



GUIDANCE NOTES ON

SAFEHULL FINITE ELEMENT ANALYSIS OF HULL STRUCTURES

DECEMBER 2004 (Updated February 2014 – see next page)

**American Bureau of Shipping
Incorporated by Act of Legislature of
the State of New York 1862**

**Copyright © 2004
American Bureau of Shipping
ABS Plaza
16855 Northchase Drive
Houston, TX 77060 USA**

Updates

February 2014 consolidation includes:

- December 2004 version plus Corrigenda/Editorials



GUIDANCE NOTES ON

SAFEHULL FINITE ELEMENT ANALYSIS OF HULL STRUCTURES

CONTENTS

SECTION 1	Introduction	1
1	Objectives	1
3	Scope of Application	1
5	Overview of Analysis Procedure	1
7	Required Analyses	2
9	Supplementary Analyses	2
FIGURE 1 Overview of the FEA Model Functions and Analyses for SafeHull TSA		2
SECTION 2	Global 3D FEA Model	4
1	General	4
3	Extent of the Global 3D FEA Model	4
5	Coordinate and Unit System of the Global Model	8
7	Element Types and Combination	9
9	Finite Element Modeling	10
9.1	Model for Responses	10
9.3	Element Size	10
11	FEA Model Verification	16
13	SafeHull Verification Tools	17
TABLE 1 Recommended Baseline Mesh Size and Mesh Order for Global Model		11
FIGURE 1 Extent of the 3D Global Model and Mesh Arrangement for Tankers		5
FIGURE 2 Extent of the 3D Global Model and Mesh Arrangement for Bulk Carriers		6
FIGURE 3 Extent of the 3D Global Model and Mesh Arrangement for Container Carriers		7
FIGURE 4 Model Coordinate System		8
FIGURE 5 Typical Mesh Arrangement of Aframax Oil Tankers		12
FIGURE 6 Typical Mesh Arrangement of VLCC Oil Tankers		13

FIGURE 7	Typical Mesh Arrangement of Cape Size Bulk Carriers	14
FIGURE 8	Typical Mesh Arrangement of Container Carriers	15
FIGURE 9	SafeHull Nastran Model Check Tool.....	17
FIGURE 10	SafeHull Groups and Displays the Model by Grouping Parts	17
SECTION 3	Loading and Boundary Conditions	18
1	General	18
3	Loading and Loading Patterns	18
5	Load Combination Factor and Load Cases	19
7	Identify Compartments	19
9	Apply Load Components to Compartment Boundaries	20
11	Pressure Superposition and Force Conversion	20
13	Hull Girder Loads and Load Equilibrium	21
15	Boundary Conditions	22
15.1	Supporting Rod and Its Property	22
15.3	Tankers.....	23
15.5	Bulk Carriers	24
15.7	Containership Models	25
15.9	Boundary Constraint Beams on the Two End Sections	26
TABLE 1	Steel Vessel Rules Specified Loading Criteria (150 m or More in Length)	18
TABLE 2	Steel Vessel Rules Specified Loading Patterns.....	19
TABLE 3	Steel Vessel Rules Specified Loading Combination Factors	19
FIGURE 1	SafeHull “View Tank Boundaries” and “Search Log File” Tools	20
FIGURE 2	SafeHull “View Tank and Frame Load” Tool.....	21
FIGURE 3	View Curves of Hull Girder Load in the SafeHull System	22
FIGURE 4	Spring Supports for Tanker Global Models.....	23
FIGURE 5	Spring Supports at Fore End of Bulk Carrier Global Models.....	24
FIGURE 6	Spring Supports at Aft End for Container Carrier Global Models.....	25
FIGURE 7	Spring Supports at Fore End for Container Carrier Global Models.....	26
SECTION 4	Evaluation	27
1	General	27
3	Checking Global Model Response	27
5	Plate Panels for Evaluation.....	29
7	Yielding Strength.....	29
7.1	Yielding Criteria for Watertight Members	29
7.3	Yielding Criteria for Non-watertight Members	30

9	Buckling Strength.....	30
9.1	Buckling Criteria for Watertight Members	30
9.3	Buckling Criteria for Non-Watertight Members	30
11	Fatigue Strength	30
TABLE 1	Steel Vessel Rules-specified Loading Criteria (150 meters or more in length)	27
FIGURE 1	View FEA Solution in SafeHull System.....	28
FIGURE 2	Overall Deformation of Three Cargo Model for Load Case 1	28
SECTION 5	Critical Areas	32
1	Tanker	32
1.1	Transverse Web	32
1.3	Horizontal Stringer Sections	34
1.5	Longitudinal Girder Sections	34
3	Bulk Carrier	34
5	Containership	38
FIGURE 1	Critical Areas – Tankers	32
FIGURE 2	Critical Areas for Typical General Bulk Carriers	36
FIGURE 3	Critical Areas for Typical Containerships, Transverse Sections	39
FIGURE 4	Critical Areas for Typical Containerships, Deck & Longitudinal Section.....	40
SECTION 6	Frequently Asked Questions.....	41
	Element types for 3D fine mesh global model	41
	Element size for 3D fine mesh global model	41
	Small access opening for 3D fine mesh global model.....	41
	Boundary condition for 3D fine mesh global model	41
	Failure Criteria (Yielding & Buckling) for Fine Mesh Models	41
	3D Buckling Criteria	42
	3D Yielding Criteria	42
	2D Yielding Criteria	42
	2D Buckling Criteria	42
	Mesh size for local 3D zooming analysis	42
	Critical Area for local 3D zooming analysis	42
	Critical Area for fatigue zooming analysis.....	43
SECTION 7	References	44

This Page Intentionally Left Blank

Copyright 2014 by ABS

SECTION 1 Introduction

1 Objectives

This document provides the guidelines of the Finite Element Analysis (FEA) approach for ABS SafeHull Total Strength Assessment (TSA) of hull structures, as specified in the *ABS Rules for Building and Classing Steel Vessels*, which are referred to herein as the “*Steel Vessel Rules*”. The analysis approach considered in this Guide uses a 3D global FEA model with sufficient mesh density to assess both global hull-girder and local main supporting structural yielding and buckling strengths. A local very fine mesh model may be required when a detail design or critical structural area needs to be analyzed. This approach is referred to as “Global 3D Fine Mesh FEA Method”. It differs from an earlier method that used separate steps for global coarse mesh and local fine mesh analyses. Structural idealization, load application, analysis procedures and evaluation of analysis results are carried out in a consistent manner based on sound engineering practice. This Guide also provides the necessary information to help the users with the ABS SafeHull TSA system.

3 Scope of Application

This Guide is provided for the finite element structural analysis of tankers*, bulk carriers and container carriers, as specified in the *Steel Vessel Rules*. The instructions given regarding load application and strength evaluation are the loading (both the load cases and loading patterns) and strength criteria specified in the *Steel Vessel Rules* for assessing these ship types.

The strength criteria in Part 5C of the *Steel Vessel Rules* are based on a “net” ship approach. Therefore, the corrosion margins specified in the *Steel Vessel Rules* are to be deducted from the design scantlings for the finite element modeling and strength assessment of the hull structures.

This Guide presents information relating to the “Global 3D Fine Mesh FEA” method, as required to receive the ABS SafeHull Notation. Information related to the critical areas, local finer mesh and very fine mesh modeling and analyses are also discussed here. The users should follow the Rules-required and supplementary analyses, as shown in Section 1, Figure 1, as well as the model special requirements to determine the FEA modeling and analysis.

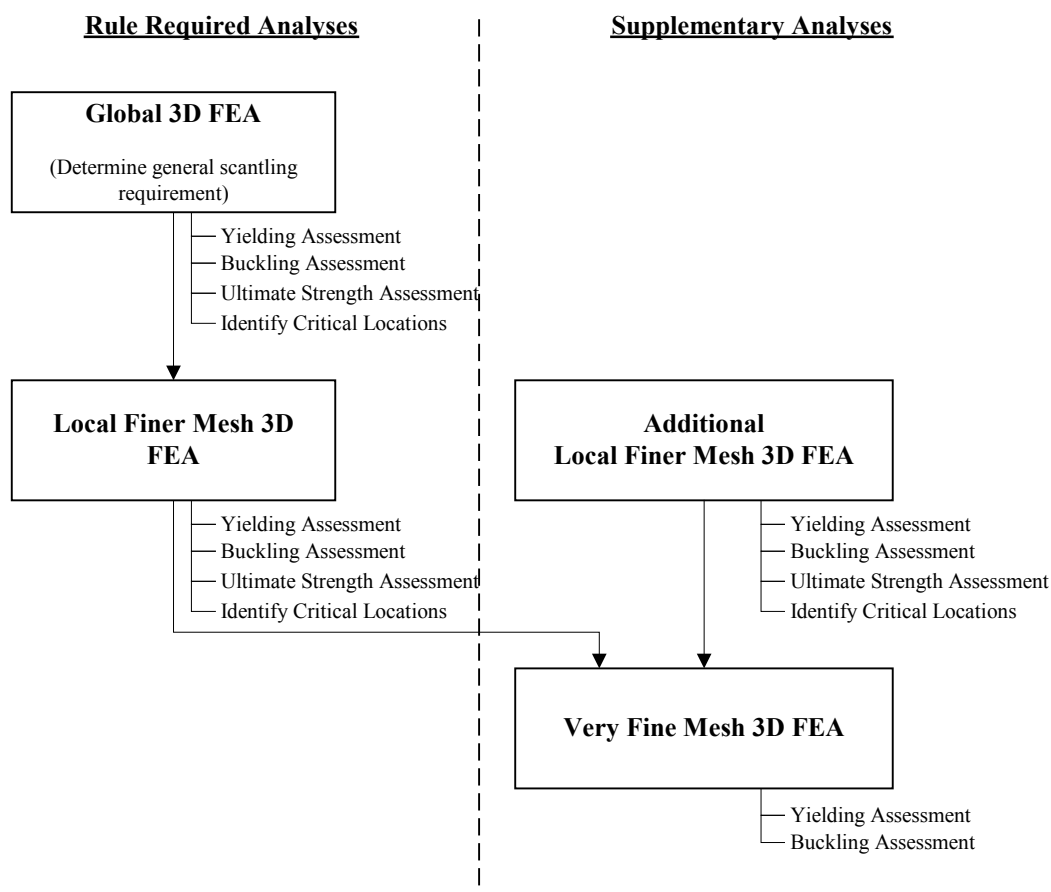
* *Note:* Similarly, the finite element structural analysis as specified in this Guide may be used for the analysis of membrane tank SH LNG carriers. The information on the LNG carriers, typical modeling and example of analysis will be provided by the ABS Engineering Office, if necessary, until the next edition of the Guide is published.

5 Overview of Analysis Procedure

The SafeHull FEA analysis is used to determine the structural responses of the vessels to the load cases specified in the *Steel Vessel Rules*. This requires developing a series of FEA models of the ship structures and applying the *Steel Vessel Rules*-specified loading conditions to determine the deflections and stresses of the structures. The solved FEA results are evaluated against the *Steel Vessel Rules* acceptance strength criteria. ABS has developed the SafeHull system to help the users perform the mentioned loading and evaluation analyses for global, local fine mesh and fatigue fine mesh models.

FEA is an open-ended, general-purpose tool that is used to varying degrees and levels of details ranging from the general global structural responses to very fine detail stress concentrations. The analysis scope and level of details must be matched to the acceptance criteria. The Rules include acceptance criteria associated with certain levels of the analysis performed on different FE models. An overview of the Rules-required and supplementary analyses, as well as the functions of the different levels of the FEA models and corresponding evaluations are shown in Section 1, Figure 1. However, it is understood that users may want to perform additional supplementary analyses for better understanding of the vessel’s structural behavior and strength.

FIGURE 1
Overview of the FEA Model Functions and Analyses for SafeHull TSA



7 Required Analyses

As shown in Section 1, Figure 1, SafeHull total strength assessment requires the global and local finite element analyses in association with the *Steel Vessel Rules*-specified loading and evaluation criteria to ensure the ship structures meet the yielding, buckling and ultimate strength requirements. The global model analysis mainly focuses on the general scantling requirements for the structural members, such as longitudinal members (watertight), transverse bulkheads (watertight or non-tight) and main supporting members (non-tight), while the local finer mesh model analysis mainly focuses on the special requirements for the “critical areas”, such as the connections, openings, bracket toes, structural knuckle points, corners of large openings and so on.

9 Supplementary Analyses

Users may perform additional finer mesh analyses to verify that the structures are adequately designed. As no two structures are exactly alike, load generation techniques and analysis objectives differ. A certain amount of engineering judgment must be used to decide the level and detail to which the analysis should be carried out.

Issues that may dictate the need for additional analysis are listed, but not limited to, as follows:

- Known problem areas of similar structural arrangements or details
- The areas where relatively high stress levels were found in Global 3D FEA
- Suspected areas or details not adequately represented in the global models

Areas that might be subjected to finer mesh analyses are listed, but not limited to, as follows:

- Access openings in horizontal girders
- Bracket toes of main supporting structures
- Hopper corner details
- Hatch corners

Section 5, “Critical Areas”, of this Guide includes a discussion of typical critical areas that might be subjected to additional finer mesh analyses.

SECTION 2 Global 3D FEA Model

1 General

The global 3D FEA model is a representative of the hull structures of three cargo holds (or tanks, in the case of an oil carrier) with the middle cargo hold within $0.4L$ amidships. It is used to determine both the global response of the hull girder and local behavior of the main supporting structures. The stress results from such models must be suitable for strength evaluation of the watertight boundaries of cargo holds (or tanks) and non-tight main supporting structures. The global 3D FE analyses establish the scantling requirements of plates and stiffeners, and they are sufficient for establishing the steel weight estimate. Structural details are evaluated by the subsequent local 3D FE analyses.

The strength assessment procedures in this Guide are based on a “net” ship approach, wherein the nominal design corrosion values are to be deducted. The “net” thickness or scantlings correspond to the minimum strength requirements acceptable for classification, regardless of the design service life of the vessel. In addition to the coating protection specified in the *Steel Vessel Rules* for all ballast tanks, the minimum corrosion values for plating and structural members, as given in the *Steel Vessel Rules* (Section 5C-1-3 for oil carriers, Section 5C-3-3 for bulk carriers, Section 5C-5-3 for container carriers) are to be applied. These minimum values are used solely for the above purpose and are not to be construed as renewal standard.

In view of higher corrosion rates for structural members in some regions, such as high-stressed areas, additional design margins are considered for the primary and critical structural members to minimize repairs and maintenance costs. The beneficial effects of these design margins on the reduction of stress and the increased effectiveness of the hull girder section modulus can be appropriately accounted for in the design evaluation. However, extra scantlings, such as owner-specified additional thickness, included in the vessel’s design specifications, should not be used in the finite element models.

3 Extent of the Global 3D FEA Model

To evaluate the vessel’s structures within $0.4L$ amidships with reasonable accuracy, the finite element models ideally place the target cargo hold in the middle and extend approximately the length of the adjacent holds fore and aft. In addition, there is a short extension beyond the transverse bulkheads at both ends.

Section 2, Figures 1, 2 and 3 show the extent of the typical finite element models for oil carriers, bulk carriers and container carriers, respectively.

Even though the ABS SafeHull system can handle both full and half-width models, it is recommended that the finite element models should be created with both the port and starboard sides of cargo hold structures, that are symmetrical with respect to the centerline, for easier review, result analysis and subsequent strength evaluation.

Within $0.4L$ amidships of some container carriers, the length of the parallel mid-body is usually less than one cargo hold length. During the initial design stage, only the midship section drawing is available. In this case, it is assumed that all three cargo holds have the same prismatic shape in order to determine the scantlings of the typical transverse sections and double bottom structures. However, the lower part of the hull structures should be evaluated using the actual hull shape. Accordingly, it is advisable to start with a simple prismatic model followed by an actual shape model.

FIGURE 1
Extent of the 3D Global Model and Mesh Arrangement for Tankers

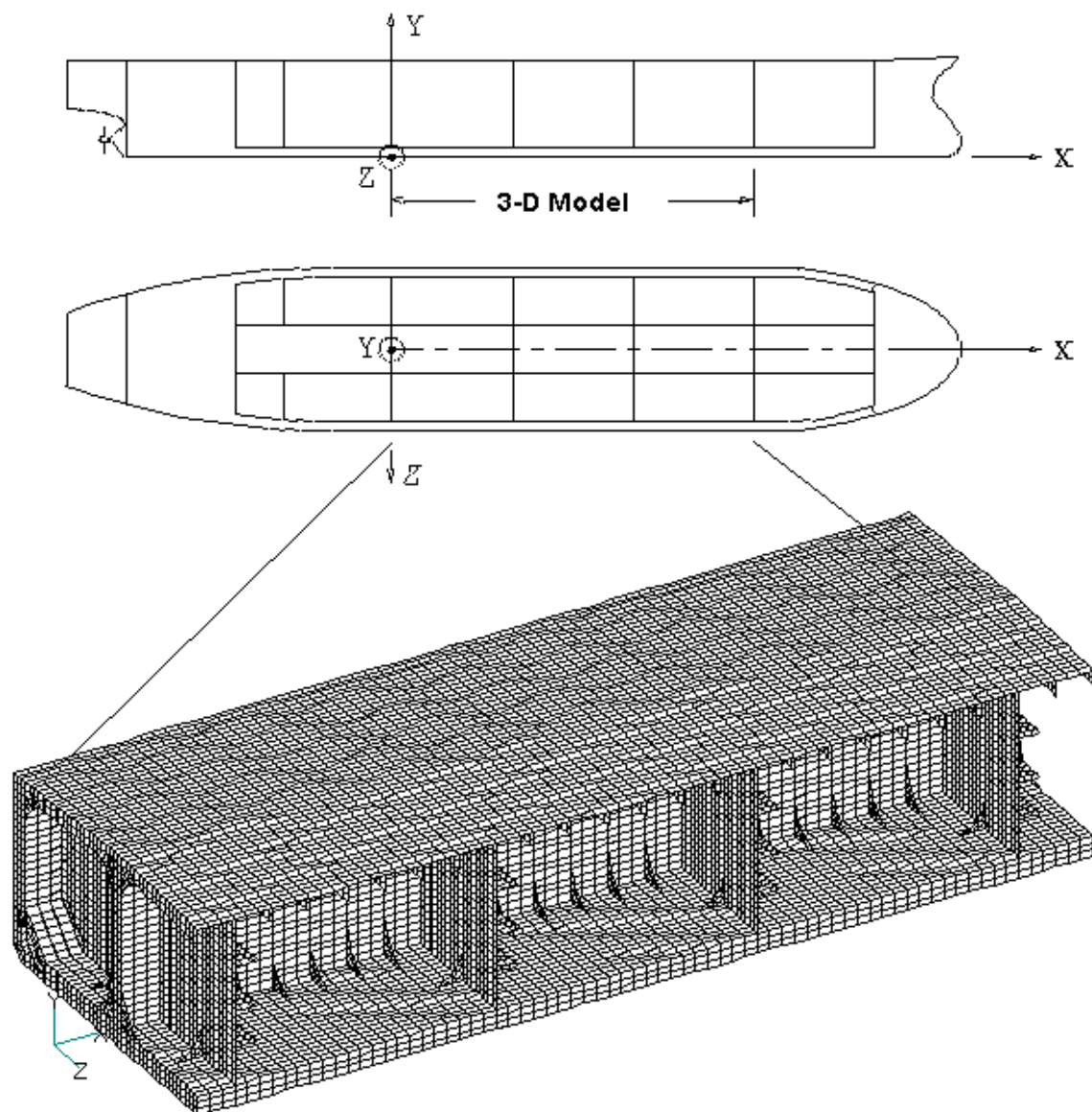


FIGURE 2
Extent of the 3D Global Model and Mesh Arrangement for Bulk Carriers

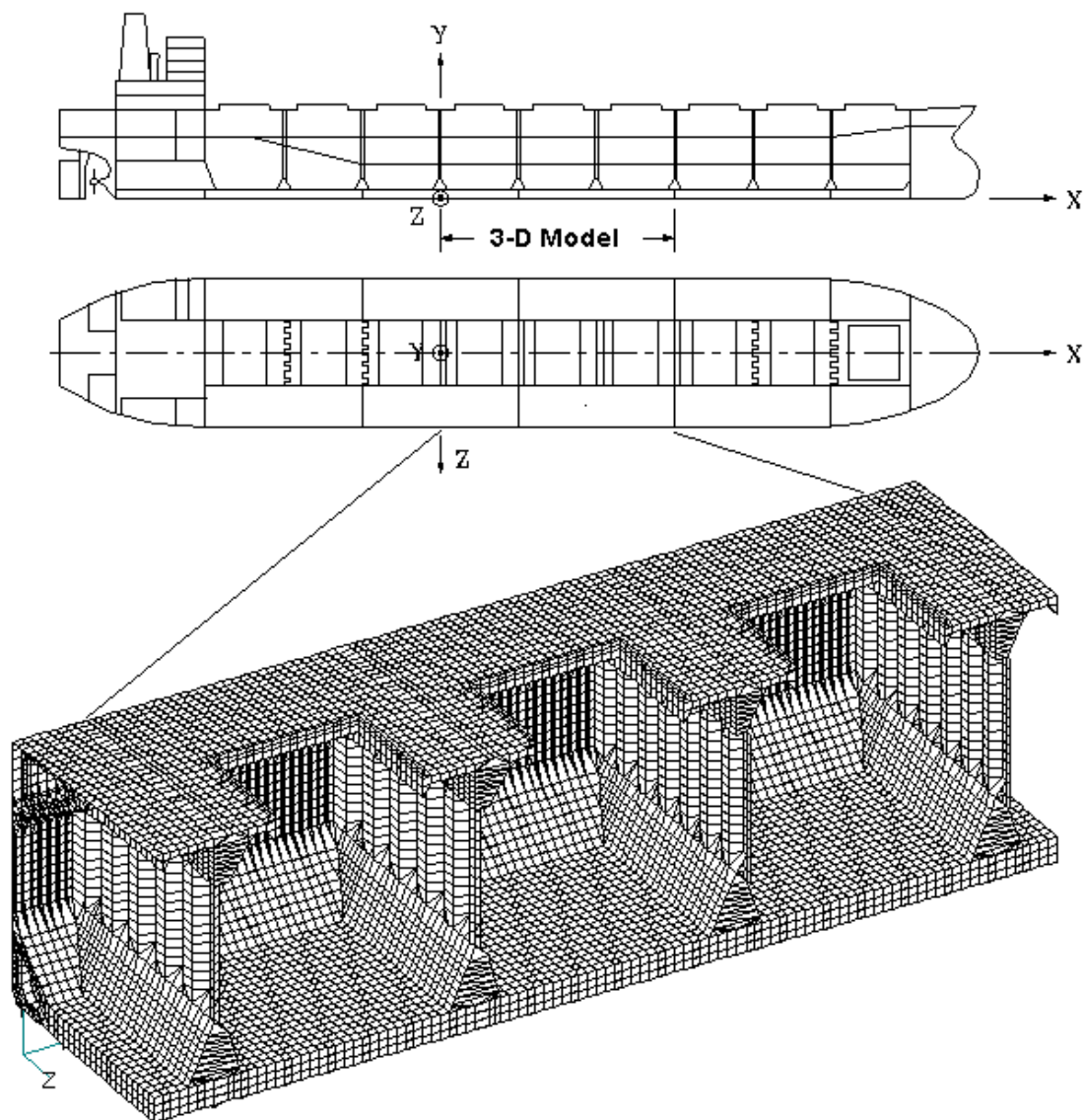
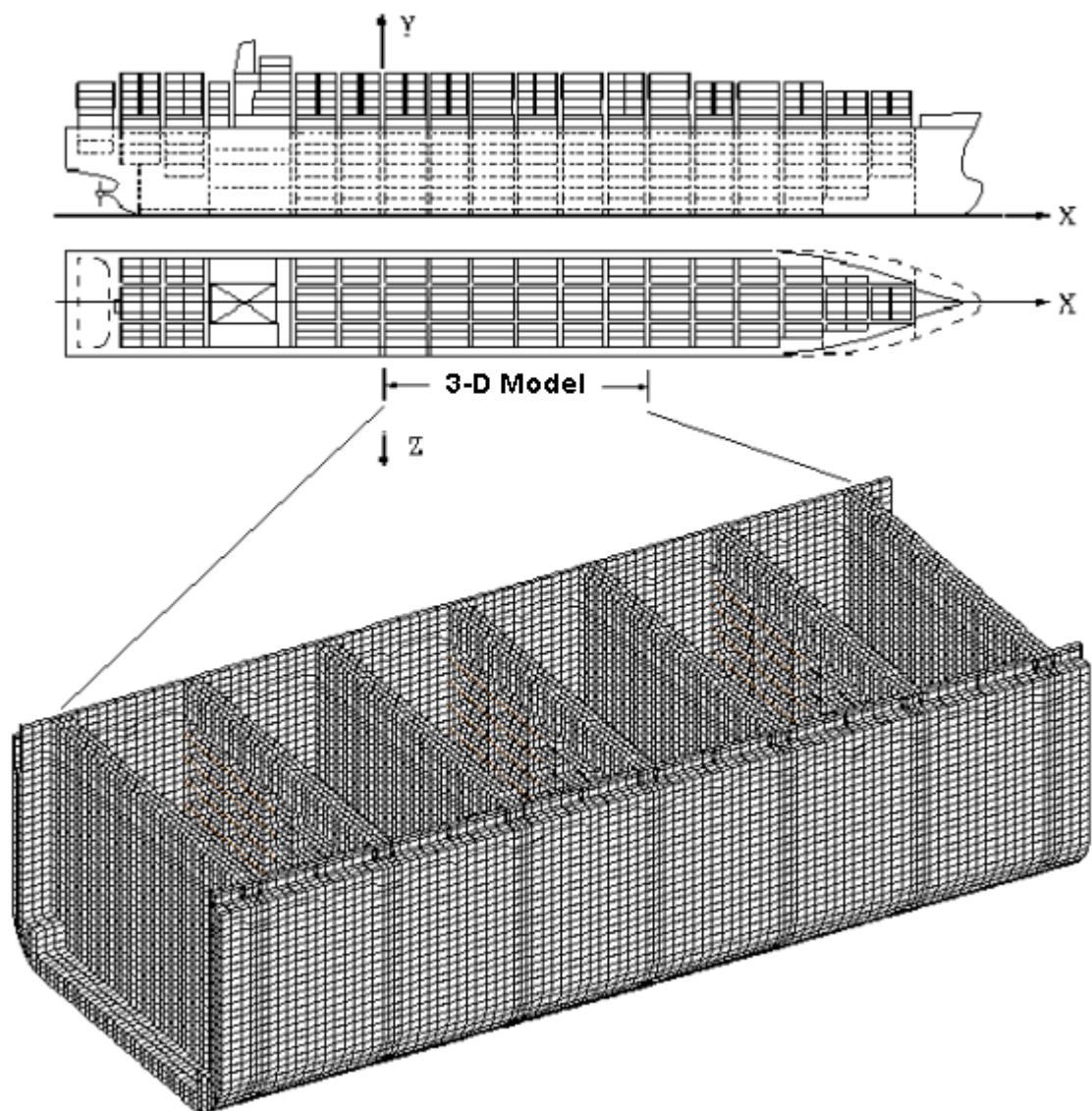


FIGURE 3
Extent of the 3D Global Model and Mesh Arrangement for Container Carriers



5 Coordinate and Unit System of the Global Model

In this Guide, as well as within the SafeHull software system, the global coordinate (right-hand) system of this reference finite element model is defined as follows and shown in Section 2, Figure 4:

- X-axis: Longitudinal, positive from aft to fore
- Y-axis: Vertical, positive upward
- Z-axis: Transverse (athwart-ship), positive toward starboard

X-origin can be at any model longitudinal location, but it is recommended that X-origin be located at the aftermost bulkhead of three cargo holds. The Y-origin and Z-origin must be located at the intersection of the baseline and centerline planes.

The SafeHull software system is using engineering metric units and they are:

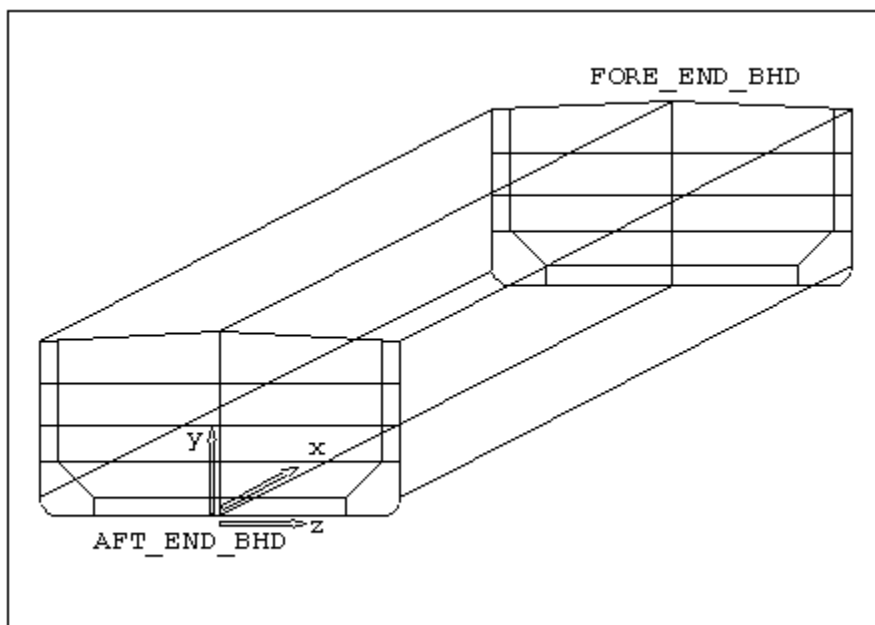
- Length: Centimeter (cm)
- Force: Kilogram force (kg force)
- Mass: Kilogram (kg)

Therefore, the stress is in kg/cm^2 .

Since the users may create FE models for other purposes, they may not match the above coordinate and unit system requirements. The SafeHull system provides a tool to convert the user's right-hand coordinate system into the SafeHull-required coordinates and units.

It should be mentioned that after the users decide the X-origin to be used for the global model, this origin is used for all of the sub-models. The SafeHull system provides a conversion tool to be used on all sub-models created outside of SafeHull.

FIGURE 4
Model Coordinate System



7 Element Types and Combination

In general, the ship structural FE model, whose geometry, configuration and stiffness approximate the actual ship hull structures, consists of four types of elements:

For stiffeners:

- Truss element (rod element) with axial stiffness only and a constant cross-sectional area along the length of the element.
- Beam element with axial, torsional and bi-directional shear and bending stiffness with constant properties along the length of the element.

For plates:

- Membrane plate element (i.e., plane-stress element) with bi-axial and in-plane shear stiffness and constant thickness.
- Bending plate element with in-plane stiffness as the membrane element plus out-of-plane bending stiffness and constant thickness.

These four simple types of elements are considered sufficient to represent the hull structures even though higher order element types exist.

Ship structures consist of various stiffened plates. These stiffened plates are represented by a combination of membrane plates and rod elements as long as only in-plane stress is calculated from the model. This is the case for the global 3D FE approach. Since rod elements lack out-of-plane load carrying capability, all loads applied on the intermediate nodal points of rod elements must be shifted to the adjacent primary supporting structures while associated singularity problems of unsupported nodal points are dealt with by slave-master constraints. These tasks are automatically handled in the SafeHull system.

The basic approach of shifting load technology is to find the nearest master node in the predetermined orientation, which has sufficient stiffness in the degree of freedom of which the slave node has zero or very small stiffness. If slave/master relationship is established, load acting on slave node is shifted to master node. The orientation determination is based on the structural member. In general, the search for the longitudinal members is in the longitudinal direction and the search for the transverse bulkheads is in the vertical direction. The radiation search is used for slave nodes of complicated structures, if their master nodes can't be found within the search range.

Combined use of membrane plates and rod elements may simplify the modeling processes and reduce the total number of degrees of freedom in the model. However, additional operations, such as shifting load, may result in less accurate results for some elements.

Combination of bending plates and beam elements is preferable, since computer technology has advanced to the point that computing time is not an issue for the FE analysis.

If beam elements are used to model the stiffeners, eccentric beams (with their neutral axis offset from the attached nodes) should not be used. SafeHull software separately estimates such an effect during strength evaluation. Appropriate properties of beam elements are assigned by considering equivalent concentric beams. This process is using the effective plate width (i.e., individual space of stiffeners) in the calculation of moment of inertia and assuming the neutral axis being located at the center layer of the attached plate. Attached plates are excluded from the calculation of sectional areas of beam elements.

Combined use of bending and membrane plate elements is not a common practice. However, this does not preclude the combined use of rod elements (faceplates) and bending plates (web plates) for main supporting structures.