An Investigation On Boundary Interpolation Errors Of The Submodeling Technique

Gokul Arasu. C **(Assistant Surveyor, Indian Register of Shipping)**
P. Kurinjivelan **(Assistant Surveyor, Indian Register of Shipping)**
A. Samanta **(Associate Vice Président, Indian Register of Shipping)**

Prediction of stress concentration is becoming inevitable task as the finite element method emerges out in the field of fatigue analysis. Stress concentration can be determined either by empirical formulae or by finite element analysis. Submodeling is one of the well established techniques in finite element analysis to compute the stress concentration. However the major and important drawbacks in submodeling technique are the problem of error associated with the cut boundary degree of freedom interpolations. In order to avoid overshooting of global polynomial interpolations, the interpolation function can be a piecewise polynomial. This piecewise interpolation technique is being adopted in the commercial finite element analysis tool 'Ansys'. Even though the piecewise interpolation is used, accuracy depends on the order of the interpolating function used since the software uses the shape function of element for the piecewise interpolations. To achieve better accuracy with reasonable number of elements, a linear element with higher order interpolation scheme can be adopted. This paper deals with the validity of the submodeling technique by checking the sensitivity of the results for various mesh sizes and extent of the submodel. A cubic spline interpolation scheme with linear element is adopted to enhance accuracy in the submodeling technique in the present paper.

KEY WORDS

Cubic Spline Interpolation; Finite Element Analysis; Hot Spot Stress; Submodeling Technique.

INTRODUCTION

The accurate determination of hot spot stress due to geometric discontinuities ensures the precise assessment of fatigue life. The hot spot stress approach for the fatigue life assessment of ship structure is a proven and well-known technique as it can be applied to any complex geometry of ship structures. Most of the classification societies recognized this method as an effective way to estimate the fatigue life.

Two methods are available for the prediction of hot spot stress. In the first method the classification societies prescribe some empirical expressions to find out the hot spot stress concentration factor which can be multiplied to the nominal stress to get the hot spot stress. Another way for the accurate determination of hot spot stress is by the use of finite element analysis. The difficulty present in the finite element analysis method is the high stress gradient of the hot spot stress, which needs very fine mesh analysis to capture the stress at the hot spot, more accurately. The prediction of hot spot stress plays a vital role in fatigue analysis. However, for a full ship analysis, it is not possible to discretize the whole ship by refined elements, as it will increase the number of equations greatly, causing difficulty in solving the problem even by computer. There are a few techniques available to determine stress concentration effect namely local refinement methods, substructuring and submodeling technique. Among these, the submodeling technique is recognized as more suitable as it can give more accurate results with lesser resources. In submodeling technique, a sub region is broken out from the original global region and analyzed separately. This sub model requires boundary conditions taken from the finite element analysis of the global model. This method is more effective in saving computational time. However the submodeling technique has some restrictions also. This method gives accurate results away from the boundary because of the imposed displacements at the cut boundary. This can be overcome by selecting the region of interest away from the cut boundary. Another major and significant drawback is the error associated with the cut boundary degree of freedom interpolations. The cut boundary interpolations are required since the cut boundary nodes of the submodel are more when compared to the global model nodes at the cut boundary section, due to the fact that the submodel is always finer than the global model. Hence there is a need of an interpolation technique to get the cut boundary degree of freedom of the submodel.

Piecewise polynomial interpolation function can be adopted to avoid the overrun of the global polynomial interpolations. The general purpose finite element analysis tool Ansys uses the shape function of the element for the piecewise interpolation. Although the piecewise interpolation is used by Ansys, the accuracy depends on the order of the interpolating function used. In Ansys, the linear elements (Shell63) use the linear piecewise interpolation and the quadratic elements (Shell93) use the quadratic piecewise polynomial interpolation. Hence the accuracy depends on the type of the element used. In order to achieve better accuracy, some higher order interpolation scheme can be adopted. In the present paper cubic spline interpolation is used to determine the cut boundary degrees of freedom. The smoothness and validity of this curve fit is accurate when compared to other interpolation techniques. This interpolation scheme can be applied for lower order elements to obtain a higher accuracy.

METHODOLOGY

In the submodeling technique, displacements are calculated on the cut boundary of the global model and are specified as boundary conditions for the sub model. This technique is studied on:

- Benchmark problem: Plate with UDL is taken and the results of the global model and the submodel are checked with the target value. (Timoshenko and Woinowsky-Krieger, 1984)
- The observations from the plate problem for the submodeling technique are verified for a transverse bulkhead of an oil tanker.
- The VB code developed for cubic spline interpolation scheme for the submodeling technique is verified for the plate and a ship model.

ANALYSIS

PLATE PROBLEM

The plate problem is taken at the initial stage of analysis, since the analytical results are available. The results are compared with Ref 1. In the present case various parameters are changed to check the various sources of errors in the submodeling technique for the plate problem. To check the validity of the observed results for different plate dimensions, the aspect ratio of the plate is varied from 1 to 2. The other parameters like extent of the model and most importantly the variation in the result with respect to the global model mesh size is also studied. The different extent of the model studied is shown in Table 1.

Meshing

The main objective of this problem is to identify the interpolation errors. The interpolation error mainly depends on the global model mesh size to submodel mesh size ratio. When the global model element size is larger (coarse mesh), the error will be more. To quantify these errors for linear and quadratic elements different mesh size are chosen for the global model. For the prediction of stress concentrations, the element size should be in the range of plate thickness as per the guideline given by the classification societies for the refined mesh analysis. In the present analysis the submodel mesh size is taken 10mm as the plate size is 10mm. Various mesh sizes adopted for the analysis are shown in Table 2.

Boundary conditions

Simply supported boundary condition is given on all the edges of the global model. In the submodel the cut boundary degrees of freedom are interpolated from the global model and applied to the submodel.

Table 1. Extent of the model

Plate Size (mm)	Extent of Submodel (mm)
1000 x 1000	500x500, 600x600, 700x700, 750x750
1000 x 1500	500x750, 600x900, 700x1050, 750x1125
1000 x 2000	500x1000, 600x1200, 700x1400, 750x1500

Table 2. Mesh sizes

Global model Mesh size (mm)	Submodel Mesh size (mm)
80 x 80	10 x 10
60 x 60	10 x 10
40 x 40	10 x 10
20 x 20	10 x 10
10 x 10	10 x 10

Loading

In all the cases a constant pressure value of 0.01 N/mm^2 is applied on the global model as well as on the submodel.

Results and discussion

The deformation pattern of the global model is shown in Fig.1. The contour plot shows the maximum deformation at the center of the plate. The results at the central location are compared. Similarly the von Mises stress is also observed for verification of the submodeling technique for various cases. The maximum stress is observed at the centre of the plate and compared in all the cases. Fig.2 shows the stress pattern of the plate when subjected to uniformly distributed load. The maximum deformation and the stresses are compared in all the cases with the analytical solution. The percentage error in all the cases is also observed. Fig.3 and Fig.4 show the comparison of errors for various extents of the linear and quadratic shell element models, for deformation and stress respectively. It is worth noting that the quadratic element gives better results as compared to the linear elements for submodeling technique. This is due to the increase in number of nodes as compared to linear elements. Also the polynomial order of shape function is higher in case of quadratic element. Since Ansys uses the shape function for the interpolation scheme, quadratic shell elements give better results as compared to linear shell elements.

When 0.2 % of error in the submodel deformation result is considered as accepted, global model mesh size can be 8 times larger than the submodel mesh size for quadratic shell elements. In other words, the submodeling can be done with a mesh size of 0.125 times the global model mesh size, provided the global model results are converged solution or the global model mesh size is justified. For the same case, global model mesh size can only be 4 times larger than the submodel mesh size for linear shell elements. In other words, the submodeling can be done with a mesh size of 0.25 times the global model mesh size provided the global model results are converged or if the global model mesh size is justified. Extent of the model from 25 % to 56 % of the global model did not vary the deformation and stress results considerably.

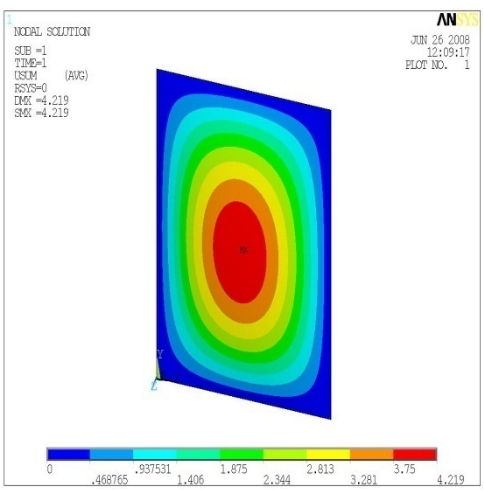

Fig.1 Total deformation

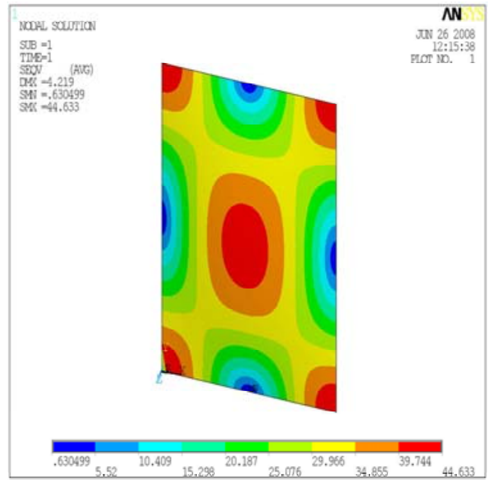

Fig.2 von Mises stress pattern

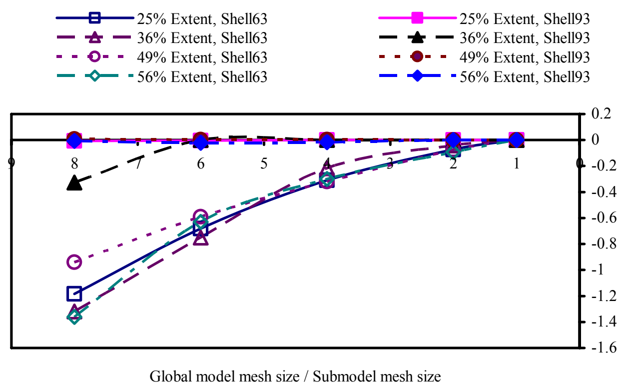

Fig.3 Error comparison for the central deformation cases

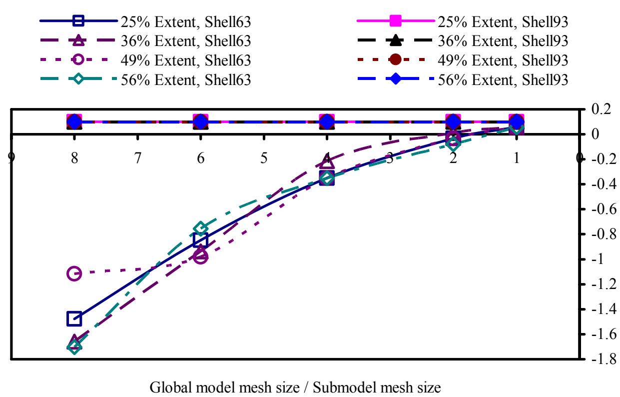

Fig.4 Error comparison for the stress cases

VERIFICATION OF THE OBSERVATIONS FOR A TRANSVERSE BULKHEAD

Modeling

A transverse bulkhead of an oil tanker is modeled to study the observations obtained from the plate problems. The tank bulkhead consists of three stringers and stiffeners as shown in Fig.5. The girders and stringers are modeled as shell element to predict the stress concentration accurately. Only half the bulkhead is modeled to take the advantage of symmetry.

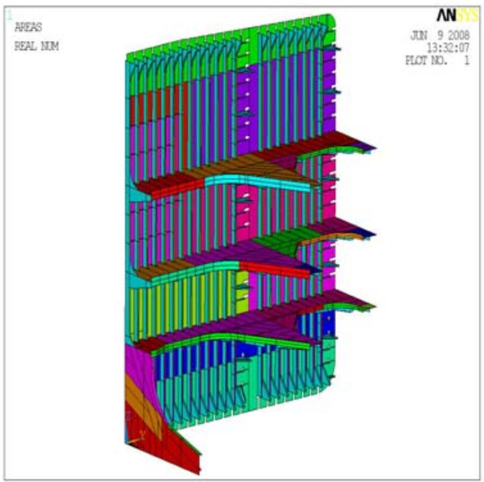

Fig.5 Transverse bulkhead of an oil tanker

Meshing

Mesh size is the varying parameter for each of the global model. Submodel mesh size is 100 mm in all the cases. But the global model mesh size is varying from 100mm to 600mm. The variation in the accuracy of the submodel with respect to the global model mesh size is studied in the present work.

Axis System

The axis system followed in modeling the bulkhead is described as follows:

X axis: Along the length of the ship
Y axis: Along the beam of the ship
Z axis: Along the depth of the ship

Boundary conditions

An idealized fixed boundary condition is applied at the edges and along the longitudinal bulkhead line of the transverse bulkhead due to the encasement of the edges to the side shell and longitudinal bulkhead respectively. At the centerline (vertical) of the bulkhead, symmetry boundary condition is given (Uy, θx, θz = 0).

Loading

A constant and uniformly distributed load of 0.1 N/mm^2 is applied for the study purpose.

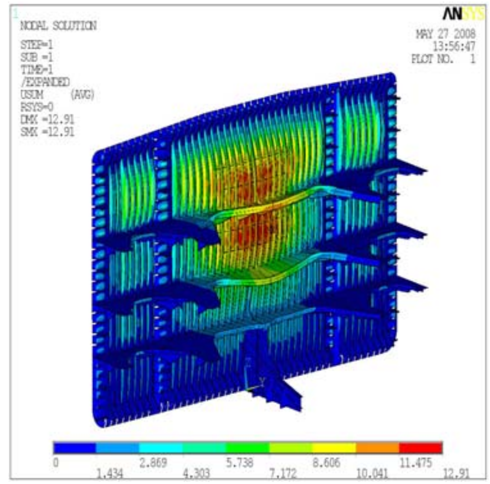

Fig.6 Total deformation of bulkhead

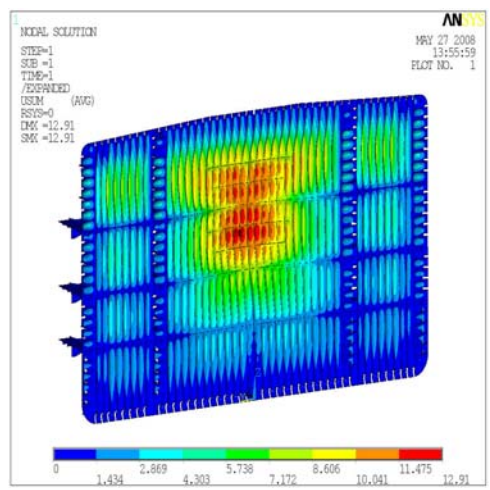

Fig.7 Total deformation-rear view

Results and discussion

The deformation and stress results are observed for all the cases. The total deformation obtained is shown in Fig.6 and Fig.7. The global model deformation and the submodel deformation are shown in Fig.8 and Fig. 9. The deformation and the hot spot stress values are taken at a bracket toe, which is considered as one of the hot spots in the fatigue analysis. The trend of the deformation and the stress values for various submodel mesh size to global model mesh size ratio is shown in Fig. 10 and Fig. 11. The percentage of error for deformation and stress as compared to the global model with a mesh size equal to the submodel mesh size (100mm) is shown in Fig.12 and Fig.13. The comparison of the percentage of error for deformation and stress values obtained in plate problem and bulkhead problem is shown in Fig.14 and Fig.15.

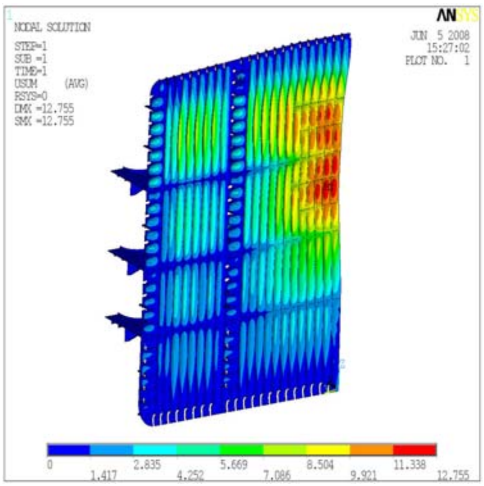

Fig.8 Global model deformation

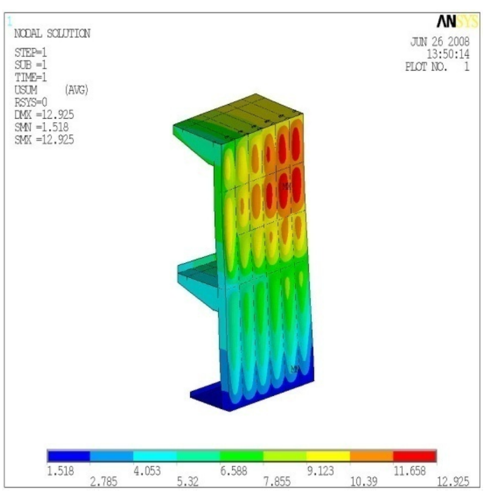

Fig.9 Submodel deformation

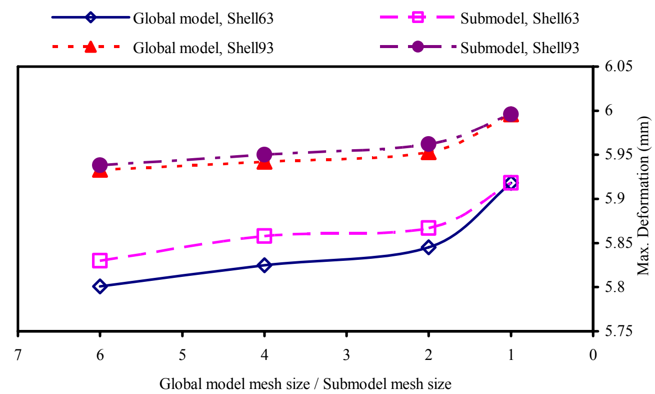

Fig.10 Maximum deformation plot for the transverse bulkhead

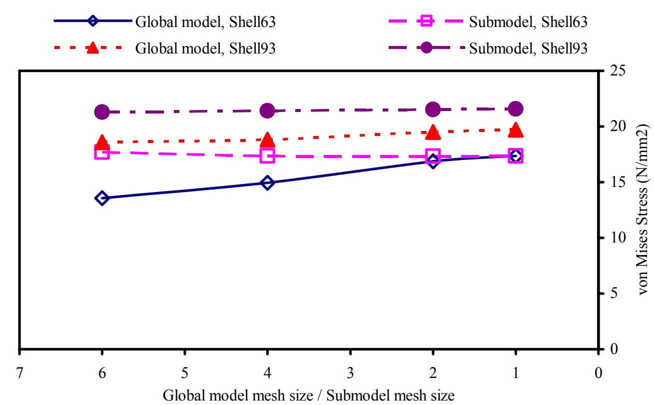

Fig.11 von Mises stress plot for the transverse bulkhead

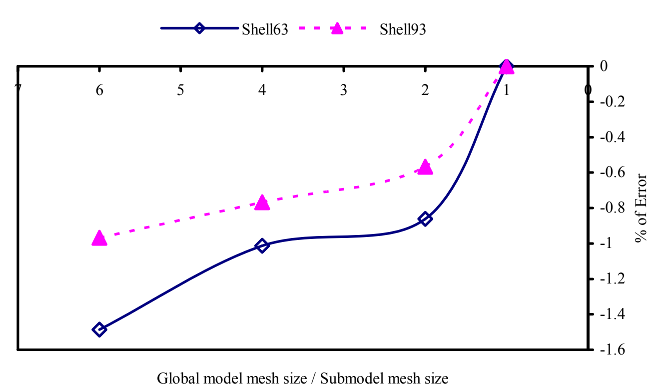

Fig.12 Comparison of error in deformation

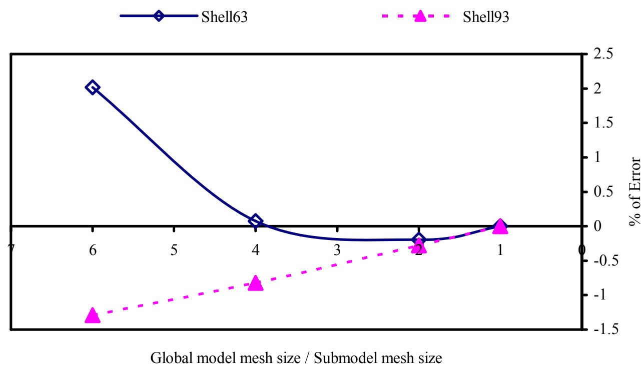

Fig.13 Comparison of error in von Mises stress pattern

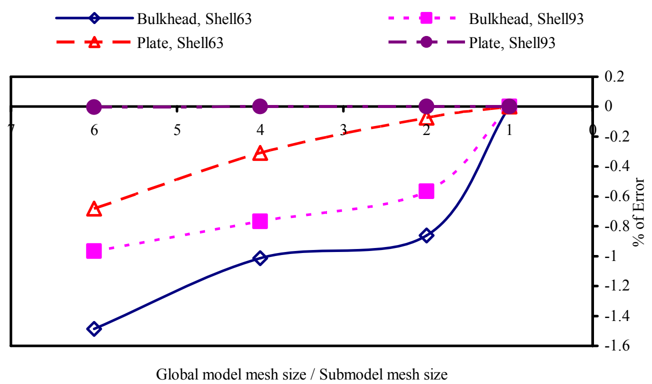

Fig.14 Comparison of error in deformation

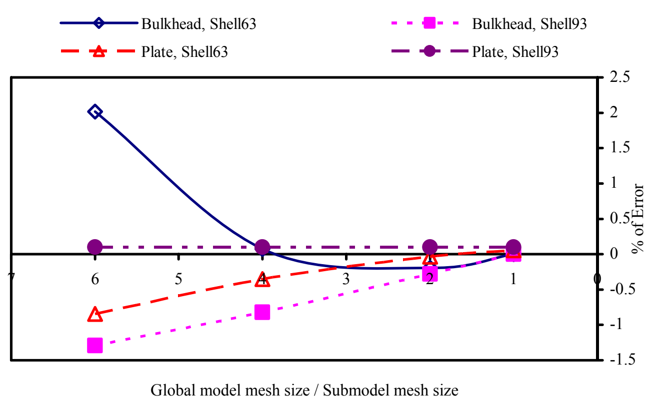

Fig.15 Comparison of error in stress

CUBIC SPLINE INTERPOLATION

While applying submodeling technique for some complex structures such as ships, it is not advisable to choose quadratic element for modeling due to the increase in computational resource. Linear element model can be opted in such cases. However a large variation between the actual result and the linearly interpolated results in submodeling technique are observed. This is due to the fact (as already mentioned) that linear elements use linear interpolation scheme. In such cases, a linear modeling with higher order polynomial interpolation scheme for submodeling can be implemented. Cubic spline interpolation fits the piecewise cubic polynomial for each couple of data points in the total set of points. It passes through all the available values and fits well to get the required value at the unknown points. The curve fit of the field variable ensures the displacement, slope and curvature continuity at all the data points. So the smoothness and validity of this curve fit is accurate compared to other piecewise linear and quadratic polynomials.

The assumed form for the cubic polynomial curve fit for each segment is,

$$y = a_i(x-x_i)^3 + b_i(x-x_i)^2 + c_i(x-x_i) + d_i \qquad (1)$$

Where the spacing between successive data point is

$$h_i = x_{i+1} - x_i \qquad (2)$$

The cubic spline ensures the continuity of the derivatives at the data points.

Case Study for the Plate

The benchmark plate problem is considered. For submodeling technique, cubic spline interpolation scheme is used for the interpolation of cut boundary degrees of freedom. The plate is analyzed and the results are presented in the Fig.16 and Fig.17.

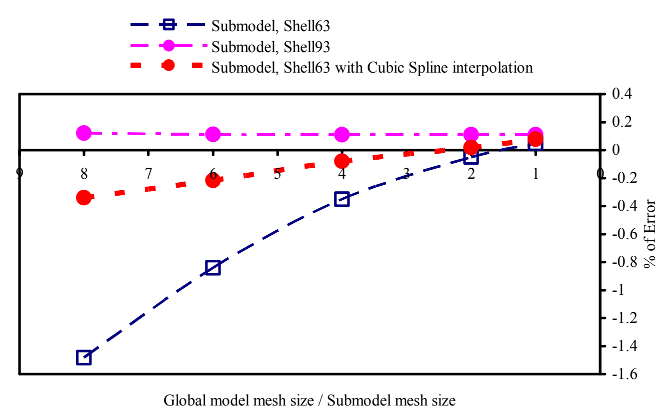

Fig.16 Comparison of various interpolation schemes for Deformation

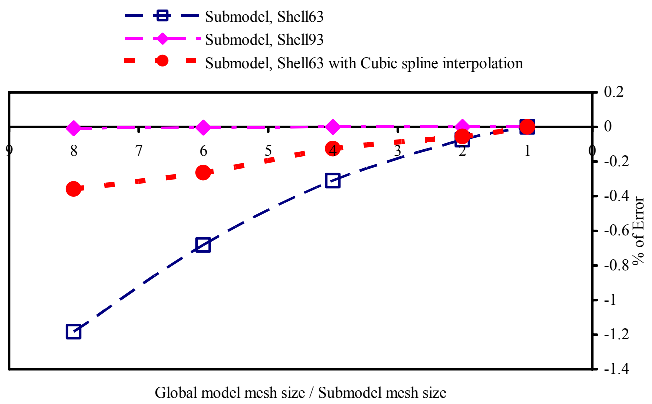

Fig.17 Comparison of various interpolation schemes for von Mises stress

Results and Discussions

Fig.16 shows the percentage of error observed in each of the cases for deformation and the Fig.17 represents the percentage of error observed for the von Mises stress (The values are compared with the value obtained for global model with mesh size = submodel mesh size). As observed from Fig.16 and Fig.17 there is a considerable improvement in the results when cubic spline interpolation is used for the linear element (Shell63).

Case Study for a Ship Structure

An offshore patrol vessel (OPV) of 93m length, 12.6m beam and 6m depth is considered for the study purpose. A detailed 3-D Shell–Beam model of the full ship is modeled based on the drawings submitted by Yard, using the Ansys. All plating and primary girders of the vessel are modeled with four noded linear Shell element. For modeling longitudinal and transverse stiffeners, 3-D Beam element is used. All columns are modeled by using Pipe element. The weights of the components that are not modeled are simulated using Mass elements. A combination of quadratic and triangular finite element mesh of the model is generated using free meshing controls. Fig. 18 represents the FE model of the OPV.

Loading Condition

The head sea 100% loading condition is considered. The static analysis results are obtained for the loading condition.

Results and Discussions

Fig.19 shows the deformation plot of the global model (mesh size 600mm) for the model extent from 34400mm (57th frame) to 76600mm (128th frame) along the x-axis. Fig.20 shows the deformation pattern for the submodel with boundary conditions interpolated from main model using Ansys interpolation scheme. The submodel for the same extent is considered. A uniform mesh size of 300mm is adopted for the submodel. The cut boundary DOFs are interpolated from the main model with the help of Ansys. The same is applied for the submodel as boundary condition and solved. Another submodel of the same extent is considered with a mesh size of 300mm. In this case, the cut boundary DOFs are interpolated using the VB code developed for cubic spline interpolation scheme. Fig. 21 shows the deformation pattern for the submodel with the boundary conditions interpolated using the cubic spline interpolation scheme. Fig. 22 shows the stress plot for the global model. Fig.23 shows the stress pattern for the submodel with the boundary condition interpolated by linear interpolation scheme. Fig. 24 shows the stress pattern for this case. . It is worth noting that for submodel with cubic spline interpolation, the deformation and stress patterns match well with that of the global model result for a linear element.

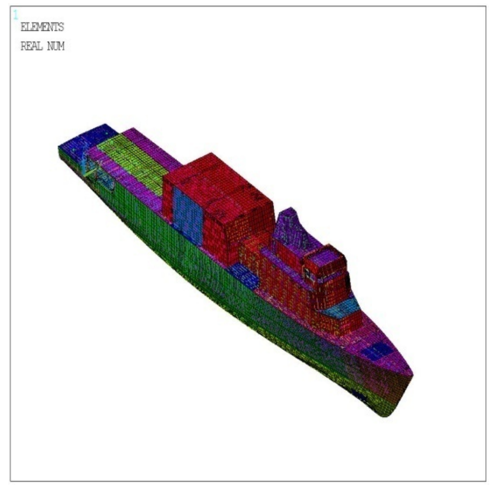

Fig.18 Finite Element model of OPV

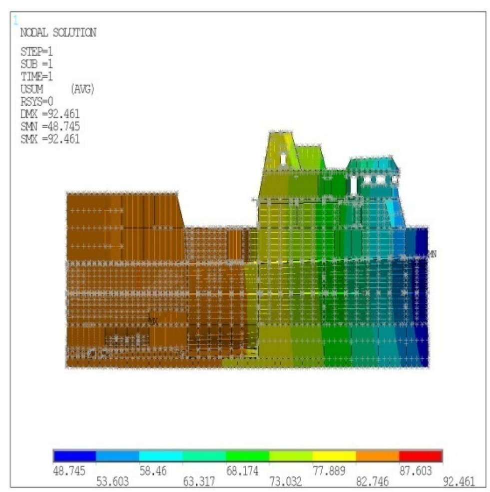

Fig. 19 Deformation pattern of Global model (600mm mesh size, δ_{max} = 92.461mm)

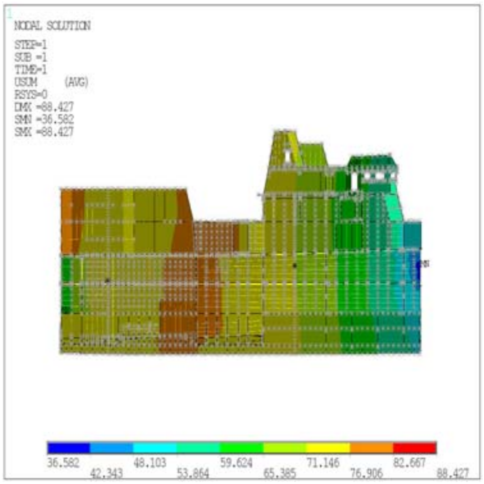

Fig. 20 Deformation pattern of Submodel
(Linear Interpolation, mesh size 300mm, δ_{max} = 88.427mm)

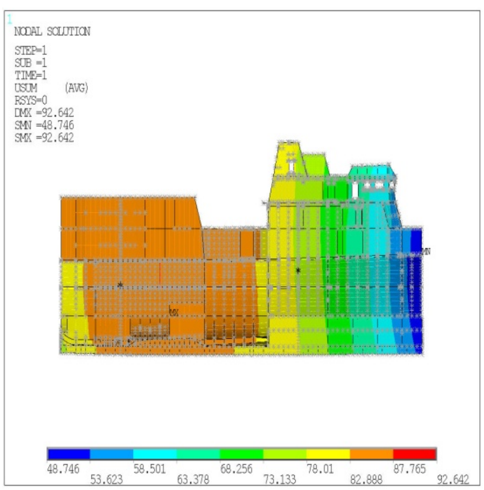

Fig. 21 Deformation pattern of Submodel
(Cubic spline Interpolation, mesh size 300mm, δ_{max} = 92.642mm)

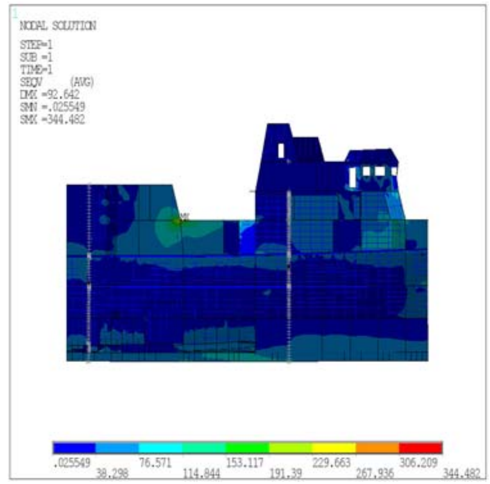

Fig. 22 Stress pattern of Global model
(600mm mesh size, σmax = 344.482N/mm2)

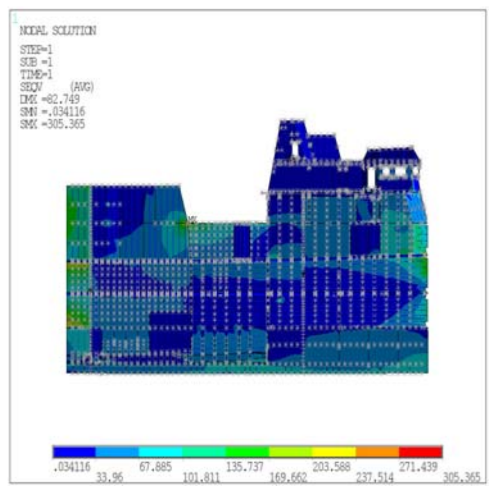

Fig. 23 Stress pattern of submodel
(Linear Interpolation, mesh size 300mm, σ_{max} = 305.365N/mm^2)

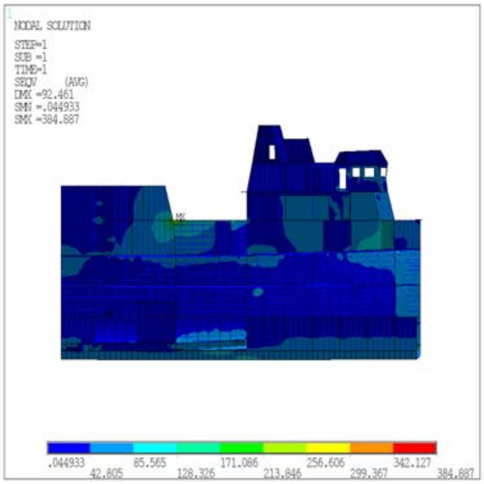

Fig. 24 Stress pattern of submodel
(Cubic spline Interpolation, mesh size 300mm, σ_{max} = 384.887N/mm^2)

CONCLUSIONS

Though the quadratic element (Shell93) is giving better result over linear element (Shell63) it is not always advisable to choose the quadratic element as it has more number of nodes, thus analysis is costly compared to the linear element. General purpose software like Ansys allows only linear interpolation when a linear element is used. In this paper an alternative method is proposed where linear element is used for the analysis and at the same time cubic spline interpolation is applied to achieve better accuracy in the result. From the results, it is observed that, cubic spline interpolation gives a better result and can be implemented for submodeling purpose, particularly when a large structure (i.e. Ship, aircraft) is analyzed.

ACKNOWLEDGEMENT

The authors acknowledge Mr. K.V.Rajagopalan from Naval Architecture Group of Indian Register of Shipping for his contributions in the VB programming for the cubic spline interpolation scheme.

REFERENCE

1. Timoshenko S, Woinowsky-Krieger S. *Theory of Plates and Shells*, Second Edition. Singapore. McGraw-Hill, 1984.