Finite element modelling and stresses analysis of moonpool area of a drillship

Ahmed MH Elhewy
Naval Architecture and Marine Engg. Dept., Faculty of Engg., Alexandria University, Alexandria, Egypt
email: Ahmed. Elhewy@engalex.edu.eg

The aim of this paper is to present the finite element modelling and stresses analysis of ship structures. Complicated structures such as ship hull need to be analysed accurately and efficiently. The finite element method is considered as one of the most powerful tools that can give satisfactory results. Various loading conditions acting on the ship hull such as still water loads and wave induced loads are briefly explained. The structural modelling of a midship section in moonpool area of drillship is created. Different loading conditions have been applied and investigated in order to assign the worst one to the ship. The structural response under the applied loads is obtained. The stresses distribution through all the members of the section is monitored to explore the areas of maximum stresses. The elastic form of the midship section is obtained. The results show the outstanding capacity of finite element method for analysis of such complex structure by obtaining the stress concentration areas and the hull deformation form under the subjected loads.

تعتبر طريقةَ العنصرِ المحدود أداة حسابية قويَّة تستخدم في تصميمِ المنشأت البحريةِ المعقّدةِ. هذه المنشأت البحريةِ المعقّدةِ مثل هيكلِ السفينةِ والذى مَنْ الضَّرُوري أَنْ تُحلّلَ بدقّة وبشكل عالى الكَفاءه. إنّ طريقةَ العنصرِ المحدود الأداةُ الوحيدةُ التي يُمْكِنُ أَنْ تَعطي نَتائِجَ مقنعةَ. إنّ الهدفَ الرئيسى من هذا البحث هو عرض طريقه العنصرِ المحدودِ وإستخدامها لتحليلِ المنشأت البحريةِ. لذلك تم شْرحُ بعض أوضاع التحميل المُخْتَلِفةِ في السفينةِ مثل أحمالِ الماء الساكن و كذلك أحمالَ الامواج باختصار. كذلك تم إختيار قطاع في سفينةِ حفر لعمل نموذج ثلاثي الابعاد لوسط السفينةِ لحساب توزيع الاجهادات و الازاحات المختلفه تحت تأثير الأحمالِ التطبيقيةِ. وأظهرت النَتائِجُ القدرةَ البارزةَ لطريقةِ العنصرِ المحدود على تحليلِ مثل هذا المنشأ المعقّدِ بالحُصُول على مناطقِ تركيزِ الإجهادَ وشكلَ التشوهِ الهيكليَ تحت تأثير الأحمال.

Keywords: Ship structures, Finite element method, Drillship

1. Introduction

The main purpose of structural analysis are to determine the distribution of stresses throughout the structure to ensure that the allowable stresses are not exceeded also to calculate the displacements at certain points of the structure to ensure that the specified clearances are not violated. Finite element analysis is generally known as the most important technological breakthrough in the field of engineering analysis of structures. The development of computer has caused the finite element method to become one of the most popular techniques for solving engineering problems [1]. The finite element method provides a tool for analysis of general types of structures. The body to be analysed can have arbitrary shape, loads and support conditions. The mesh can mix elements of different types, shapes and physical properties. User-prepared input data controls the selection of problem type, geometry, boundary conditions and element type [2]. Hull structures of ships consist of a steel framework surrounded by steel plating. A hull module is a three-dimensional framework of beams and stiffened panels [3]. The finite element method had been used in analysing a bulk carrier hull girder [4, 5]. The finite element method basics and methodology are summarized in appendix I.

2. Midship structural analysis

The primary purposes of the drill ship are to drill and perform extended well tests, both of which are conducted while the vessel is stationary. Therefore, the hull form was developed to have characteristics that are advantageous while the vessel is stationary instead of in transit. The ship has a large parallel middle body (200 m) and large beam (38 m) which provide larger water plane area increasing the stability of the ship due to the increase of the moment of area. There are six centreline cargo tanks with combined volume of over 200,000 m^3. A 2 m double bottom extends the length of the ship, as well as 7 m wing tanks port and starboard. The general arrangement of the ship is shown in fig. 1. These tanks along with the forepeak tank contain over 76,000 m^3. The drill ship has a very complex construction due to its designed functions. A midship section of a drill ship with specifications shown in table 1 is modeled in order to be analyzed by using the finite element method. The loads acting on the ship are the sum of the still water and the wave induced loads.

Table 1
Ship specifications

Main particulars of the drillship under consideration	
Length OA (m)	260
Length BP (m)	256.8
Breadth (m)	38
Depth (m)	18.5
Design draft (m)	12.73
Young's modulus of hull material (N/m^2)	2×10^{11}
Yield stress of hull material (N/m^2)	3.15×10^{08}
Ultimate tensile strength (N/m^2)	4.4×10^{08}

2.1. Still water loads

When a ship is at sea, it is subjected to forces which cause the structure to distort and the correct assessment of the magnitude of the forces is difficult. The forces may be divided into static forces and dynamic forces. Still water forces are static in nature, by considering ship to be floating in equilibrium. Still water load curve is obtained as the algebraic sum of weight curve and buoyancy curve.

Different loading conditions are assigned to the ship in order to determine the worst one. Table 2 summarized these loading conditions. In order to determine the distribution of bending moment along the ship, the weight and buoyancy, load, shear and moment distribution for the worst loading condition are shown in fig. 2. The load, shear and bending moment diagrams are obtained as shown in fig. 2.

2.2. Wave-induced loads

Wave induced loads are usually classified as slowly varying loads which consist of the dynamic pressure distribution on the hull due to the combination of wave encounter and the resulting ship motion, sloshing of liquid cargo, shipping of green seas on deck, wave slap on sides and on foredecks, inertia loads, launching and berthing loads, ice breaking loads, etc.

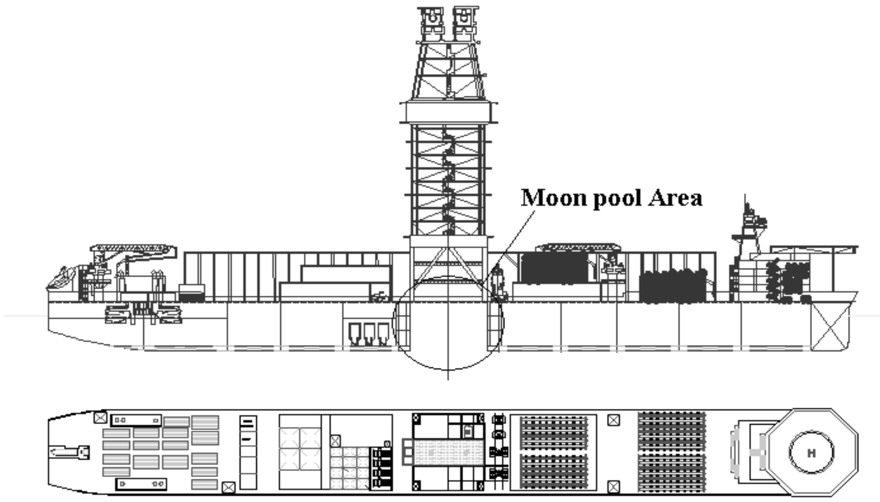

Fig. 1. Drillship general arrangement.

Table 2
The loading conditions acting on the ship

Loading condition	Description
Lightship Condition	The ship has no personnel, consumables or cargo on board but it has its special outfitting.
Transit condition	The vessel has only the ship's crew on board and has one hundred percent of its fresh water. There is no cargo or variable deck loads on board but the vessel is full of lube oil and has fifty percent of its fuel and provisions.
Departure to oil field.	The ship has the ship's and drilling crew on board. She is carrying one hundred percent of its provisions, water, fuel and lube oil. The ship is carrying all of the variable deck loads including: drilling mud, sack material, cement, riser joints, completion riser, drill pipe, casing, blow-out preventer and subsea gear, 3rd party equipment, and drill water but it has no crude oil on board.
Starting the drilling operations condition	The ship is carrying seventy-five percent of its fuel and provisions. Ballast water has been taken on board to reach the drilling draft.
End of drilling operations condition	The ship is carrying one hundred percent of its water and lube oil while it is carrying thirty-five percent of its fuel and provisions. The ship has already commenced drilling operations, therefore it has no drill pipe, riser pipe or casing on board but it has yet to extract any crude oil from the well. Half of the drill mud, cement, drill water, and sack storage have been used. Also, the blow-out preventer is on the wellhead while the Christmas tree is still on board. There is enough ballast water to maintain the desired drilling draft.
Departure from oil field condition	The ship is carrying one hundred percent of its water and lube oil while it is carrying thirty-five percent of its fuel and provisions. The ship is carrying one hundred percent of its drilling mud, risers and drill string but it has no cement, drill water, casing, completion riser or sack material. The ship is carrying its capacity of crude oil.
Arrival at port condition	The ship is carrying one hundred percent of its water and lube oil while it is carrying ten percent of its fuel and provisions. The vessel is carrying one hundred percent of its drilling mud, risers and drill string but it has no cement, drill water, casing, completion riser or sack material. The ship is carrying its capacity of crude oil

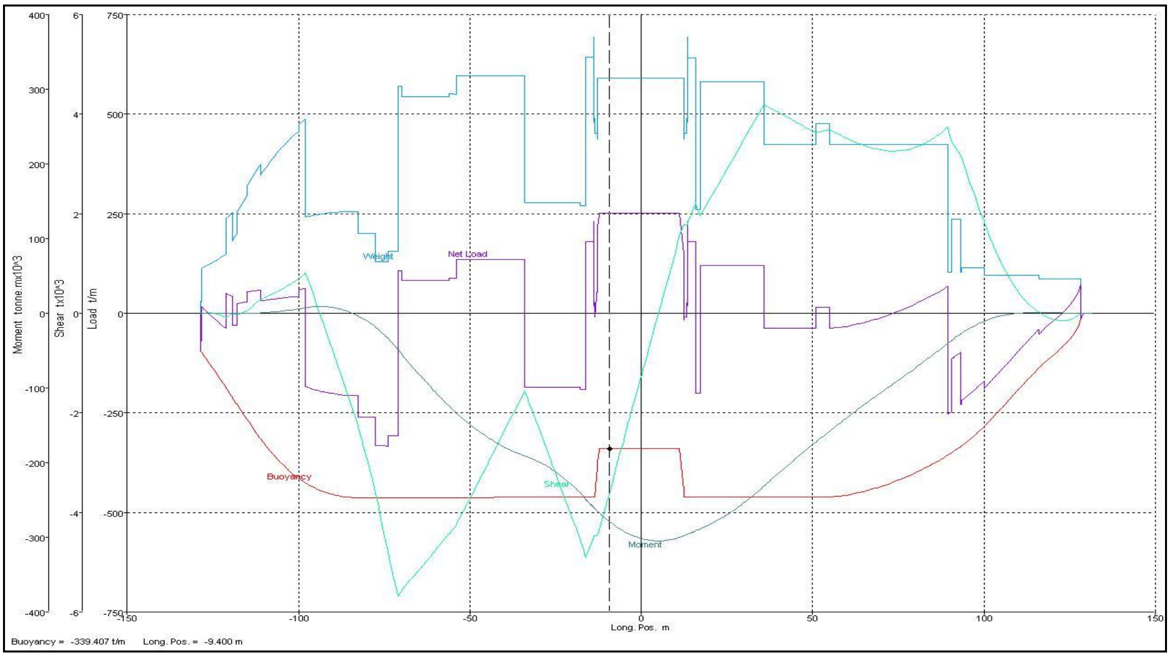

Fig. 2. Weight, buoyancy, load, shear and bending moment distributions.

The midship section of the ship has been modelled and created by using MAESTRO program. The midship section is shown in fig. 3. The three dimensions model is created for 32 m part of the parallel middle body is shown in fig. 4. The section has a centre line bulkhead in the crude oil tank for the sloshing and structural considerations. The double bottom height is 2 m. the moonpool is 25 m long and 10 m wide. There is a cofferdam that encircles the moonpool. Outboard and forward of the cofferdam are the base oil, brine and drilling water used during drilling operation.

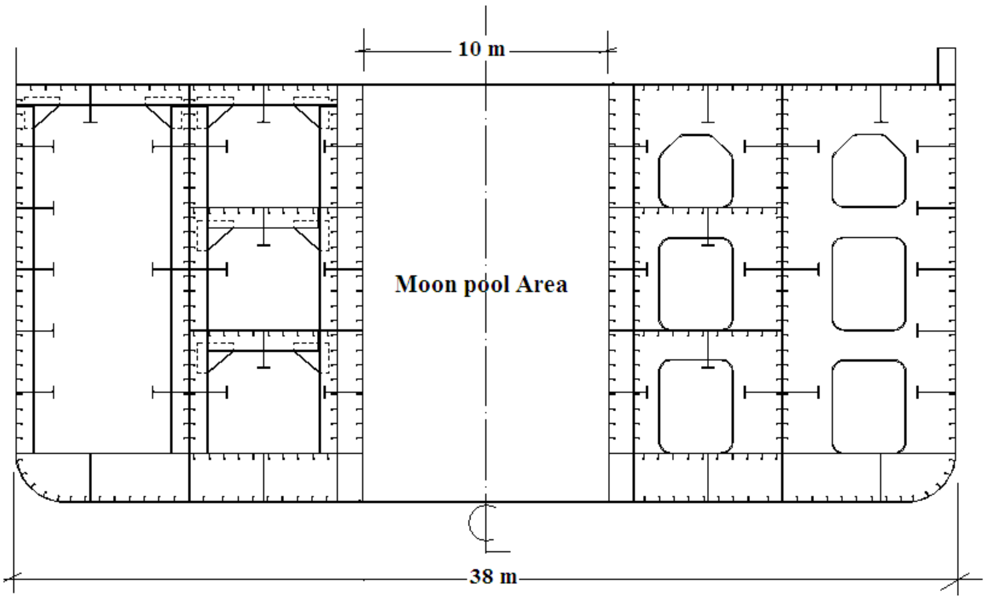

Fig. 3. Midship section in moonpool area.

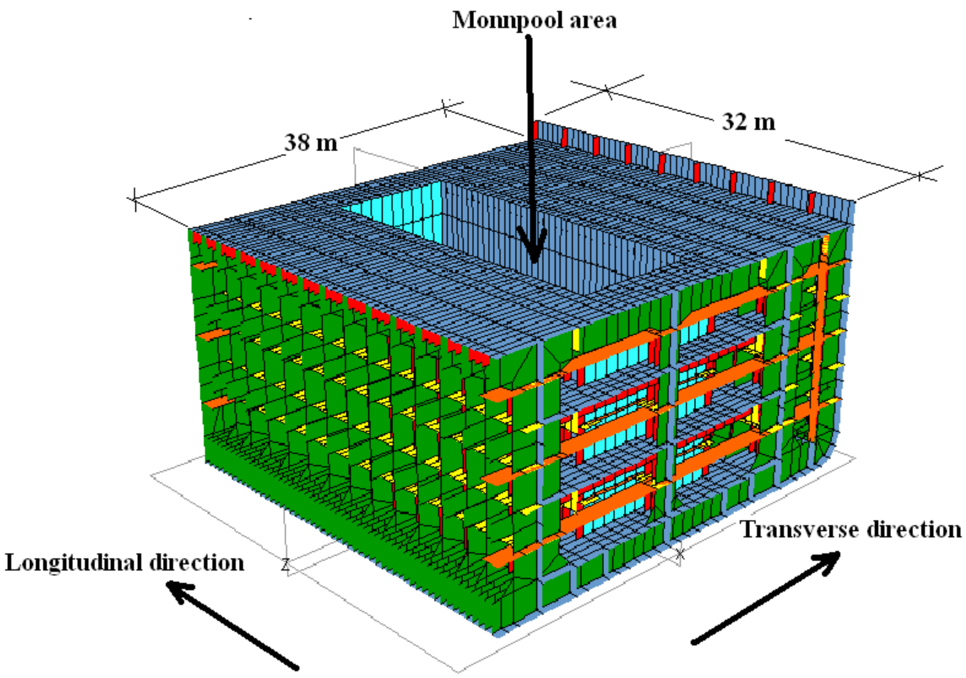

Fig. 4. The three dimensions model for 32 m part of the parallel middle body.

3. Results

After the section is modelled the worst loading condition is recognised among different loading conditions. The maximum load is assigned to the section members in order to obtain the structure response under the assigned load. The stresses in the longitudinal and transverse directions are calculated to show the contours of the maximum stresses produced on the section. The stresses contours are shown in fig. 5 for longitudinal direction and fig. 6 for transverse direction. The deformations of all structure members are also obtained as shown in fig. 7.

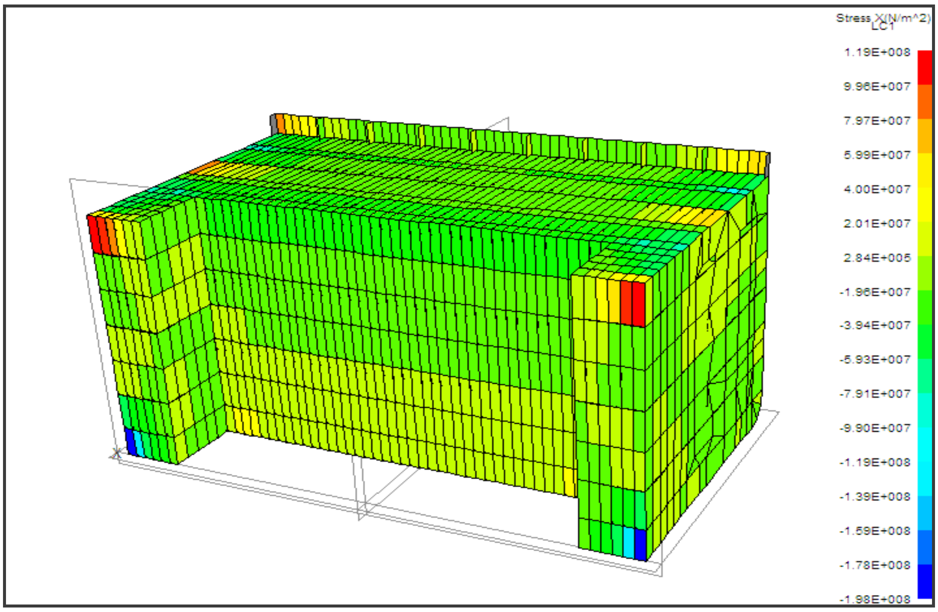

Fig. 5. Stresses in longitudinal direction.

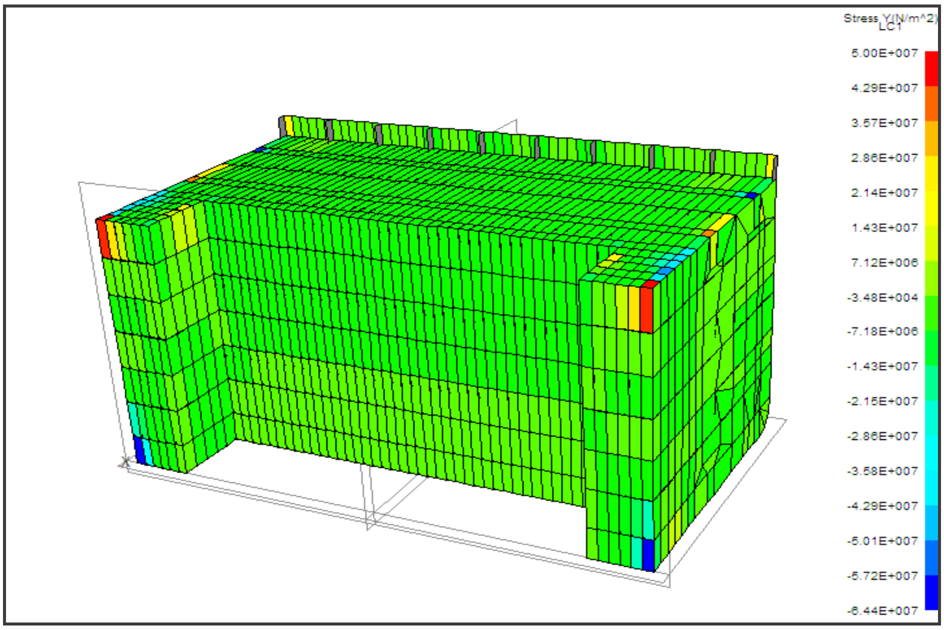

Fig. 6. Stresses in transverse direction.

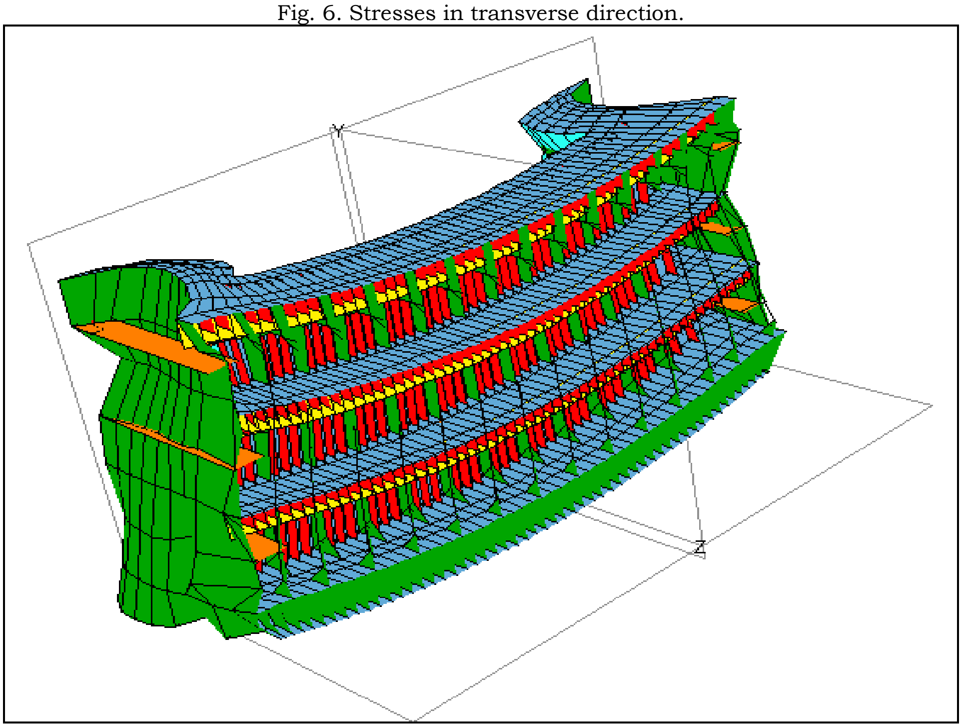

Fig. 7. Midship section deflection form.

4. Conclusions

From the current study it can be concluded that the power of finite element method resides principally in its versatility. The main advantages of FEM are that physical problems which are so far intractable and complex for any closed form solution can be analysed. It can take care of any type of boundary, material anisotropy and any type of loading. Another attractive feature of FEM is the close physical resemblance between the actual structure and its finite element model. The calculated stresses in the longitudinal and transverse directions show no unexpected values that may exceed the yield stress of the material used. However, this is not enough to ensure that the structure can withstand the applied loads more investigations are needed to be done. The deflected form has a symmetric shape which means that the boundary conditions that assigned to the section are appropriate. There is no violated value of the deflection at any member through out the section. The finite element method does not only present good approximated results but also a time saving in calculating the stresses and deflection through the whole structure. In the future vision, this work may be followed by applying the structural reliability methods in order to introduce all failure modes in the design of the section as well as optimizing the structural steel weight.

Appendix I

Finite element method basics and methodology

I.1. Finite element method

The objective of finite element method is to develop a model which simulates the elastic behaviour of the continuous structure as closely as required. In the finite element method the original structure is idealized as a notional assemblage of finite elements connected one to another only at discrete points called nodes and the boundaries at a finite set of nodes. The loads applied to the structure are also idealised as a set of loads which act at the nodes only. Many of the nodes are free, i.e. they will be able to translate or rotate to meet the demands of the elastic deformation of the elements to which they are attached. Some of the nodes will no be free. They will be constrained. Generally, the constraint will be complete. The total structure must satisfy the three fundamental requirements: equilibrium of forces, compatibility of displacements and laws of material behaviour. These conditions can be formulated in matrix notation in stiffness analysis. When dealing with complicated ship and offshore structures, it is necessary to determine the displacements of all the free nodes in order to be able to evaluate the stresses occurring in the members.

I.2. Structural modelling

Structural modelling is the first step in the analysis of a structure. A hull can be modelled as a beam or 3D structure. For local level analysis, different structural components like decks, bulkheads, etc, can be modelled by using 2D membrane or plate element. Super element technique can be effectively used for modelling ship structures which has repetitive structural components [6]. The super element may be a combination of bar, beam, triangle and quadrilateral elements [7]. Nodes that are wholly within the super element and are not required for attaching it to the rest of the structure may be deleted by static condensation. If such super elements occur repeatedly, this deletion provides a saving in computation. All elements and nodes must be numbered so that we can set up a matrix of connectivity. Fig. I.1 shows a modelling of a transverse frame into beam elements and plane stress problem into quadrilateral elements.

It is important to remember that the order in which the nodes and elements are numbered greatly affects the computing time. This is because we get a symmetrical, banded stiffness matrix, which bandwidth is dependent on the difference in the node numbers for each element, and this bandwidth is directly connected with the number of calculations the computer has to do. Computer finite element method programs have internal numbering that optimizes this bandwidth to a minimum by doing some internal renumbering of nodes if they are not optimal.

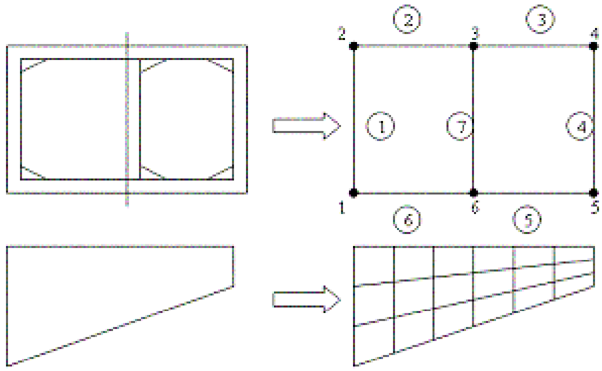

Fig. I.1. Modelling of frame and plane stress problems [8].

I.3. Element analysis

The element analysis has two key components; expressing the displacements within the elements, and maintaining equilibrium of the elements. In addition, stress-strain relationships are needed to maintain compatibility. The final result is the element stiffness relationship:

$S = kv$

where:
S = Generalized nodal point forces
k = Element stiffness matrix
v = Nodal point displacements
For beam elements this relationship was obtained using the exact relationships between forces and moments and the corresponding displacements. These results could therefore be interpreted as being obtained by the governing differential equation and boundary condition of the beam elements. For e.g. a plane stress problem it is not possible to use an exact solution. The displacements within the elements are expressed in terms of shape functions scaled by the node displacements. Hence, by assuming expressions for the shape functions, the displacements in an arbitrary point within the element are determined by the nodal point displacement.

The section of the structure that the element is representing is kept in place by the stresses along the edges. In the finite element analysis it is convenient to work with nodal point forces. The edge stresses may in the general case be replaced by equivalent nodal point forces by demanding the element to be in an integrated equilibrium using work or energy considerations. This technique is often referred to as to "lump" the edge forces to nodal forces.

After structural modelling the element characteristics like stiffness matrix [k] and nodal load vector [p] are calculated for each element. These [k] and [p] are transformed to global level by multiplying with transformation matrix. These element characteristics are assembled to get the stiffness matrix and load vector for the structure in the global system.

This requirement results in a relationship between the nodal point displacements and forces to be given as

$S = kv + S^0$

Where,
S^0 = nodal point forces for external loads

Computer programs usually have many options for types of elements to choose among [9]. Fig. I.2 shows the most usual elements.

I.4. System analysis

A relationship between the load and the nodal point displacements is established by demanding equilibrium for all nodal points in the structure

$$R = Kr + R^0$$
$$K = \sum_j a_j^T K_j a_j$$
$$R^0 = \sum_j a_j^T S_j^0$$

Where,
R = Load vector
r = Global displacements

The stiffness matrix is established by directly adding the contributions from the element stiffness matrices. Similarly the load vector R is obtained from the known nodal forces.

I.5. Boundary conditions

Boundary conditions are introduced by setting nodal displacements to known values or spring stiffness are added.

I.6. Finding global displacements

The global displacements are found by solving the linear set of equations stated above:

$r = K^{-1}(R - R^0)$

I.7. Calculation of stresses

The stresses are determined from the strains by Hooke's law. Strains are derived from the displacement functions within the element combined with Hooke's law. They may be expressed generally by

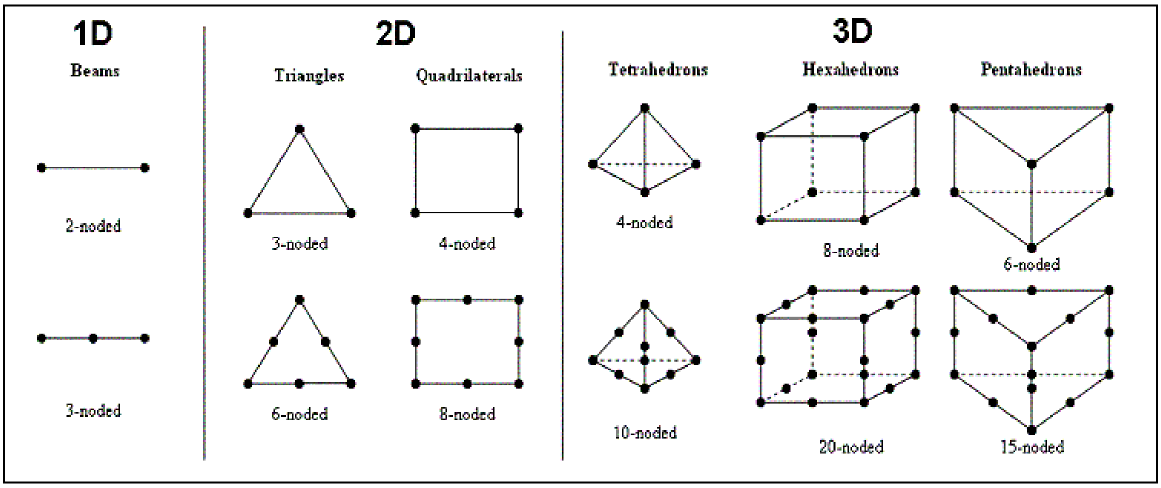

Fig. I.2. Different 1D, 2D and 3D basic elements.

$\sigma(x,y,z) = D\,B(x,y,z)v$

Where,
$v = a\,r$
D - Hooke's law in matrix form
B - Derived from u(x, y, z) Output interpretation programs, called postprocessors, help the user sort out the output and display it in graphical form.

A Finite Element Analysis (FEA) consists of three separate stages; preprocessing, processing, and postprocessing. A complete finite element analysis is a logical interaction of these three stages.

References

[1] K.J. Bathe and E.L. Wilson, Numerical Methods in Finite Element Analysis, Prentice Hall Inc, (1996).

[2] R.D. Cook, Concepts and Applications of Finite Element Analysis, John Wiley and Sons, Inc, Third Edition (1989).

[3] W. Jiang et al., "Finite Element Modelling of Stiffened and Unstiffened Orthotropic Plates", Computers and Structures, Vol. 63 (1), pp. 105-117 (1997).

[4] D. Servis et al., "Finite Element Modelling and Strength Analysis of Hold No. 1 of Bulk Carriers", Marine Structures 16 (2003), pp. 601–626 (2004).

[5] Hadi K. Amlashi and T. Moan, "Ultimate Strength Analysis of a Bulk Carrier Hull Girder under Alternate Hold Loading Condition – A Case Study Part 1: Nonlinear Finite Element Modelling and Ultimate Hull Girder Capacity", Marine Structures 21, pp.327–352 (2008).

[6] T.S. Koko and M.D. Olson, "Nonlinear Analysis of Stiffened Plates using Super Elements", International Journal for Num Methods in Engineering, Vol. 31, pp. 319-343 (1991).

[7] J.K. Park, A.K. Thayambillil and Y.E.I. Park, "Local Buckling of Stiffeners in Ship Plating", Journal of Ship Research, Vol. 42, pp. 56-67, March (1998).

[8] C.S. Smith et al., "Strength of Stiffened Plating under Combined Compression and Lateral Pressure", RINA, pp. 131-142 (1991).

[9] M.A. Daye and T.G. Tordis, "Elasto Plastic Algorithms for Plates and Shell under Static and Dynamic Loads" Computers

and Structures, Vol. 39 (12), pp. 195-205 (1991).
[10] C.J. Chen, W. Liu and S.M. Chern, "Vibration Analysis of Stiffened Plates", Computers and Structures, Vol. 50 (4), pp. 471-480 (1994).
[11] M. Mukopadhyay, "Stiffened Plates in Bending", Computers and Structures, Vol. 50 (4), pp. 541-548 (1994).

Received April 21, 2009
Accepted August 4, 2009