Fatigue Assessment of Aluminum Ship Details by Hot-Spot Stress Approach

Bård W. Tveiten (V), Xiaozhi Wang (M), Stig Berge (V)

Presented at SNAME 2007, held in Fort Lauderdale, Florida, 13-17 November 2007, and Reprinted with the kind permission of the Society of Naval Architects and Marine Engineers (SNAME)

ABSTRACT

Fatigue tests of full scale models of an aluminum ship structural detail were carried out to obtain a design S-N curve. The test detail was also analyzed by the finite element method, using several modeling techniques and element types. The results from both experimental tests and finite element analysis are discussed. Recommendations on the procedure of fatigue assessment of aluminum ships including S-N curve to be used are also presented.

KEY WORDS:
Fatigue strength, fatigue testing, S-N data, hot-spot stress, finite element analysis, welded joints, aluminum structures, fatigue assessment, ship structures.

NOMENCLATURE

\( a \)    Weld throat section
\( E \)    Young's modulus
\( K_s \)  Hot-spot (structural) stress concentration factor
\( \log(a) \) Intercept of S-N curve (log-log format)
\( m \)    Slope of S-N curve (log-log format)
\( N \)    Cycles to failure
\( R \)-ratio \( R = S_{min}/S_{max} \) (min. stress/max. stress)
\( S_{33} \) Longitudinal stress
\( SCF \)  Stress concentration factor
\( SP_{Max} \) Maximum principal stress perpendicular to the weld toe
\( t \)    Flange plate thickness
\( t_w \)  Shell thickness used in shell weld idealization
\( \Delta \sigma_{nominal} \) Nominal stress range
\( \Delta \sigma_s \) Hot-spot stress range
\( \epsilon_{nominal} \) Nominal longitudinal strain (elastic)
\( \epsilon_{22} \) Longitudinal strain, shell element model
\( \epsilon_{33} \) Longitudinal strain, finite solid element model
\( \nu \)  Poisson ratio
\( \sigma_{nominal, elastic-beam} \) Nominal stress obtained from elastic beam theory
\( \sigma_s (K_s \cdot \sigma_{nominal}) \) Hot-spot (structural) stress
\( \sigma_{0.4} \) Stress value at extrapolation points
\( \phi \)  Weld flank angle
\( \lambda \)  Weld leg length
\( \rho \)  Weld toe radius

INTRODUCTION

Unlike large commercial ships, high speed vessels are often constructed of aluminum. Compared with steel structures, aluminum alloys structures provide more flexibility in the design of complex weld details such as aluminum alloy extrusions which offer designers a large freedom from restrictions often given by the “standard” shapes. However, this freedom brings more difficulties in standardizing the fatigue design and analysis methodology of welded aluminum structures.

Some work has been done to assess the fatigue strength of aircraft and civil engineering structures. However, limited work has been performed with regard to welded joints typically used in aluminum ships. The main objective of the study presented in this paper is to investigate the existing design and analysis methodologies and thereby develop a reliable fatigue design and analysis procedure for aluminum ship structures through numerical evaluation and experimental verification.

The study consists of three main parts; a review of existing methodologies for fatigue assessment of welded aluminum ship structures based on the hot-spot stress approach, numerical analyses using the finite element method (FEM) to determine and validate the hot-spot stress assessment procedures, and full scale fatigue tests of a typical welded aluminum ship detail for assessment of design S-N curves to be used together with the hot-spot stress results to be derived from the numerical stress analyses.
FATIGUE OF WELDED ALUMINIUM STRUCTURES

The Hot-Spot Stress Concept

A key issue addressed in this paper is the definition and the calculation of the hot-spot stress, $\sigma_s$, used in fatigue design of welded structures.

Fatigue design codes for welded aluminum structures are in general based on a nominal stress range S-N curve approach where the design stress is the local nominal stress range, $\Delta \sigma_{\text{nominal}}$, neglecting the effect of any stress raisers due to local weld geometry. However, the nominal stress approach is not practical for welded structures characterized by relatively complex geometry, or combinations of loads. As an alternative, a design principle based on a hot-spot stress range S-N curve has been proposed. This reduces the number of design S-N curves since the geometrical features of the structural details are included in the design stress range rather than in the design S-N curves. However, the local notch effects are still embedded in the design S-N curves. Note that the hot-spot stress range approach applies only to fatigue failures initiating at the weld toe location.

The hot-spot stress, $\sigma_s$ in Fig. 1, is defined as the surface stress at the weld toe location that includes the stress raising effects due to structural (geometrical) discontinuities, but excludes the stress concentration due to the presence of the weld notch. The non-linear stress peak due to the presence of the weld notch may be excluded by means of an extrapolation procedure of stress from outside the region that is influenced by the local notch.

Generally, all hot-spot stress extrapolation procedures proposed in the open literature are based on surface stress extrapolation, with the assumption that the hot-spot stress varies linearly through the plate thickness and the effect of the weld notch is localized within a distance close to the weld, where the distance is normally expressed as a function of the main plate thickness. It has been shown that the extrapolation methods may depend on joint type, see Tveiten (1999), Tveiten and Moan (2000). That is, the method may be calibrated to a given joint geometry but may fail to get an accurate assessment of different joint configurations and dimensions.

![Fig. 1. Definitions of stress used in fatigue analysis.](image)

The through-thickness stress field will also govern crack growth rate. Different details with the same hot-spot stress value may show different stress fields through the plate, and the crack growth life is affected. Thus, the concept of a uniquely determined design S-N curve based on a hot-spot stress is an approximation. For tubular joints a two-parameter hot spot stress has been proposed, including a degree-of-bending component, Berge, Haswell and Engesvik (1994).

Several methods have been proposed for assessment of the hot-spot stress on the basis of the stress distribution through the thickness. It has been shown that these methods could be promising in order to cope with problems related to mesh sensitivity and geometry dependency that have been experienced with surface extrapolation methods: Lotsberg (2003), Doerk, Fricke and Weissenborn (2003), Dong (2004), Poutiainen, Tanskanen and Marquis (2004). Dong (2004) has proposed a method that has been claimed to be mesh-size insensitive and can be used in combination with both solid and shell element models. However, it has been demonstrated that the method in some cases is mesh-sensitive or even sensitive to how element stress values are determined during the post-processing: Doerk, Fricke and Weissenborn (2003), Wang, Sun and Cheng (2004), Poutiainen, Tanskanen and Marquis (2004).

Modeling Principles for FE Analysis

Several possibilities are available for modeling of the structural geometry, ranging from solid elements, thick or thin shell elements, or a combination of these. Shell element models as well as coarse solid element models are characterized by a linear stress distribution over the plate thickness. Therefore, both types of finite element modeling are suitable for the calculation of the hot-spot stress since the nonlinear stress distribution due to the presence of the weld notch is excluded. The main problem that arises when shell elements are used is that the shell element formulation only provides a model for the mid-plane of the plates (the element thickness is given as an element property) and thus the local change of the stiffness associated with the weld shape can not be modeled, see Sumi (1997), Tveiten and Moan (2000).
Existing surface extrapolation methods require mesh refinement, to the point where further refinement does not result in significant change of the stress distribution between the extrapolation points. Alternatively, element mesh requirements can be specified with respect to size and order (linear or quadratic shape functions). Hence, the stress extrapolation procedures are linked to specific requirements for different finite element types and finite element meshes. For practical design there is a conflict of interests between the required accuracy of the resolved hot-spot stress and model complexity and computing time.

Design S-N Curves for Welded Aluminum Structures - Hot-Spot Stress

Limited S-N data for validation of the hot-spot stress approach for welded aluminum ship structures is available. Partanen and Niemi (1999) compiled fatigue test results for a variety of aluminum test specimens with moderate thickness (up to 6mm) subjected to pulsating tension (stress ratio $R > 0$). The hot-spot stress was based on the extrapolation procedure proposed by IIW (2005). The authors concluded that the hot-spot stress range approach can safely be used with an S-N curve of fatigue class FAT40 (FAT40 refers to the characteristic stress range at $2 \times 10^6$ cycles) for butt and fillet welded aluminum joints of relatively thin plates and extrusions (up to 6mm) with crack initiation from the weld toe (Fig. 2).

Tveiten and Moan (2000) published S-N data on flat bars with fillet welded in-plane brackets. The hot-spot stress was calculated using the extrapolation methods proposed by Tveiten and Moan (2000) and Niemi (1993). The S-N data obtained for the flat bar/bracket connections seemed to correspond well with the findings of Partanen and Niemi (1999) with respect to an appropriate hot-spot stress design S-N curve of fatigue class FAT 40 (Fig. 3). Aluminum box-stiffener/lap joints with wall thickness 3mm were studied by Ye, Moan and Tveiten (2001). The results showed that using a hot-spot stress range approach, design S-N curve of fatigue class 44 proposed by CEN (1999) seems to give conservative fatigue assessments, Fig. 3. Additional experimental studies by Macdonald, Haagensen and Søvik (1998) and Tveiten (2003) have proposed a fatigue design methodology for welded aluminum space frames made of rectangular hollow section joints using a linear (or quadratic) extrapolation of the hot-spot stress according to recommendations given by the IIW (2005) and a design S-N curve with a fatigue class of FAT 40 as shown in Fig. 3.

IIW (2005) has recently included a design S-N curve of fatigue class 40 for use with extrapolated hot-spot stress values for a large range of welded aluminum details. However, for some specific details including cruciform joints with load-carrying fillet welds the use of a design S-N curve of fatigue class 36 is proposed, IIW(2005).

STRESS ANALYSIS

Test Model

A schematic outline of the specimen geometry is given in Fig. 4. The transverse web plate and end plates were made of aluminum alloy 5083- H22/32, while the extruded longitudinal stiffener was made of aluminum alloy 6082-T6. The test specimens were designed and manufactured at a Norwegian shipyard according to normal workmanship standards. All the fillet welds were produced manually and tested in the as-welded condition.
Test Rig and Boundary Conditions

The test rig used for the fatigue tests was designed in order to simulate the effect of lateral load transfer from a longitudinal stiffener into a transverse web. At the end of the models a 12mm aluminum plate was welded and fitted with a fixed cylindrical shaft that acted as a bearing against the test ramp. On one side a ball bearing was fitted, to eliminate any frictional forces from the contact with the test ramp. The loading was provided by a servo-hydraulic actuator with a load capacity of ±50 kN, operated in load control.

Finite Element Models

In this study, the numerical stress analyses were carried out using a commercial finite element program (ABAQUS 2004) and using a Dell LATITUDE X300 Lap-top with an Intel Pentium M processor of 1.40GHz, and 1.11GB of RAM. All analyses were linear elastic.

The finite elements used in the solid models were 20-node quadratic elements (C3D20/R) with full and reduced integration and 8-node linear brick elements with full and reduced integration (C3D8/R) for all main elements. Note that the weld was modeled without a radius fillet at the weld toe in the solid element models. Hence, a numerical singularity will exist at the weld toe location.

The finite elements used in the shell element models were 4-node double curved general-purpose shell elements with full and reduced integration (S4/R) with five integration points through the shell thickness, and 8-node doubly curved thick shell with reduced integration (S8R).

Pinned-end boundary conditions were assumed, corresponding to the test conditions (Fig. 5). The finite element models had two symmetry planes (the [1-2]-plane and the [2-3]-plane), thus, only ¼ of the finite element model was modeled (Fig. 6).

The finite element analyses were linear-elastic assuming:

- Young modulus of $E = 0.70 \times 10^5$ MPa
- Poisson ratio of $\nu = 0.3$

All dimensions are given in millimeter (mm), and loads in Newton (N).

In the case of the shell element models, some of the analysis models used an ‘idealized weld’ in order to include the stiffness of the weld. It has been proposed by IIW (2005) and Fricke (2001) to include the weld stiffness when using shell elements, by e.g. inclined shell elements or constraint equations.
Fatigue Assessment of Aluminum Ship Details by Hot-Spot Stress Approach

Fatigue Assessment of Aluminum Ship Details by Hot-Spot Stress Approach 259

ABS TECHNICAL PAPERS 2007

a) Tied constraints between the surfaces located “inside” the weld, where the flange is the master and the transverse plate is the slave. Tie constraints tie together two surfaces where each node on the slave surface is constrained to have the same displacements as the point on the master surface.

b) Inclined shell elements with increased thickness, \( t_w = 10 \cdot t (100 \text{mm}) \).

c) Solid elements connected to the shell surfaces at the flange and the transverse plate, using a surface-based option in the FEM code that allows for a transition from shell element modeling to solid element modeling.

Finite Element Analysis Results - Comparison of FE Models

Fig. 7 shows the distribution of the normalized longitudinal stress (\( S_{33}/\sigma_{\text{nominal, elastic-beam}} \)) results along the centre line of the stiffener for a set of different solid and shell element analyses using full integration elements and reduced-integration solid and shell elements. In the subsequent figures presented in this paper the notation “red” means ‘reduced integration’, while the notation “full” means “full integration”.

The normalized stresses are based on the derived nodal stress, which is an averaged nodal stress that is extrapolated to nodes from integration points using element shape functions and averaged over all elements that contribute to the node. The nominal stress, \( \sigma_{\text{nominal, elastic-beam}} \), was calculated using elastic beam theory, neglecting effects of shear lag and the small fillet of the stiffener cross section.

For ship structures characterized by relatively complex geometry subjected to combinations of loads, defining a nominal stress becomes rather cumbersome or very often impossible. For the structural detail subjected to 3-point bending investigated in this study, the nominal stress was easily defined and could also be calculated using elastic beam theory. Hence, for the discussion regarding the hot-spot stress calculations, the hot-spot stress has been normalized for two main reasons; a) to verify the validity and applicability of the finite element models compared to elastic beam theory, and b) to obtain a stress concentration factor that indicates the stress increase caused by the structural geometry of the intersection of the transverse frame/longitudinal stiffener. For the subsequent discussion of the hot-spot stress range approach, stress concentrations relative to beam theory are given.

IIW (2005) presented rather comprehensive recommendations (see Section 2.2.3.4 in IIW 2005) with respect to recommended meshing of coarse and fine element models, element formulation and weld characterization (for shell element models) together with recommended extrapolation procedures. For relatively fine element models, IIW (2005) proposes that the distance of the centre point of the first integration point of the elements to the weld is no more than 0.4·\( t \).

ABS (2005) and IACS (2006) propose an element mesh density approximately equal to the representative net thickness in the assessed area for oil tankers (applies to hopper knuckle connections only) and bulk carriers.

Fig. 7 shows that with respect to the global stress distribution, there is a good correspondence between the solid element and the shell element analyses, and the calculations based on elastic beam theory. As expected, the 8-node solid element model is stiffer, thus giving around 3% lower nominal stresses compared to the 20-node solid element model. Note that for the solid element model, the stress is plotted from the weld toe, while for the shell element models (without the weld) the stress is plotted from the element intersection line.

Fig. 8 shows the normalized longitudinal stress along the centre line of the stiffener obtained from various solid element analyses using full-integration and reduced-integration elements.

As shown in Fig. 8, a continued refinement of the element mesh of the finite solid element model using 20-node quadratic brick elements with full integration close to the weld will confine the effect of the singularity at the notch (no weld toe radius was modeled) to an increasingly smaller region. Convergence in the region where it is assumed that the effect of the hot-spot stress can be derived from (0.4·\( t \) - 1.5·\( t \)) was obtained for element sizes of the order of approximately 0.2·\( t \) (1.92x3.75x3.00 (mm)); however, for the analyses using 20-node quadratic brick elements with reduced integration it is seen that convergence in the region between 0.4·\( t \) and 1.5·\( t \) was obtained for element sizes in the order of approximately 0.4·\( t \) (4.175x3.75x5.00 (mm)). The 8-node brick elements with reduced integration provide results similar to the 20-node reduced-elements for same mesh densities.

Figure 7. Normalized longitudinal stress distribution along centre line of stiffener for various finite element models.

Distance from weld toe intersection line, [mm]
The results presented in Fig. 9 show that by reducing the integration and the order of the element (from quadratic to linear), the effect of the mesh size with respect to the hot-spot stress distribution is somewhat eliminated and the finite element models with much less refined mesh give approximately equal stress results within the extrapolation zone (0.4t - 1.5t). It is important to bear in mind that the definition of hot-spot stress is the linear through-thickness stress, and by using lower order elements or reduced-integration elements, the stresses become linearized, which effectively reduces the effect of the non-linear stress peak at the weld toe.

Even though the stress distribution obtained from the 20-node and 8-node solid element models are rather coincident, the difference in CPU time between the 20-node finite solid element analysis using full-integration and the 8-node reduced-integration analyses are quite large, from 1596 seconds to 87 seconds (a factor of ≈18). Hence, tremendous computing time can be saved by using lower order, reduced finite element models. However, it is important to note that at the weld toe, the higher order elements describe the stress concentration better as the stress is derived using higher order polynomials. The linear 8-node reduced-integration elements provide good results for many applications if the mesh density is sufficiently refined, which is particularly important when the structure is carrying bending loads. For instance a single linear element with reduced integration through the thickness of a beam, plate or pipe section is unable to resist bending loads as all the integration points are laying on the neutral axis.

Although the solid and shell element models show excellent agreement for the global nominal stress distribution, the stress distribution at the hot-spot location is somewhat different resulting in higher nodal stresses for the shell element models than for the solid element models, Fig. 9. This is most likely due to the absence of the weld in the shell models. It has been proposed by Fricke (2001) that vertical or inclined elements with appropriate stiffness, or constrained equations to couple node displacements to model, the weld stiffness need to be considered for cases where the
results are affected by a high degree of local bending. However, the various methods for including the weld stiffness should be used with caution. Tveiten (1999) has shown by means of shell and solid element analyses combined with strain gage measurements of longitudinal stiffener/bracket connections that extrapolated hot-spot stress values were rather sensitive to how the weld stiffness had been accounted for in the shell element model. Fig. 10 shows that for this particular detail, various models for including the weld in the shell models actually lead to less accurate stress distributions close to the weld toe when compared to the results obtained with the solid element models. The tied constraint model gave the largest deviation.

The use of 4-node full integration shell elements with rather dense mesh in the order of 1.0·t is not able to properly describe the stress increase at the notch (element intersection line). However, by using 8-node reduced integration shell elements the stress results are comparable to the much denser 4-node shell element models. As shown in Figure 9, reducing the mesh size of the shell 8-node reduced-integration elements, does not change the stress distribution significantly.

Figs. 11 and 12 present the finite element results from the “converged” models, showing no significant change in the stress distribution with increased mesh density.

Fig. 11. Normalized longitudinal stress distribution close to weld toe, weld throat thickness: a = 3.50mm

Fig. 12. Normalized longitudinal stress distribution along centre line of stiffener close to weld toe location for the “converged” shell element models

Based on the recommendations given by: Niemi (1993), Tveiten and Moan (2000), Wang, Sun and Cheng (2004), ABS (2005), IIW (2005) and IACS (2006) with respect to mesh refinement and alternatives with respect to modeling, the following finite element models have proved to be the most applicable in order to derive the hot-spot stress:

1. Solid, 20-node, red, element at hot-spot: 4.175x3.75x5.0 (mm), longitudinal element length ≈ 0.4·t
2. Solid, 8-node, red, element at hot-spot: 3.13x3.5x2.5 (mm), longitudinal element length ≈ 0.3·t
3. Shell, 8-node, red, element at hot-spot: 10.0x11.875 (mm), longitudinal element length = 1.0·t
4. Shell, 4-node, red, element at hot-spot: 3.63x4.75 (mm), longitudinal element length = 0.3·t
5. Shell, 8-node, red, element at hot-spot: 5.0x4.75 (mm) longitudinal element length = 0.5·t

Model 3 corresponds to the guidelines given by ABS (2005) for oil tankers and IACS (2006) (element size approximately equal to the representative net thickness in the assessed area), while the models 1, 2, 4 and 5 are selected on the basis of recommendations given by Niemi (1993), Tveiten and Moan (2000), and IIW (2005) where it is observed that further mesh refinement does not significantly change the stress distribution close to the weld toe. Figs. 11 and 12. Note that model 5 deviates from the requirements given by IIW (2005), which states that the first element should not be larger than 0.4·t. However, as seen in Fig. 12, further refinement of the mesh from 0.5·t does not significantly change the stress distribution.
Some aspects influencing the hot-spot stress calculations

In tubular joints, for which the hot-spot stress extrapolation methods were originally developed, the crown or saddle points are normally the locations of the hot-spot under most loading conditions. As opposed to tubular joints the extrapolation path is not obvious for welded plate connections. In the case of the specimen geometry used in this study the maximum hot-spot stress is not at the centre line but closer to the longitudinal stiffener’s edge, Fig. 13.

In the FE analyses that were discussed above the weld was idealized as a ‘wedged cylinder’ at the intersection of the longitudinal stiffener/web-frame connection. In a design process this is the most likely choice due to its simplicity with respect to modeling. For this model the variation in longitudinal nominal stress across the width of the flange was insignificant (less than 4% in the extrapolation region). However, as shown in Fig. 14 the actual welds were highly irregular across the stiffener and had a fillet also at the side of the flange. Hence, in order to calculate the effect of a more realistic weld shape, the solid element models were modified, adding a ‘wedge shaped cylinder cell’ at the side of the upper flange, Fig. 13. The introduction of a more realistic weld geometry lead to a significant increase in the stress close to the weld toe towards the edge of the stiffener. Fig. 15 shows stress profiles at the centre line and at the edge of the stiffener flange from the 20-node solid element model. The difference between the stress distribution along the centre line and the edge of the longitudinal stiffener range from 5% to 15% in the ‘extrapolation’ region. At the centre line the effect of this modification of the weld is insignificant.

For the shell element models (with no weld included), the variations were somewhat larger, with around 15% to 20% difference, Fig. 16. Obviously, this is not due to a modification of the weld shape as the weld is not included in the model. The stress increase seen in the shell element model is most likely related to the rather complex interaction of the web frame/stiffener and the cut-out in the transverse web plate.

Fig. 13. Solid element model, including a weld shape modeling the as-built geometry shown in Fig. 14.

Fig. 14. Sample of weld shape at hot-spot.

Fig. 15. Normalized longitudinal stress distribution along the centre line and at the edge of the upper flange close to weld toe location.
The maximum stress is not located exactly at the edge, but at some distance from the edge corresponding to approximately one element size. Hence, the extrapolated maximum values of the hot-spot stress presented in Table 2 are based on the stresses derived at one element length from the edge of the longitudinal stiffener.

Hot-Spot Stress Calculations Based on FE Analysis

The hot-spot stress extrapolation schemes applied in this study for deriving the hot-spot stress were:
1. Linear extrapolation, using points at 0.4·t and 1.0·t, (IIW 2005)
2. Linear extrapolation, using points at 0.5·t and 1.5·t, IACS (2006) for Bulk Carriers
3. Hot-spot stress derived using one single point at 0.5·t, IACS (2006) for tankers

Note that for post-processing purposes when basing the extrapolation schemes on recommendations given by 1-3 above, the linear extrapolated hot-spot stress can be derived directly from the averaged nodal surface stresses at the ‘reference points’, using extrapolation equations given below as:

a) Linear extrapolation, using points at 0.4·t and 1.0·t:
$$\sigma_{S} = 1.67 \cdot \sigma_{0.5\cdot t} - 0.67 \cdot \sigma_{1.0\cdot t}$$

b) Linear extrapolation, using points at 0.5·t and 1.5·t:
$$\sigma_{S} = 1.50 \cdot \sigma_{0.5\cdot t} - 0.50 \cdot \sigma_{1.5\cdot t}$$

The subscript refers to the specific location of the ‘reference points.’

Normal practice in hot-spot stress extrapolation procedures is to carry out the hot-spot stress extrapolation to the weld toe when using solid finite elements and to the element intersection line when using shell elements. However, ABS (2005) has proposed to carry out the extrapolation to the weld toe, even though the weld has not been included in the shell element model. In this study, a weld leg length of 4.95mm has been used (which corresponds to a weld throat thickness of 3.5mm). (The applicable surface stress used in the extrapolation scheme proposed by ABS (2005) is the stress derived at the centroid of the elements, Fig. 17.

Fig. 16. Normalized longitudinal stress distribution along the centre line and at the edge of the upper flange close to weld toe location - shell element models.

Independent of the finite element model, element type and extrapolation method, a difference of approximately a factor 1.3 (1.75/1.31) to 1.4 (1.63/1.18) between the highest and lowest value is observed, Tables 1 and 2. Note that even though a thorough evaluation of the various finite element models was carried out initially, large differences in hot-spot stress results were experienced. For instance, in an initial design process this will have resulted in a difference in predicted fatigue lives by a factor of approximately 2.2 to 2.7.

The hot-spot stress concentration factors derived from the extrapolation points at 0.4·t and 1.0·t gives the largest values for all models since the extrapolation is performed in a region close to the weld, where the stress gradient is steepest. Except for the 8-node shell element model with element length at hot-spot of 5.0mm where a difference of 1.13 (1.54/1.36) at the centre location is seen, the difference is rather small, i.e. between 0.99 (1.75/1.76) to 1.07 (1.37/1.28). This is due to a rather linear trend of the stress distribution as seen in Figs. 11, 12, 20 and 21.

It should be noted that the “single point” definition of the hot-spot stress (at 0.5·t) shows the smallest variability. Increasing the element size to 1.0·t (recommended by IACS 2006) and using quadratic 8-node shell elements do not significantly change the derived hot-spot stress compared to the denser 4-node element model. It is worth noting that the hot-spot stress concentration factor derived from the 8 node solid element model gives the overall lowest calculated hot-spot stress values.

The values obtained by the ABS (2005) extrapolation method using shell elements are close to the values found by solid elements with extrapolation to the weld toe.

Table 3 presents the ratio between the extrapolated hot-spot stress derived at the edge of the stiffener (one element length from the edge) and the extrapolated hot-spot stress derived at the centre line. It is seen that ‘edge-

Fatigue Assessment of Aluminum Ship Details by Hot-Spot Stress Approach

263
Based’ extrapolation of the hot-spot stress gives a range 1.11 - 1.18 for the solid element models, while for the shell element models the range is somewhat larger (1.05 - 1.30).

Fig. 17. Reference points at centroid of elements used for the assessment of hot-spot stress according to ABS (2005), element size in hot-spot region ~1.0 t.

Experimental Stress Analysis - Verification of FE Analysis

For measurement of strain close to the hot-spot, strip gauges were used. Along each line, four measurements of strain was obtained covering the region 0.5t - 1.5t. The strips were located to measure strain at the centre line and at the two edges of the flange, corresponding to the extrapolation paths used in the FE analysis. The instrumentation is shown in Figure 18.

In the solid element analyses, the weld was idealized with a weld flank angle, $\phi = 45$ degrees, a weld throat section, $a = 3.5$ mm (leg lengths, $\lambda = 4.95$ mm), and with a weld toe radius, $\rho = 0$ mm. As shown in Fig. 14, the welds were highly irregular across the stiffener with much larger weld leg lengths at the edges of the stiffener. In the subsequent plots of the measured and calculated strains, the distance from the weld toe are taken as the actual distance from the weld along a line perpendicular to the weld.

Fig. 19 shows a comparison between the strain gage measurements and the finite element results at the centre line, based on the same finite element models used for the derivation of the hot-spot stress, Tables 1 and 2. As seen from Fig. 19, there is an excellent agreement between the nominal stresses, defined as the stress at a distance more than approximately 80mm from the weld toe. The difference between the finite element models is around 3%. It is also seen that the nominal strains measured at the centre line at each side of the transverse web plate (named L and R) are in excellent agreement. Thus, the measured strains show that the rig arrangement provided a symmetrical 3-point bending loading condition.

In Fig. 20 data from the region close to the weld is shown, at a larger scale. Data for two models are shown, from both sides of the transverse web plate, in effect for four models of the weld. The measured data are seen to fall in a relatively narrow scatter-band. Considering the relatively large variability of the weld profile (Fig. 14) this may be taken as a confirmation that at a distance of 0.4t or more from the weld toe, the stress distribution is essentially unaffected by the local notch of the weld.

The shell models are overpredicting the stress in this region, most likely due to the extrapolation to the shell intersection line. For the solid element models the agreement with measured stress is very good. For these models the definition of the weld toe location is the same as in the experiments.

Fig. 18. Fatigue cracking was initiated along weld toe on top of stiffener, in most cases close to one of the edges of the stiffener. the fatigue crack would grow along the weld toe and through the flange of the stiffener (end of fatigue life).
Effect of fabrication tolerances and weld distortions

The test models were instrumented to measure strain at the centre-line and at the edges of the stiffener, Fig. 18. The results for test #5 are shown in Fig. 21. The difference between edge and centre line for the measured strains is much larger than the difference that was calculated from the model shown in Figure 13. In the tests a bending gradient across the width of the flange is apparent. It is also seen that the stress at the edges is nonlinear, i.e. the SCF varies with load level. The stress at the centre-line is unaffected by the load level, and in good agreement with the FE results. Bending component occurred due to weld distortions that had lead to some warping of the models, and subsequently an uneven contact between the models and the fixtures that were used in the testing.

Shown in Fig. 22 is an FE model that represents the warping as a torsional displacement of the test models. Non-linear analysis was carried out to simulate the load-dependent contact between the model and the fixture. Two different finite element analyses were carried out with contact between the test model and the ramp of respectively 25% and 100% of the total width of the model, Fig. 23. The first case corresponds to low load levels with little contact between the test ramp and the end plate, while the second case corresponds to high load levels with full contact between the test ramp and the end plate.

Fig. 19. Normalized longitudinal stress distribution from FE analysis and strain gauge measurements, at centre line of stiffener. Stress is normalized to nominal stress from beam theory.

Fig. 20. Normalized longitudinal stress distribution obtained from FE analysis (centre line) and strain gauge measurements (centre line and edges), at various load levels.

Fig. 24 shows the finite element results using a combination of the two models. The model using 25% contact was applied to loads up to 5kN, while the model using 100% contact was applied to loads larger than 5kN. The plot in Fig. 24 clearly shows the effect of the load dependency with respect to uneven contact between the test ramp and the end plate.
Fig. 22. Initial displaced mesh, angular displacement caused by a torsional moment (end plate fixed). Two levels of angular displacement, displacement of corner nodes: $U = 13.5\text{mm}$.

Fig. 23. Contact of test specimen and test ramp for warped specimen, at low loads.

Fig. 24. Calculated strain at edges of flange, using nonlinear FE model to represent warping deformations (cf. Figs. 22 and 23).

It should be noted that along the centre line of the stiffener the measured and calculated stresses are linear, and apparently unaffected by the warping (Fig. 21). At the edges there is an additional stress concentration due to fabrication factors, in this case welding distortions. Although the test models would have different boundary conditions in a ship structure, similar effects may be expected, and need to be taken into account.

Welded structures are in general subject to fabrication tolerances that may lead to significant secondary stresses. FE analysis with idealized geometry may in many cases give nonconservative results, due to the neglect of effects of as-built geometry.

Discussion of the FE Analyses and Assessment of Hot Spot Stress

The results presented in Tables 1 and 2 demonstrate the variability in stress results close to the hot-spot locations for different finite element models and extrapolation paths. The results are consistent with previous studies on structural details (plate joints) presented by e.g. Sumi (1997), Tveiten (1999), Tveiten and Moan (2000) and Tveiten (2003), in that the stress distribution close to the weld toe (extrapolated stresses) are highly dependent on whether a shell element model or a solid element model have been used in the stress analysis, and in particular how the weld has been modeled.

Analysis using a solid element model provides stress results that are most consistent with strain gage measurements. However, the use of finite element tools in structural design is a trade-off with respect to the desired accuracy and time (or actually cost). Hence, in practical design the use of simplified shell element analyses with rather dense element meshes is preferred for analysis of complicated geometries. A reliable hot-spot stress analysis procedure is dependent on a well-defined link between the calculation of the hot-spot stress and the stress implicitly used as the design stress in the S-N curve. Assuming that the hot-spot stress is consistently predicted and reproducible regardless of detail configuration, the same calculation procedure should be used to calculate the hot-spot stress at an actual detail that was used when the hot-spot stress design S-N curve was derived from test data.

Assessment of an extrapolated hot-spot stress using FE models is subject to significant uncertainties. The large variability in hot-spot stress demonstrated here, using the recommendations given in literature, demonstrates the possible pitfalls involved in the process of deriving hot-spot stress values used in design. New methods for hot-spot stress extrapolation which have been claimed to be mesh-insensitive have been proposed, Dong (2004), Poutiainen and Marquis (2004). These methods may substantially improve the methodology for hot-spot stress calculations. However, there are still open questions related to these new methods (see e.g.: Doerk, Fricke and Weissborn 2003, Wang, Sun and Cheng 2004, Poutiainen, Tanskanen and Marquis 2004), and they are indeed highly laborious and complicated when it comes to post-processing and result derivation.
S-N Curves

Introduction

Based on the work presented in the previous section, full scale fatigue tests were carried out. The objective of the tests was to verify appropriate design S-N curves to be used together with the hot-spot stress derived from the numerical stress analyses and strain gage measurements.

Fatigue test results

A total number of nine models were tested at a constant amplitude loading (load control) with an R-ratio equal to 0.1. Cycles to failure, \( N \), was defined as a complete loss of the load carrying capacity of the longitudinal stiffener. The testing was performed in laboratory air at ambient temperature with a loading frequency of 2.5 Hz. The frequency of 2.5 Hz was used as this was the maximum frequency possible in order to obtain a stable and well controlled sinusoidal loading with defined maximum and minimum. It has been reported in literature that no frequency effects on crack growth can be detected in ambient air, Davidson, Griffiths and Machin (2002).

The crack growth of test specimen #1 was closely monitored to determine the residual fatigue strength of the detail after a visible fatigue crack had been observed. A visible “engineering sized” surface crack of about 2mm length was first observed close to the edge of the longitudinal stiffener, at the point of maximum stress range. Thereafter, the observed crack grew through the thickness of the flange within three thousand. After that, within one thousand cycles, a complete loss of the load carrying capacity of the longitudinal stiffener occurred. Fatigue life from a visible crack to failure was thus negligible.

The fatigue test results based on a nominal stress range are presented in Table 4. The stress range, \( \Delta \sigma_{\text{nominal, elastic-beam}} \), is the nominal stress value at the weld toe location calculated by means of elastic beam theory.

The S-N data were analyzed on the basis of a log-linear S-N curve

\[
\log N = \log a - m \cdot \log S
\]

The test results are shown in Fig. 25, with a mean life curve and a parallel curve at minus 2 standard deviations (notional 97.7% lower confidence band). The slope parameter of the S-N curve from regression analysis is \( m = 3.42 \).

Comparison With Proposed S-N Curve

In Fig. 26 the fatigue test data are plotted on the basis of the measured hot spot stress, using linear extrapolation 0.5t-1.5t. The stress profile for the most highly stressed edge of the stiffener was used, including the effect of secondary stress due to the warping. Also plotted are regression lines, and IIW design curves. The regression analysis was carried out with a slope parameter \( m = 3.0 \) in agreement with IIW S-N curves.

The FAT 40 S-N curve has been recommended as a design curve for hot spot stress, Partanen and Niemi (1999), IIW (2005). Studies by Tveiten and Moan (1997) on an aluminum stiffener/girder connection have shown that the use of a FAT 30 curve together with a linear procedure to derive the structural stress concentration provides a safe fatigue design.

The data shown in Fig. 25 indicate that the FAT 40 curve may be unconservative for design of this type of detail, and that FAT 32 or 36 may be more appropriate.
extrapolated hot spot stress. In this case the FAT 32 curve appears to be most relevant for design.

![Figure 27](image)

**Fig. 27.** S-N data plotted on the basis of hot spot stress 0.5\(t\) from the weld toe, which was measured for each model. Regression lines and IIW design curves are shown.

**CONCLUSIONS**

Fatigue tests of full scale models of an aluminum ship detail were carried out to obtain an S-N curve. The test detail was also analyzed by the finite element method, using several modeling techniques and element types. The results from both experimental tests and finite element analysis were discussed. Recommendations on the procedure of fatigue assessment of aluminum ships including S-N curve to be used were presented.

The following recommendations based on this study are given:

- Preferably shell element models should be used. However, no weld idealization should be included and the extrapolation should be carried out to the element intersection line.
- Solid element models can also be used. In that case the weld should be modeled with a flank angle of 45 degrees, no weld toe radius, and with a weld throat thickness (and overall weld geometry) according to expected sizes seen in the structure. The extrapolation should be carried out to the weld toe.
- The element size in the hot-spot region should be sufficiently small to ensure convergence.
- If an extrapolated hot spot stress is used, the extrapolation path should be verified by an initial evaluation of possible locations of the hot spot stress, since the extrapolation path is not obvious for welded plate connections.
- The hot spot stress from FE analysis could be taken at a point 0.5\(t\) from the weld toe, or by linear extrapolation 0.5\(t\) - 1.5\(t\). The former is simpler, and is somewhat less sensitive to details of the FE model.
- Due to fabrication tolerances significant secondary stresses may be present in a welded structure. In a design based on idealized FE models these stresses need to be taken into account.
- On the basis of measured hot spot stress using linear extrapolation 0.5\(t\) - 1.5\(t\) the fatigue test data indicate a design curve in the range FAT 32 to FAT 36.
- On the basis of measured hot spot stress using the stress at 0.5\(t\) the fatigue test data indicate a design curve corresponding to FAT 32.
- The overall recommendation from this work is to use a hot spot stress defined at 0.5\(t\) from the weld toe and a FAT 32 design curve for this type of welded aluminum details.

**ACKNOWLEDGEMENT**

This work was carried out as commissioned work for American Bureau of Shipping (ABS). Financial support from ABS is gratefully acknowledged.

Mr. Sheng Wei, National University of Singapore and Mr. Asmund Saelevik, Norwegian University of Science and Technology (NTNU) are acknowledged for their significant contributions to the experimental program.

**REFERENCES**


ABS, “Rules for Building and Classing Steel Vessels”, Part 5 – Specific vessel Types, Chapter 1 – Vessels Intended to Carry Oil in Bulk (150m (492ft) or more in length), Appendix 1 – Guide for Fatigue Assessment of Tankers, Chapter 13, USA, 2005.


Fatigue Assessment of Aluminum Ship Details by Hot-Spot Stress Approach

Proceedings of OMAE Specialty Conference on Integrity of Floating Production, Storage & Offloading (FPSO) Systems, Houston, USA, August 30 - September 2, 2004

### Table 1. Calculated hot-spot stress concentration factors, $K_s$, based on normalized longitudinal stress, $\sigma_{\text{Max}}/\sigma_{\text{nominal}}$, for an elastic-beam at the centre line of the upper flange obtained from finite element models

<table>
<thead>
<tr>
<th>Extrapolation Points</th>
<th>Shell, 4-node, red, element at hot-spot: 3.33x4.75o2)</th>
<th>Shell, 8-node, red, element at hot-spot: 10x10o2)</th>
<th>Shell, 8-node, red, element at hot-spot: 5x4.75o2)</th>
<th>Solid, 20-node, red, element at hot-spot: 4.175x3.75x5.00o3)</th>
<th>Solid, 8-node, red, element at hot-spot: 3.13x3.75x5.00o3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4t</td>
<td>1.41</td>
<td>-</td>
<td>1.32</td>
<td>1.26</td>
<td>1.24</td>
</tr>
<tr>
<td>1.0t</td>
<td>1.07</td>
<td>-</td>
<td>1.00</td>
<td>1.06</td>
<td>1.04</td>
</tr>
<tr>
<td>$K_s$</td>
<td>1.63</td>
<td>-</td>
<td>1.54</td>
<td>1.40</td>
<td>1.37</td>
</tr>
<tr>
<td>0.5t</td>
<td>1.34</td>
<td>1.34</td>
<td>1.21</td>
<td>1.21</td>
<td>1.18</td>
</tr>
<tr>
<td>$K_s$</td>
<td>0.96</td>
<td>0.91</td>
<td>0.90</td>
<td>1.00</td>
<td>0.99</td>
</tr>
<tr>
<td>0.5t</td>
<td>1.34</td>
<td>1.34</td>
<td>1.21</td>
<td>1.21</td>
<td>1.18</td>
</tr>
<tr>
<td>$K_s$</td>
<td>1.34</td>
<td>1.34</td>
<td>1.21</td>
<td>1.21</td>
<td>1.18</td>
</tr>
<tr>
<td>$X_{1} = 0.5t$</td>
<td>-</td>
<td>1.19</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$X_{R} = 1.5t$</td>
<td>-</td>
<td>0.89</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$K_s, \text{ABS}$</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

1)Element sizes too large to derive nodal stresses accurately
2)Hot-spot stress has been derived carrying out the stress extrapolation to the element intersection line, except for the extrapolation scheme proposed by ABS (2004) where the extrapolation has been carried out to a ‘fictitious’ weld toe location
3)Hot-spot stress has been derived carrying out the stress extrapolation to the weld toe

### Table 2. Calculated hot-spot stress concentration factors, $K_s$, based on normalized longitudinal stress, $\sigma_{\text{Max}}/\sigma_{\text{nominal}}$, for an elastic-beam at the edge (at one element from the edge) of the upper flange obtained from finite element models

<table>
<thead>
<tr>
<th>Extrapolation Points</th>
<th>Shell, 4-node, red, element at hot-spot: 3.33x4.75o2)</th>
<th>Shell, 8-node, red, element at hot-spot: 10x10o2)</th>
<th>Shell, 8-node, red, element at hot-spot: 5x4.75o2)</th>
<th>Solid, 20-node, red, element at hot-spot: 4.175x3.75x5.00o3)</th>
<th>Solid, 8-node, red, element at hot-spot: 3.13x3.75x5.00o3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4t</td>
<td>1.56</td>
<td>-</td>
<td>1.56</td>
<td>1.45</td>
<td>1.37</td>
</tr>
<tr>
<td>1.0t</td>
<td>1.26</td>
<td>-</td>
<td>1.28</td>
<td>1.21</td>
<td>1.14</td>
</tr>
<tr>
<td>$K_s$</td>
<td>1.75</td>
<td>-</td>
<td>1.75</td>
<td>1.60</td>
<td>1.52</td>
</tr>
<tr>
<td>0.5t</td>
<td>1.50</td>
<td>1.45</td>
<td>1.54</td>
<td>1.40</td>
<td>1.31</td>
</tr>
<tr>
<td>1.5t</td>
<td>1.10</td>
<td>1.10</td>
<td>1.10</td>
<td>1.09</td>
<td>1.04</td>
</tr>
<tr>
<td>$K_s$</td>
<td>1.71</td>
<td>1.63</td>
<td>1.76</td>
<td>1.55</td>
<td>1.45</td>
</tr>
<tr>
<td>0.5t</td>
<td>1.50</td>
<td>1.45</td>
<td>1.54</td>
<td>1.40</td>
<td>1.31</td>
</tr>
<tr>
<td>$K_s$</td>
<td>1.50</td>
<td>1.45</td>
<td>1.54</td>
<td>1.40</td>
<td>1.31</td>
</tr>
<tr>
<td>$X_{1} = 0.5t$</td>
<td>-</td>
<td>1.32</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$X_{R} = 1.5t$</td>
<td>-</td>
<td>0.92</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$K_s, \text{ABS}$</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

1)Element sizes too large to derive nodal stresses accurately
2)Hot-spot stress has been derived carrying out the stress extrapolation to the element intersection line, except for the extrapolation scheme proposed by ABS (2004) where the extrapolation has been carried out to a ‘fictitious’ weld toe location
3)Hot-spot stress has been derived carrying out the stress extrapolation to the weld toe
Table 3. Ratio - extrapolated hot-spot SCF, $K_s$, derived at the edge of the stiffener and at the centre line of the stiffener

<table>
<thead>
<tr>
<th>Extrapolation scheme</th>
<th>Shell, 4-node, red, element at hot-spot: 3.33x4.75</th>
<th>Shell, 8-node, red, element at hot-spot: 10x10</th>
<th>Shell, 8-node, red, element at hot-spot: 5x4.75</th>
<th>Solid, 20-node, red, element at hot-spot: 4.175x3.75x5.00</th>
<th>Solid, 8-node, red, element at hot-spot: 3.13x3.75x5.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear extrapolation, 0.4·t and 1.0·t</td>
<td>1.05</td>
<td>1.14</td>
<td>1.15</td>
<td>1.12</td>
<td>1.12</td>
</tr>
<tr>
<td>Linear extrapolation, 0.5·t and 1.5·t</td>
<td>1.12</td>
<td>1.05</td>
<td>1.30</td>
<td>1.18</td>
<td>1.13</td>
</tr>
<tr>
<td>Hot-spot stress, one single point at 0.5·t</td>
<td>1.13</td>
<td>1.09</td>
<td>1.28</td>
<td>1.16</td>
<td>1.11</td>
</tr>
<tr>
<td>Linear extrapolation, ABS (2004)</td>
<td>-</td>
<td>1.14</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

1) Element sizes too large to derive stresses accurately
2) Hot-spot stress has been derived carrying out the stress extrapolation to the element intersection line, except for the extrapolation scheme proposed by ABS (2004) where the extrapolation has been carried out to a ‘fictitious’ weld toe location
3) Hot-spot stress has been derived carrying out the stress extrapolation to the weld toe

Table 4. Test data, nominal stress range calculated from beam theory.
$SCF, K_s$, from strain gauge measurements, extrapolated (0.5·t-1.5·t) and at 0.5·t from weld toe, at edge of stiffener flange.

<table>
<thead>
<tr>
<th>Test spec. no.</th>
<th>$\Delta\sigma_{\text{nominal, elastic-beam}}$ [MPa]</th>
<th>N</th>
<th>$K_s$ 0.5·t-1.5·t</th>
<th>$K_s$ 0.5·t</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>74.65</td>
<td>264,012</td>
<td>1.47</td>
<td>1.34</td>
</tr>
<tr>
<td>2</td>
<td>74.65</td>
<td>109,917</td>
<td>2.25</td>
<td>1.80</td>
</tr>
<tr>
<td>3</td>
<td>74.65</td>
<td>68,609</td>
<td>1.61</td>
<td>1.51</td>
</tr>
<tr>
<td>4</td>
<td>39.19</td>
<td>1,625,397</td>
<td>1.39</td>
<td>1.26</td>
</tr>
<tr>
<td>5</td>
<td>39.19</td>
<td>1,132,858</td>
<td>1.95</td>
<td>1.80</td>
</tr>
<tr>
<td>6</td>
<td>39.19</td>
<td>Overloaded</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>7</td>
<td>39.19</td>
<td>990,897</td>
<td>1.33</td>
<td>1.22</td>
</tr>
<tr>
<td>8</td>
<td>39.19</td>
<td>1,596,500</td>
<td>1.95</td>
<td>1.81</td>
</tr>
<tr>
<td>9</td>
<td>67.19</td>
<td>271,863</td>
<td>1.52</td>
<td>1.40</td>
</tr>
<tr>
<td>10</td>
<td>67.19</td>
<td>297,170</td>
<td>1.33</td>
<td>1.24</td>
</tr>
</tbody>
</table>