Dokument: SASAK-RAP-DE-DTU-KEH-0007-00

Quick Calculation of Hot-spot Stresses

SASAK
Projekt 1 - Designregler

Martin Rosander

Pelmatic Knud E. Hansen, marts 2000

ABSTRACT

A quick calculation of the “Hot-Spot” stresses in a proposed test set-up is performed using the ANSYS general purpose finite element program and some macros previously developed by ISH. Results indicate that subjecting the test-piece to a fluctuating load of approx. 2.5 kN will result in a noticeable fatigue crack after a number of cycles in the order of 10^6.

TABLE OF CONTENTS

INTRODUCTION

In order to test the fatigue life of welded joints in aluminium structures test specimens were developed by Pelmatic Knud E. Hansen and produced at BILCON. These are to be tested in a fatigue test machine by the Department of Naval Architecture (ISH) at the Danish Technical University (DTU). Prior to tests an approximate level of loading is to be established, in order to ensure a realistic life-span of the specimens. Too low loading will result in specimens lasting too long, increasing the cost of the tests significantly. Too high loading on the other hand will result in a too quick crack development, making it difficult to evaluate the results.

In order to evaluate the so-called “hot-spot stresses” several macros were developed at ISH earlier in the project. These were used in the current analysis to generate and evaluate the finite element model.

The specimens were already manufactured at the time of the analysis. The dimensions were taken from drawings, see appendix[1]. The test set-up was also taken from drawings, see appendix[1]. A test set-up using a “push” design was used, where the specimen is subject to a compressive load resulting in a dominantly tensile load at the hot-spot.

One test-geometry has been evaluated.

The hot-spot stress calculated in the finite element model using the ISH-macros was instead used to read directly in the appropriate S-N diagrams. An appropriate number of cycles was calculated based on that the test should take approximately one day (8 hrs) to perform at 100 Hz:

$$\text{No. of cycles} = 100 \cdot 60 \cdot 60 \cdot 8 = 2.88\text{E}6.$$

Thus a total number of cycles in the region of 1-3 million should be an appropriate design criterion.

The stress concentration factor K_W was not included in the analysis. The weld quality was thus not accounted for. Any grinding of a weld typically increases fatigue life with a factor 2 or more. Neither was any variation in the stress in the analysis, as it was assumed that the design load was achieved in every cycle.

It should be clear that the calculated "hot-spot stress" is only the result of a given numerical routine, and has very little resemblance with what is happening at the weld root in the "real world". The current work is completely based on the findings in ref.[1].

Given the limitations of the analysis, as an indicator of which initial load level should be used during the test it is considered sufficient. This was the overall purpose of the present work.

MATERIAL DATA

The material used in the test specimens were stated as 5023-W23.

To determine the life-span an appropriate S-N curve must be chosen. In EuroCode No.9 Part 2 several fatigue curves are presented. Lars Tofte Johansen , see ref. [1] has explained the code in Danish. Approximately appropriate curve is 44-4.5, at 10^6 cycles 52 MPa is indicated.

Previous preliminary calculations have used ref.[2], using curve no. III. A stress range of 55 MPa is here recommended for 10^6 cycles.

Effect on the experiment

Using the above information we conclude that the experiment should be set up in such a way that to give a stress range in the order of 50-60 MPa.

It is assumed that the stresses in the SN-curves given for the welded material are determined using FEM more or less the same way as done in this report, which is based on ref.[1]. Since ref.[1] uses DNV recommendations as basis when constructing the finite element mesh and post-processing evaluation macros, it is considered likely that the same norm is applied in at least ref.[2]. If the stress-range calculation during the tests were calculated in any other way, for example by using a different mesh, using a different type of stress extrapolation, or even just using the nominal stress as range measure, the above conclusion will not hold

FINITE ELEMENT MODEL

The finite element model of the test specimen was modelled as text files using ANSYS, a quick and flexible way of generating models. The model was completely parameterised, meaning that the dimensions could quickly be changed for future analyses, given that the overall geometry does not change significantly.

The calculations were performed in steps, were first a course model was used to provide correct boundary conditions for a finer meshed model. This technique is commonly called sub-modelling. The course model can be seen in figure 1 below.

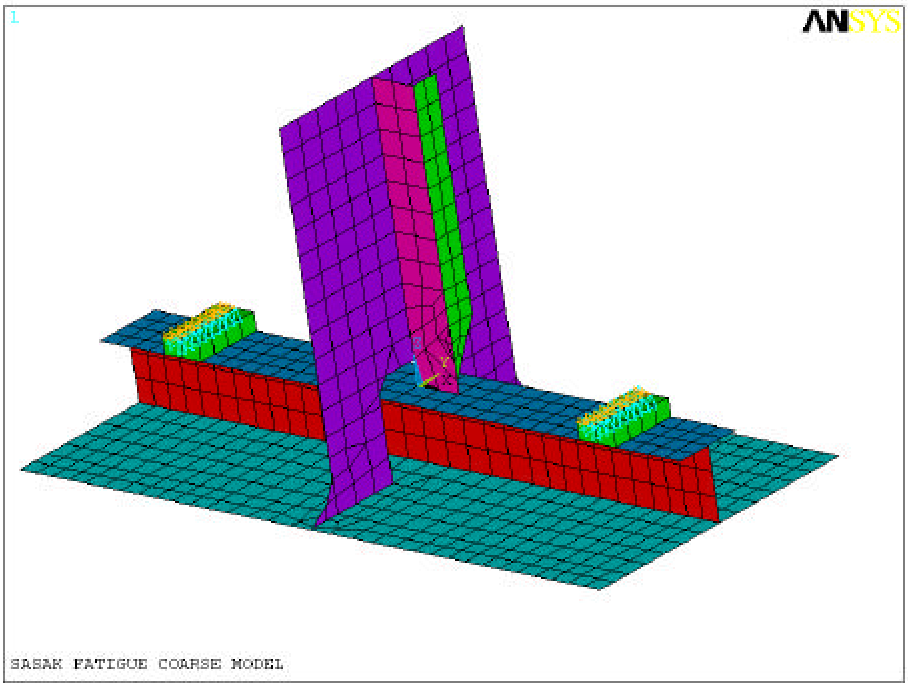

Figure 1. The course finite element model, with boundary conditions.

Co-ordinate Systems

A global co-ordinate system was placed at the hot-spot point. The x-axis pointed along the T-profile, the z-axis upwards along the web plate. The y-axis was thus parallel to the bottom plate; see figure 1 above. Several local co-ordinate systems were used to orient the elements in the same direction, these were not shown above.

Boundary Conditions

The boundary conditions were modelled using solid elements at the locations where the test-rig was to be bolted to the specimen. At the centre line of the solid steel block displacement restrictions were enforced. The specimen was also constrained from rotating around the global x-axis. In reality a contact condition should be more realistic, it was however considered to be beyond the scope of the current analysis to include this feature.

Loads

Loads were applied as pressure on the elements on the bottom plate, at the locations indicated by the drawings in appendix[1]. See also figure 2 below.

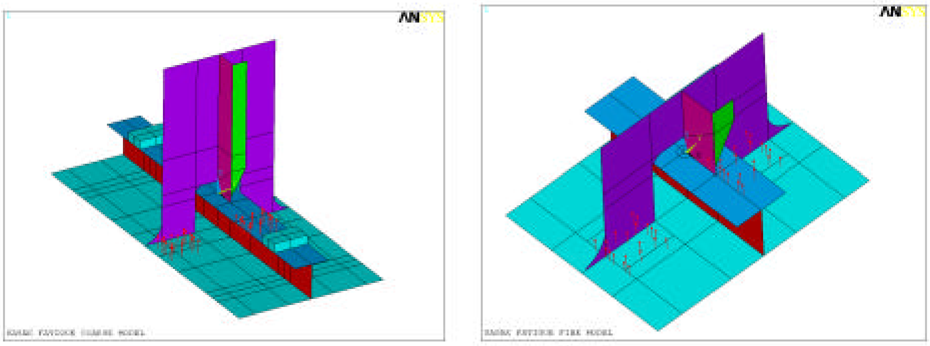

Figure 2. The load application on the coarse and fine models.

A pressure load totalling 1000

kg was used for all calculations.

Elements

The model used 8-node shell elements, SHELL93 in ANSYS. The solid elements were compatible 20-node SOLID95.

Sub-Model

The finer mesh required for accurate calculations was achieved using a technique called sub-modelling. This means that at an appropriate distance from the “hot-spot” of interest a “cut” is performed in the course model. The nodal displacements of the course model are then used as boundary conditions for the finer meshed sub-model. Using this technique solution-time can be saved on large finite element models. The sub-model can be seen below in figure(3).

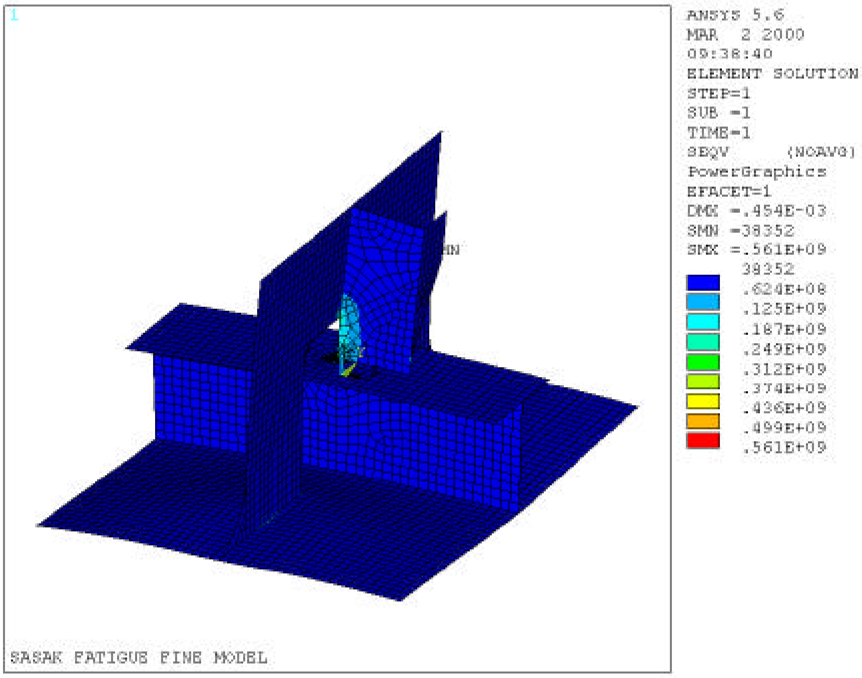

Figure 3. Equivalent von Mises stress plot of the submodel, 1000 kg load.

The mesh around the “hot-spot” was generated using the macro “GEN” provided by ISH.

Solution

The model was solved using a standard 450 MHz Intel PC running WindowsNT. ANSYS rev. 5.6 was used. Solution time was typically around 5 minutes. It is doubtful whether the use of sub-modelling resulted in any timesaving, since the specimen size was small. All calculations were linear, using the pre-conjugated gradient solver in ANSYS.

Element checks were performed to ensure that no nodes were doubly defined. Element thickness was checked, and the reaction forces were listed to ensure that the correct load was applied.

POST-PROCESSING AND RESULTS

The post processing was performed using the macros “BERA”, “BHUVUD” and “RPATH” provided by ISH. The macros “BERA” and “BHUVUD” readily provide the user with the stress extrapolated using the DNV recommended method, while “RPATH” plots the stresses in a graph.

In order to calculate K_G the nominal stress must be found. This stress **cannot** be taken from the finite element calculations, instead it **must** be calculated using the methods mentioned in DNV, see ref.[4]. If this is not done the calculated K_G is not valid, see ref.[1]. In the current report the nominal stress was not calculated, due to the fact that it was the load leading to the hot-spot stress which was demanded. The hot-spot stress was used directly as indicator of actual stress range. This is of course not completely correct, and it is not recommended to use this method in the future. The lack of time for the report prevented further analysis.

Results

As all results are linear simple scaling of the stress levels apply. It was seen that using the proposed test set-up a concentration of stress occurs at the point where the heel, where the weld ends against the top flange of the T-profile, see figures 4, 5 and 6 below:

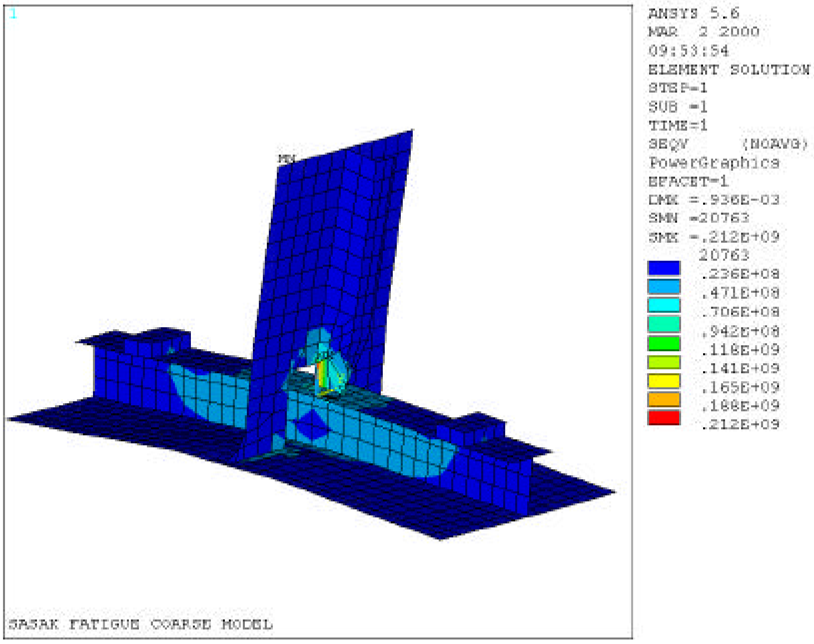

Figure 4. Coarse model results, showing the equivalent von Mises stress.

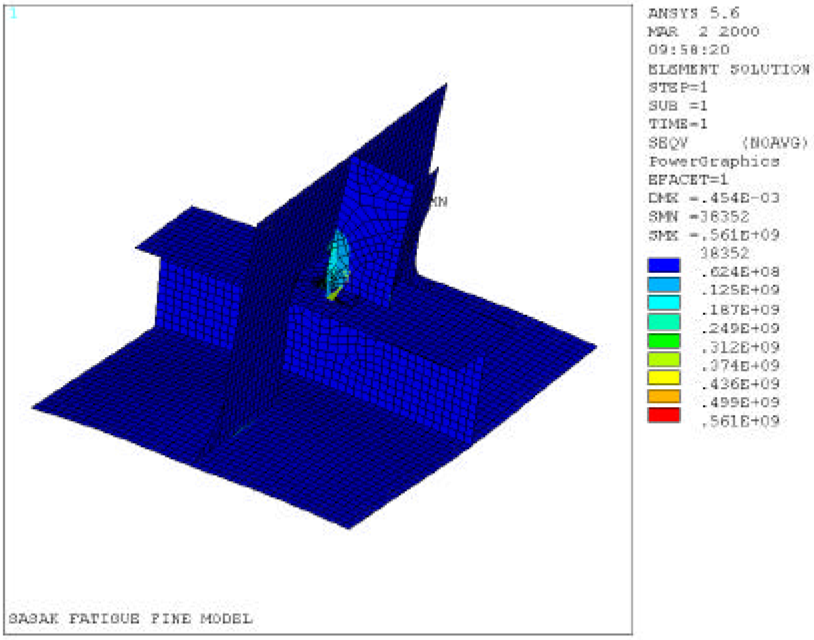

Figure 5. Fine model von Mises equivalent stress

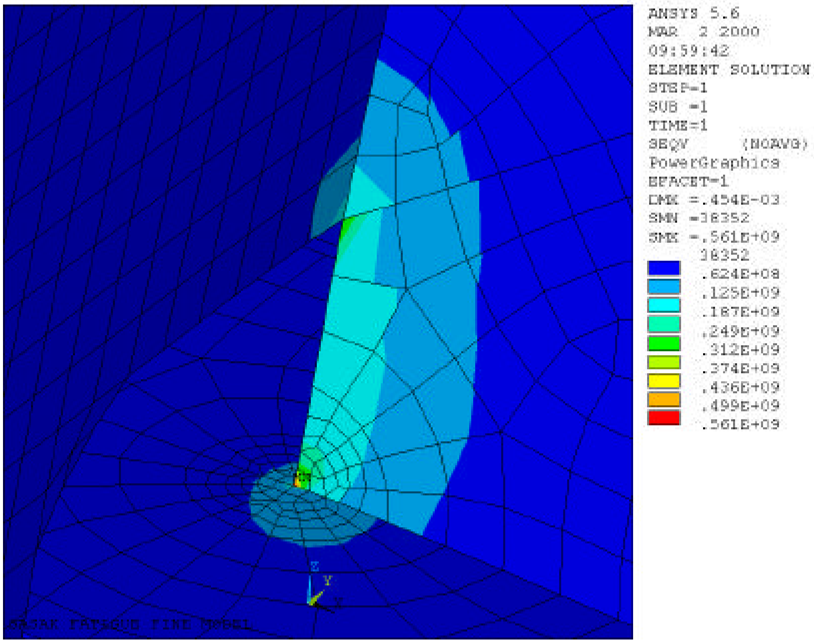

Figure 6. Zoom of hot-spot, von Mises equivalent stress.

The weld is here subject to both to the tensile stress from the L-profile as well as the bending stress at the top of the T-profile. It is thus a mixed mode stress field (both mode I and mode II), theoretical analysis on the fatigue properties of the specimen will be complicated to perform.

It is not completely clear in which direction the extrapolation procedure should be performed, and how to interpret the resulting hot-spot stresses. It was chosen to evaluate the stresses in the vertical z-axis direction along the L-profile as these were the highest. Stresses in other directions were neglected.

A plot of the effective stresses using the "RPATH" macro developed at ISH is shown in figure 7 below. The path is along a vertical axis running upwards from the hot-spot.

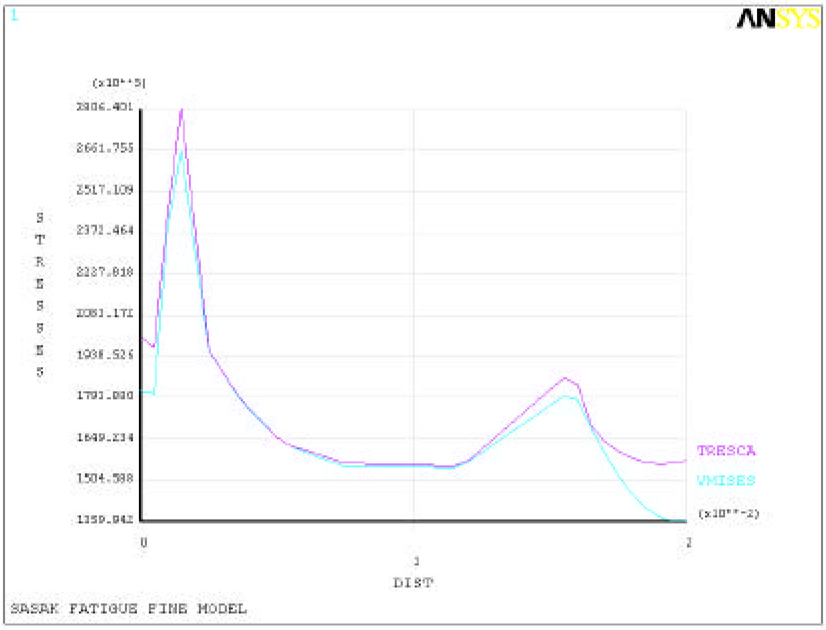

Figure 7. Effective stresses along a vertical path upwards from the hot-spot.

The extrapolated hot-spot stress in the global z-direction (in [MPa]) can be seen below as a function of applied load (in[kg]), see figure 8 below:

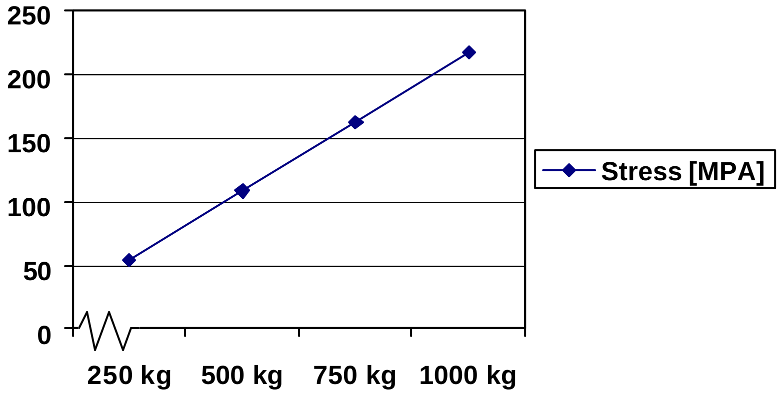

Figure 8. Extrapolated hot-spot stress as function of applied load in kg

As the results are completely linearly dependent only one load was actually calculated using ANSYS (at 1000 kg), the rest of the results in figure 8 were extrapolated.

Remarks

Due to the placement of the vertical web plate it might be difficult to visually detect the crack initiation when it occurs.

There is also a possibility that it is the upper weld connecting the web plate to the L-profile that fails first, depending on the weld quality compared to the lower weld attaching the L-profile to the T-profile. This has not been further investigated due to time constraints.

CONCLUSION

From the results in figure 8 and the section materials (see above) we conclude that using a load of 250-300 kg and 100 Hz ought to result in a visible fatigue crack during a day of testing.

Remarks

From what has been said earlier it should be clear that this is only an initial load setting recommendation. Experienced lab personnel will know that the first tests are always used to tune the test set-up. Any changes in specimen quality will have immediate and dramatic effects on a fatigue test. Increase or decrease of load might be necessary in such case.

DISCUSSION

It is important to bear in mind that all fatigue testing is stochastic in its nature. Even using smooth specimens of raw base material that has not been welded, large variations in fatigue life can be experienced. This stochastic behaviour is enhanced when specimens are post-treated, like when welded. Not only is the base material heated and cooled to melting temperature in a little controlled manner, but also the welder's skill is of utmost importance. Defects, weld size etc. all influence the fatigue life with several orders of magnitude of stress cycles until crack initiation. Any post processing of the welds in the form of grinding and shot-peening will also have large influences on the fatigue life. In order to get a minimal statistical base at least 3 to 5 (preferably 5) identical specimens should be tested.

The primary intention with the macros developed at ISH was to consistently compare different proposed joint designs without having to perform lengthy real-life fatigue tests on every proposal. Finite element calculations in general and specifically the macros cannot be used to exactly predict the number of cycles until crack initiation of any given individual specimen, such accuracy is not feasible within the time-scope given to this report.

The best use of these first tests is as tuning and verification of the calculation procedure developed at ISH, providing the necessary data to calculate the K_W for example. As with all testing it is of utmost importance that each specimen is properly documented; measured, photographs before and after the test with zooms of the crack initiation region, welding parameters carefully documented, etc, etc. In a later stage further tests can be used to document increase of fatigue life on newly developed designs.

Another fundamental issue is the test-machine, is it load-driven or displacement-driven? The practical difference is that as a defect propagates through a specimen normally it's compliance (flexibility) increases. A load-driven machine will try to keep the same load, increasing the displacement during the test. This can lead to very quick crack propagation. A displacement-driven machine keeps the same displacement regardless of the load. The effect of such a set-up can be that after initial specimen cracking the crack stops propagating, as the load becomes too small to drive the crack forward. At the time of writing it was not completely clear to the author which type of machine would be used.

REFERENCES

[1] SASAK-RAP-DE-AKS-DTU-0001-02, SASAK: Procedure for Calculating Hot-Spot Stresses in Aluminum Constructions, ISH/DTU, 1999-11-16

[2] Lars Tofte Johansen, Dimensionering av udmattelsebelastade svejste samlingar I aluminium efter EUROCODE No. 9 Part 2, Force Instituttet

[3] DNV, Fatigue Analysis of High Speed Craft – Preliminary Report – Confidential

[4] DNV, Fatigue Assessment of Ship Structures, Classification Notes No. 30.7, September 1998.

APPENDIX

1. Appendix; Drawings