# CALCULATION OF THE HOT SPOT STRESSES BY THICK SHELL ELEMENTS FOR LONGITUDINAL STIFFENER CORNER

# Timo P. J. Mikkola Technical Research Centre of Finland (VTT), Ship Laboratory, Tekniikantie 12, Espoo, Finland

Rakenteiden Mekaniikka, Vol. 26 No 2 1993, ss. 3 - 16

#### ABSTRACT

The hot spot stress approach can be used for more accurate fatigue analysis of welded plate structures. The required structural stress concentration at the critical location can be calculated by the finite element method. Thick shell elements were tested for this task by analysing a simple case of a longitudinal stiffener. A program HOREX was developed for the required stress extrapolation from the finite element stress results. In practical applications a very fine mesh is required at the fatigue critical area. The present results gave a relative element side length requirement of < 0.4 times the plate thickness. The thick shell elements gave accurate hot spot stress concentration factor results for the present simple test case. The local mesh refinement of shell element models was automated. The programs developed were applied for a post test analysis of variable amplitude fatigue tests of aluminium ship hull structure. The P/FATIGUE® program was applied for the numerical fatigue analysis.

#### INTRODUCTION

The design against fatigue failure has become increasingly important in shipbuilding due to the use of new high strength steels and aluminium. The new designs are also being more and more optimised for tighter and/or case specific structural specifications. New fatigue design practices need to be developed and applied for full utilisation of the new materials and for the fulfilment of the required structural specifications. The fatigue analysis must be based on realistic load history data which can be obtained by direct measurements or numerical simulation, see e.g. ref. [1]. Effective equipment

and numerical procedures are required for transforming the measured, often complex service load histories to stress cycle data. The hot spot approach seems to be an ideal tool for the fatigue analysis of welded steel structures [2,3]. It uses a clearly defined stress measure applicable both in experimental measurements and in numerical analyses. The fatigue failure properties of the material are defined by simple welded test pieces and only the structural stress concentration, so called hot spot stress, is required for each analysed structural detail. The complexity of practical engineering designs often make direct measurement of the hot spot stresses difficult or impractical. The numerical stress analysis methods e.g. the finite element (FE) method can be utilised for scaling the measured nominal stress levels to hot spot stresses at the fatigue critical structural details. Numerical analysis methods can be applied during the whole product life cycle and in modern design practice the numerical and experimental analysis methods should be tightly connected.

The FE method is well established and robust method for stress analysis of complex structural components and it suits well for the analysis of hot spot stresses. The difficulties in using the FE method are associated with the interpretation of the peak stresses. The hot spot stress is defined as the structural stress at the hot spot which excludes the local non-linear peak stresses due to the weld geometry [2]. In the FE model the hot spot is usually a sharp corner and thus a singular point. The analyst has several possibilities for modelling the joint area ranging from solid three dimensional (3D) elements, thick or thin shell elements or a combination of these. The 3D solid elements are better suited for modelling the weld geometry. The shell elements on the other hand exclude the nonlinear stress distribution over wall thickness per definition whereas the 3D elements do not. All element types require very fine mesh at the hot spot which easily results in unpractically large FE models.

The use of shell elements is advantageous for two reasons when compared to the use of solid elements. Firstly the model generation is easier and secondly the number of degrees of freedom per element is only 2/3 when comparing an eight noded five degree of freedom per node shell element to the usual 20 noded solid element. The difficulties are connected to shell elements structural model and to the post processing of the stress results. The shell element models the structure by surfaces with translational and rotational stiffnesses. This model may give rise to additional singularities at areas where structural members having different surface orientations are connected. A typical problematic case is the longitudinal stiffener in a plate. In the result post-processing typically nodal averaged values are used and these may be meaningless at such locations. Further, the model for the welded connection is too flexible when the weld itself is not modelled but it may be difficult to define a suitable shell or beam element for modelling the weld. Another difficulty associated with the shell element models are the possible offsets at the joints connecting different structural members.

In the present study the applicability of the thick shell element was tested for calculating the hot spot stress concentration factor. Special attention was given for developing FE model pre- and post-processing programs to enable easy model generation and hot spot stress calculation.

### FINITE ELEMENT MODELLING

The use of commercially available general purpose FE model pre- and postprocessing programs seems unavoidable when practical engineering work is considered. In the present work the PATRAN<sup>®</sup> program was applied. The calculation of the hot spot stresses sets special requirements for the model and results post-processing which are, at least at present, not well supported by general purpose programs. At least three different problems can be identified which are connected to mesh refinement at the hot spot area, result post-processing and the use of different types of elements.

The FE model around the hot spot must be very fine and the mesh design at this area should support calculation of the hot spot stresses. Outside this area a much coarser mesh is usually sufficient. Direct use of general purpose programs for the mesh generation gives easily an excessively large model or a compromise must be made between the model size and the required mesh density at the hot spot area. The mesh refinement towards the hot spot is typically done either by irregular sized and oriented triangulars or irregular shaped quadrilaterals both of which are undesirable at this area. The present work applied an h-version local model refinement procedure which was originally developed for local refinement of 2D plane FE models [4]. The modifications required for the shell elements were implemented to the Post3D program [4]. The FE model refinement is made by selecting a group of elements around the critical location and dividing each element to four new elements, see initial and refined FE model details in Fig. 1. The continuity of the displacement field over the element edges is sustained by generating constraint equations. The initial mesh can be designed to account for the nominal stresses in the structure and near the hot spot it is advantageous to use rectangular shaped elements.



Fig. 1 Local model refinement.

The general purpose programs support usually only the use of nodal point stress results. Typically the integration point results for each element are extrapolated to the nodal points separately for each element. The nodal values from different elements are averaged and this continuous stress field is then used for post-processing. This approach is straightforward and suits well for practical work although in some special cases it may produce meaningless results. For shell elements problems arise at the joints where shell elements with different surface normals are connected. For 3D solid elements the same problem is encountered when elements with different materials are connected. In the present study this problem was avoided by modelling the structure with several FE model parts and connecting these by constraint equations. The ACR program [5] was slightly modified for automatically generating the required constraint equations.

The modelling of the weld may cause additional problems. In calculating the hot spot stresses the local stress raising effects due to the weld should be excluded by definition. For this reason modelling of the weld with solid elements is usually done with only one element over the wall thickness, see e.g. [6]. When shell elements are used, the definition of the shell elements modelling the weld is problematic. Sometimes 3D solid elements are used for modelling the weld and shell elements are used elsewhere. The connecting of these two parts is usually not supported by the general pre-processing programs. If transition elements are used then also the post-processing may be difficult. In the present work the weld was totally excluded from the model and this approach seemed to be sufficient for the present cases.

#### HOT SPOT STRESS EXTRAPOLATION

The hot spot stress should be extrapolated to the hot spot using the stress results along a line perpendicular to the weld [2]. Linear extrapolation from the stress values at the distances of  $0.4\times t$  and  $1.0\times t$  is recommended. For cases with strong stress gradients, parabolic extrapolation can be used and for this the use of the stress values at the distances of  $0.4\times t$ ,  $0.9\times t$  and  $1.4\times t$  is recommended. When shell elements are applied, an additional problem for the calculation can be caused by the structural model. A problematic case is e.g. a longitudinal stiffener in a plate, see Fig. 2, as the stresses are transmitted between the plate and the stiffener through a line instead of a surface defined by the stiffener and weld seam thicknesses. In such cases the calculated stress peak at the hot spot location can become too high and narrow. Direct use of the nodal point stresses leads to overestimation of the hot spot stresses, see e.g. ref. [2].

In the present analyses the hot spot stresses were calculated using the nodal point stresses by the HOREX (Hot spot stress extrapolation) program. The stress at the hot spot location was extrapolated from the stress values at the distance of  $0.4\times t$  and  $1.0\times t$  or  $0.4\times t$ ,  $1.2\times t$  and  $2.0\times t$  with linear or parabolic

7

extrapolation respectively. The difference between the present points used for the parabolic extrapolation and those recommended in ref. [2] is due to historic reasons and the HOREX program has been changed in this respect. The extrapolation points don't have to coincide with the nodal points as the HOREX program interpolates the stresses along the extrapolation line using the nodal point values and element shape functions. The HOREX program calculates the hot spot stresses also by averaging the stresses in the transverse direction for a user specified width before the hot spot stress extrapolation. The stress averaging in transverse direction is done using the continuous stress field defined by the element shape functions and the nodal point stresses.



Fig. 2 A plate with a welded longitudinal stiffener and its FE model.

#### PLATE WITH WELDED LONGITUDINAL STIFFENER

A plate with welded longitudinal stiffener, see Fig. 2, was analysed in order to verify the present approach for the hot spot stress calculation. The dimensions of the structure were: the plate width 120 mm, the plate and stiffener thicknesses 12.5 mm, the stiffener length 150 mm and the stiffener height 30 mm on both sides of the plate. Symmetric half of the structure was modelled, see Fig. 2. The stiffener was welded symmetrically on both sides of the plate. The plate was loaded by an axial force giving nominal stress of 100 MPa. The FE analyses were made by the ADINA<sup>®</sup> program using eight noded thick shell elements. The analysis was repeated with four FE models by gradually refining the mesh and with a fifth model with refined mesh only at the plate and stiffener corner, Table 1 and Fig. 3. The weld was not modelled.



Fig. 3 Different FE models for the longitudinal stiffener case. Only model details at the stiffener corner are shown. Detail dimensions were plate width 16 mm, plate length 21 mm and stiffener height 20 mm.

Table 1 FE model data for the plate with longitudinal stiffener, dof is degrees of freedom and h/t is the relative element side length in axial direction at the corner.

Model	elements	nodes	dof	h/t
1	104	370	1686	0.91
2	368	1214	5676	0.45
3	530	1742	7932	0.23
4	1178	3770	17304	0.11
5	152	546	2264	0.23

9

The longitudinal stiffener created a very strong and narrow peak in the axial stress distribution in front of the corner formed by the plate and the stiffener end, Fig. 4. The highest peak stress was found in the plate at the first node away from the corner as the nodal averaging lowered the axial stress at the corner even though the nodal averaging was done separately for the stiffener and the plate. As the FE mesh was refined the stresses were radically changed only at the elements with one corner node at the hot spot location. Thus the mesh refinement affected the hot spot stress as long as the first extrapolation point was located in the first element. This gave an element side length requirement of h/t < 0.4 for the first element at the hot spot location. The present size requirement is connected to the singularity at the hot spot location and larger elements could be acceptable for non-singular cases.



Fig. 4 Axial stresses a) at plate mid section and b) at plate cross section at the stiffener corner. Note that results for models 3 and 5 fit exactly.

The hot spot stresses were extrapolated to the stiffener corner by the HOREX program using both linear and parabolic extrapolation. Both the nodal point stresses and the stresses averaged in transverse direction for the stiffener thickness were used. The extrapolated hot spot stresses are given in Table 2 as stress concentration factors. In Fig. 5 the extrapolation of the hot spot stresses and the FE results for the axial stresses are shown for the model 5. The parabolic extrapolation gave slightly higher results compared to the linear extrapolation. The nodal point stresses gave significantly higher hot spot stresses when compared to the stresses averaged in transverse direction for the stiffener thickness. The element size requirement was met by the

models 3 to 5 and correspondingly the hot spot stress results varied very little. The result calculated using the stresses averaged for the stiffener thickness coincided well with the result  $K_s=1.4$  given in ref. [6] and even the nodal point stresses gave realistic results of around 1.6.

Model	K <sub>s</sub>	Ks	K <sub>S</sub>	Ks
	nodal $\sigma$	nodal $\sigma$	aver. o	aver. $\sigma$
extrap.	linear	parab.	linear	parab.
1	1.97	2.06	1.60	1.65
2	1.92	1.99	1.51	1.53
3	1.58	1.62	1.41	1.43
4	1.62	1.67	1.43	1.46
5	1.58	1.61	1.41	1.43

Table 2 Hot spot stress concentration factors,  $K_s$ . Results calculated with models fulfilling the element side length requirement are shaded.



Fig. 5 Extrapolation of the hot spot stresses for the model 5.

The transverse averaging of the stresses was made for the stiffener thickness when calculating the hot spot stress concentration factors in Table 2. In the real structure the weld seam makes the area larger through which the stresses are transmitted between the stiffener and the plate. To study this effect the transverse averaging was done for double the stiffener width (25 mm). The resulting hot spot stress concentration factor  $K_s=1.25$  for the model 3 which was 11% lower than the value given in Table 2. The real effect of the weld seam can be estimated to be less than this result for two reasons. Firstly the actual weld dimensions were not reported in ref. [6], but it is reasonable to expect that they were less than double the stiffener thickness. Further the hot spot was located at the plate side weld toe at the stiffener corner. At this location the weld seam was obviously round which doesn't support clear definition of the averaging width. The use of the stiffener thickness as the stress averaging width is advantageous as it is clearly defined and the resulting error is on the safe side.

The FE model five was created from the FE model one by refining the mesh only at the six elements with one corner node at the hot spot location. Two steps of mesh refinement was made and the resulting element size at the corner corresponded to that of the model 3. The results for the models 3 and 5 were practically identical which demonstrates that the very fine mesh is required only at a small area near the singularity.

In the present example the hot spot stress result converged above the correct value and coarse meshes gave always conservative results. All the FE models underestimated the axial stress value at the singularity but at the same time overestimated the distance to which the singularity at the corner would raise the stresses. As a result of this the axial stress at the point  $x = 0.4 \times t$  was overestimated which resulted in overestimation of the hot spot stress. The situation would be different with even coarser mesh as a sufficiently large element would not notice the singularity and would give a too small hot spot stress.

# APPLICATION TO A SHIP HULL DETAIL

The same techniques was applied to post analysis of fatigue tests of large aluminium test specimens modelling a ship hull detail. The specimens were loaded in a test rig by a variable amplitude axial load [1,7]. The load histories for the tests were defined by a numerical slamming simulation. The test series resulted in a nominal stress fatigue failure curve for the actual design.

In the present work the test structures were analysed by the FE method using thick shell elements and the ADINA<sup>®</sup> program [8]. The hot spot stresses were calculated by the HOREX program for various hot spot locations in the structure, see Fig. 6 for example. Most of the failures initiated from a water

hole corner and this area was analysed by several gradually refined FE models. The results showed that the stresses at the water hole corner behave very similarly to the stresses at the longitudinal stiffener corner. The calculated hot spot stress concentration factor for the water hole corner was  $K_s = 1.4$ . In ref. [7] a mean experimental stress concentration factor of 1.88 was given. The difference can be explained by the initial imperfections of the test structures and the differences in the actual weld quality and the weld quality of the test pieces used in defining the material Wöhler curve.

The hot spot stress results were used for fatigue analysis of the structure by the P/FATIGUE<sup>®</sup> program. The material data was taken from fatigue tests for a butt welded small specimen [9]. One of the analysed load histories is shown in Fig. 7 together with its damage profile for the water hole corner calculated by the P/FATIGUE<sup>®</sup> program.



Fig. 6 FE model for the ship hull test structure a) basic FE model and refined meshes at hot spot locations 1-4 b) at a water hole corner (1) and c) at transverse frame cut-out for longitudinal stiffener (2 - 4).



Fig. 7 Fatigue analysis of the ship hull test structure: a) load history and b) its damage profile for the water hole corner.

#### CONCLUSIONS AND DISCUSSION

The present results show that the thick shell element is capable of calculating the hot spot stresses. The FE mesh at the hot spot location must be fine enough to allow hot spot stress extrapolation without using the stress results from the element nearest to the hot spot location. The present results gave an element side length requirement of h/t < 0.4 at the hot spot location for the first element. This very fine mesh was required only for a very small area around the hot spot location. The FE model could easily be created using the local FE model refinement by the Post3D program. The h-version local model refinement keeps the model size acceptable. Several hot spot locations can easily be analysed by the same model. The P/FATIGUE<sup>®</sup> program was applied for the fatigue analysis of aluminium structures modelling a ship hull detail. The experimental and numerical results for the fatigue life became closer by the application of the hot spot approach. The remaining differences were explained by the initial imperfections and weld quality.

The hot spot is a local stress raiser in the structure. It seems reasonable to expect that the required mesh density at this area is determined by the joint geometry. The FE model in the surrounding areas should simply be capable of calculating the nominal stresses. The required degree of accuracy at the hot spot is not self-evident as only the structural stress concentration is sought. In the present analyses the weld was not modelled and this may not be adequate for all joint types e.g. a doubler plate or a load carrying partially penetrated fillet weld. The research in this area will be continued by analysing other joint types aiming at the definition of robust and effective modelling techniques.

# REFERENCES

- Hakala, M.K. et al. Structural design of an aluminium missile boat. In: N.N. (ed.), Proceedings of the First International Conference on Fast Sea Transportation, Vol. 2. Trondheim, Norway 1991. Pp. 727-741.
- 2 Niemi, E. Recommendations concerning stress determination for fatigue analysis of welded components. IIW Doc. XIII-1458-92, March 1993 version, 69 p. (Under preparation).
- 3 Härkönen, T. & Tervola, T. Lujat hitsattavat teräkset (Weldable high strength steels). Tampere 1993. Metalliteollisuuden Keskusliitto, Tekninen Tiedotus 1/93. 124 p.
- 4 Mikkola, T.P.J. & Niemi, H. Quality assurance for fracture mechanical finite-element analyses. Computers & Structures 40(1991)2, pp. 271-279.
- 5 Mikkola, T.P.J. & Raiko, H. Development of an automated fracture assessment system for nuclear structures. International Journal of Pressure Vessel & Piping, 52(1992)3, pp. 357-377.
- 6 Fricke, W. & Petershagen, H. Detail design of welded ship structures based on hot spot stresses. In: J. B. Caldwell & G. Ward (eds.), Practical Aspects in Designing Ship and Offshore Structures. 1989. Vol. 2. Elsevier Applied Science. Pp. 1087-1100.

- 7 Kuitunen, R., Solin, J. & Juntunen, J. Ohjusveneen hitsausliitosten väsymiskestävyyden määritys (Fatigue strength of welded joints of a missile boat). Espoo, 1990. Technical Research Centre of Finland, Metals Laboratory, VTT-MET C-144, 36 p. + app. 54 p.
- 8 Mikkola, T.P.J. Fatigue analysis of aluminium test structures. Espoo 1992. Technical Research Centre of Finland, Ship Laboratory, Technical Report, LAI92105. 22 p. + app. 51 p.
- 9 Sormunen, M. Murtovenymän vaikutus hitsattujen alumiinien väsymislujuuteen (The effect of the ultimate elongation on the fatigue strength of aluminium). Espoo 1989. Diplomityö, Helsingin Teknillinen korkeakoulu. 107 p. + liitt. 18 p.