공학 연구논문집 제 30권 1호 pp. 227~237. 울산대학교
Journal of Engineering Research Vol. 30, No. 1, pp. 227~237. 1999
University of Ulsan

Efficient Three Dimensional Modelling Technique for The Transverse Strength Analysis of Ship Structures

Park, Chi-Mo
School of Transportation Systems Engineering

<Abstract>

Conventional transverse strength analysis of ship structures has resorted to two dimensional frame analysis method or three dimensional finite element method using mostly membrane shell elements in modelling a ship structure. However, the former is too simple to correctly incorporate the interactions between longitudinal strength members and transverse frames, and the latter requires a great deal of time and efforts and involves a high possibility of making errors in the modelling process, even with well-developed pre-processors, due to its complexity.

This paper offers a simple and efficient three dimensional finite element modelling technique together with some basic studies providing theoretical background to the present modelling technique. Finally developed modelling technique is applied to the transverse strength analysis of 160 ton high speed catamaran to show its usefulness.

선체 횡강도 해석을 위한 효율적 3차원 유한요소 모델링 기법

박 치모
수송시스템공학부

<요 약>

종래의 선체 횡강도 해석은 대부분 2차원 골조구조 해석이나 주로 판 요소를 사용하는 3차원

유한요소해석에 의해 이루어져 왔으나, 전자는 종강도 부재와 횡늑골 사이의 상호작용을 정확히 고려하기에는 너무 단순하고 후자는 그 모델링 과정의 복잡성 때문에 막대한 시간과 노력을 요할 뿐만 아니라 모델링 과정에서 실수를 범할 가능성도 매우 높은 것으로 잘 알려져 있다. 본 논문에서는 비교적 간단하면서도 효율적인 3차원 유한요소 모델링 기법을 이론적인 기초 연구와 함께 제시하였고, 개발된 기법을 160톤 급의 고속쌍동선의 횡강도 해석에 적용하여 그 유용성을 보여주고 있다.

1. Introduction

From the viewpoint of ship structural characteristics, transverse strength analysis has no simple approximate method to use, like simple beam theory which is suitable for longitudinal strength analysis, and has resorted to two dimensional frame analysis accounting for the effect of longitudinal girders on the transverse strength by means of elastic springs of equivalent stiffness[1,2]. But this method has following two major shortcomings.

- The analysis can not be performed at a stroke because it requires preceding separate beam or grillage analyses to get the equivalent spring constants of longitudinal girders.
- The two dimensional finite element model can not be used in common for the longitudinal strength analysis, and separate finite element model would be necessary for it.

Along with the improvement of computer hardware and finite element related softwares, three dimensional finite element analysis, where most structural components, even up to transverse frames, are represented by two dimensional membrane shell elements, has become popular among ship structural engineers. But this conventional three dimensional analysis requires a great deal of time and efforts and involves a high possibility of making errors in the modelling process, even with well-developed pre-processors, due to its complexity.

In order to solve those problems, this paper develops a simple and efficient finite element modelling technique which greatly reduces the number of elements and simplifies the modelling process by representing frames and girders as beam[sometimes called bar] elements while stiffened panels are represented as the elements of same type but of smaller number compared to the conventional three dimensional method. In order to clarify the difference between the two methods, conventional and present, Table 1.1 shows the comparison of the types of finite elements used in both methods. This paper also gives some basic studies providing theoretical background to the present modelling technique. Finally developed modelling technique is applied to the transverse strength analysis of 160 ton high speed catamaran to show its usefulness. MSC/NASTRAN[6] and MSC/XL are used as a solver and pre- and post-processor, respectively in the analysis.

Table 1.1 Types of Finite Elements Used in Both Modelling Methods

Structural Components \ Methods	**Conventional Modelling**	**Present Modelling**
Transverse Frame (attached to plating)	Flange → **ROD** Web → Membrane **SHELL**	(Hybrid) **BEAM**
Pillar, Cross Tie, etc.	**BEAM**	**BEAM**
Bracket Joint	Membrane **SHELL**	**Rigid BAR (RBAR)**
Stiffened Panel	Plate → Membrane **SHELL** Longi. Stiffener → **ROD**	Membrane **SHELL**(Larger) **ROD**(Lumping several Longi. Stiffeners)

2. Finite Element Modelling for Various Structural Components

2.1 Beams attached to plating

In order to represent the effect of beam bending on in-plane response of plating to which the beams are attached, hybrid beam element[3] is adopted, where the sectional area A is the sectional area of only the beam itself(flange and web) while the neutral axis position and the moment of inertia I are calculated for the combined section formed by the beam plus a plate flange of effective breadth b_e. While there are many different guidances for b_e[3,4,5], this paper is to adopt b_e=0.2l(l: effective span of beam) as recommended by KR[4].

The structure nodes are located at the neutral axis of the combined section. The plate's contribution to the membrane stiffness is accounted for by locating the plate elements, which have only membrane stiffness, such that their stiffness acts in the plane of the combined neutral axis, as illustrated schematically in Fig.2.1.

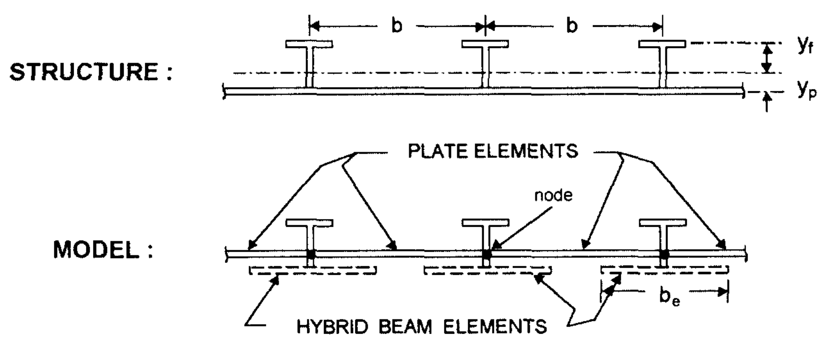


Fig.2.1 Hybrid Beam Element

2.2 Pillar, cross tie, longitudinal girder, etc.

General one dimensional members subjected to axial force and bending are modelled as ordinary beam elements[called Bar in MSC/NASTRAN].

2.3 Brackets

To represent the strengthening effect of the brackets attached to the ends of beams, rigid bar[called RBAR in MSC/NASTRAN] elements are used as shown in Fig.2.2.

For simplicity, the length of RBAR is to be determined as shown in Fig.2.2 using the span point concept proposed by DNV[5].

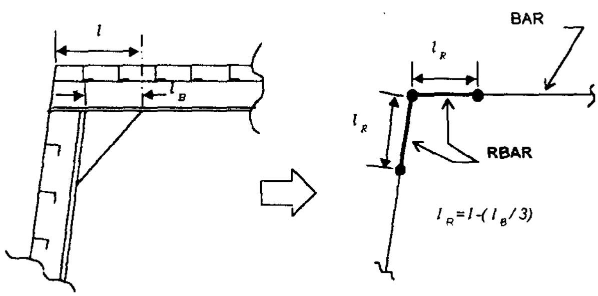


Fig.2.2 Finite Element Modelling of a Bracketed Part

2.4 Stiffened panels [3]

A rectangular part of stiffened plating surrounded longitudinally by two neighboring transverse frames and transversely by two longitudinal lines connecting the facing nodal points on the neighboring transverse frames is treated as a stiffened panel element.

As stiffened panels are assumed to have only in-plane stiffness in the hull module analysis, each stiffened panel element is represented as a combination of a four node membrane shell element[QUAD4] and two ROD elements which lump all longitudinal stiffeners at two longitudinal sides of the shell element as shown in Fig.2.3.

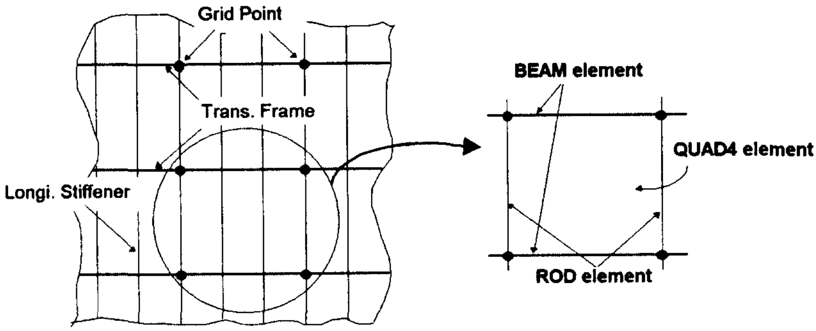


Fig.2.3 Finite Element Modelling of Stiffened Plate

3. Comparison of BAR Element Modelling and SHELL-ROD Element Modelling for the Frame Analysis

Since the most significant difference between the two modelling methods, conventional and present, is in the way to model the transverse frames, that is, as SHELL and ROD elements in the former and as BAR elements in the latter, I-section beam[frame segment] shown in Fig.3.1 is modelled in those two ways, in order to compare the efficiency of both modelling methods. These two modellings are depicted in Fig.3.2 and Fig.3.3. which are, respectively, representative of the conventional and the present modelling.

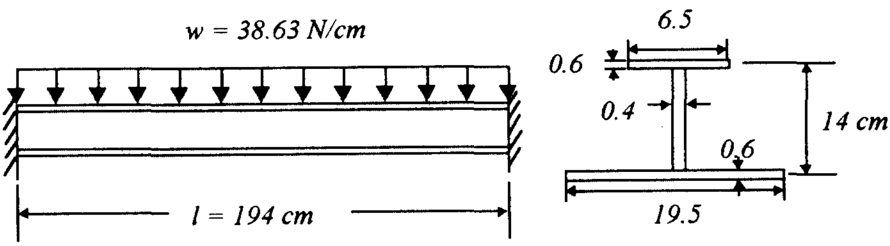


Fig.3.1 I-Section Fixed-end Beam Subject to Uniform Pressure

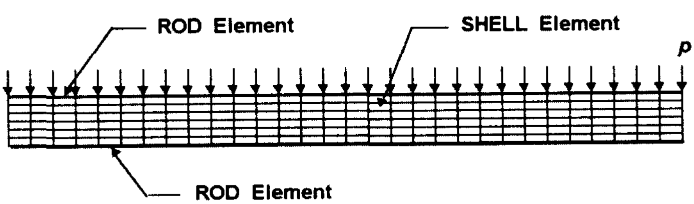


Fig.3.2 SHELL-ROD Element Modelling

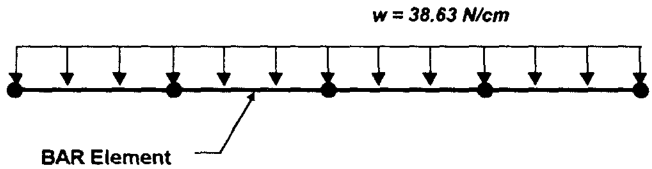


Fig.3.3 BAR Element Modelling

3.1 SHELL-ROD element modelling

For analyzing the I-section beam, upper and lower flanges are modelled as ROD elements and the web is modelled as QUAD4 membrane shell elements.

In order to figure out the effect of mesh fineness on the accuracy of analysis, the web is divided into 2(12), 4(24), 8(48), 16(96) equal parts vertically(horizontally) and each model is named Model-S2, Model-S4, Model-S8, Model-S16, respectively.

3.2 BAR element modelling

To make a good comparison of two modelling methods on an equal basis, following assumption needs to be made in BAR element modelling; Sectional areas of upper and lower flange are lumped at upper and lower ends of the web, respectively, and have no moment of inertia about their own neutral axes[Fig.3.4].

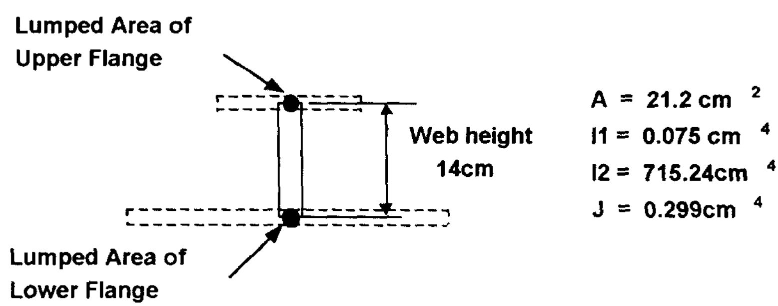


Fig.3.4 Assumed Beam Section

3.3 Comparison of two analysis results

Analysis results are put in Table 3.1. which shows the comparisons of deflections and stresses at two positions, an end and mid-section, for the two models.

Table 3.1 Comparisons of Max. Deflections and Stresses of The Analysis Model for Two Modelling Methods

Model	Mid-Section				End-Section	
	Deflection [cm]		Stress[N/cm^2]		Stress[N/cm^2]	
	Without Shear Effect	With Shear Effect	Top	Bottom	Top	Bottom
Model-B2	-2.846E-2	-4.338E-2	-8.110E2	3.748E2	1.622E3	-7.500E2
Model-B16	-2.846E-2	-4.338E-2	-8.110E2	3.748E2	1.622E3	-7.500E2
Model-S2	-3.990E-2		-7.841E2	3.622E2	1.231E3	-5.650E2
Model-S4	-4.065E-2		-8.052E2	3.711E2	1.439E3	-6.500E2
Model-S8	-4.085E-2		-8.108E2	3.734E2	1.556E3	-6.938E2
Model-S16	-4.090E-2		-8.122E2	3.740E2	1.624E3	-7.160E2

Following conclusions can easily be drawn from Table 3.1 ;

Considering that the analysis results of 2 BAR element model and 16 BAR element model are the same, only a few BAR elements seem enough in case of BAR element analysis. On the other hand, in the case of SHELL-ROD element analysis, the results show the remarkable change with the increase of element number and improving results nearly approach the ones of BAR element analysis. These show that, for the frame anlysis, the BAR element modelling is more efficient than the SHELL-ROD element modelling.

4. An Example of Application

4.1 Selection and modelling of the hull module

Present modelling technique is applied to the hull module analysis of 160 ton high speed catamaran. The hull module is taken as a complete segment of the hull girder extending over one longitudinal bulkhead space amidships and including the bounding bulkheads of half the actual scantlings. Transverse symmetry of the ship makes it possible to analyze only half the segment, that is, one side of the center plane and Fig.4.1 represents the finite element model of it.

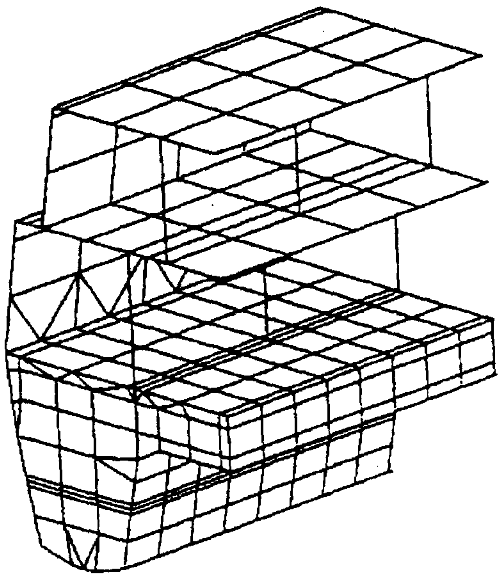


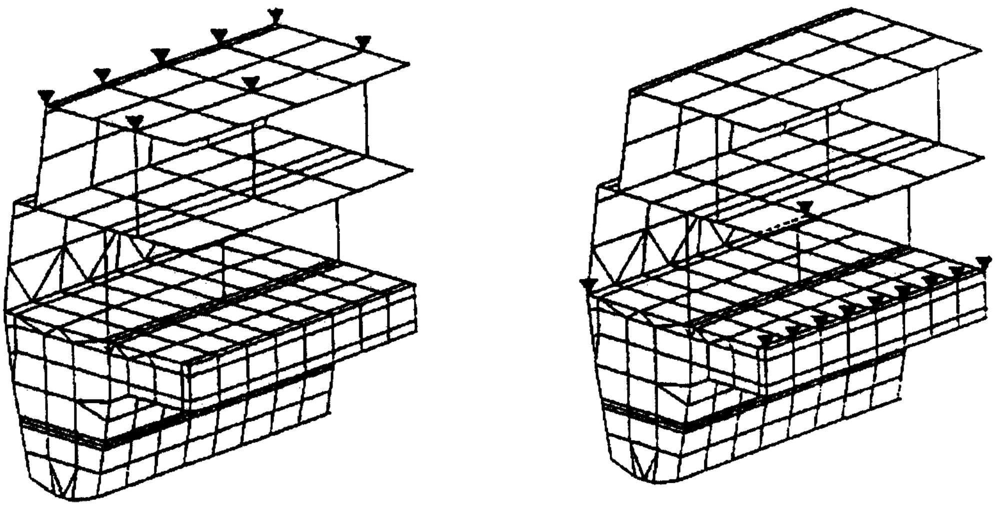
Fig.4.1 Finite Element Model of a Catamaran Hull Module Constructed by The Present Modelling Technique

4.2 Load and boundary conditions

DNV rules[5] for high speed catamaran demand to apply following five combinations of load for transverse web frame beam element analysis;

- Sea pressure on all elements
- Slamming pressure on bottom and sea pressure on decks
- Slamming pressure on bottom from outside and sea pressure on hull outer side and decks
- Slamming pressure on bottom from inside and sea pressure on tunnel side, tunnel top and decks
- Slamming pressure on tunnel top and sea pressure on tunnel side, bottom from inside and decks

Among the above five combinations, only the first combination is to be chosen for the analysis because the purpose of this paper is just to demonstrate the analysis using developed modelling technique. While specifying the detailed load conditions, DNV rules only demand due attention to boundary conditions without any specified guidance. This paper has chosen two boundary conditions, as depicted in Fig.4.2, which are assumed to be suitable in checking the stresses, respectively below and above the main deck. In Fig.4.2 only the vertical constraints are marked which are more controversial than the longitudinal and the transverse constraints because the imbalance of commonly given loads is most severe in the vertical direction.



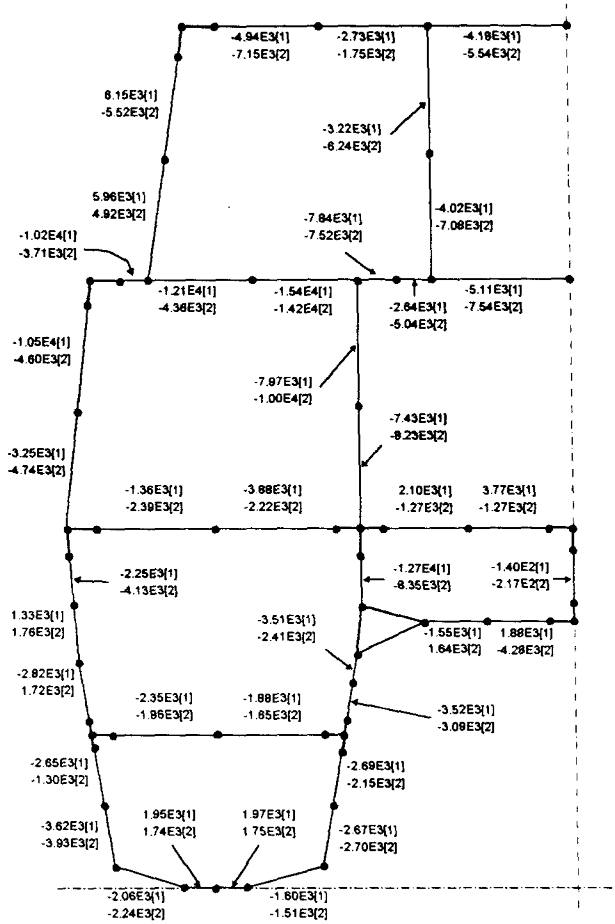
(a) B.C. 1 (b) B.C. 2

▼ denotes the constraint of vertical displacements

Fig.4.2 Applied Boundary Conditions

4.3 Analysis results and investigation

Hull module analysis has been carried out according to the present modelling technique and the results are put in order in Fig.4.3. The figures in Fig.4.3 represent the calculated maximum stresses in each BAR element of the frame located halfway between the two end bulkheads. Great difference is observed in the stress distributions for the two boundary conditions, which tells the importance of choosing appropriate boundary conditions.



NOTE :
- Unit of stresses is N/cm^2.
- Numbers in brackets denote the case of boundary conditions.

Fig.4.3 The Distributions of Maximum Stress in Each BAR Element of the Mid-Frame

5. Conclusions

The results of the present research for developing simple and efficient finite element modelling technique of ship structures are summarized as follows.

1) The required number of elements of the proposed modelling technique is less beyond comparison(1/40 or below) than the one of the conventional shell element modelling

technique for the same structure. So its simplicity and time-saving effect make this technique especially useful for optimum structural design, which involves iterative structural analyses.

2) There seem to be many engineers who believe that, in a finite element analysis, more elements unconditionally lead to more exact results, but it is valid only for the same type of elements. This paper has made clear that only a few beam elements can offer better results than a large number of shell elements in the case of frame or girder analysis.

3) Considering that analysis results can be greatly affected by boundary conditions, classification rules are recommended to include the clear guidance on the boundary conditions as well as load conditions for the direct structural analysis.

4) The present modelling technique is applicable to various types of ship structural analysis like ;

- Transverse strength analysis
- Longitudinal strength analysis
- Preliminary global analysis for the following local fine mesh analysis

REFERENCES

[1] Hardy-Cross, "Analysis of Continuous Frames by Distributing Fixed-End Moments", Proceedings ASCE, 1932.

[2] S. S. Na et al., "Minimum Weight Design of Transverse Strength Member by Using Finite Element Method", Journal of The Society of Naval Architects of Korea, Vol.22, No.3, Sep., 1985.

[3] O. H. Hughes: Ship Structural Design- A Rationally-Based, Computer-Aided, Optimization Approach, John Wiley & Sons, 1983.

[4] KR, "Tentative Rules for High Speed and Light Craft", Nov. 1995.

[5] DNV, "Tentative Rules for Classification of High Speed and Light Craft", Jan. 1995.

[6] Mark P. Miller: MSC/NASTRAN User's Guide, MSC, Dec., 1993.