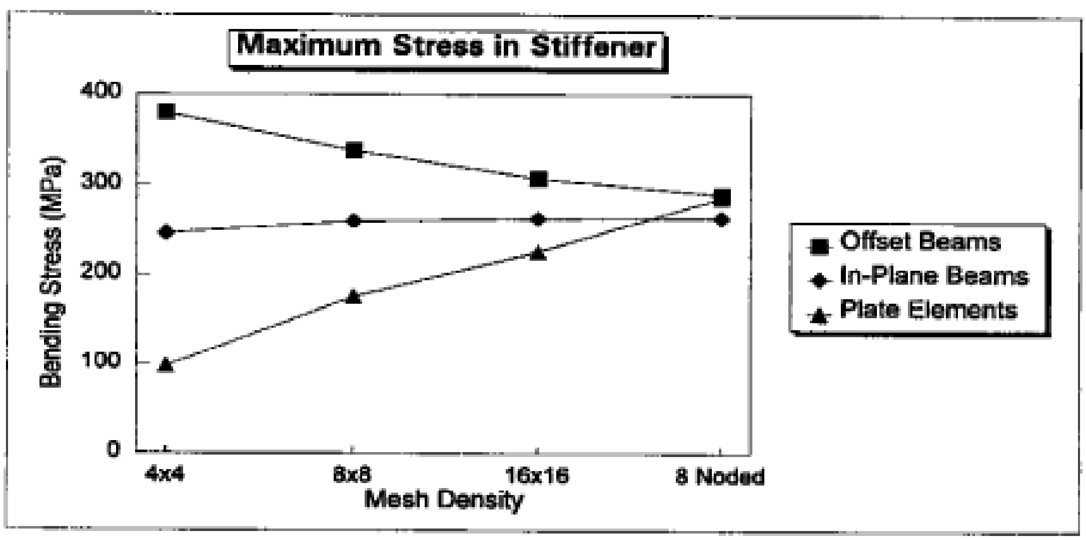


Figure C1.4 Summary of Deflection and Stress Results

FEA Example No. 1	Title : Stiffened Panel - In-Plane Loading

In-Plane Loading :

The second part to this example considers the same stiffened panel subjected to in-plane loading. The problem was modelled in two ways :

1) Using ordinary membrane elements but with orthotropic material properties; and
2) Explicit modelling of stiffeners using 4 node membrane elements as per Example 1j.

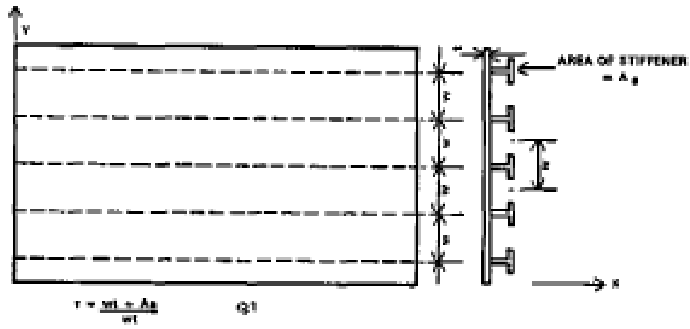
Description :

To model membrane action of stiffened plate structure advantage can be taken of the facility, available in most general purpose FEA packages, to model material orthotropy. Using an approach presented below (adapted from Hughes, see Reference on Page C-9), it is possible to simulate structural orthotropy by material orthotropy. The appropriate expressions are:

$E_x = r\,E$

$E_y = r\,E\,/\,[r - v^2(r-1)]$

$G_{xy} = G = E\,/\,[2(1 + v)]$



The value of "r" is defined in the figure above. With this approach the stiffened plate structure is modelled using ordinary membrane elements but with orthotropic material properties. The expressions given above assume that the stiffeners are aligned in the "x" direction. The expressions can be altered to reflect stiffener alignment in the "y" direction. Care must be taken to ensure that the local coordinate system for the element corresponds with that assumed for defining the material properties. A further assumption implicit in the approach is that the stiffeners are assumed to have identical properties and to be equally spaced.

Results :

Table C1.2 presents the results for the two cases investigated under in-plane loading. The case with orthotropic material properties predicts plate stresses and displacement reasonably accurately. It is important to bear in mind that the plate stresses obtained directly from the FEA for the orthotropic plate are incorrect. However, the actual stress can be derived from the predicted stress by factoring it by 1/r.

TABLE C1.2 Comparison of Finite Element Model Results

Description		Orthotropic material properties	Stiffeners modelled explicitely with plate elements
Stress in plate (MPa)		346.00*	350.00
Displacements	U_x	-1.50	-1.51
	U_y	7.51	7.52
	U_z	0.00	-0.08

* Obtained by dividing the predicted FEA stress by the factor r

C2.O Multiple Deck Openings

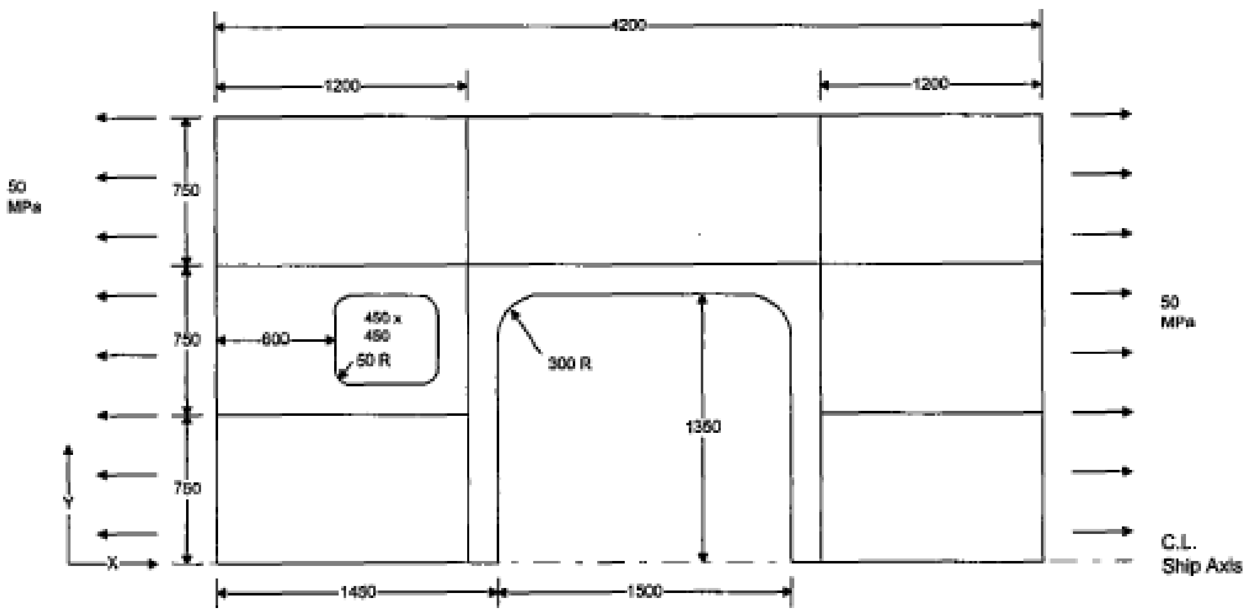
A deck with multiple openings is used as an example to illustrate the influence of mesh density and the element type on deflection and stress results. The mesh density is gradually increased from coarse to fine, Two types of elements, 4-node membrane elements and 8-node shell elements were used. The example also illustrates the effect of varying element aspect ratio. The results obtained from the various trials are tabulated and compared with the converged solution.

FEA Example No. 2	Title : Multiple Deck Openings

Problem Description:

A deck with multiple openings is used to illustrate the influence of mesh density, element aspect ratio, and type of element on deflection and stress results. The density of the mesh is gradually increased from coarse to fine. The use of two types of elements, four node linear and eight node quadratic shells, are illustrated. In addition, dummy line elements with very small area are used along the edge of the opening to extract maximum principle stresses. The latter may be used to overcome errors resulting from extrapolation of stresses from the shell element integration points to the nodes along the edge of the opening.

Engineering Model :



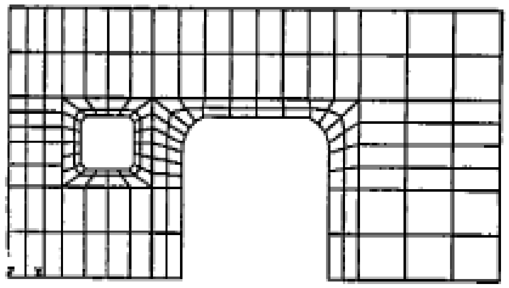
Material Properties :	Geometric Properties :	Loading :
$E = 207 \times 10^3$ MPa $v = 0.3$	Deck Plate t = 6.35 mm Long. Stiff. 152 x 102 Tee Trans. Stiff. 127 x 102 Tee Major Access Coaming 50 x 6.35 mm FB	Uniform Tension = 50 MPa Symmetry BC on +/- Y Boundaries

Modelling Features :

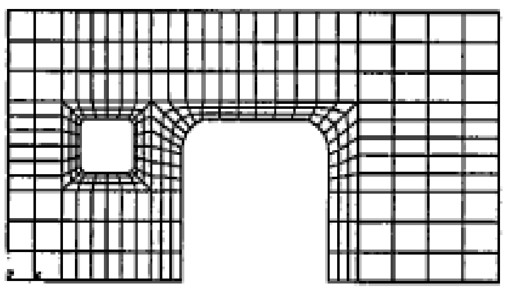
- modelling around stress concentrations
- selection of element type
- effect of varying the mesh density
- use of higher order elements
- effect of aspect ratio in the area of stress concentrations

FEA Example No. 2	Title : Multiple Deck Openings

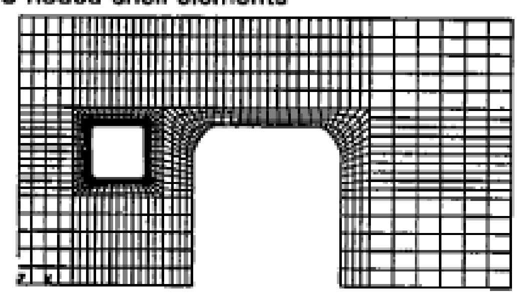
Finite Element Models :



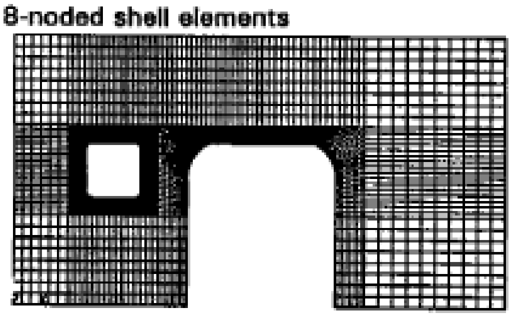
	Nodes	Elements	Degrees of freedom
2a : 4-noded membrane shell elements	214	235	642
2e : 8-noded shell elements	995	465	5970



2b : 4-noded membrane shell elements	351	379	1053
2f : 8-noded shell elements	3044	1256	18264



2c : 4-noded membrane shell elements	1213	1104	3639
2g : 8-noded shell elements	4842	1924	29052



2d : 4-noded membrane shell elements	3186	3272	9558
2h : 8-noded shell elements	9368	3540	56208

FEA Example No. 2	Title : Multiple Deck Openings

DISCUSSION OF RESULTS

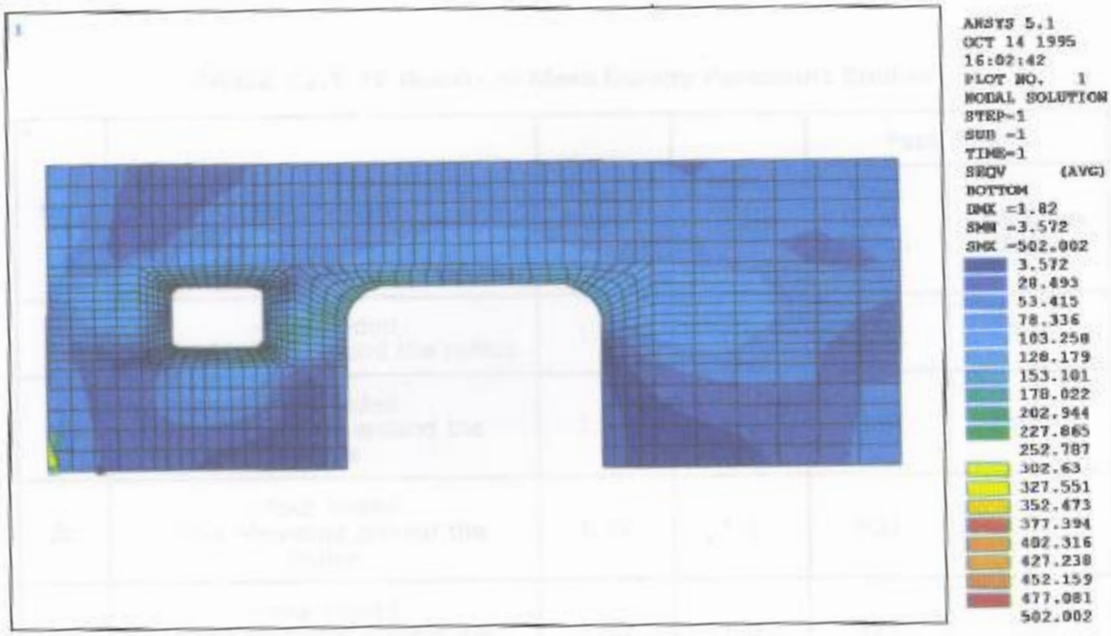
The analyses revealed peak stresses at the lower left corner of the smaller opening as shown in Figure C2.1 (the top figure shows stress contours for the full model and the bottom figure provides a close-up view of stress contours around the smaller opening). The stress concentration near the larger opening was relatively insignificant due to the presence of the coaming.

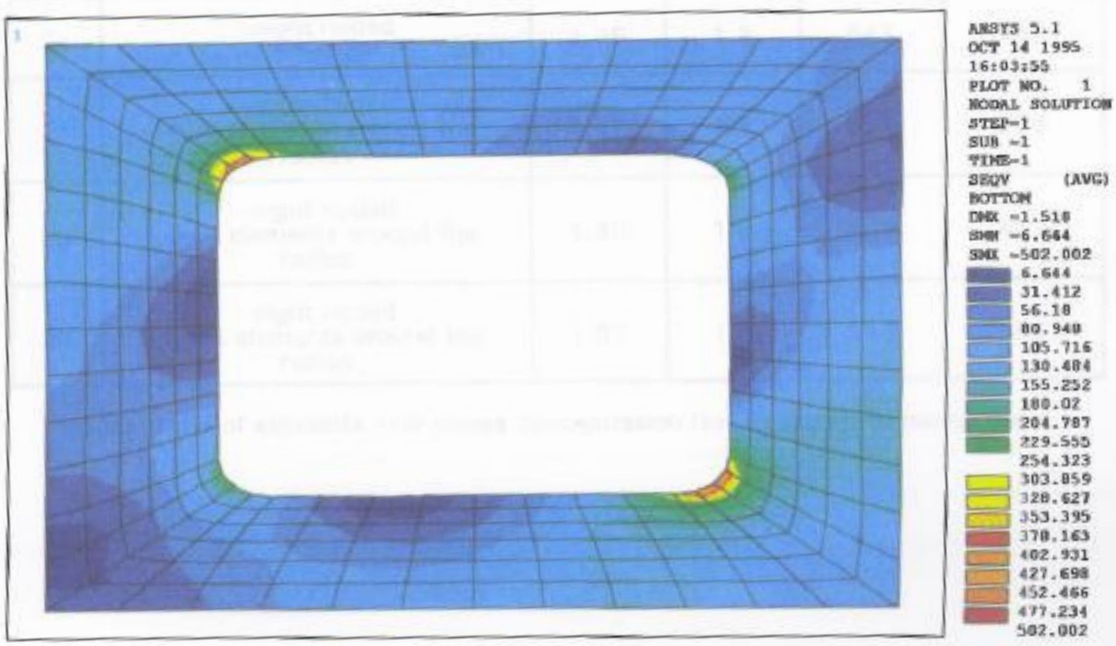
When the mesh density around the openings was increased, with the aspect ratio held constant, the results indicate a progressive increase in the magnitude of peak stress. The results listed in Table C2.1 indicate a converging trend in the magnitude of peak stress with mesh refinement. Although the peak stress always occurs at the same corner, it should be noted that the precise location of the peak stress varies slightly with the refinement of the mesh (number of nodes around the corner radius). Some of the differences in the results may also be due to different mesh transitioning (from areas of coarse mesh density away from the openings to areas of high mesh density at the openings) in the different models.

The results in Table C2.1 indicates the rate of convergencence of the stress results is greater for the line elements (truss or spar elements with only one degree of freedom per node placed along the edge of the openings) than it is for the plate elements. The use of line elements for obtaining stresses also overcomes stress extrapolation errors that arise in shell elements. Note that the stress results for shell elements must be extrapolated from the element integration points to the node locations at the edge of the opening.

Parametric studies were conducted to evaluate the effect of aspect ratio in predicting stress concentrations. The mesh density of Example 2d was used as the basis for this investigation. The aspect ratio of elements around the smaller opening was varied from 1.05 to 3.00. The results, Table C2.2, indicate that the best values for stress concentrations are obtained when the aspect ratio is close to one. The difference in the stress results when the aspect ratio is changed from 1.05 to 3.00 is about 8%.

FEA Example No. 2	Title : Multiple Deck Openings





:

TABLE C2.1 FE Results of Mesh Density Parametric Studies

No.	Description	Aspect Ratio*	Max. Disp. (mm)	Peak Stress	
				Shell Elem. (MPa)	Line Elem. (Mpa)
2a	–four noded –one element around the radius	1.29	1.8	300	399
2b	–four noded –two elements around the radius	1.38	1.8	369	453
2c	–four noded –four elements around the radius	1.37	1.8	502	556
2d	–four noded –eight elements around the radius	1.37	1.9	572	593
2e	–eight noded –one element around the radius	1.38	1.9	543	557
2f	–eight noded –two elements around the radius	1.37	1.9	570	606
2g	–eight noded –four elements around the radius	1.36	1.9	583	607
2h	–eight noded –eight elements around the radius	1.37	1.9	591	609

* Aspect ratio of elements near stress concentration (see figure on following page)

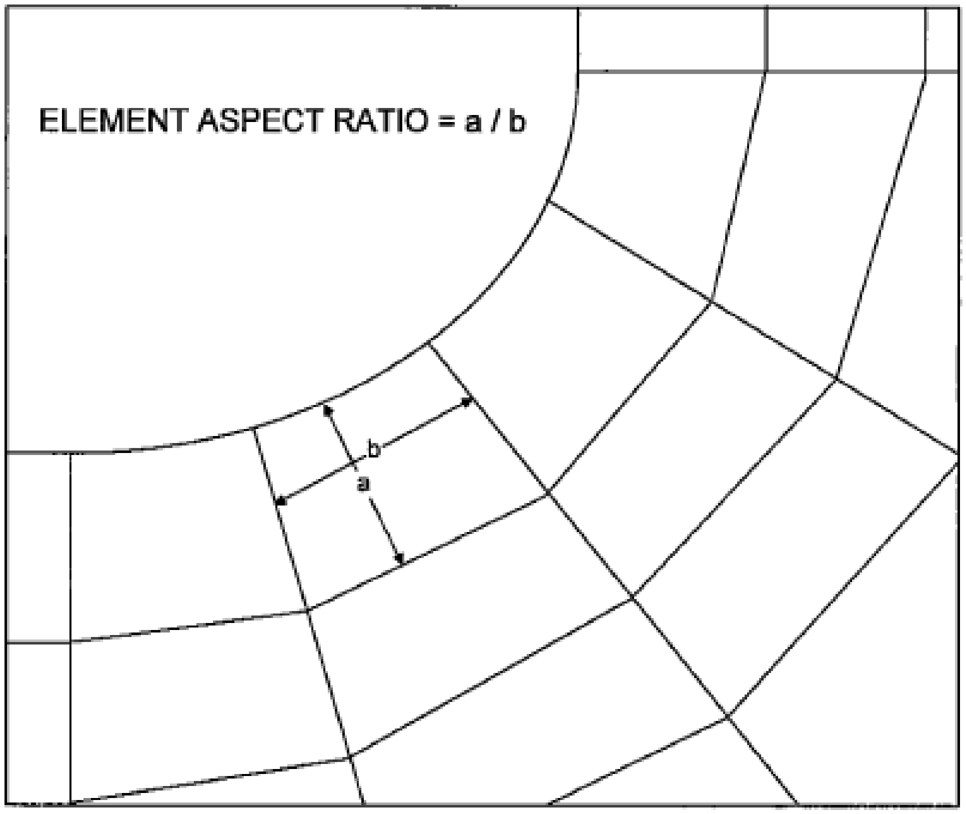


TABLE C2.2 Results from Aspect Ratio Parametric Studies

Trial No.	Aspect Ratio*	Peak Stress in Plate Elem. MPa	Relative ** Peak Stress Ratio
1	3.00	537	0.92
2	1.98	561	0.96
3	1.37	572	0.98
4	1.05	585	1.00

* Aspect ratio of elements near stress concentration

** Ratio of peak stress to that for trial No. 4 (plate element aspect ratio of 1.05, i.e. 585 MPa)

C3.0 MAST

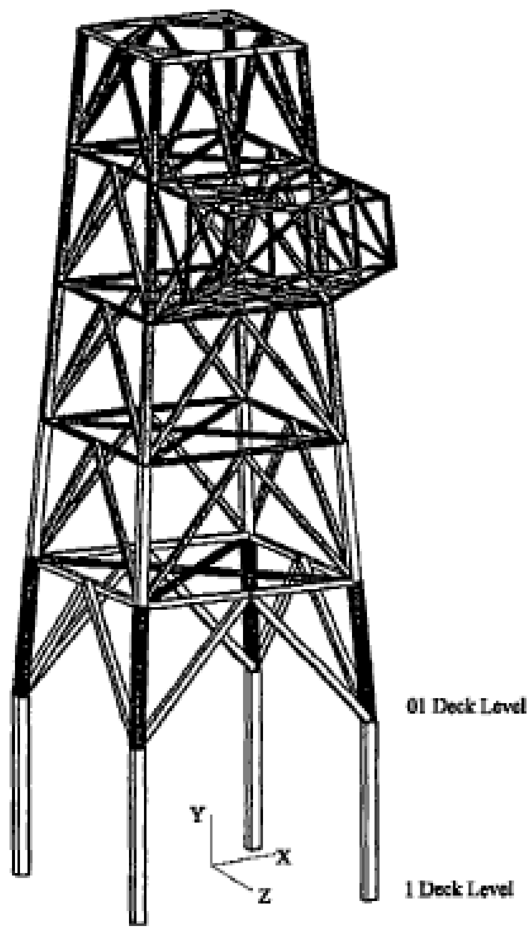
A major factor in the modeling of lattice masts is the modeling of the connection details, Depending on the type of connection, the joints can be modeled with full rigidity at the joint, or some or all members can be modeled as pinned (hinged) joints. A simple truss-type mast structure is used to illustrate both these options. In the case of a rigid jointed structure, the mesh density (i. e., the number of elements per member of the mast) was varied to investigate the influence on the results. Both static and dynamic analyses were performed on all these models.

FEA Example No. 3	Title : Mast

Problem Description:

The truss-type mast structure shown below, consisting of steel pipe sections, is to be analyzed for shock accelerations loading and to calculate frequencies and mode shapes.

Engineering Model :



Material Properties :	Geometric Properties :	Loading :
E = 207x10³ MPa v = 0.3	see Table C3.1	Base Accelerations: 8 g in X 18 g in Y 8 g in Z

Modelling Features :

- pinned and rigid connections
- model refinement
- static and dynamic analyses

FEA Example No. 3	Title : Mast

Finite Element Models :

The finite element models of the mast are as shown below.

Example 3a is modelled with all joints pinned. However, if the member is continuous and has nodes between the two ends (viz. two or more elements per member) then rotations are restrained at such nodes to simulate the continuity of the member. The following is a list of members that are treated continuous:

- Main legs
- Horizontal members
- One out of the two cross braces at every level
- Principal members of the spur frame

Examples 3b and 3c are modelled with all rigid joints.

The three-dimensional beam element (BEAM44) of ANSYS is used in modelling mast members. This element has six degrees of freedom per node, and has the option of suppressing rotational degrees of freedom at nodes to simulate pinned connections. The various payloads and other dead loads were represented by mass elements (MASS21). The coordinate system used in the finite element model is as follows (also shown in the figures below):

X - Athwartship (positive in port direction)
Y - Vertical (positive upwards)
Z - Longitudinal (positive in forward direction)

The boundary conditions applied to the mast are as follows:

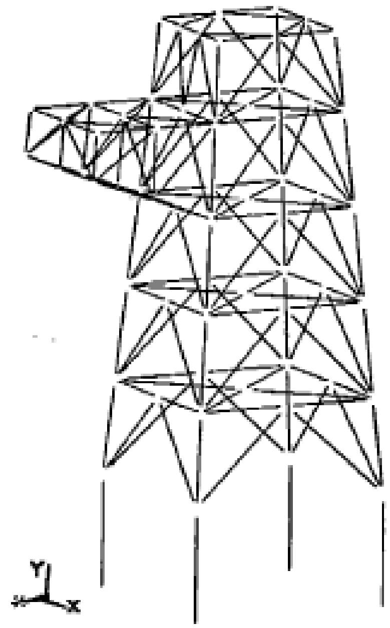
Main Legs:	UX = UY = UZ = 0	at 1 deck level
	UX = UZ = 0	at 01 deck level

The static analysis consisted of three load cases of base accelerations in the X, Y, and Z directions. The accelerations applied are as follows:

Case i.	8 g Athwartship Shock (m/s^2):	$a_x = 78.48$	$a_y = 9.81$	$a_z = 0$
Case ii	18 g Vertical Shock (m/s^2):	$a_x = 0$	$a_y = 186.39$	$a_z = 0$
Case iii	8 g Longitudinal Shock (m/s^2):	$a_x = 0$	$a_y = 9.81$	$a_z = 78.48$

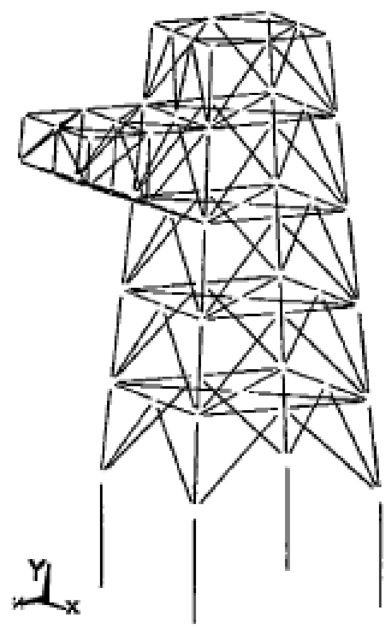
For the dynamic load case, translational master degrees of freedom are selected at the corner nodes of each level and the first 5 natural frequencies and the corresponding mode shapes are extracted.

FEA Example No. 3	Title : Mast



65 Nodes
217 Elements
370 Degrees of Freedom

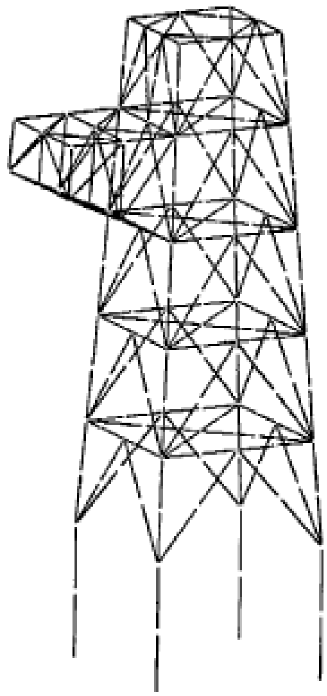
Example 3a - Pinned Joints; Typically one element per member



65 Nodes
217 Elements
370 Degrees of Freedom

Example 3b - Rigid Joints; Typically one element per member

FEA Example No. 3	Title : Mast



200 Nodes

352 Elements

1180 Degrees of Freedom

[redacted]

FEA Example No. 3	Title : Mast

DISCUSSION OF RESULTS

The displacements for the three static load cases are summarized in Table C3.2. When the two modelling approaches (pinned joint versus rigid joint models) are compared, the model with pinned joints predicts the most flexible structure with the most displacements for every load case. Also, in some cases, the maximum displacement is predicted at a location different from the one predicted by the rigid joint model. In the second load case (Vertical shock) the displacement in Y direction, although at the same location for all three models, is excessively overpredicted by the pinned joint model. The maximum vertical deflections occur at the centre of the horizontal cross braces. Under vertical shock loading, these members act similar to beams subject to a unform distributed load (ie. inertial loading) for which the maximum deflection in the simply supported case (ie. pinned ends) is five times that for the fixed ends case.

Table C3.3 lists peak stresses. As expected, the axial stresses are approximately the same for the two approaches. However, the bending stresses at mid-span of horixontal members and cross braces are significantly more in the pinned joint model. This is again due to the different end conditions in the two modelling methods. The model with simply supported end conditions naturally predicts higher moments at mid-span.

Among the two models with fully rigid connections, the predicted maximum stresses are similar. The probable disadvantage with the one element per member model is that the stress at the centre of the member will not be calculated. It is possible that some members might have peak stresses at the centre as opposed to the ends if the members are also subject to local transverse loads (eg. wind loads, high inertial loads, equipment support loads).

The natural frequencies and mode shapes for the two approaches are similar (see Table C3.4). Figure C3.1 shows the first five mode shapes obtained from example 3b.

The variations in deflection and some stress results between the pin jointed and rigid jointed models are significant. Hence, extreme care and proper judgement is needed in deciding on the right modelling approach for the problem.

FEA Example No. 3	Title : Mast

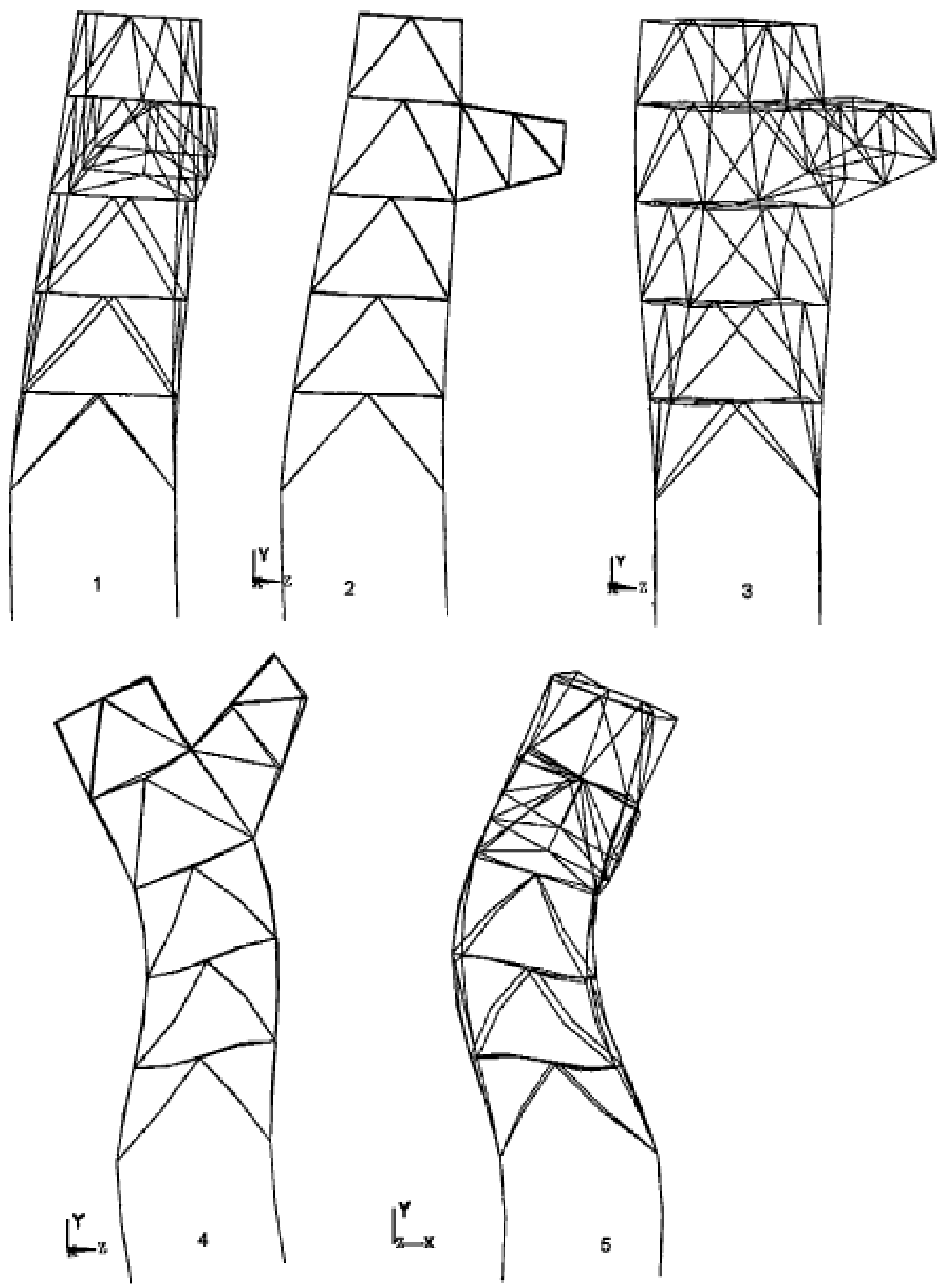


Figure C3.1 The first five mode shapes

Table C3.1: Geometry Properties

Real Constant Set No.	Member or Component Description	Cross Section or Size	Real Constants				
			Area (10^{-6} m^2)	I_{ZZ} (10^{-6} m^4)	I_{YY} (10^{-6} m^4)	TKZB1 (10^{-3} m)	TKYB1 (10^{-3} m)
1	Main Legs - 1 Deck to 02 Deck	7.25" OD x 6.0" ID	8392.0	29.9700	29.9700	92.10	92.10
2	Main Legs - 02 Deck to Level B	7.1" OD x 6.25" ID	5750.0	20.7400	20.7400	90.13	90.13
3	Main Legs - Level B to Level D	7.0" OD x 6.375" ID	4236.0	15.3100	15.3100	88.90	88.90
4	Main Legs - Level D to Level F	5.0" OD x 4.25" ID	3520.0	6.1000	6.1000	63.50	63.50
5	Main Legs - Level F to Top	4.875" OD x 4.375" ID	2344.0	4.0540	4.0540	61.91	61.91
6	"V" Braces - 02 Deck to Level D	4.875" OD x 4.5" ID	1780.0	3.1600	3.1600	61.91	61.91
7	"V" Braces - Level D to Level G	3.625" OD x 3.25" ID	1306.0	1.2490	1.2490	46.00	46.00
8	"V" Braces - Level G to Top	4.0" OD x 0.226" t	1730.0	1.9900	1.9900	50.80	50.80
9	Horizontals - Level A to Level D	4.0" OD x 3.625" ID	1450.0	1.7000	1.7000	50.80	50.80
10	Horizontals - Level E to Level G	3.0" OD x 2.635" ID	1069.0	0.6840	0.6840	38.10	38.10
11	Horizontals - Level MG	2.875" OD x 0.203" t	1100.0	0.6370	0.6370	36.51	36.51
12	Horizontals - Level MG	4.0" OD x 0.226" t	1730.0	1.9900	1.9900	50.80	50.80
13	"X" Braces - Level A to Level D	3.625" OD x 3.25" ID	1306.0	1.2490	1.2490	46.00	46.00
14	"X" Braces - Level E to Level G	3.0" OD x 2.635" ID	1069.0	0.6840	0.6840	38.10	38.10
15	"X" Braces - Level MG	2.875" OD x 0.203 t	1100.0	0.6370	0.6370	36.51	36.51
16	Platform	2.375" OD x 0.154" t	693.0	0.2771	0.2771	30.20	30.20

TABLE C3.2 Comparison of displacements for the Mast finite element analyses

Description	Max. Displacement (mm)			Location
	Example 3a –pinned joints	Example 3b –rigid joints with 1 element per member	Example 3c –rigid joints with 2 elements per member	
Athwartship (Z) shock				
δ_x	-15.13[1]	-15.07	-15.07	outer tip of spur frame
δ_y	-1.60[2]	-1.59	-1.60	outer tip of spur frame
δ_z	1.42	1.40	1.41	spur frame at main leg junction
Vertical (Y) shock				
δ_x	-3.46	-0.76	-0.76	middle of horizontal member - level 2
δ_y	-74.24	-16.59	-16.75	centre of X brace - level 2
δ_z	3.09[3]	3.07	3.08	horizontal member at mid span (top of mast)
Longitudinal (Z) shock				
δ_x	-0.37	-0.37	-0.37[4]	outer tip of spur frame
δ_y	3.94	3.93	3.94	outer tip of spur frame
δ_z	-27.74	-14.56	-14.56	spur frame at main horizontal at mid span

1 The maximum is -26.7 at the middle of horizontal member - level 2

2 The maximum is -3.91 at the centre of cross brace member - level 2

3 The maximum is -3.67 at the middle of horizontal member - level 4

4 The maximum is 0.76 at the middle of V-brace - level 2

TABLE C3.3 Comparison of stresses for the Mast finite element analyses

Description	Stress (MPa)			Location
	Example 3a -pinned joints	Example 3b -rigid joints with 1 element per member	Example 3c -rigid joints with 2 elements per member	
<u>Athwartship (Z) shock</u> Axial stress (σ_x) Bending stress (σ_{by}) Bending stress (σ_{bz})	 ±105 ±36[1] ±158	 ±104 ±39 ±61	 ±104 ±39 ±58	 Lower V braces Lower V braces at main leg junction Horizontal members at mid-span
<u>Vertical (Y) shock</u> Axial stress (σ_x) Bending stress (σ_{by}) Bending stress (σ_{bz})	 -81, +41 ±245[2] ±34[3]	 -81, +40 ±163 ±41	 -81, +40 ±163 ±41	 Main legs, spur frame diagonals X braces at main leg junction Spur frame at main leg junction
<u>Longitudinal (Z) shock</u> Axial stress (σ_x) Bending stress (σ_{by}) Bending stress (σ_{bz})	 ±88 ±17[4] ±193	 ±88 ±31 ±60	 ±87 ±31 ±58	 Lower V braces Spur frame at main leg junction Horizontal members at mid-span

1 Main Legs at level 1

2 Cross Braces at mid-span

3 Main Legs at mid-span

4 Main Legs at mid-span

TABLE C3.4 Comparison of frequencies for the Mast finite element analyses

Mode	Frequency (Hz)			Mode Shape
	Example 3a –pinned joints	Example 3b –rigid joints with 1 element per member	Example 3c –rigid joints with 2 elements per member	
1	13.30	13.31	13.30	Bending about Z- axis (1st mode)
2	13.76	13.77	13.76	Bending about X-axis (1st mode)
3	21.56	21.53	21.53	Twisting about Y-axis
4	34.51	34.39	34.41	Bending about X-axis (2nd mode)
5	38.33	38.13	38.16	Bending about Z- axis (2nd mode)

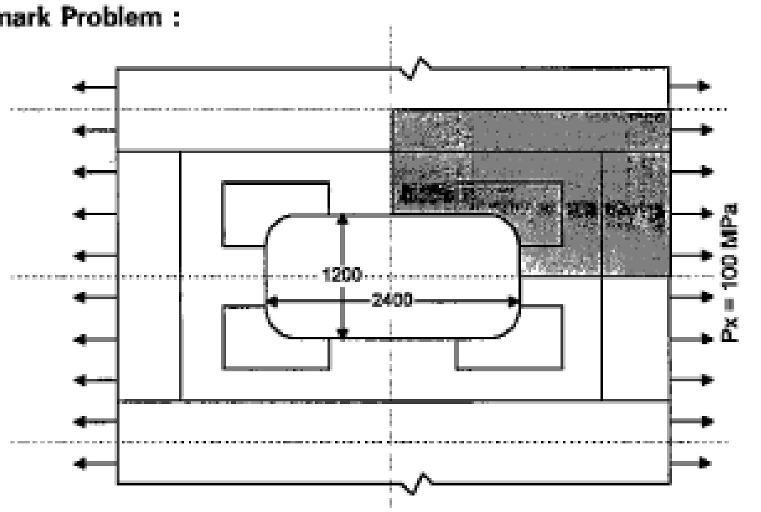
Appendix D
Ship Structure Benchmark Problems for Assessing FEA Software

Benchmark No. :	BM-1-a	**Benchmark Title :**	Opening with Insert Plate
Analysis Type :	2D Static	**Element Type(s) :**	4-Node Plane Stress 2-Node Line (Axial Stress)

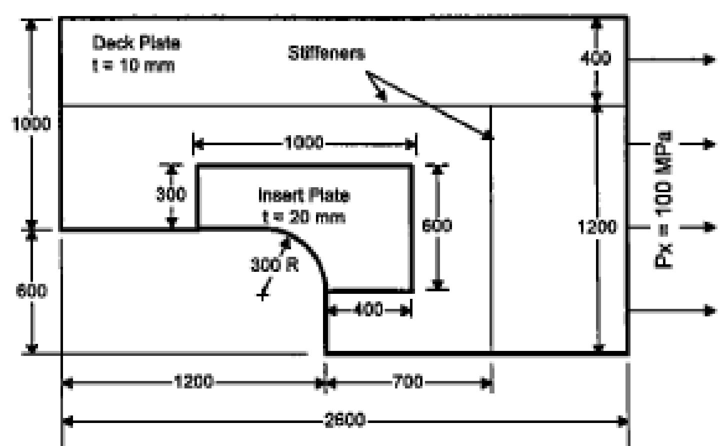
Problem Description:

A rectangular deck opening with rounded corners is reinforced with insert plates at each corner. Determine the maximum von Mises stress in the 20 mm insert plate and the 10 mm deck plate.

Sketch of Benchmark Problem :



a) Deck Opening With Insert Plate



b) Detail of Shaded Region of Deck Opening

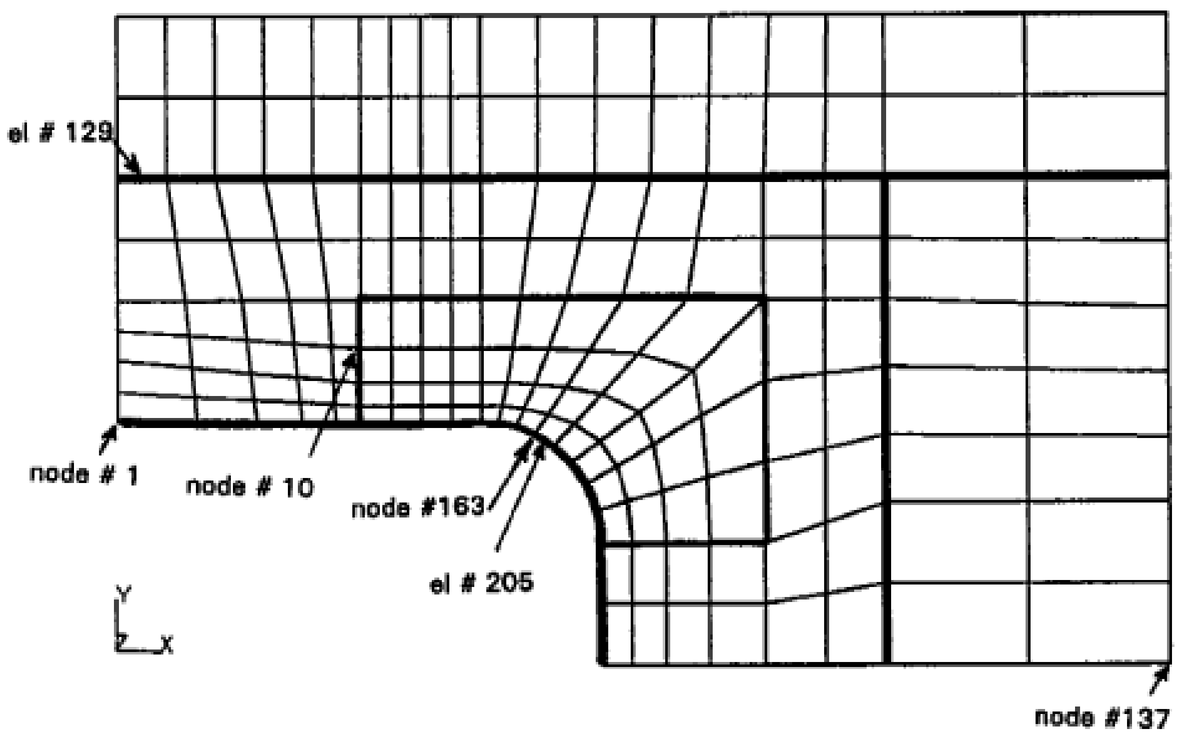
Material Properties :	**Geometric Properties :**	**Loading :**
$E = 207000\ N/mm^2$ $v = 0.3$	Deck Plate $t = 10$ mm Insert Plate $t = 20$ mm Stiffeners $A = 1575\ mm^2$ Line Elements $A = 1\ mm^2$	$P_x = 100\ N/mm^2$ (Applied as nodal force loading)

Benchmark No. : BM-1-a	Benchmark Title : Opening with Insert Plate

Analysis Assumptions :

Due to symmetry, only one-quarter of the opening is modeled. The deck stiffeners are modelled using axial stress line elements since only in-plane loading is considered.

Finite Element Model :



No. of Nodes : 200

No. of Elements : 212

1. Deck Plate 120 4-Node Plate Elements $t=10$ mm
2. Insert Plate 48 4-Node Plate Elements $t=20$ mm
3. Stiffeners 25 2-Node Line Elements $A=1575$ mm^2
4. Line Elements 19 2-Node Line Elements $A=1$ mm^2 (for stresses at free edge)

Boundary Conditions :

$U_x = 0$ at $X=0$
$U_y = 0$ at $Y = 0$ and $Y = 1600$
$U_z = 0$ at ($X=0;Y=0$), ($X=0;Y=1600$), ($X=2600;Y=0$), and ($X=2600,Y=1600$)

Benchmark No. : BM-1-a	Benchmark Title : Opening with Insert Plate			
Finite Element Software Results	**ANSYS 5.1**	**MSC / NASTRAN Windows 1**	**ALGOR 3.14**	**Converged Solution [4] (ANSYS 5.1)**
FEA Software Element Types :	SHELL63 LINK8	CQUAD4 CROD	TYPE 6 TYPE 1	SHELL93 LINK8
Maximum Stresses (MPa)				
1. Deck Plate σ_{eqv} [1] (node # 10)	192.8	193.5	192.3	196.9
2. Insert Plate σ_{eqv} [1] (node #163)	198.3	189.2	199.3	206.3
3. Stiffeners σ_a [2] (el # 129)	139.8	139.8	139.8	140.3
4. Edge Elements σ_a [3] (el # 205)	204.4	203.3	204.4	209.0
Maximum Deflections (mm)				
Ux (node #137)	1.496	1.496	1.496	1.506
Uy (node # 1)	0.157	0.157	0.157	0.157

Comments on Benchmark Results :

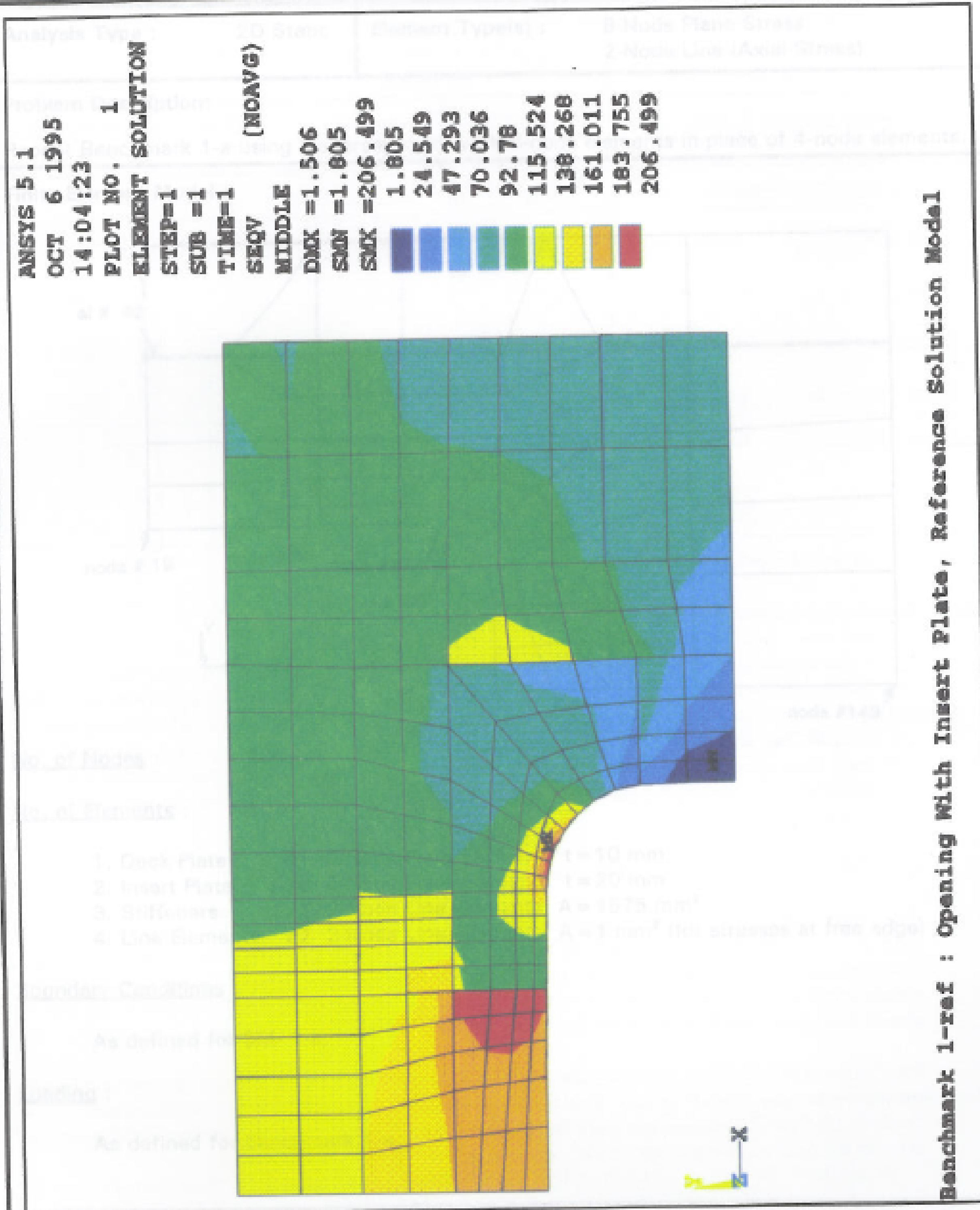
1. σ_{eqv} is the maximum von Mises or equivalent stress reported for the plate elements (section properties 1 and 2). The values presented are the nodal averaged stresses within each group of elements of the same section property. The nodal averaged stresses are obtained by extrapolating stresses at the element integration points to the node locations, and then averaging the values at each node. Different FEA software may use different extrapolation and averaging methods which can lead to slight differences in the nodal stress results.

2. σ_a is the maximum axial or direct stress in the line elements.

3. The benchmark FE model includes line elements of small arbitrary area (section property 4 with A = 1 mm^2) which are used to obtain stresses around the free edge of the opening. The maximum axial stress reported in the line elements corresponds approximately to the maximum principal and von Mises stress at the edge of the opening, irrespective of the stress extrapolation method used for the plate elements.

4. The "converged solution" for this benchmark was obtained using a more refined model of the same problem consisting of 8 node shell elements with ANSYS 5.1. The stress contour plot for the converged solution is shown on the following page. Note that the plot shows element stresses, not nodal averaged stresses, so as to permit presentation of the results for the two plate thicknesses on the same plot. Although the plot shows slight discontinuities in the stress contours, these are mainly away from the areas of interest. The difference between the maximum element stresses and the nodal averaged stresses is minimal at the two locations reported in the above table. There is a real stress discontinuity at the border between the insert plate and the deck plate due to the abrupt change in plate thickness. The stress contour values are in units of MPa. The "MX" on the plot signifies the location of maximum stress.

Benchmark No. :	BM-1-a	**Benchmark Title :**	Opening with Insert Plate

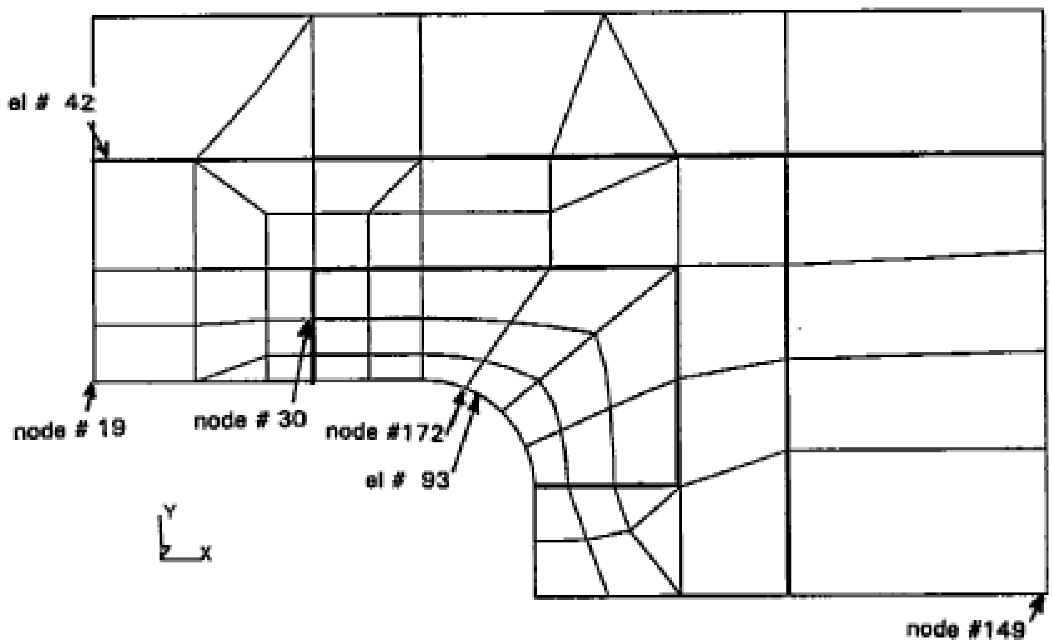


Benchmark No. :	BM-1-b	**Benchmark Title :**	Opening with Insert Plate
Analysis Type :	2D Static	**Element Type(s) :**	8-Node Plane Stress 2-Node Line (Axial Stress)

Problem Description:

Repeat Benchmark 1-a using a coarser mesh with 8-node elements in place of 4-node elements.

Finite Element Model :



No. of Nodes : 200

No. of Elements : 103

1. Deck Plate 41 8-Node Plate Elements t=10 mm
2. Insert Plate 18 8-Node Plate Elements t=20 mm
3. Stiffeners 22 2-Node Line Elements A=1575 mm^2
4. Line Elements 22 2-Node Line Elements A=1 mm^2 (for stresses at free edge)

Boundary Conditions :

As defined for BM- 1-a.

Loading :

As defined for Benchmark 1-a.

Benchmark No. : BM-1-b	**Benchmark Title :** Opening with Insert Plate			
Finite Element Software Results	**ANSYS 5.1**	**MSC / NASTRAN Windows 1**	**ALGOR**	**Converged Solution[4] (ANSYS 5.1)**
FEA Software Element Types :	SHELL93 LINK8	CQUAD8 CROD	NA*	SHELL93 LINK8
Maximum Stresses (MPa)				
1. Deck Plate σ_{eqv} [1] (node # 30)	195.6	195.6	-	196.9
2. Insert Plate σ_{eqv} [1] (node #172)	207.8	204.5	-	206.3
3. Stiffeners σ_a [2] (el # 42)	140.3	140.3	-	140.3
4. Edge Elements σ_a [3] (el # 93)	207.8	207.8	-	209.0
Maximum Deflections (mm)				
Ux (node #149)	1.505	1.505	-	1.506
Uy (node # 19)	0.157	0.157	-	0.157

Comments on Benchmark Results :

*ALGOR does not include 8-node plate elements for stress analysis.

1. σ_{eqv} is the maximum von Mises or equivalent stress reported for the plate elements (section properties 1 and 2). The values presented are the nodal averaged stresses within each group of elements of the same section property. The nodal averaged stresses are obtained by extrapolating stresses at the element integration points to the node locations, and then averaging the values at each node. Different FEA software may use different extrapolation and averaging methods which can lead to slight differences in the nodal stress results.

2. σ_a is the maximum axial or direct stress in the line elements.

3. The benchmark FE model includes line elements of small arbitrary area (section property 4 with A = 1 mm^2) which are used to obtain stresses around the free edge of the opening. The maximum axial stress reported in the line elements corresponds approximately to the maximum principal and von Mises stress at the edge of the opening, irrespective of the stress extrapolation method used for the plate elements.

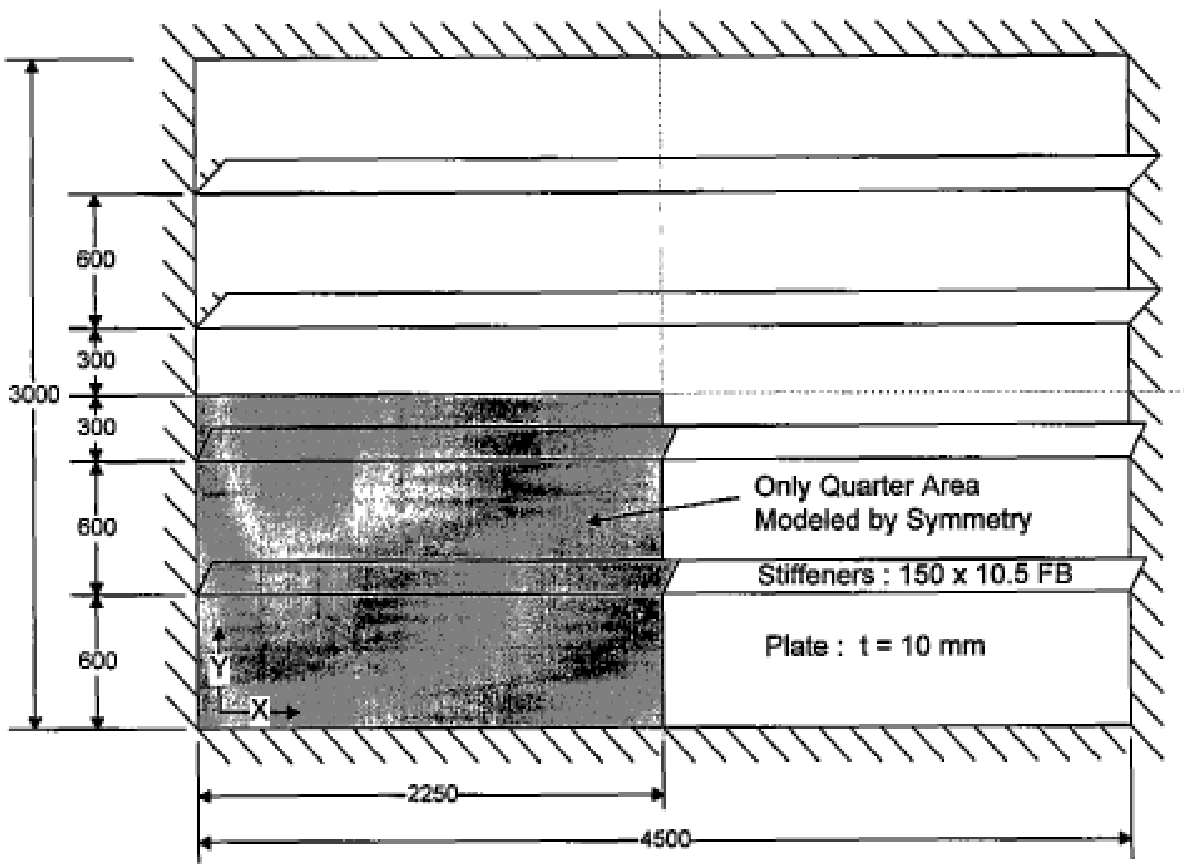
4. The "converged solution" for this benchmark was obtained using a more refined model of the same problem consisting of 8 node shell elements with ANSYS 5.1. The stress contour plot for the converged solution is shown on Page D-5. Refer to the BM-1-a results for further discussion of the converged solution.

Benchmark No. :	BM-2-a	**Benchmark Title :**	Stiffened Panel
Analysis Type :	3D Static 3D Modal	**Element Type(s) :**	4-Node Shell 2-Node Beam (In plane of plate)

Problem Description:

A rectangular stiffened panel is subject to a uniform pressure load applied to its surface. Determine the maximum deflection, stresses and natural frequencies for the panel.

Sketch of Benchmark Problem :

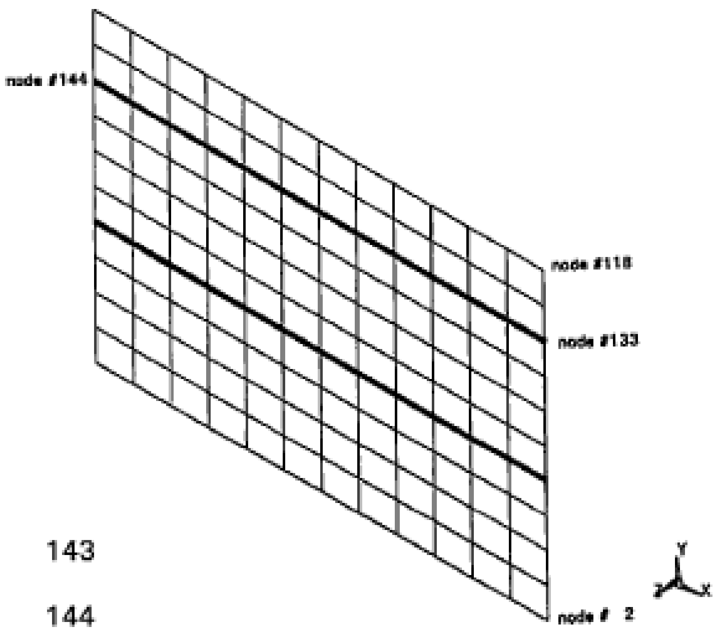


Benchmark Problem 2 : Stiffened Panel

Material Properties :	**Geometric Properties :**		**Loading :**
$E = 207 \times 10^9$ N/m^2	Plate	t = 10 mm	P_z = 9810 Pa
$\nu = 0.3$	Stiffeners	150 x 10.5 FB	
$\rho = 7850$ kg/m^3			

Benchmark No. : BM-2-a	Benchmark Title : Stiffened Panel

Finite Element Model :



<u>No. of Nodes</u> : 143

<u>No. of Elements</u> : 144

1. Panel 120 4-Node 3-D Plate Elements $t = 10$ mm

2. Stiffeners 24 2-Node 3-D Beam Elements

$A = 0.001575\ m^2$	$Y_t = 0.1352$ m
$I_{zz} = 53.35 \times 10^{-6}\ m^4$ **	$Y_b = 0.0148$ m
$I_{yy} = 10.19 \times 10^{-6}\ m^4$	$Z_t = 0.00525$ m
$I_{xx} = 0.0553 \times 10^{-6}\ m^4$ (Torsion)	$Z_b = 0.00525$ m

** In-Plane Beam elements I_{zz} includes 40 t effective plate width.

<u>Boundary Conditions</u> :

1. <u>Static Analysis</u>
   - All nodes fixed at edges along $x=0$ and along $y=0$.
   - Symmetry about YZ plane along edge at $x = 2.250$ m
   - Symmetry about XZ plane along edge $y = 1.500$ m

2. <u>Modal Analysis</u>*
   - All nodes fixed at edges along $x=0$ and along $y=0$.
   - Symmetry about YZ plane along edge at $x = 2.250$ m
   - Antisymmetry about XZ plane along edge $y = 1.500$ m

* This benchmark test only requires calculation of the first four natural frequencies for symmetry / antisymmetry boundary conditions. In order to capture all modes of vibration, the modal analysis of the quarter model would also have to consider symmetry / symmetry, antisymmetry / symmetry, and antisymmetry / antisymmetry boundary conditions.

Benchmark No. : BM-2-a			Benchmark Title : Stiffened Plate		
Finite Element Software Results		**ANSYS 5.1**	**MSC / NASTRAN Windows 1**	**ALGOR 3.14**	**Converged Solution [1] (ANSYS 5.1)**
Element Types :	Plate Stiffeners	SHELL63 BEAM4	CQUAD4 CBAR	TYPE 6 TYPE 2	SHELL93 SHELL93
Maximum Stresses 1. Plate σ_{eqv} [2]	(MPa) (node # 2)	39.3	38.2	36.5	42.1
2. Stiffeners σ_x [3] Tension Compression	(MPa) (node #133) (node #144)	 69.0 -135.8	 69.0 -135.8	 69.0 -135.0	 61.3 -126.5
Maximum Deflections Uz [4]	(mm) (node #118)	3.30	3.29	3.29	3.50
Natural Frequencies [5] : 1st Mode 2nd Mode 3rd Mode 4th Mode	 (Hz) (Hz) (Hz) (Hz)	 36.5 60.9 100.1 110.2	 36.5 61.1 100.4 111.4	 36.6 61.2 102.4 111.9	 35.9 61.0 96.5 106.5

1. The "converged solution" results were obtained using a refined mesh model with 8-node shell elements on ANSYS 5.1. The von Mises Stress contours for the converged model are shown on Page D-13. The stress contours are in units of Pa (N/m^2).

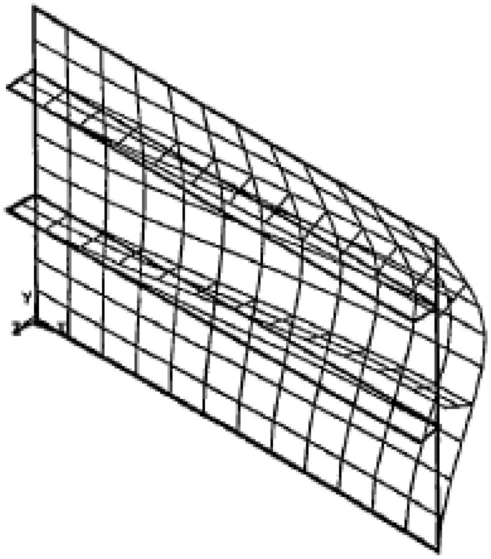
2. The maximum stress in the plate occurs at the middle of the long fixed edges (node 2). Reported are the maximum nodal averaged von Mises stress of the top or bottom surface of the plate elements. Note that different FEA programs may use different conventions for defining the top and bottom surfaces of plate elements. Also, different FEA programs use different extrapolation and averaging techniques for computing plate / shell element stresses which may lead to slight differences (refer to BM-1-a for discussion).

3. Reported are the maximum stresses in the beam elements (axial stress + bending stress). The maximum tensile stress occurs at the centre of the middle stiffeners (node 133). The maximum compressive stress occurs at the fixed ends of the middle stiffeners (node 144).

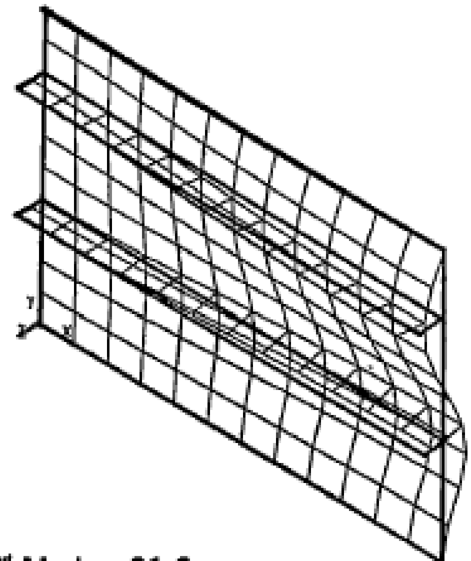
4. The maximum out-of-plane deflection (Uz) occurs at the centre of the panel (node 118). Differences in deflection and stress results relative to the converged model are due mainly to the simplifying assumption of 40 t effective plate width used in defining the beam properties.

5. The frequencies and mode shapes for symmetry / antisymmetry boundary conditions from the converged model are shown on Page D-12. The mode shapes predicted by the BM-2-a FEA models are the same as those for the converged model. The frequencies predicted by the BM-2-a model deviate slightly from those predicted by the converged model, particularily for the 3rd and 4th modes. These are more complex modes involving torsion of the stiffeners for which the beam + plate element model is probably too simplified. However, the plate + beam model gives very good predictions for the first two modes.

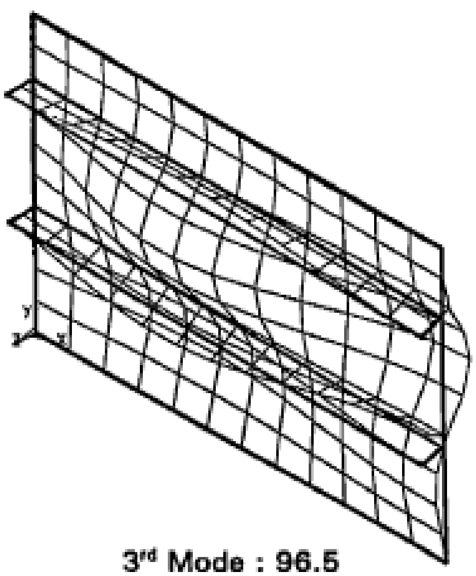
Benchmark No. :	BM-2-a	Benchmark Title :	Stiffened Plate



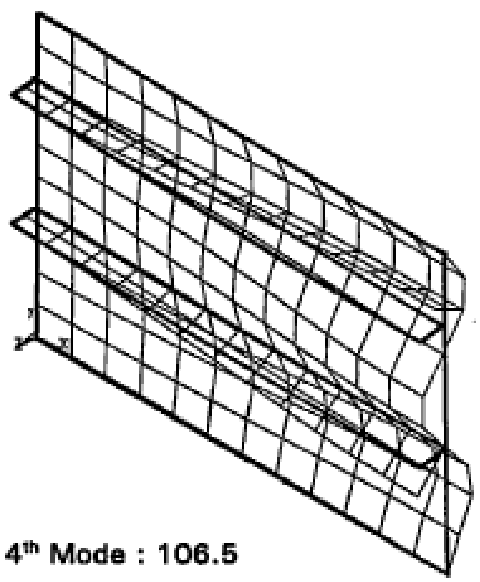
1st Mode : 35.9 Hz



2nd Mode : 61.0



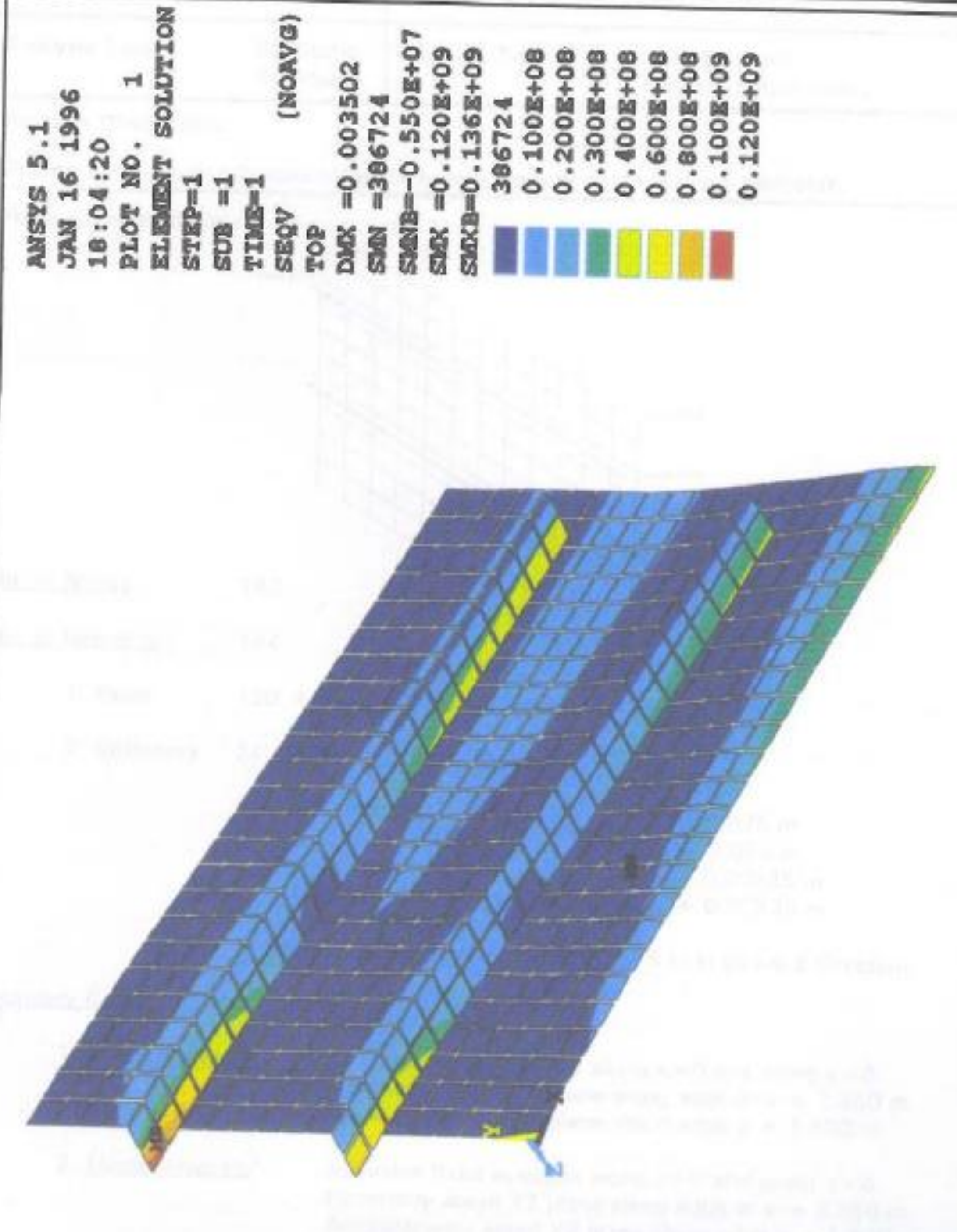
3rd Mode : 96.5



4th Mode : 106.5

Modal Analysis Results of Converged Model for BM-2 (ANSYS 5.1)

Benchmark No. :	BM-2-a	Benchmark Title :	Stiffened Plate



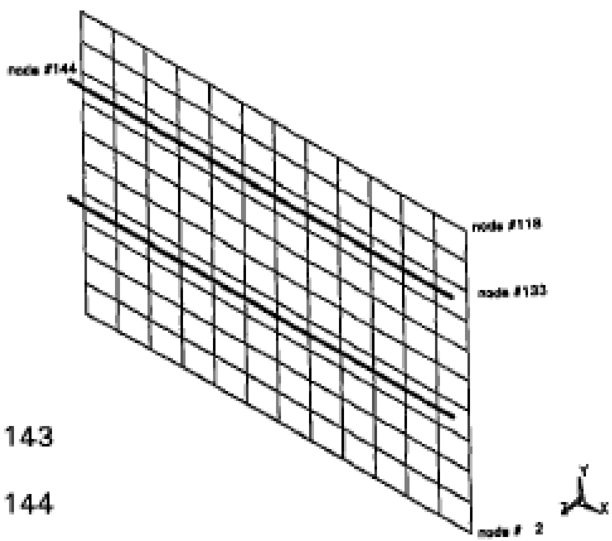
Benchmark 2 - Reference Solution

Benchmark No. :	BM-2-b	Benchmark Title :	Stiffened Panel
Analysis Type :	3D Static 3D Modal	Element Type(s) :	4-Node Shell 2-Node Offset Beam

Problem Description:

Repeat BM-2-a using 2-node offset beams in place of in-plane beam elements.

Finite Element Model :



No. of Nodes : 143

No. of Elements : 144

1. Panel — 120 4-Node 3-D Plate Elements — $t = 0.010$ m

2. Stiffeners — 24 2-Node 3-D Beam Elements**

$A = 0.001575$ m^2	$Y_t = 0.075$ m
$I_{zz} = 0.0145 \times 10^{-6}$ m^4	$Y_b = 0.075$ m
$I_{yy} = 2.95 \times 10^{-6}$ m^4	$Z_t = 0.00525$ m
$I_{xx} = 0.0553 \times 10^{-6}$ m^4 (Torsion)	$Z_b = 0.00525$ m

** Beam element centroid off-set 0.075 m in global Z direction.

Boundary Conditions :

1. Static Analysis
   - All nodes fixed at edges along x=0 and along y=0.
   - Symmetry about YZ plane along edge at x = 2.250 m
   - Symmetry about XZ plane along edge y = 1.500 m

2. Modal Analysis*
   - All nodes fixed at edges along x=0 and along y=0.
   - Symmetry about YZ plane along edge at x = 2.250 m
   - Antisymmetry about XZ plane along edge y = 1.500 m

* This benchmark test only requires calculation of the first four natural frequencies for symmetry / antisymmetry boundary conditions.

Benchmark No. : BM-2-b			Benchmark Title : Stiffened Plate		
Finite Element Software Results		**ANSYS 5.1**	**MSC / NASTRAN Windows 1**	**ALGOR 3.14**	**Converged Solution [1] (ANSYS 5.1)**
Element Types :	Plate Stiffeners	SHELL63 BEAM44	CQUAD4 CBEAM	TYPE 6 TYPE 2	SHELL93 SHELL93
Maximum Stresses	(MPa)				
1. Plate σ_{eqv} [2]	(node # 2)	42.1	38.2	34.4	42.1
2. Stiffeners σ_x [3]	(MPa)				
Tension	(node #133)	70.3	70.4	70.3	61.3
Compression	(node #144)	-153.7	-154.0	-153.7	-126.5
Maximum Deflections	(mm)				
Uz [4]	(node #118)	3.42	3.41	3.41	3.50
Natural Frequencies [5] :					
1st Mode	(Hz)	36.3	36.3	36.5	35.9
2nd Mode	(Hz)	61.1	61.2	61.7	61.0
3rd Mode	(Hz)	97.0	95.7	101.9	96.5
4th Mode	(Hz)	107.0	106.8	111.9	106.5

1. The "converged solution" results were obtained using a refined mesh model with 8-node shell elements on ANSYS 5.1. The von Mises Stress contours for the converged model are shown on Page D-13.

2. The maximum stress in the plate occurs at the middle of the long fixed edges (node 2). Reported are the maximum nodal averaged von Mises stress of the top or bottom surface of the plate elements. Note that different FEA programs may use different conventions for defining the top and bottom surfaces of plate elements. Also, different FEA programs use different extrapolation and averaging techniques for computing plate / shell element stresses which may lead to slight discrepancies (refer to BM-1-a for discussion).

3. Reported are the maximum stresses in the beam elements (axial stress + bending stress). The maximum tensile stress occurs at the centre of the middle stiffeners (node 133). The maximum compressive stress occurs at the fixed ends of the middle stiffeners (node 144). The off-set beam element introduces an artificial moment into the problem which results in over prediction of the stresses and under prediction of deflections. This effect also influences stress results for the plate elements. Refer to Example 1, Appendix C for further discussion of this effect.

4. The maximum out-of-plane deflection (Uz) occurs at the centre of the panel (node 118).

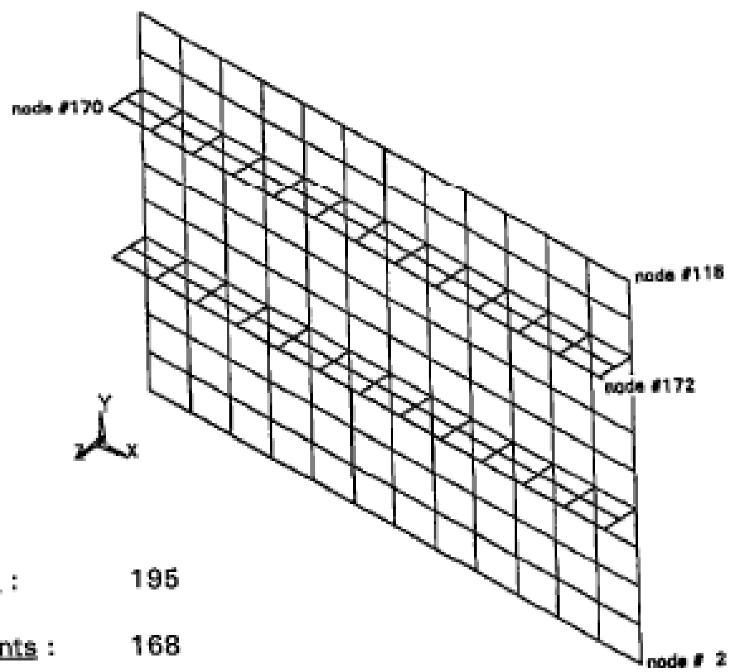
5. The frequencies and mode shapes for symmetry / antisymmetry boundary conditions from the converged model are shown on Page D-12. The mode shapes predicted by the BM-2-b FEA models are the same as those for the converged model.

Benchmark No. : BM-2-c	**Benchmark Title :** Stiffened Panel
Analysis Type : 3D Static 3D Modal	**Element Type(s) :** 4-Node Plate

Problem Description:

Repeat BM-2-a using 4-node plate elements to model the stiffeners and plate explicitly.

Finite Element Model :



No. of Nodes : 195

No. of Elements : 168

Panel	120	4-Node 3-D Plate Elements	t = 10 mm
Stiffeners	48	4-Node 3-D Plate Elements	t = 10.5 mm

Boundary Conditions :

1. Static Analysis
   - All nodes fixed at edges along x=0 and along y=0.
   - Symmetry about YZ plane along edge at x = 2.250 m
   - Symmetry about XZ plane along edge y = 1.500 m

2. Modal Analysis*
   - All nodes fixed at edges along x=0 and along y=0.
   - Symmetry about YZ plane along edge at x = 2.250 m
   - Antisymmetry about XZ plane along edge y = 1.500 m

* This benchmark test only requires calculation of the first four natural frequencies for symmetry / antisymmetry boundary conditions.

Benchmark No. : BM-2-c			Benchmark Title : Stiffened Plate		
Finite Element Software Results		**ANSYS 5.1**	**MSC / NASTRAN Windows 1**	**ALGOR 3.14**	**Converged Solution [1] (ANSYS 5.1)**
Element Types :	Plate Stiffeners	SHELL63 SHELL63	CQUAD4 CQUAD4	TYPE 6 TYPE 6	SHELL93 SHELL93
Maximum Stresses	(MPa)				
1. Plate σ_{eqv} [2]	(node # 2)	42.3	41.3	39.3	42.1
2. Stiffeners σ_x [3]	(MPa)				
Tension	(node #172)	68.9	69.0	68.2	61.3
Compression	(node #170)	-126.0	-126.0	-124.0	-126.5
Maximum Deflections	(mm)				
Uz [4]	(node #118)	3.47	3.43	3.42	3.50
Natural Frequencies [5] :					
1st Mode	(Hz)	36.1	36.2	36.1	35.9
2nd Mode	(Hz)	60.8	61.1	61.2	61.0
3rd Mode	(Hz)	95.0	94.9	97.4	96.5
4th Mode	(Hz)	104.9	105.8	106.3	106.5

1. The "converged solution" results were obtained using a refined mesh model with 8-node shell elements on ANSYS 5.1. The von Mises Stress contours for the converged model are shown on Page D-13.

2. The maximum stress in the plate occurs at the middle of the long fixed edges (node 2). Reported are the maximum nodal averaged von Mises stress of the top or bottom surface of the plate elements. Note that different FEA programs may use different conventions for defining the top and bottom surfaces of plate elements. Also, different FEA programs use different extrapolation and averaging techniques for computing plate / shell element stresses which may lead to slight discrepancies (refer to BM-1-a for discussion).

3. Reported are the maximum nodal averaged stresses, σ_x, in the stiffener plate elements (maximum of top or bottom surface stress). The maximum tensile stress occurs at the centre of the middle stiffeners (node 172). The maximum compressive stress occurs at the fixed ends of the middle stiffeners (node 170).

4. The maximum out-of-plane deflection (Uz) occurs at the centre of the panel (node 118).

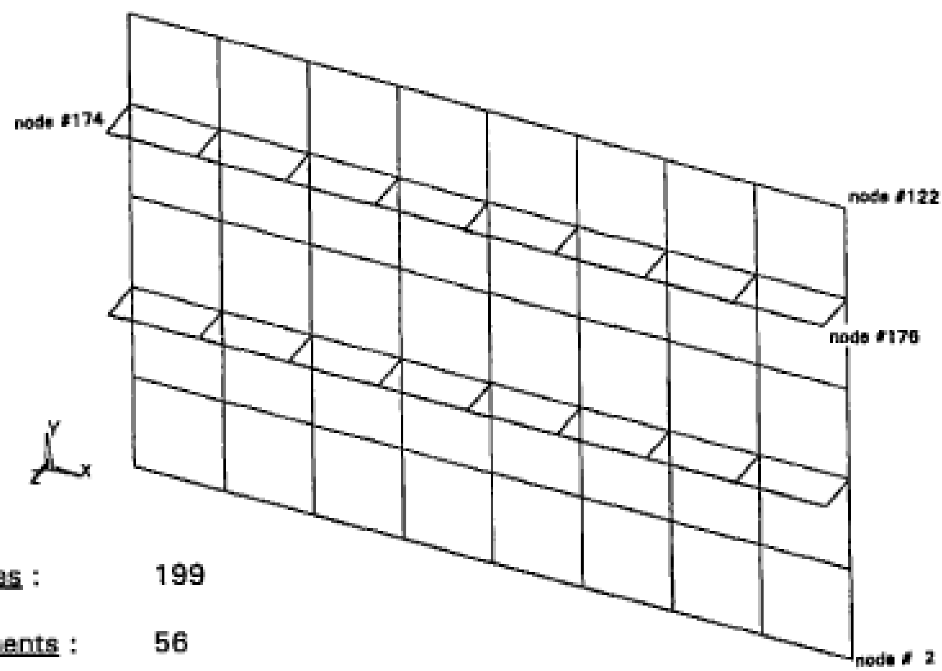
5. The frequencies and mode shapes for symmetry / antisymmetry boundary conditions from the converged model are shown on Page D-12. The frequencies and mode shapes predicted by the BM-2-c FEA models are very similar to those from the converged model.

Benchmark No. :	BM-2-d	**Benchmark Title :**	Stiffened Panel
Analysis Type :	3D Static 3D Modal	**Element Type(s) :**	8-Node Plate

Problem Description:

Repeat BM-2-a using 8-node plate elements to model the stiffeners and plate explicitly.

Finite Element Model :



No. of Nodes : 199

No. of Elements : 56

Panel	40	8-Node 3-D Plate Elements	t = 10 mm
Stiffeners	16	8-Node 3-D Plate Elements	t = 10.5 mm

Boundary Conditions :

1. Static Analysis
   - All nodes fixed at edges along x=0 and along y=0.
   - Symmetry about YZ plane along edge at x = 2.250 m
   - Symmetry about XZ plane along edge y = 1.500 m

2. Modal Analysis*
   - All nodes fixed at edges along x=0 and along y=0.
   - Symmetry about YZ plane along edge at x = 2.250 m
   - Antisymmetry about XZ plane along edge y = 1.500 m

* This benchmark test only requires calculation of the first four natural frequencies for symmetry / antisymmetry boundary conditions.

Benchmark No. : BM-2-d			Benchmark Title : Stiffened Plate		
Finite Element Software Results		**ANSYS 5.1**	**MSC / NASTRAN Windows 1**	**ALGOR**	**Converged Solution [1] (ANSYS 5.1)**
Element Types :	Plate Stiffeners	SHELL93 SHELL93	CQUAD8 CQUAD8	NA*	SHELL93 SHELL93
Maximum Stresses	(MPa)				
1. Plate σ_{eqv} [2]	(node # 2)	41.7	41.7	-	42.1
2. Stiffeners σ_x [3]	(MPa)				
Tension	(node #176)	69.9	69.9	-	61.3
Compression	(node #174)	-143.0	-143.0	-	-126.5
Maximum Deflections	(mm)				
Uz [4]	(node #122)	3.49	3.49	-	3.50
Natural Frequencies [5] :					
1st Mode	(Hz)	36.0	36.0	-	35.9
2nd Mode	(Hz)	61.0	61.0	-	61.0
3rd Mode	(Hz)	96.6	96.1	-	96.5
4th Mode	(Hz)	105.9	105.6	-	106.5

*ALGOR does not include 8-node plate elements for stress analysis.

1. The "converged solution" results were obtained using a refined mesh model with 8-node shell elements on ANSYS 5.1. The von Mises Stress contours for the converged model are shown on Page D-13.

2. The maximum stress in the plate occurs at the middle of the long fixed edges (node 2). Reported are the maximum nodal averaged von Mises stress of the top or bottom surface of the plate elements. Note that different FEA programs may use different conventions for defining the top and bottom surfaces of plate elements. Also, different FEA programs use different extrapolation and averaging techniques for computing plate / shell element stresses which may lead to slight discrepancies (refer to BM-1-a for discussion).

3. Reported are the maximum nodal averaged stresses, σ_x, in the stiffener plate elements (maximum of top or bottom surface stress). The maximum tensile stress occurs at the centre of the middle stiffeners (node 176). The maximum compressive stress occurs at the fixed ends of the middle stiffeners (node 174).

4. The maximum out-of-plane deflection (Uz) occurs at the centre of the panel (node 122).

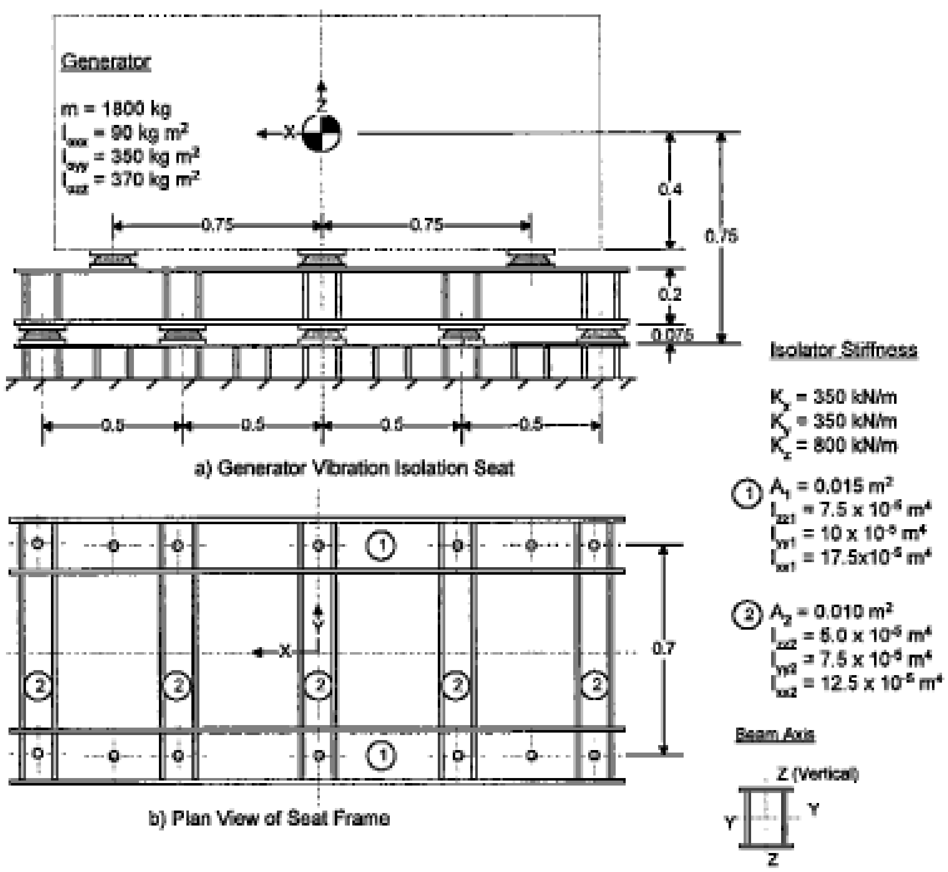
5. The frequencies and mode shapes for symmetry / antisymmetry boundary conditions from the converged model are shown on Page D-12. The frequencies and mode shapes predicted by the BM-2-d FEA models are very similar to those from the converged model, despite the relative coarseness of the mesh of the former.

Benchmark No. :	BM-3	**Benchmark Title :**	Machinery Vibration Isolation System
Analysis Type :	3D Modal	**Element Type(s) :**	3D Beams 1 DOF Springs (in X, Y, Z directions) Mass (with Rotational Inertia)

Problem Description:

Determine the natural frequencies for this generator vibration isolation system.

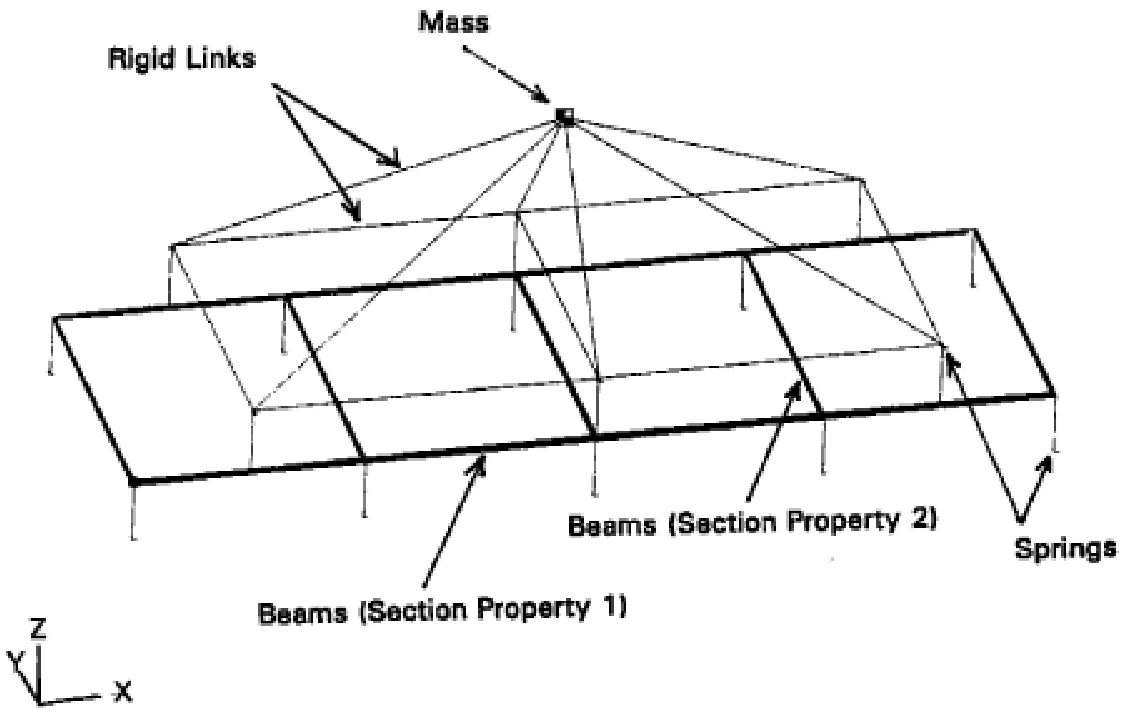
Sketch of Benchmark Problem :



Material Properties :		**Geometric Properties :**	**Loading :**
1. Steel	E = 207x10^3 MPa v = 0.3 ρ = 7850 kg/m^3	Refer to above sketch. Generator modelled as rigid link elements and point mass at centroid.	Not Applicable.
2. "Rigid" Links	E = 207x10^{4}MPa v = 0.3 ρ =0 kg/m^3		

Benchmark No. : BM-3	Benchmark Title : Machinery Vibration Isolation System

Finite Element Model :



No. of Nodes : 81

No. of Elements : 90

14	Beams (Section Property 1)
5	Beams (Section Property 2)
14	Springs (X-Direction)
14	Springs (Y-Direction)
14	Springs (Z-Direction)
1	Mass
51	Rigid Links

Boundary Conditions :

Isolator springs fixed at deck seating level.

Benchmark No. : BM-3	Benchmark Title : Machinery Vibration Isolation System			
Finite Element Software Results		**ANSYS 5.1**	**MSC / NASTRAN Windows 1**	**ALGOR 3.14**
FEA Software Element Types :		BEAM4 MASS21 COMBIN14	CBAR CONM2 CROD	TYPE 2 - TYPE 1 & 7
Total Mass and C of G Location :				
Total Mass (kg)		2545.7	2545.7	2545.7
C of G X (m)		1.0000	1.0000	1.0000
Y (m)		0.3500	0.3500	0.3500
Z (m)		0.4066	0.4066	0.4065
Modes and Frequencies (Hz)				
1 Translation in Y direction	(1st)	2.85	2.85	2.80
2 Translation in X direction	(1st)	3.60	3.60	3.66
3 Translation in Z direction	(1st)	6.30	6.30	6.30
4 Rotation about Z axis	(1st)	6.62	6.62	6.98
5 Rotation about Y axis	(1st)	9.61	9.61	10.04
6 Rotation about X axis	(1st)	11.12	11.12	11.45
7 Translation in X direction	(2nd)	14.76	14.76	14.89
8 Rotation about Z axis	(2nd)	15.28	15.28	16.61
9 Translation in Y direction	(2nd)	16.92	16.92	16.79
10 Translation in Z direction	(2nd)	21.51	21.51	21.51
11 Rotation about Y axis	(2nd)	22.86	22.86	23.60
12 Rotation about X axis	(2nd)	23.12	23.12	24.44

Comments on Benchmark Results :

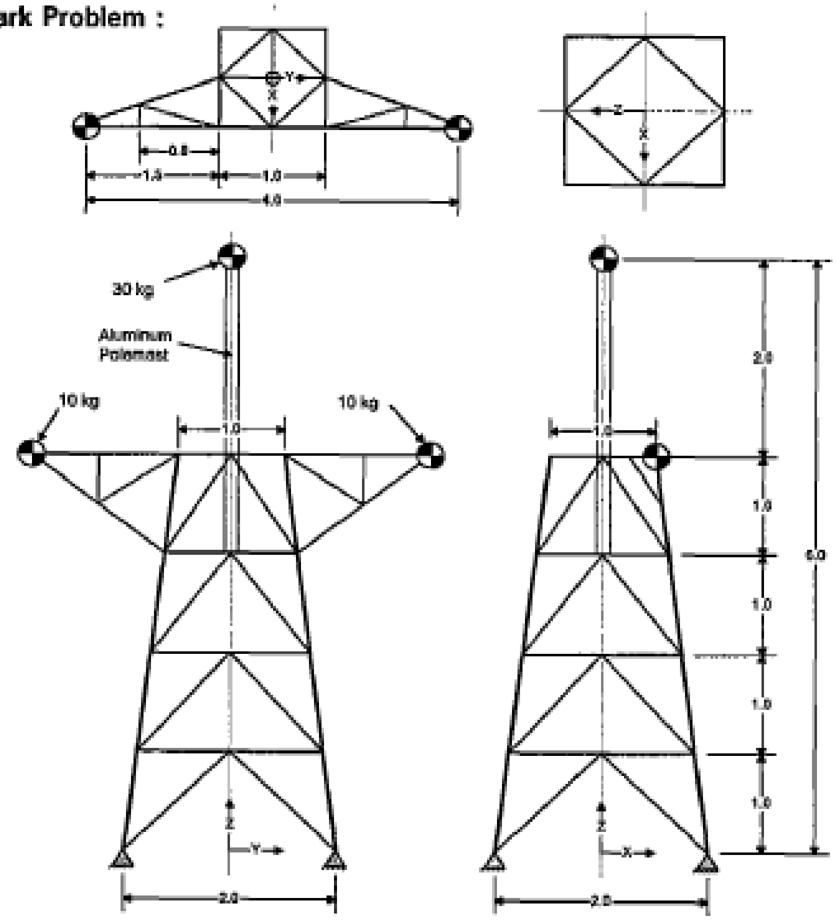
Modes 1 to 6 involve vibration modes with the generator and raft masses moving in phase, while the two masses are out-of phase for modes 7 to 12.

Benchmark No. :	BM-4	**Benchmark Title :**	Mast Structure
Analysis Type :	3D Static 3D Modal	**Element Type(s) :**	3D Beam 3D Spar Mass

Problem Description:

Determine the stresses, displacements, natural frequencies and modes under the specified loading conditions for the mast structure shown in the sketch below.

Sketch of Benchmark Problem :



Material Properties :	**Geometric Properties :**	**Loading :**
1. Steel $E = 207 \times 10^9 N/m^2$, $\nu = 0.3$, $\rho = 7850\ kg/m^3$ 2. Aluminum (pole mast) $E = 70 \times 10^9 N/m^2$, $\nu = 0.3$, $\rho = 2900\ kg/m^3$	Refer to table of section properties.	Accelerations $a_x = 5\ m/s^2$, $a_y = 5\ m/s^2$, $a_z = 15\ m/s^2$ Nodal Forces $F_x = 3000\ N$ (Applied on all nodes)

Benchmark No. : BM-4	Benchmark Title : Mast Structure

Member Section Properties

Section No.	Description	O. Dia. (m)	Area ($\times 10^{-3}$ m^2)	Izz & Iyy ($\times 10^{-6}$ m^4)	Ixx ($\times 10^{-6}$ m^4)	Element Type	No. Elems
1	Main Legs	0.12700	3.520	6.100	12.2	Beam	32
2	Pole Mast Support	0.09200	1.306	1.249	2.50	Beam	8
3	Vertical Braces 0.09200	1.306				Spar	32
4	Main Horizontals	0.07620	1.069	0.684	1.37	Beam	32
5	Pole Mast (Aluminum)	0.24130	4.887	33.70	67.4	Beam	5
8	Horizontal Braces	0.07302	1.100			Spar	16
9	Platform Braces	0.06040	0.693			Spar	10
10	Platform Chords	0.06040	0.693	0.2771	0.554	Beam	12

Finite Element Model :

The main legs, polemast, main horizontals and platform frame chords are modelled as continuous beams (ie. with full continuity), while the various brace members are modelled as spars with pinned ends.

No. of Nodes : 67

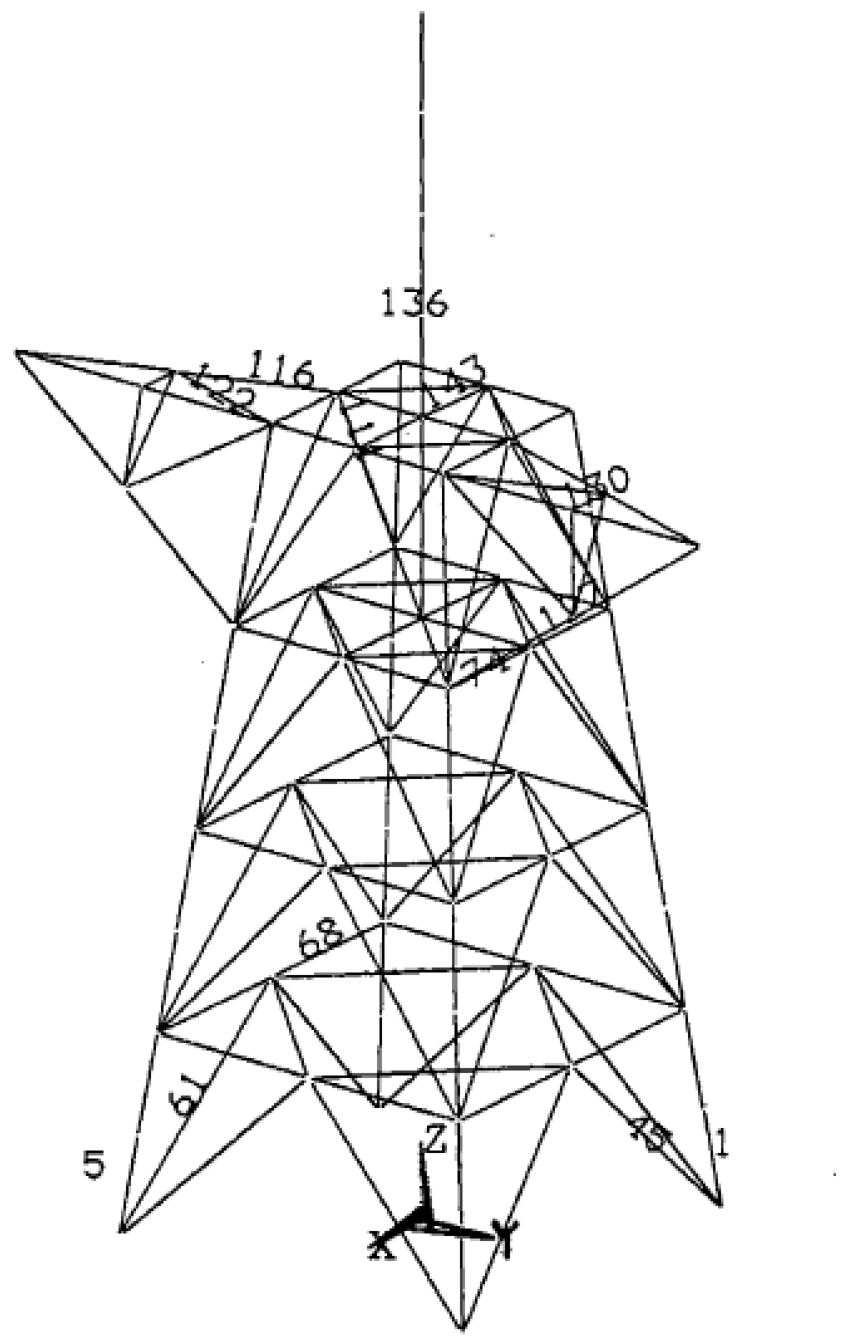
No. of Elements : 150

Boundary Conditions : UX, UY, & UZ translations of node at base of each leg restrained.

Static Analysis Loads : Nodal force of 3000 N in X direction (Fx) at every node.
Accelerations $a_x = 5$ m/s^2, $a_y = 5$ m/s^2, $a_z = 15$ m/s^2.

Benchmark No. : BM-4	**Benchmark Title :** Mast Structure

Plot of Finite Element Model Showing Critical Element Numbers :



Benchmark No. : BM-4				Benchmark Title : Mast Structure		
Finite Element Software Results				**ANSYS 5.1**	**MSC / NASTRAN Windows 1**	**ALGOR 3.14**
FEA Software Element Types :				BEAM4 LINK8 MASS21	CBAR CROD CONM2	TYPE 2 TYPE 1
Total Mass :	(kg)	m		1415.8	1415.8	1418.7
Centre of Gravity:	(m)	X		0.0336	0.0336	0.0335
		Y		0.0003	0.0003	0.0003
		Z		2.3797	2.3797	2.3841
Max. Deflections [1] :	(mm)	U_X	(node #63)	12.00	12.00	12.65
		U_Y	(node #63)	-0.36	-0.37	-0.41
		U_Z	(node #56)	-0.62	-0.62	-0.65
Total Reaction Forces :	(N)	F_X		-190920	-190921	NA*
		F_Y		7079	7079	
		F_Z		21236	21237	
Stresses (MPa) [2] :						
1. Main Legs	Max. Tensile		(el #1)	33.70	33.67	33.72
	Max. Compressive		(el #5)	-36.09	-36.11	-31.35
2. Pole Mast Support	Max. Tensile		(el #143)	99.42	99.41	95.85
	Max. Compressive		(el #142)	-108.96	-108.95	-97.76
3. Vertical Braces	Max. Tensile		(el #45)	34.94	34.94	38.15
	Max. Compressive		(el #61)	-35.54	-35.54	-37.78
4. Main Horizontals	Max. Tensile		(el #74)	48.41	48.40	47.81
	Max. Compressive		(el #68)	-38.11	-38.09	-39.61
5. Pole Mast	Max. Tensile		(el #136)	53.53	53.54	49.98
	Max. Compressive		(el #136)	-53.88	-53.86	-50.01
8. Horizontal Braces	Max. Tensile		(el #111)	10.77	10.77	10.97
	Max. Compressive		(el #109)	-4.32	-4.32	-4.29
9. Platform Braces	Max. Tensile		(el #130)	4.60	4.61	4.73
	Max. Compressive		(el #122)	-15.64	-15.64	-16.40
10. Platform Chords	Max. Tensile		(el #116)	71.90	71.92	75.97
	Max. Compressive		(el #127)	-73.43	-73.41	-74.85

Benchmark No. : BM-4	Benchmark Title : Mast Structure		
Finite Element Software Results	**ANSYS 5.1**	**MSC / NASTRAN Windows 1**	**ALGOR 3.14**
Modes and Frequencies : [3] (Hz)			
1 Pole Mast Cantilever Bending	20.75	20.76	20.72
2 Pole Mast Cantilever Bending	20.79	20.80	20.76
3 Local Bending of Main Horizontals	41.13	41.13	41.13
4 Platforms Bending in X Direction	47.46	47.46	47.45

Comments on Benchmark Results :

1. The maximum deflections in the X and Y directions occur at the top of the polemast. The maximum vertical deflection occurs at the starboard spur frame.

2. The stresses listed are the maximum combined (axial + bending) stress in the beam elements. Note that the maximum combined stresses are calculated assuming that the beam element section is rectangular, although the members in this particular problem are actually of circular cross section. This is a limitation of most FEA programs, although some programs may allow calculation of element stresses assuming a circular cross section. The simplified assumption of a square or rectangular cross section is conservative.

3. The first two vibration modes involve cantilever bending of the polemast in the X and Y directions (they are in fact identical modes due to the symmetry of the polemast). The third mode involves local bending of the main horizontal members of the mast. The fourth mode involves bending of the two platforms at the top of the mast in the X direction.

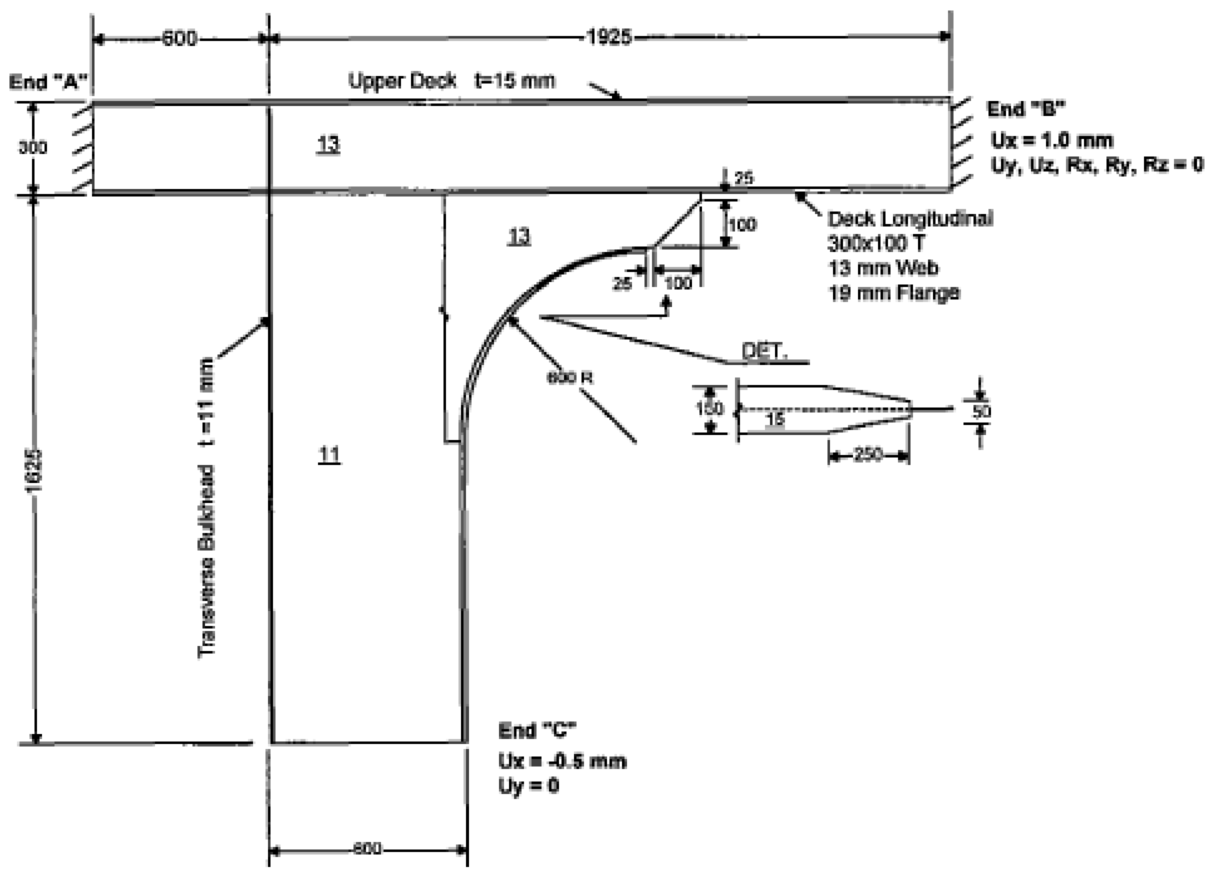
4. The ALGOR program requires a separate module to output reaction forces which is not included with the basic solution module and, as such, reaction results were not available. In addition, the ALGOR program does not include mass elements for linear static analysis. Instead, the inertia loads due to the payload masses were modelled by applying nodal forces at the appropriate locations. The difference in modelling approach and the inability to confirm the total applied loads may explain, in part, the differences in the ALGOR results compared to those obtained by ANSYS and NASTRAN.

Benchmark No. :	BM-5	Benchmark Title :	Bracket Detail
Analysis Type :	3D Static	Element Type(s) :	4-Node Thick Shell (With Transverse Shear)

Problem Description:

Determine the maximum stress for the VLCC Top Bracket detail shown in the sketch below.

Sketch of Benchmark Problem :

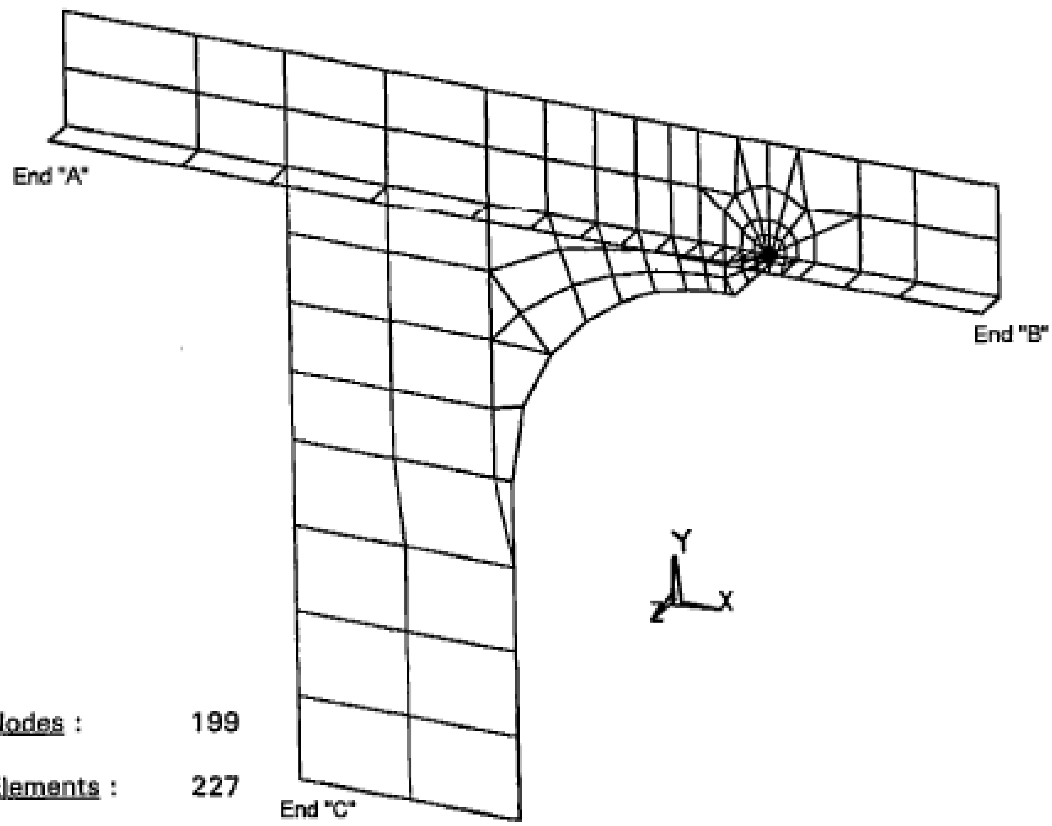


Material Properties :	Geometric Properties :	Loading :
$E = 207 \times 10^3$ MPa $\nu = 0.3$	As defined in above sketch.	Applied displacement constraints.

Benchmark No. : BM-5	Benchmark Title : Bracket Detail

Finite Element Model :

Thick shell / plate elements with transverse shear flexibility are used to model the bracket, deck longitudinal, and the web of the bulkhead stiffener. The transverse bulkhead, and upper deck are modelled using line elements of 40 t^2 section area (9000 mm^2 for deck, 4850 mm^2 for bulkhead). The flange of the bulkhead stiffener is modelled with line elements using the 2250 mm^2 area of the flange. The areas of the flange line elements taper down to 923 mm^2 at the end of the bracket. Line elements of a small arbitrary area (0.01 mm^2) are included at the toe of the bracket for obtaining stresses.



No. of Nodes : 199

No. of Elements : 227

Boundary Conditions :

Translation in Z direction restrained at nodes representing main deck and transverse bulkhead.

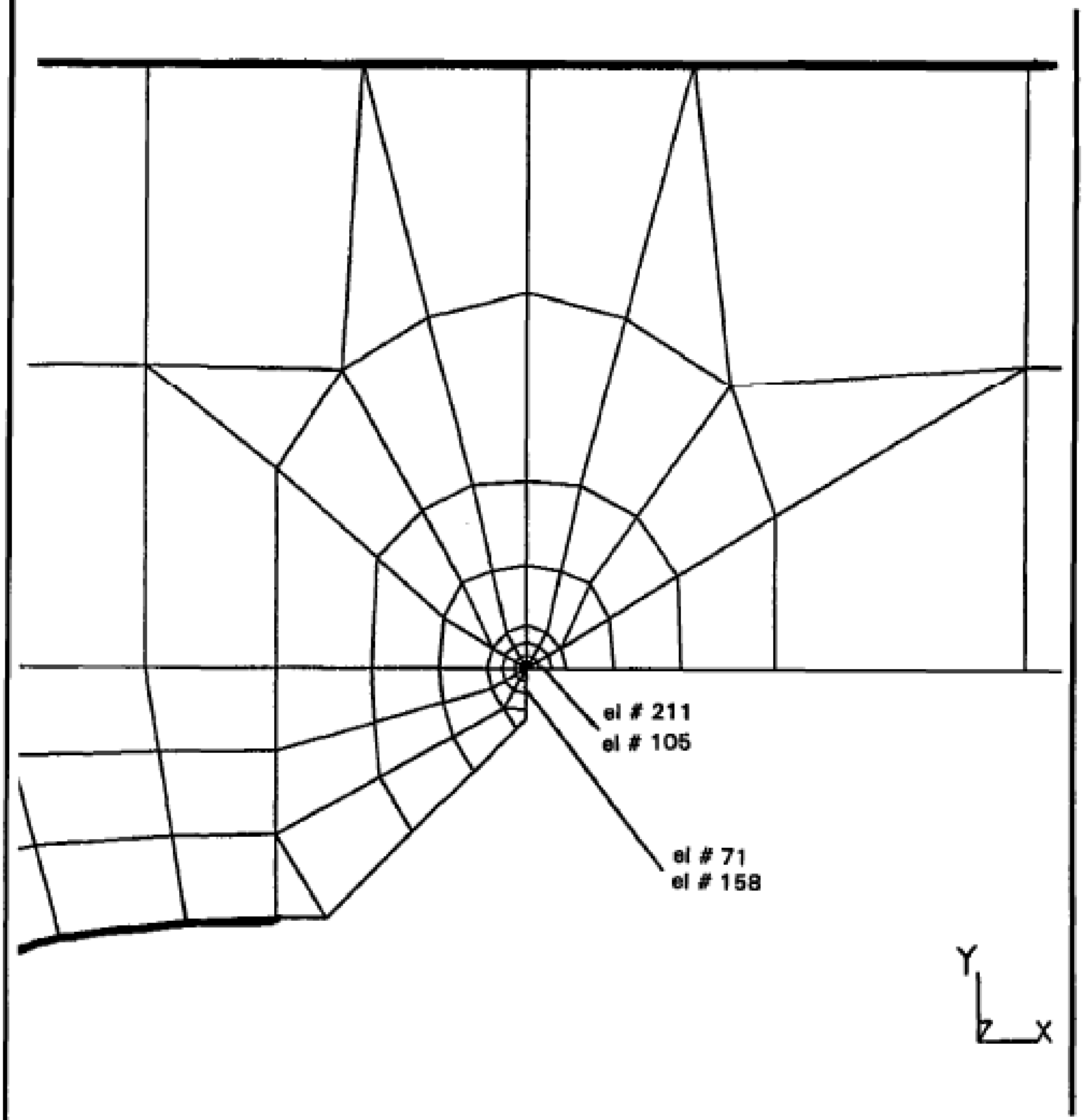
At end "A" of the model, all nodal degrees of freedom are fixed.

At end "B" of the model, a 1 mm displacement is applied in the positive X direction and all other nodal degrees of freedom are fixed.

At end "C" of the model, a 0.5 mm displacement is applied in the negative X direction and the vertical displacement in the Y direction is constrained to zero.

Benchmark No. : BM-5	Benchmark Title : Bracket Detail

Plot showing Critical Element Locations at Toe of Bracket :



Benchmark No. : BM-5	Benchmark Title : Bracket Detail			
Finite Element Software Results	**ANSYS 5.1**	**MSC / NASTRAN Windows 1**	**ALGOR**	**Converged Solution [1] (ANSYS 5.1)**
Element Types :	SHELL43 LINK8	CQUAD4 CROD	*NA	SHELL93 LINK8
Plate Element Stresses σ_{eqv} [2] (MPa)				
1. Bracket (el # 71)	209.3	209.6	-	203.5
2. Deck Long. Web (el # 105)	248.9	247.6	-	243.4
Edge Element Stresses σ_{x} (MPa)				
1. Bracket (el # 158)	119.8	121.5	-	133.0
2. Deck Long. Web (el # 211)	235.5	236.0	-	240.1
Maximum Deflections (mm)				
Ux (node # 86)	1.000	1.000	-	1.000
Uy (node #185)	-0.339	-0.336	-	-0.348
Uz (node #106)	-0.366	-0.354	-	-0.388
Reaction Forces at A : (N)				
Fx	-1194400	-1194700	-	-1191800
Fy	-28343	-28302	-	-26414
Fz	5967	6019	-	-5064

* ALGOR does not provide a thick shell element with transverse shear.

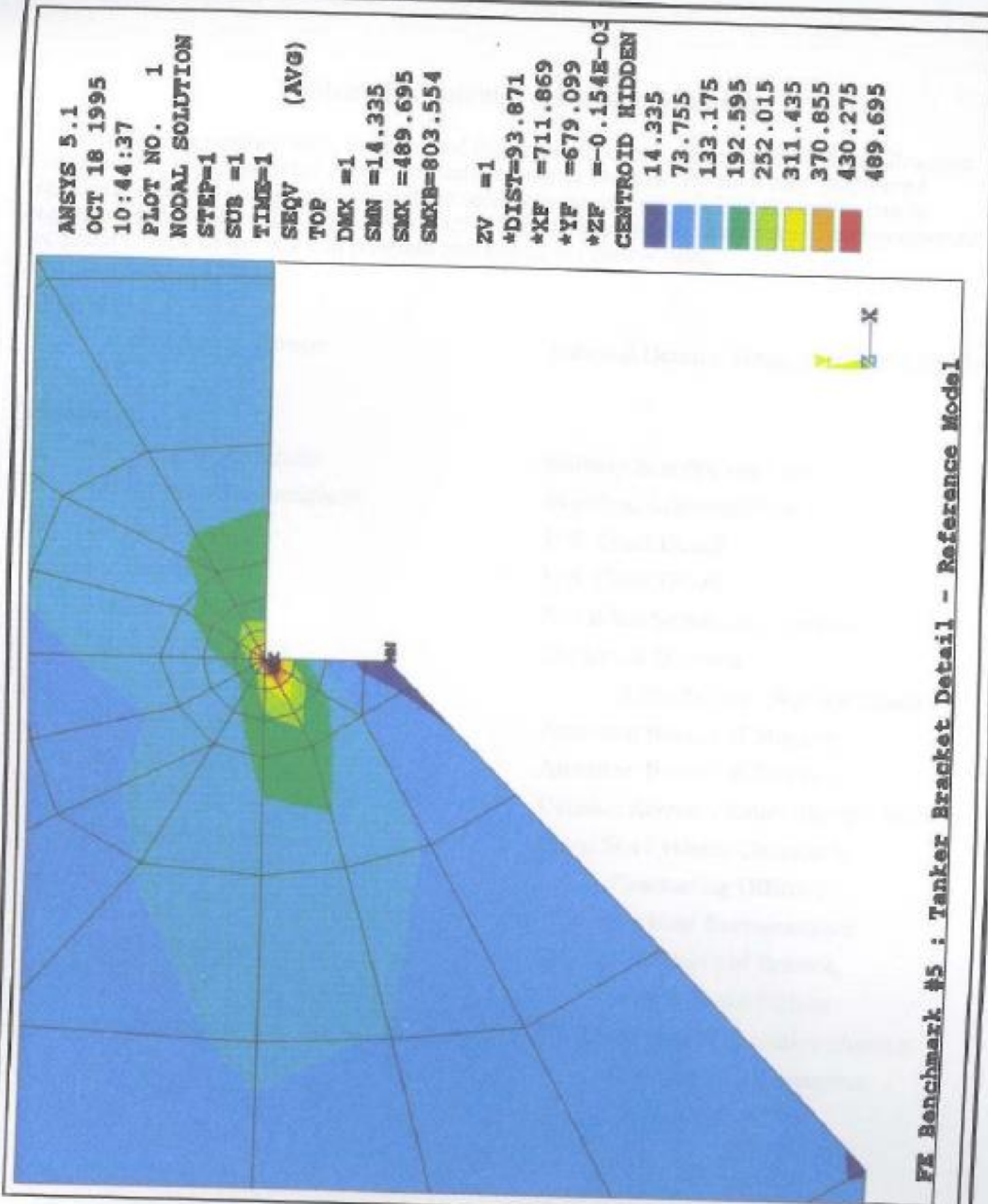
1. The "converged solution" results were obtained using a refined mesh model with 8-node shell elements on ANSYS 5.1. The von Mises Stress contours of the toe of the bracket for the converged model are shown on Page D-31. The stress contours are in units of MPa (N/mm^2).

2. This particular bracket detail problem is complicated by the existence of a stress singularity at the end corner or toe of the bracket. In a linear elastic analysis, the stress at this point is theoretically infinite. Refining the finite element mesh gives progressively higher stresses which are meaningless. One method which is commonly used to get around this problem is to use the so called "hot spot" stress. In calculating the hot spot stress no account is taken of the weld geometry, and in an idealised finite element representation (ignoring the weld), the stress is equal to the value at about one plate thickness from the corner (Chalmers, 1993). In this benchmark, the hot spot stress is calculated two ways :

   a) Using element centroidal von Mises stresses at the elements 10 mm from the corner (elements 71 and 105, see figure on Page D-29); and
   b) Using line element stresses at 10 mm from the corner (elements 158 and 211).

   The advantage of these methods are that they do not rely on the techniques used to extrapolate stresses to the node points which may vary for different FEA programs.

Benchmark No. : BM-5	Benchmark Title : Bracket Detail



REFERENCES

[1] NAFEMS. NAFEMS - Quality System Supplement to BS EN ISO 9001 Relating to Engineering Analysis in the Design and Integrity Demonstration of Engineered Products. 1999.

[2] Noor AK. Bibliography of Books and Monographs on Finite Element Technology. Appl Mech Rev 1991;44:307–17. doi:10.1115/1.3119505.

[3] Morris AJ. A practical guide to reliable finite element modelling. John Wiley & Sons; 2008.

[4] Mac Donald BJ. Practical stress analysis with finite elements. Glasnevin Publishing; 2011.

[5] Brauer JR. What every engineer should know about finite element analysis. M. Dekker; 1993.

[6] Meyer C, American Society of Civil Engineers. Structural Division. Task Committee on Finite Element Idealization. Finite element idealization for linear elastic, static, and dynamic analysis of structures in engineering practice. The Society; 1987.

[7] NAFEMS. NAFEMS Guidelines to finite element practice 1984.

[8] Steele JM. Applied finite element modeling : practical problem solving for engineers. M. Dekker; 1989.

[9] DNV GL. DNVGL-CG-0127 Finite element analysis. 2015.

[10] Altair. Altair OptiStruct 2019 Verification Problems. 2019.

[11] Altair. Altair HyperWorks 2019 Hardware Recommendations and Certifications, Version 1.5. 2019.

[12] BSI. BS 7910: Guide to methods for assessing the acceptability of flaws in metallic structures. BSI; 2015.

[13] American Petroleum Institute. API 579-1 / ASME FFS-1. 3rd ed. 2016.

[14] ABS. Guidance Notes on Ship Vibration #147. 2018.

[15] Altair University. Practical Aspects of Finite Element Simulation - A Study Guide. 2019.

[16] Siemens. Superelement User’s Guide. 2019.

[17] Sönnerlind H. Applying and Interpreting Saint-Venant’s Principle | COMSOL Blog 2018. https://www.comsol.com/blogs/applying-and-interpreting-saint-venants-principle/ (accessed June 4, 2019).

[18] Fung YC, Tong P. Classical and computational solid mechanics. World Scientific; 2001.

[19] Park G-J. Equivalent Static Loads Method for Non Linear Static Response Structural Optimization. 2010.

[20] Haufe A, Feucht M, Neukamm F, Dubois P. Recent Enhancements to the GISSMO Failure Model in LS-DYNA. 2011.

[21] Basaran M, Weichert D. An Extension of the GISSMO Damage Model Based on Lode Angle Dependence. 2010.

[22] Callister WD, Rethwisch DG. Materials Science and Engineering: An Introduction. 9th ed. 2013. doi:10.1007/3-540-29970-X_12.

[23] ASTM. E8/E8M − 16a Standard Test Methods for Tension Testing of Metallic Materials 1 2018. doi:10.1520/E0008_E0008M-16A.

[24] Dieter GE, Bacon DJ. Mechanical metallurgy. McGraw-Hill; 1988.

[25] Bonet J, Wood RD. Nonlinear continuum mechanics for finite element analysis. Cambridge: Cambridge University Press; 2008. doi:10.1017/CBO9780511755446.

[26] Ramberg W, Osgood WR. Technical Note No. 902: Description of stress–strain curves by three parameters. Washington DC.: 1943.

[27] Lemaître J, Chaboche J-L. Mechanics of solid materials. Cambridge University Press; 1990.

[28] Hallquist JO. LS-DYNA ® Theory Manual. 2006.

[29] Lemaitre J. A Continuous Damage Mechanics Model for Ductile Fracture. J Eng Mater Technol 1985;107:83. doi:10.1115/1.3225775.

[30] Lemaitre J. How to use damage mechanics. Nucl Eng Des 1984;80:233–45.

[31] Gurson AL. Continuum Theory of Ductile Rupture by Void Nucleation and Growth: Part I—Yield Criteria and Flow Rules for Porous Ductile Media. J Eng Mater Technol 1977;99:2. doi:10.1115/1.3443401.

[32] Opera Magistris. Opera Magistris (Elements of Applied Mathematics). 2016.

[33] Conle FA. Correcting Elastic FEA Results for Local Plasticity 2019. http://fde.uwaterloo.ca/Fde/Notches.new/neuber.html (accessed June 11, 2019).

[34] Neuber H. Theory of Stress Concentration for Shear-Strained Prismatical Bodies With Arbitrary Nonlinear Stress-Strain Law. J Appl Mech 1961;28:544. doi:10.1115/1.3641780.

[35] Topper TH, Wetzel RM, Morrow J. Neuber's Rule Applied to Fatigue of Notched Specimens. 1967.

[36] Tipton SM. A Review of the Development and Use of Neuber's Rule for Fatigue Analysis. SAE Trans 1991;100:143–8. doi:10.2307/44581113.

[37] Chu CC. A three-dimensional model of anisotropic hardening in metals and its application to the analysis of sheet metal formability. J Mech Phys Solids 1984;32:197–212. doi:10.1016/0022-5096(84)90008-5.

[38] Conle A, Oxland T, Topper TH. Computer-Based Prediction of Cyclic Deformation and Fatigue Behavior. Low Cycle Fatigue, 100 Barr Harbor Drive, PO Box C700, West Conshohocken, PA 19428-2959: ASTM International; 1988, p. 1218-1218–9. doi:10.1520/STP24548S.

[39] Forrest PG. Fatigue of metals. 1962.

[40] DNV GL AS. FE Model Library for Collision Analysis: Description and Results (2015-

0984.Rev1). 2016.

[41] Zerbst U, Ainsworth RA, Beier HT, Pisarski H, Zhang ZL, Nikbin K, et al. Review on fracture and crack propagation in weldments - A fracture mechanics perspective. Eng Fract Mech 2014. doi:10.1016/j.engfracmech.2014.05.012.

[42] NAFEMS. Fatigue and Fracture Mechanics in FEA - Part-1. 2001.

[43] NEFEMS. Fatigue and Fracture Mechanics in FEA - Part-4 2001.

[44] Turnquist M, Smith I, Eng P. A Life-Cycle Approach to the Assessment of Pipeline Dents. 2016.

[45] Lee Y-L, Barkey ME, Kang H-T. Metal fatigue analysis handbook : practical problem-solving techniques for computer-aided engineering. Butterworth-Heinemann; 2011.

[46] Hui CY, Ruina A. Why K? High order singularities and small scale yielding. Int J Fract 1995;72:97–120. doi:10.1007/BF00042823.

[47] Anderson TL. Fracture mechanics : fundamentals and applications. 4th ed. 2018.

[48] Irwin GR. Onset of fast crack propagation in high strength steel and aluminum alloys. 1956.

[49] Tada H, Paris PC, Irwin GR. The Stress Analysis of Cracks Handbook, Third Edition. Three Park Avenue New York, NY 10016-5990: ASME; 2000. doi:10.1115/1.801535.

[50] ASME. ASME Boiler and Pressure Vessel Code Complete Set. 2019.

[51] Hutchinson JW. Singular behaviour at the end of a tensile crack in a hardening material. J Mech Phys Solids 1968;16:13–31. doi:10.1016/0022-5096(68)90014-8.

[52] Rice JR, Rosengren GF. Plane strain deformation near a crack tip in a power-law hardening material. J Mech Phys Solids 1968;16:1–12. doi:10.1016/0022-5096(68)90013-6.

[53] ASTM. ASTM E1820 - 18ae1 Standard Test Method for Measurement of Fracture Toughness. 2018.

[54] ASTM. ASTM E399 - 17 Standard Test Method for Linear-Elastic Plane-Strain Fracture Toughness KIc of Metallic Materials. 2017.

[55] ASTM. ASTM E1290 - 08e1 Standard Test Method for Crack-Tip Opening Displacement (CTOD) Fracture Toughness Measurement (Withdrawn 2013). 2013.

[56] BSI. BS 7448: Fracture Mechanics Toughness Tests Part 1, 2, 3 and 4. 2005.

[57] ISO. ISO 12737: Metallic materials: determination of plane-strain fracture toughness. British Standards Institution; 2011.

[58] Williams ML. On the Stress Distribution at the Base of a Stationary Crack. J Appl Mech 1935;24:109–14.

[59] O’Dowd NP, Shih CF. Family of crack-tip fields characterized by a triaxiality parameter—I. Structure of fields. J Mech Phys Solids 1991;39:989–1015. doi:10.1016/0022-5096(91)90049-T.

[60] O’Dowd NP, Shih CF. Family of crack-tip fields characterized by a triaxiality parameter—

II. Fracture applications. J Mech Phys Solids 1992;40:939–63. doi:10.1016/0022-5096(92)90057-9.

[61] Yang S, Chao YJ, Sutton MA. Higher order asymptotic crack tip fields in a power-law hardening material. Eng Fract Mech 1993;45:1–20. doi:10.1016/0013-7944(93)90002-A.

[62] Chao YJ. J-A 2 Characterization of Crack-Tip Fields: Extent of J-A 2 Dominance and Size Requirements. Int J Fract 1998;89:285–307.

[63] EDF Energy. R6-Revision 4, Assessment of the integrity of the structures containing defects, Amendment 11. Gloucester: 2015.

[64] ABS. Rules for Building and Classing Marine Vessels. Houston,TX: 2019.

[65] ABS. Rules for Building and Classing Steel Vessels. Houston, TX: 2019.

[66] Hughes OF, Paik JK, Béghin D. Ship structural analysis and design. Society of Naval Architects and Marine Engineers; 2010.

[67] Cook RD, Malkus DS, Plesha ME. Concepts and applications of finite element analysis. 1989.

[68] Jayadevan KR, Thaulow C, Østby E, Berg E, Skavllerud B, Holthe K, et al. Structural integrity of pipelines: T- stress by line- spring. Fatigue Fract Eng Mater Struct 2005;28:467–88. doi:10.1111/j.1460-2695.2005.00889.x.

[69] MAESTRO. MAESTRO Version 11.7. Manual. 2018.

[70] NAPA. Structural Design 2019. https://www.napa.fi/software-and-services/ship-design/structural-design/ (accessed September 11, 2019).

[71] Lee Y-L, Barkey ME, Kang H-T. Metal Fatigue Analysis Handbook: Practical Problem-solving Techniques for Computer-aided Engineering. vol. 0. 2011.

[72] NAVSEA. Handbook for The Design and Assessment of Naval Surface Ship Structure, MIL-HDBK-519. 2009.

[73] Michaelson RW. User’s Guide for SPECTRA: Version 8.3 (NSWCCD-65-TR-2000/07) 2000.

[74] Sikora JP, Michaelson RW, Ayyub BM. Assessment of Cumulative Lifetime Seaway Loads for Ships. Nav Eng J 2002;114:167–80. doi:10.1111/j.1559-3584.2002.tb00129.x.

[75] Sikora JP. Cumulative lifetime loadings for naval ships 1998;56:299–305.

[76] Ma M, Zhao C, Hughes O. A practical method to apply hull girder sectional loads to full-ship 3D finite-element models using quadratic programming. Ships Offshore Struct 2014;9:257–65. doi:10.1080/17445302.2013.770725.

[77] SINTEF. MARINTEK, VERES Version 3.01.0096 2012. https://www.sintef.no/en/software/shipx/ (accessed September 11, 2019).

[78] WAMIT. Wamit, Inc. - The State of the Art in Wave Interaction Analysis. 2019.

[79] Bureau Veritas. HydroStar 2019. https://marine-offshore.bureauveritas.com/software (accessed September 11, 2019).

[80] Daalen E van, Sireta F. PRECAL_R User Manual 2014.

[81] ABS. ‘Dynamic Loading Approach" for Floating Production, Storage and Offloading (FPSO) Installations. 2017.

[82] Malenica S, Stumpf E, Sireta F-X, Chen X-B. Consistent Hydro-Structure Interface for Evaluation of Global Structural Responses in Linear Seakeeping. Vol. 6 Nick Newman Symp. Mar. Hydrodyn. Yoshida Maeda Spec. Symp. Ocean Sp. Util. Spec. Symp. Offshore Renew. Energy, ASME; 2008, p. 61–8. doi:10.1115/OMAE2008-57077.

[83] Ma M, Zhao C, Danese N. A Method of Applying Linear Seakeeping Panel Pressure to Full Ship Structural Models, Conference on Computer Applications and Information Technology in the Maritime Industries, Leige, Belgium, 2012.

[84] Ochi MK, Cambridge University Press. Ocean Waves : the Stochastic Approach. Cambridge University Press; 1998.

[85] Paik JK, Thayamballi AK. Ultimate limit state design of steel plated structures. J. Wiley; 2003.

[86] ABS. High-Speed Naval Craft 2017.

[87] Gerrit Visser. Frequency Response Analysis - What is It? 2018.

[88] Joseph Servidio RA, Quinn Director JP, Gascon J, Baetsen K, Lantz J, Curry A, et al. SSC-470 Finite Element Modeling Methods: Vibration Analysis for Ships. 2015.

[89] Cook RD (Robert D. Finite element modeling for stress analysis. Wiley; 1995.

[90] Thompson MK, Thompson JM. ANSYS Mechanical APDL for Finite Element Analysis. 1st ed. Butterworth-Heinemann; 2017.

[91] ADINA R&D. ADINA Handbook. 2017.

[92] Kuna M (Meinhard). Finite elements in fracture mechanics : Theory--Numerics--Applications. Springer; 2013.

[93] Kardestuncer H, Norrie DH, Brezzi F (Franco). Finite element handbook. McGraw-Hill; 1987.

[94] ADINA R&D. Theory and Modeling Guide Volume II: Thermal. vol. II. 2010.

[95] Siemens. FEA - Powering today’s most advanced engineering analysis environment. 2019.

[96] Dassault Systems. ABAQUS Unified FEA Simulation Realistic Performance with Advanced Multiphysics Solutins. 2019.

[97] BETA Simulation Solution. ANSA - The Advanced CAE Pr-Processing Software for Complete Model Build Up. 2019.

[98] Altair University. Introduction to Nonlinear Finite Element Analysis using Optistruct. 2018.

[99] Kuna M. Finite Elements in Fracture Mechanics. vol. 201. 2013. doi:10.1007/978-94-007-6680-8.

[100] Petti JP, Dodds RH. Ductile tearing and discrete void effects on cleavage fracture under small-scale yielding conditions. Int J Solids Struct 2005. doi:10.1016/j.ijsolstr.2004.11.015.

[101] Healy B, Gullerud A, Koppenhoefer K, Roy A, Roychowdhury S, Walters M, et al. WARP3D-Release 17.9.3 3-D Dynamic Nonlinear Fracture Analyses of Solids Using Parallel Computers. 2018.

[102] Koppenhoefer KC, Dodds RH. Ductile crack growth in pre-cracked CVN specimens: Numerical studies. Nucl Eng Des 1998. doi:10.1016/S0029-5493(97)00218-5.

[103] Neukamm F, Feucht M, Haufe A. Considering damage history in crashworthiness simulations. 2009.

[104] Connor J a. WG. Computer-Aided Teaching of Finite Element Displacement Method (MIT Report 69). 1969.

[105] Bala S, Day J. General Guidelines for Crash Analysis in LS-DYNA ® Modeling Guidelines for Crash Analysis. 2003.

[106] Det Norske Veritas. DNV-RP-C208 Determination of Structural Capacity by Non-linear FE analysis Methods. 2013.

[107] As DG. DNVGL-RP-C204 Design against accidental loads. 2017.

[108] ADINA R&D. Theory and Modeling Guide Volume IV: EM. vol. IV. 2005.

[109] Wang E, De Waele W, Hertelé S. A complementary η_pl approach in J and CTOD estimations for clamped SENT specimens. Eng Fract Mech 2015.

[110] BSI. BS 7608: Guide to fatigue design and assessment of steel products. BSI; 2015.

[111] Hadley I, Horn A. Treatment of constraint in BS 7910:2013, ISO 27306 and DNVGL-RP-F108. Int J Press Vessel Pip 2019;169:77–93. doi:10.1016/J.IJPVP.2018.11.015.

[112] ASME. ASME B31G - 2012 Manual for Determining the Remaining Strength of Corroded Pipelines. ASME International; 2017.

[113] American Petroleum Institute. API RP 1176: Recommended Practice for Assessment and Management of Cracking in Pipelines. 1st ed. API; 2016.

[114] American Petroleum Institute. API 579-2 / ASME FFS-2. 1st ed. API; 2009.

[115] Zerbst U, Ainsworth RA, Madia M. Reference load versus limit load in engineering flaw assessment: A proposal for a hybrid analysis option. Eng Fract Mech 2012;91:62–72. doi:10.1016/J.ENGFRACMECH.2011.10.018.

[116] Pyttel B, Varfolomeyev I, Luke M, Berger C, Siegele D. FKM guideline “fracture mechanics proof of strength for engineering components” - overview and extension topics. Weld World 2007. doi:10.1007/BF03266576.

[117] Wang E, De Waele W, Hertelé S. A complementary ηpl approach in J and CTOD estimations for clamped SENT specimens. Eng Fract Mech 2015;147. doi:10.1016/j.engfracmech.2015.07.043.

[118] Brocks W, Cornec A, Scheider I. Computational Aspects of Nonlinear Fracture Mechanics. 2002.

[119] Brocks W. Elastic-plastic fracture mechanics. Solid Mech Its Appl 2018:49–84. doi:10.1007/978-3-319-62752-6_5.

[120] Siegele D, Schmitt W. Determination and simulation of stable crack growth in ADINA. Comput Struct 1983;17:697–703. doi:10.1016/0045-7949(83)90083-4.

[121] Brocks W, Yuan H. Numerical investigations on the significance of J for large stable crack growth. Eng Fract Mech 1989;32:459–68. doi:10.1016/0013-7944(89)90317-2.

[122] Brocks W, Eberle A, Fricke S, Veith H. Large stable crack growth in fracture mechanics specimens. Nucl Eng Des 1994;151:387–400. doi:10.1016/0029-5493(94)90183-X.

[123] Jayadevan KR, Berg E, Thaulow C, Østby E, Skallerud B. Numerical investigation of ductile tearing in surface cracked pipes using line-springs. Int J Solids Struct 2006;43:2378–97. doi:10.1016/J.IJSOLSTR.2005.06.098.

[124] Berg E, Skallerud B, Thaulow C. Two-parameter fracture mechanics and circumferential crack growth in surface cracked pipelines using line-spring elements. Eng Fract Mech 2008;75:17–30. doi:10.1016/J.ENGFRACMECH.2007.03.023.

[125] Needleman A. An analysis of decohesion along an imperfect interface. Int J Fract 1990;42:21–40. doi:10.1007/BF00018611.

[126] Needleman A. Some Issues in Cohesive Surface Modeling. Procedia IUTAM 2014;10:221–46. doi:10.1016/J.PIUTAM.2014.01.020.

[127] Yuan H, Lin G, Cornec A. Verification of a Cohesive Zone Model for Ductile Fracture. J Eng Mater Technol 1996;118:192. doi:10.1115/1.2804886.

[128] ADINA R&D. Theory and Modeling Guide Volume I : ADINA Solids & Structures. vol. I. 2017.

[129] Hao S, Brocks W. The Gurson-Tvergaard-Needleman-model for rate and temperature-dependent materials with isotropic and kinematic hardening. Comput Mech 1997;20:34–40. doi:10.1007/s004660050213.

[130] Sun X, Wimmer SA, Karrt DG. Shear Band Initiation of Brittle Damage Materials. Int J Damage Mech 1996;5:403–21. doi:10.1177/105678959600500404.

[131] Tvergaard V. On localization in ductile materials containing spherical voids. Int J Fract 1982;18:237–52. doi:10.1007/bf00015686.

[132] As DG. CLASS GUIDELINE Fatigue assessment of ship structures. 2018.

[133] DNV GL. Recommended Practice DNV GL AS - Fatigue design of offshore steel structures. 2016.

[134] DNV GL. Plus-extended fatigue analysis of ship details. 2016.

[135] Basu RI, Kirkhope KJ, Srinivasan J. SSC-387 Guideline for Evaluation of Finite Element and Results. 1996.

[136] United States Naval Academy. EN358 :: Naval Architecture & Ocean Engineering :: USNA. Annapolis: 2009.

[137] NSWCCD. Modeling Guidelines Full Ship FEM Development (Revision G). 2013.

[138] Avi E, Lillemäe I, Romanoff J, Niemelä A. Equivalent shell element for ship structural design. Ships Offshore Struct 2015;10:239–55. doi:10.1080/17445302.2013.819689.

[139] SSC. FINITE ELEMENT MODELING METHODS: VIBRATION ANALYSIS FOR SHIPS. 2015.

[140] Altair University. Learn Dynamic Analysis with Altair Optistruct. 2018.

[141] Irons B, Ahmad S. Techniques of finite elements. E. Horwood; 1980.

[142] Steele JM. Applied finite element modeling : practical problem solving for engineers. M. Dekker; 1989.

[143] Crisfield MA. Non-linear finite element analysis of solids and structures. Vol. 1, Essentials. Wiley; 2000.

[144] Belytschko T, Liu WK, Moran B. Nonlinear finite elements for continua and structures. Wiley; 2000.

[145] Bathe K-J. Finite element procedures in engineering analysis. Prentice-Hall; 1982.

[146] Bathe K-J, Bathe K-J. Finite element procedures. Prentice Hall; 1996.

[147] Bucalém ML, Bathe K-J. The mechanics of solids and structures : hierarchical modeling and the finite element solution. Springer; 2011.

[148] Chapelle D (Dominique), Bathe K-J. The finite element analysis of shells. Fundamentals. Springer; 2011.

[149] Bathe K-J. Finite element procedures. 1996.

[150] Holzapfel G. Nonlinear solid mechanics: a continuum approach for engineering science. Meccanica 2002. doi:10.1023/A:1020843529530.

[151] De Souza Neto EA, Perić D, Owen DRJ. Computational Methods for Plasticity: Theory and Applications. Comput Methods Plast Theory Appl 2008. doi:10.1002/9780470694626.

[152] Implicit: Checklist — Welcome to the LS-DYNA support site 2019. https://www.dynasupport.com/howtos/implicit/implicit-checklist (accessed June 11, 2019).

[153] Hilber HM, Hughes TJR, Taylor RL. Improved numerical dissipation for time integration algorithms in structural dynamics. Earthq Eng Struct Dyn 1977;5:283–92. doi:10.1002/eqe.4290050306.

[154] Paik JK. Practical techniques for finite element modeling to simulate structural crashworthiness in ship collisions and grounding (Part I: Theory). Ships Offshore Struct 2007;2:69–80. doi:10.1533/saos.2006.0148.

[155] Paik JK. Practical techniques for finite element modelling to simulate structural crashworthiness in ship collisions and grounding (Part II: Verification). Ships Offshore Struct 2007;2:81–5. doi:10.1533/saos.2006.0149.

[156] Abambres M, Arruda MR. Finite element analysis of steel structures – a review of useful guidelines. Int J Struct Integr 2016;7:490–515. doi:10.1108/IJSI-07-2015-0020.

[157] Amdahl J, Ehlers S, Leira BJ. Collision and grounding of ships and offshore structures : proceedings of the 6th International Conference on Collision and Grounding of Ships and Offshore Structures, ICCGS, Trondheim, Norway, 17-19 June 2013. CRC Press/Balkema; 2013.

[158] Storheim M. Structural Response in Ship-Platform and Ship-Ice Collisions. NTNU; 2016.

[159] Benson DJ. The History of LS-DYNA®. 2007.

[160] ADINA R&D. ADINA Primer. 2017.

[161] Altair University. Introduction to Explicit Analysis unsing RADIOSS - A Study Guide. 2018.

[162] Hourglass — Welcome to LS-DYNA Examples 2019. https://www.dynaexamples.com/introduction/process_simulation/hourglass (accessed July 22, 2019).

[163] LSTC. Mass scaling. LSDYNA Support 2019. https://www.dynasupport.com/howtos/general/mass-scaling (accessed July 23, 2019).

[164] LSTC. What is mass scaling? DYNA Support 2019. https://www.dynasupport.com/faq/general/what-is-mass-scaling (accessed July 23, 2019).

[165] Bala S. Overview of Mass-Scaling in LS-DYNA. D3view 2006. https://www.d3view.com/2006/10/overview-of-mass-scaling/ (accessed July 23, 2019).

[166] Altair University. Learn Fatigue Analysis with Altair OptiStruct. 2018.

[167] Goodman J. Mechanics Applied to Engineering. Longmans, Green & Company; 1899.

[168] Hertzberg RW, Vinci RP, Hertzberg JL. Deformation and Fracture Mechanics of Engineering Materials. John Wiley & Sons, Inc; 2013. doi:10.1093/nq/s6-XII.297.185-c.

[169] Wang E. A study on CMOD-based compliance calibration functions for clamped single-edge tension specimens. J Test Eval 2015;43. doi:10.1520/JTE20130333.

[170] Wang E, Zhou W, Shen G, Duan D. An experimental study on J(CTOD)-R curves of single edge tension specimens for X80 steel. Proc. Bienn. Int. Pipeline Conf. IPC, vol. 4, 2012. doi:10.1115/IPC2012-90323.

[171] Xue Thinkviewer L, Xue L, Wierzbicki T. Verification of a New Fracture Criterion Using LS-DYNA Sandia fracture challenge View project Ductile fracture modeling and formulation of damage plasticity theory View project Verification of a New Fracture Criterion Using LS-DYNA. 2006.

[172] SSC. SSC-472-Survivability of Hull Girder in Damaged Condition. 2015.

[173] SSC. SSC-459-Reliability-based Performance Assessment of Damaged Ships. 2011.

[174] SSC. SSC-409-Guide to damage toelrance analysis of marine structures. 2000.

[175] SSC. SSC-386-Ship Maintenance Project - Vol. 2 Corrosion Damage Evaluation. 1995.

[176] SSC. SSC-386-Ship Maintenace Project - Vol.1 Fatigue Damage Evaluation. 1995.

[177] Poudret J, Huther M, Jean P, Transport G, Vauhan H. SSC-81symp10-Grounding of a Membrane Tanker; Correlation Between Damage Predictions and Observations. 1981.

[178] Ainsworth RA, Hooton DG. R6 and R5 procedures: The way forward. Int J Press Vessel Pip 2008;85:175–82. doi:10.1016/J.IJPVP.2007.10.003.

[179] Kanninen MF, Broek D, Marschall CW, Rybicki E. Mechanical fracture predictions for sensitized stainless steel piping with circumferential cracks [NP-192]. 1976.

[180] CSA. CSA Z662-15. 6th ed. 2015.

[181] CEPA. CEPA Recommended Practices for Managing Near-neutral pH Stress Corrosion Cracking 3 rd edition. 2015.

[182] Hogström P, Ringsberg JW, Johnson E. Survivability analysis of a struck ship with damage opening – influence from model and material properties uncertainties. Ships Offshore Struct 2011;6:339–54. doi:10.1080/17445302.2010.500052.

[183] Hogström P, Ringsberg JW. An extensive study of a ship's survivability after collision – A parameter study of material characteristics, non-linear FEA and damage stability analyses. Mar Struct 2012;27:1–28. doi:10.1016/J.MARSTRUC.2012.03.001.

[184] Storheim M, Notaro G, Johansen A, Amdahl J. Comparison of ABAQUS and LS-DYNA in simulations of ship collisions. Iccgs-2016 2016:239–46.

[185] Storheim M, Amdahl J. Design of offshore structures against accidental ship collisions. Mar Struct 2014;37:135–72. doi:10.1016/J.MARSTRUC.2014.03.002.

[186] Storheim M, Amdahl J. On the sensitivity to work hardening and strain-rate effects in nonlinear FEM analysis of ship collisions. Ships Offshore Struct 2017. doi:10.1080/17445302.2015.1115181.

[187] Cowper GR, Symonds PS. Strain-Hardening and Strain-Rate Effects in the Impact Loading of Cantilever Beams. 1957. doi:10.21236/AD0144762.

[188] Johnson GR, Cook WH. Fracture characteristics of three metals subjected to various strains, strain rates, temperatures and pressures. Eng Fract Mech 1985;21:31–48. doi:10.1016/0013-7944(85)90052-9.

[189] Storheim M, Alsos HS, Hopperstad OS, Amdahl J. A damage-based failure model for coarsely meshed shell structures. Int J Impact Eng 2015;83:59–75. doi:10.1016/J.IJIMPENG.2015.04.009.

[190] Storheim M, Amdahl J. Non-Conservative Consequences of "Conservative" Assumptions in Ship-Platform Collision Analysis. Vol. 4B Struct. Saf. Reliab., ASME; 2014, p. V04BT02A034. doi:10.1115/OMAE2014-24457.

[191] MARPOL73-78: Brief history - list of amendments to date and where to find them. MARPOL73-78 Br Hist - List Amend to Date Where to Find Them 2012.

[192] BMT. FlawCheck Structural Integrity Assessment Tool. Ottawa, Canada: 2019.