# Analysis of High Cycle Fatigue Cracks in the Welds of Large Diesel Engines

T. Lucht

MAN Diesel & Turbo, Marine Low speed, Copenhagen

*Abstract: High cycle fatigue cracks may develop in large two stroke diesel engines used in ships and stationary power plants as the engines run at a constant cyclic loading for most of their lifetime. The dimension of the failed structure does in general exclude replacement and a repair solution is needed. The best repair solution can be found from a given quantity of solutions by relative comparison of calculated stress amplitudes and mean stress levels. The singular nature of the weld geometry does however complicate the absolute evaluation of the final welded design with the linear analysis giving unphysical high stress level at the welds. In this respect it is very interesting to investigate the recent introduction of the Extended Finite Element Method (XFEM) in Abaqus. This paper demonstrates and discusses how a real case of fatigue crack growth in a large diesel engine can be analyzed by the use of the XFEM in Abaqus.*

*Keywords: Crack Propagation, Fatigue, Fatigue Life, Fracture, Heat Transfer, Residual Stress, Welding, XFEM.*

## 1. Introduction

The development of large two-stroke engines is based on combined empirical knowledge and numerical calculations. The size of the several storey high engine shown in figure 1.a. can be illustrated from the smallest to the largest engine by the bore size of the cylinder spanning between 0.26 m and 0.98 m and the power output between 2000-75000 kW. As a consequence of the size, the prototype of each new engine series is sold to operate in ordinary service before start of production. After installation in a ship or a power plant a typical large two-stroke engine has a lifetime of 30 years, operating approximately 6000 hours per year at a constant speed around 100 RPM resulting in $10^9$ revolutions on full design load. In the general analysis of the main structure one can neglect other loads from thermal changes between start/stop of the engine, loads from the ship hull or other transient loads from vibrations of engine parts. Thus from the moment when the drawings are completed it is crucial that every part of the engine is designed with sufficient safety against fatigue loads from the combustion and inertia of the moving parts as fatigue failures in service require very expensive off-hire of the ship or power station for repair.

The assessment of safety against fatigue failures of welds is complicated because of unphysical high stress level due to the singular nature of the weld geometry. Interpolation of the unphysical high stress level will require additional investigations of specific parts of the weld which will be loaded at the most. In the recommendations by the International Institute of Welding (Hobbacher, 2007) three different methods are described in connection with numerical evaluation of welds and that is hot spot stress (only weld toes), effective notch stress and the fracture mechanics. All three methods focus on the weld toes and root which from a fatigue point of view are the weakest points due to unavoidable defects at these locations combined with typical stress concentrations. By the fracture mechanical approach typical defects in weld toe and root are assumed to be cracks for which it is determined whether the crack will propagate or not. This type of investigation on welded components of large two stroke engines has previously been performed as described in the paper (Hansen A. V., 2004). The applied meshing of the crack is very time consuming as a special mesh is required along the crack front. This type of analysis can however be more applicable due to time savings in the modelling stage by the recent introduction of XFEM. The application of XFEM to evaluate the welds resistance to fatigue failure will be evaluated in the present paper. The method is very interesting from an industrial point of view as this new feature enables fast crack evaluation in the known frame work of Abaqus in which the advanced numerical models of the engines already exist. Fracture mechanical calculations were previously performed by secondary software based on the boundary element method as described in (Lucht, 2009).

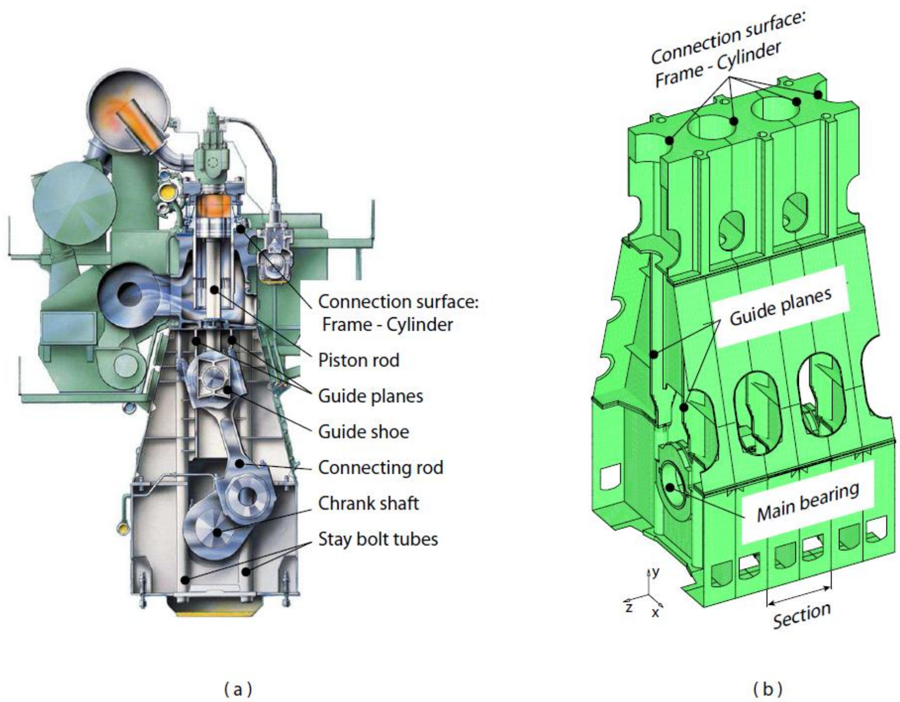


**Figure 1. Large two stroke engine a) Cross section of engine illustrating the basic principles and parts b) Abaqus model of the main structure.**

Besides stress concentrations in the evaluation of welds it is also essential to take the mean stress level into account. The heat input from the welding process introduces a significant local residual stress state in the area of the weld and thus the mean stress level is changed. As heated material is cooled tensile stresses will build up decreasing the safety against fatigue failure. Opposite material surrounding the heated material will come under compression and the safety against fatigue failure will increase in this material. Post Weld Heat Treatment removes residual stresses and it is in general considered to increase the safety against fatigue failure. Because heat treatment of large structures is costly it is not all welded parts of the engine which are heat treated and for these parts the effect of the residual stresses need to be evaluated as in (Hansen A. V., 2004). The residual stress level can be calculated by numerical simulation in Abaqus as demonstrated in (Hansen J. L., 2003) and verified by temperature and residual stress measurements. Besides the residual stress one also have to consider mean stress levels from assembly and running of the engine.

## 2. Evaluation of welded design

The recommendations by the International Institute of Welding (Hobbacher, 2007) describe three different approaches to evaluate welding as illustrated in figure 2. Common for the three methods is that they avoid direct evaluation on the non-linear stress peak due to the structural detail of the weld. In the well know hot spot method the structural stresses of the weld toes are calculated by extrapolating the stress results to the toe from points with recommended distances to the weld. The non-linear stress peak is removed from the calculation and the structural stress estimated at the weld toe can be compared with limit values for the given type of welding. Additional compensation is however required for thick plate structures as in large diesel engines due to the influence of bending stresses. In the effective notch stress method both the root and toe of the weld can be analyzed. The schematic sharp edges of the weld toe and root are rounded and thus replaced by a notch as described in the guidelines of the recommendation (Hobbacher, 2007). This modelling strategy is introduced in order to take into account the statistical nature and scatter of weld shape parameters, as well as of the non-linear material behaviour at the notch root. Limit values of the notch stress are given in the recommendations and it is independent of the structural weld detail. The method is based on empirical knowledge for which the recommended notch radius has been shown to give consistent results (Hobbacher, 2007)

The present paper uses the third and last method, fracture mechanics, to evaluate the welds based on the recommendations by the International Institute of Welding. In this method the fatigue properties of the welds are evaluated by considering the defects introduced by the welding process as cracks. The use of Linear Elastic Fracture mechanic (LEFM) to analyze high cycle fatigue is a method well known in the literature. The limit values described in the recommendations reflect general threshold values for steel below which the crack will not propagate and that is equivalent to tests performed for the welding material described later in the paper. In the case where an actual defect exists in the weld the size of the crack is obviously chosen equal to the defect. When evaluating welded joints without detected cracks it is chosen to insert a crack of equal size to what can be expected by a general welding process. For analysis of roots as illustrated in figure 2 the root gap represents an initial defect and it is thus modelled as a crack. At the weld toes small cracks can be inserted as described in the recommendation representing typical defects like undercuts. The recommended procedure for analysis of the weld roots is adopted in the present paper and the results are described later in the paper.

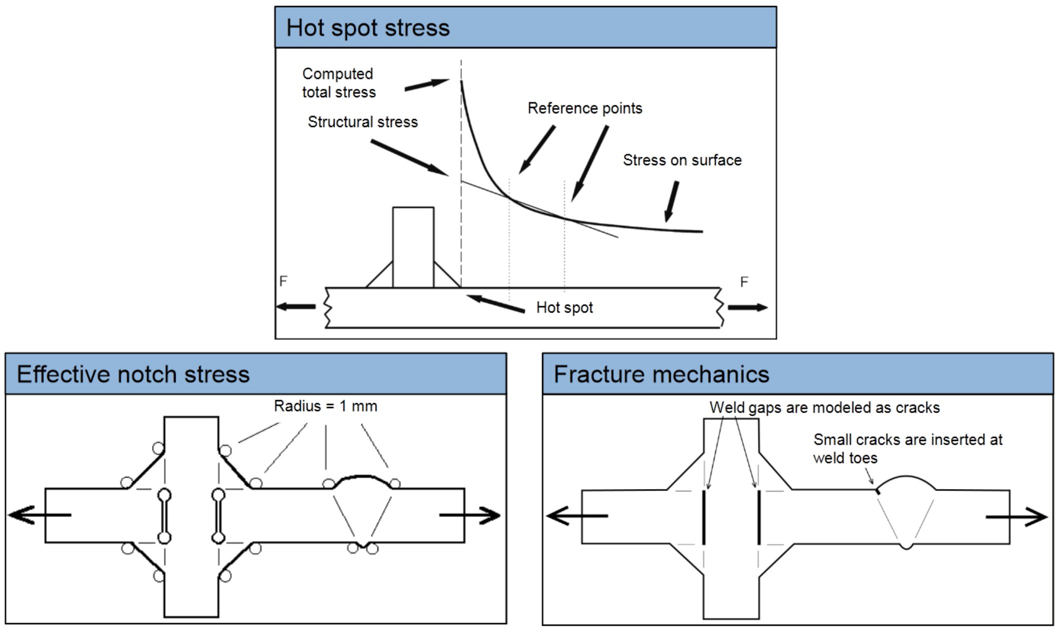


**Figure 2. Recommended procedures for numerical evaluation of welded design against fatigue failure by IIW.**

## 3. Simulation of the welding process in Abaqus

In the calculation of the residual stress field from welding of the structures in the large two stroke engines the use of analytical solutions will only be applicable to a limited extend. Variables like the geometry of the structure, weld sequences and size of the welded components will soon require three dimensional numerical calculations. This type of calculations can be performed in Abaqus as described in (Hansen J. L., 2003). The presented method aims at an welding tool to calculate deformations and residual stresses in large complex welded structures. In the verification of the method the numerical results are compared to several measurements of the type thermo-couple measurements, neutron diffraction measurements and hole-drilling strain gauge measurements. Good agreements are found in the comparisons and it is concluded that the method is reasonable and adequate for capturing the general global trend of residual stresses induced by welding.

In the simulation of the welding process a central problem is how the moving heat source is modelled. In the work by (Hansen J. L., 2003) the energy input is divided into contributions from added weld filler, body flux and surface flux as shown in figure 3. The weld profile is effectively adjusted by changing the power distribution between these three parts, thus with knowledge of welding or typical weld shape characteristics of the welding process it is possible to get at fairly good agreement with a specific weld pool shape as demonstrated in (Hansen J. L., 2003).

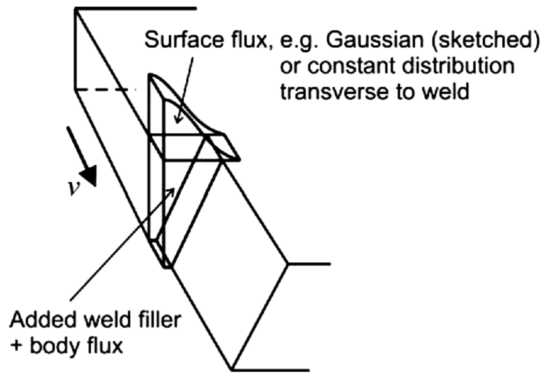


**Figure 3. Principle of moving heat source for welding, modelled with addition of weld filler, a body flux and a surface flux shown on symmetric half of a butt weld.**

The described methodology for a moving heat source can be implemented in a transient three dimensional Abaqus simulation by adding weld filler and body flux in increments as illustrated in figure 4. In the modelling stage an adequate fine mesh is specified in the total length of the weld. The elements representing the filler material are given an initial temperature above the melting temperature. The filler elements are then removed/deactivated and then reactivated incrementally with the Abaqus command, "*Model change add". The surface flux and body flux are modelled as constants throughout each step and with the commands "*Film" and "*DFlux".

Subsequent to the thermal analysis a quasi static analysis is carried out. The static analysis is loaded by the resulting temperature field calculated in each step of the transient heat transfer analysis. As elements are activated in the mechanical analysis they will have a temperature about the specified initial temperature of the filler material. When the heat source moves on the filler material will solidify and cool. The weld restrains and in this way the stress and deformation develop.

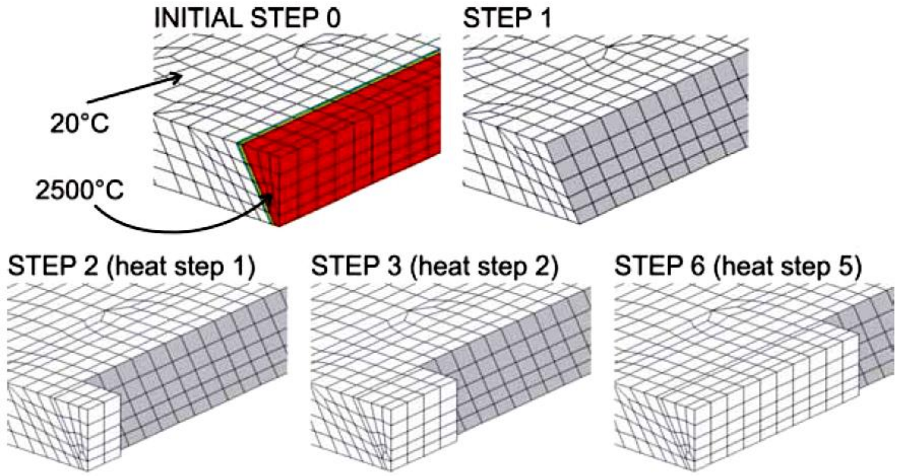


**Figure 4. Principle of moving heat source modelled with successive activation of elements serving as filler material shown on symmetric half of a butt weld.**

## 4. Cracks in welds of the second order compensator face plate

The large two stroke engines with a limited number of cylinders have second order moment compensators installed in the engine in order to limit the vibrations of the main engine structure. If the vibrations of the engine are not reduced they will transfer to the hull of the ship and result in discomfort for the crew of the ship. The moment compensation is obtained by inserting large rotating masses in the chain drive of the engine. Lack of attention on this structural detail in the complex engine structure led to reports on cracks in the welded connections between the engine frame and the face holding the shaft for the second order moment compensator.

Initial finite element investigation of the failed design was performed on shaft, face, and a small part of the engine structure loaded by the centrifugal forces of the rotating mass. Even though calculations performed in the original design phase showed reasonable relation between the total weld area and force of the compensator the simple finite element calculations clearly demonstrated four stress concentrations at the location of the interrupted weld as illustrated in figure 6. In addition to the rotation force, later analysis shows that also the combustion of the engine apply additional load on the end of the welds from where the cracks are observed to propagate.

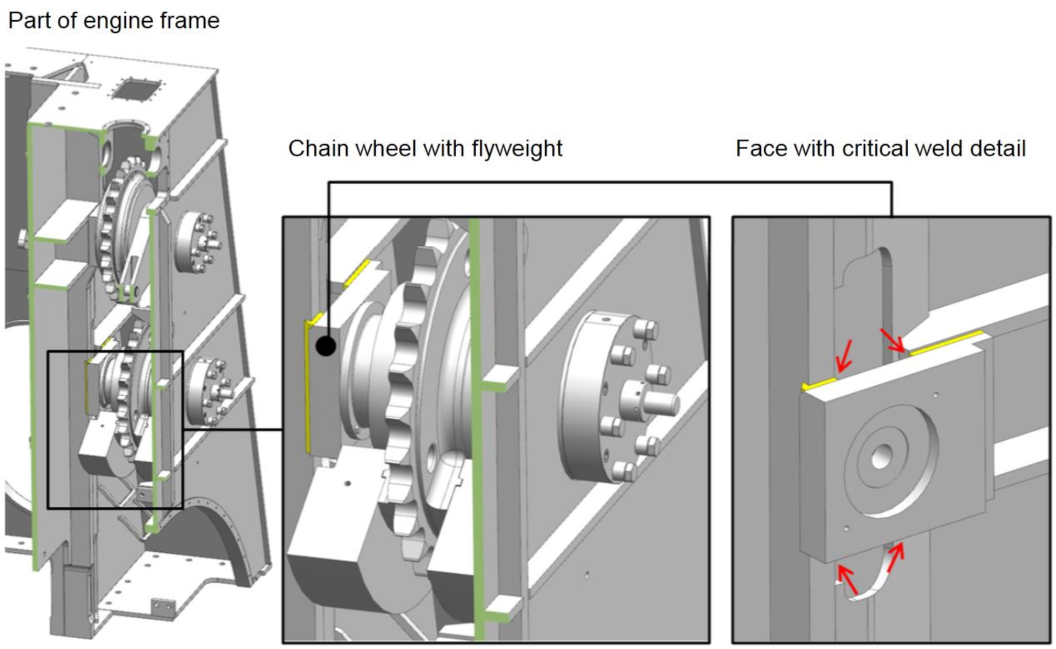


**Figure 5: Critical weld detail illustrated on chain wheel with flyweight for the second order moment compensator.**

## 5. General evaluation of the safety against fatigue failure

The model used for simulation of the running engine is simplified to an extend where only the main structure is included as illustrated in figure 1. As none of the moving parts are included, reaction forces are instead transferred to the model. Even though the loads are easily added by an automated in house software each load represents extensive research. The combustion force is transferred through the piston rod to the connecting rod and further to the crank shaft resulting in a rotation of the crank shaft. Due to the long stroke of the engine a guide shoe supports the bearing connecting the piston rod and the connecting rod resulting in horizontal forces on the guide bars. The combustion forces are in general prescribed on the connection surface between cylinder frame and cylinder and thus the cylinder can be omitted from the calculation. The main bearing forces and guide forces are calculated by an in house software based on the combustion forces and the dynamics of the moving parts. The calculated forces are transferred to the engine structure as pressure loads on the bearing surfaces using hydro or elastohydrodynamic oil film calculations. In addition to the basic loads described above a rotating load of the flyweight is included in the analysis. The loads are added to the engine model in a number of quasi static load steps described by the rotation of the crankshaft.

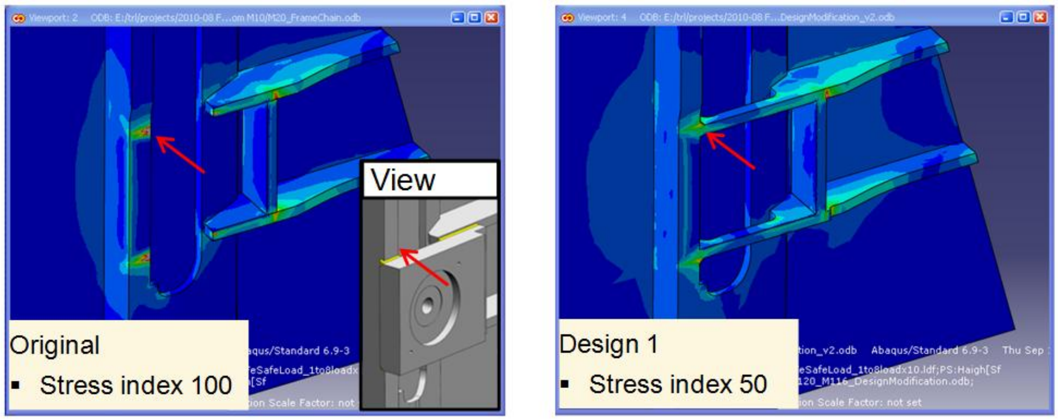


**Figure 6. Comparisons of designs based on relative stress amplitudes.**

In the post processing stage of the numerical analysis, the running engine is analyzed by secondary software in order to evaluate mean stresses and stress amplitudes from a critical plane approach. Evaluating the points against the limit curves in a Haigh diagram can in the general case give an absolute value of the safety against fatigue failure. However when dealing with welded structures the geometry of the weld generates singular stress fields which cannot be used in the Haigh diagram due to their unphysical nature. Even though the singularities in the stress fields limit the possibility of calculating an actual safety factor the stress amplitude calculations are however very useful for design comparisons as illustrated in figure 6. The peak amplitude of the original design is set to a stress index of 100 percent and the stress index of the design alternatives are calculated as a percentage of the original peak amplitude. Equivalent mesh density is however required in the comparison as the peak stress levels are highly mesh dependent. Proving that the design alternatives can decrease the largest stress amplitude does however not prove that the design alternatives are safe. Thus a secondary method is required in order to get an absolute evaluation of the safety against fatigue failure.

## 6. Evaluation of welds with XFEM stress intensity factor calculations

The advantage of XFEM in modelling of cracks is that the crack can cut the enriched elements arbitrarily and thus no special crack mesh is required. But in order to obtain reasonable calculations of the stress intensity factors it is recommended to setup modelling guidelines. Based on a thorough pre-study of the method consistency of the results can be obtained even for different analyses. In a non published study of stress intensity factor calculations in Abaqus version 6.11 it was found that

- Use hexagonal elements (C3D8) in the region of the crack to calculate stress intensity factors. Both linear and quadratic tetrahedral elements result in unstable and general poor accuracy.

- Place the crack in the centre of the elements and parallel to element edges for less deviations in the $K_I$ values along the crack front as illustrated in figure 7. The average precision is however just as good with any other orientation.
- If the crack face hits nodes or the front hits element borders the analysis might terminate prematurely but small changes in crack size, orientation and placement solves the problem.
- The element size close to the front is selected so that there is at least 10 elements on the shortest part of the crack.
- In the region surrounding the crack both linear and quadratic tetrahedral elements can be used to obtain accurate results and faster meshing of advanced geometries.
- Use five contours to calculate the J-integral and stress intensity factors based on the developed Python script for treating the raw results of the output files. The size of the contour is set to automatic adjustment and it should be in the size of one half of the smallest crack dimension.

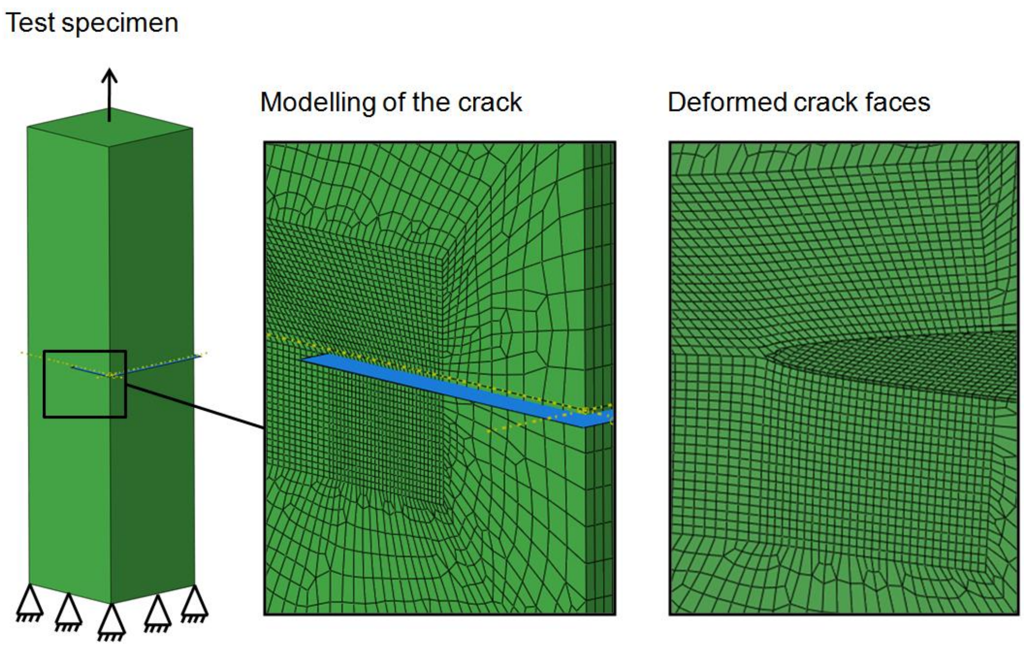


**Figure 7. Modelling of through crack in Abaqus XFEM.**

The investigation of XFEM calculations performed in Abaqus v6.11 shows that improvements can be reported relative to previous tests performed in Abaqus v6.9EF1. The investigation was performed on test problems based on a circular crack placed with different angles in a uniaxial test specimen. It was revealed that deviations below 5 % to the analytical solution can be expected for problems dominated by mode-I and deviations below 10 % to the analytical solution for problems dominated by mode II and III. Having these deviations in mind the Abaqus XFEM formulation can be used for stress intensity factor calculations but the deviations do certainly leave margin for further improvements of the accuracy.

Investigation of whether a crack of few millimetres will grow at some position in a four storey high engine does without doubt require mesh refinement in the area of the crack in order to fulfil the guidelines described above. Performing three dimensional mesh refinement on hexagonal elements in an advanced weld geometry of the engine is however a very time consuming task. One way to ease the task is by the use of a sub modelling strategy. In the case described in this paper the crack is represented in the global model as a weld gap and in the sub model as a crack shown in figure 8. The method can also be used for more advanced cases of crack growth (Lucht, 2009).

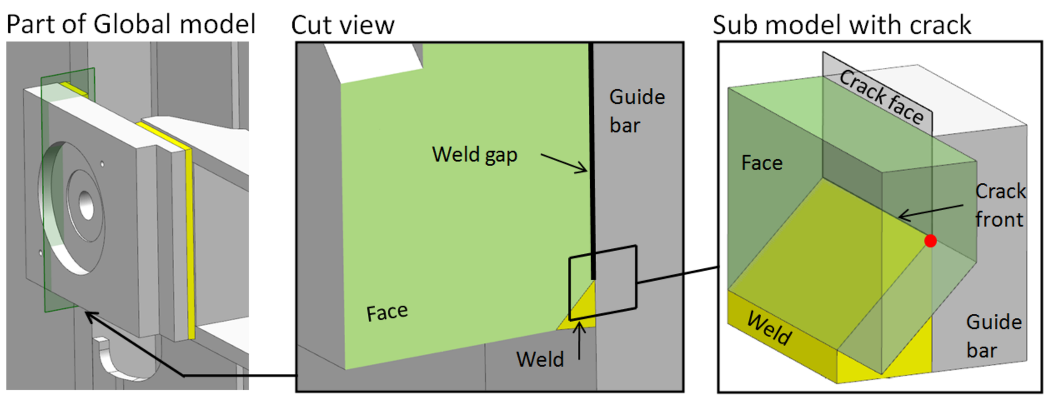


**Figure 8. Sub modelling strategy used to model part of weld gap as a crack.**

The sub model is created as one connected mesh representing the weld and the two different structures that are welded together. The Abaqus CAE is used to define the crack surface representing the weld gap between the two structures. In order to calculate the most severe dynamic loading on the crack it is chosen to exclude contact in the crack formulation as the crack (weld gap) might be open due to residual stresses of the welding process or due to a slight misalignment gap between the two parts. Displacement boundary conditions from the displacement field of the global model are now prescribed on the boundaries of the sub model. The sub model is solved and a Python script is used to extract, pick out and average the stress intensity factor results from the text files written during the pre-processing stage and execution of the simulation. The processed stress intensity factors of the analyzed problem is shown in figure 9. Each curve represents the stress intensity factors along the crack front for a specific position of the crank shaft.

Based on the calculated stress intensity factors it is obvious that all three modes of fracture is present i.e. opening ($K_I$), shearing ($K_{II}$) and tearing ($K_{III}$). As expected the largest opening and closing of the crack occurs at the free end of the crack represented by the nominal distance of zero along the crack front in figure 9. Because the contact formulation was excluded from the calculation $\Delta K_I$ values below zero are seen for part of the load cycle and they are a result of crack over closure. If a possible tensile residual stress field from the welding process was included in the calculation it would shift all curves above zero. Thus it is important not to skip the negative values in the threshold evaluation of the weld without investigation of the residual stress field. The $\Delta K_I$ values at the free end of the crack are thus around 8 MPa√m which is above the design limit found in the recommendation by the International Institute of Welding. The ratio between $K_I/K_{II}$ is used in the calculation of the crack propagation angle. Since the $K_{II}$ values are negative and in the same order as the $K_I$ factor for all steps the crack can be expected to change direction. The predicted change of direction is in agreement with observations of crack propagation in a plane 45 degrees to the initial plane of the weld gap. The $K_{III}$ factor is also present and can be expected to increase the driving force of crack propagation.

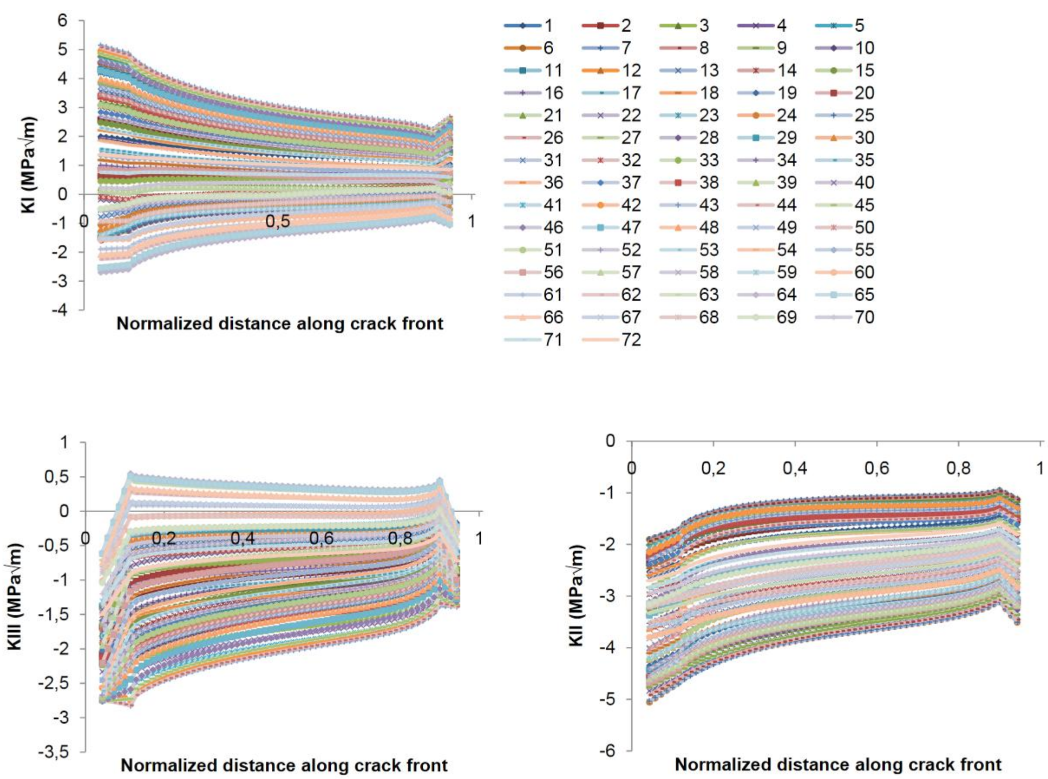


**Figure 9. Stress intensity factors calculated along the crack front for all load steps representing one rotation of the crank shaft.**

In order to evaluate the most critical point with highest ΔK value figure 10 illustrates the stress intensity factors at a point on the crack front close to the free end of the weld. The figure illustrates the oscillating values of the stress intensity factors while the engine makes one revolution. It is seen that centrifugal load from the flyweight rotation opens and closes the crack twice per revolution of the crank shaft. As the present problem clearly represents a case of non-proportional mixed mode loading it is necessary to calculate an effective stress intensity factor $\Delta K_{eff}$ which take into account the multiaxiallity of the problem when comparing to the threshold values in the IIW recommendation. A discussion of the effective stress intensity factors can be found in (Lucht, 2008). For this problem it is in the present paper chosen to use the most conservative estimate of $\Delta K_{eff}$, which is a $\Delta K_I$ with both the negative and positive values of $K_I$ included.

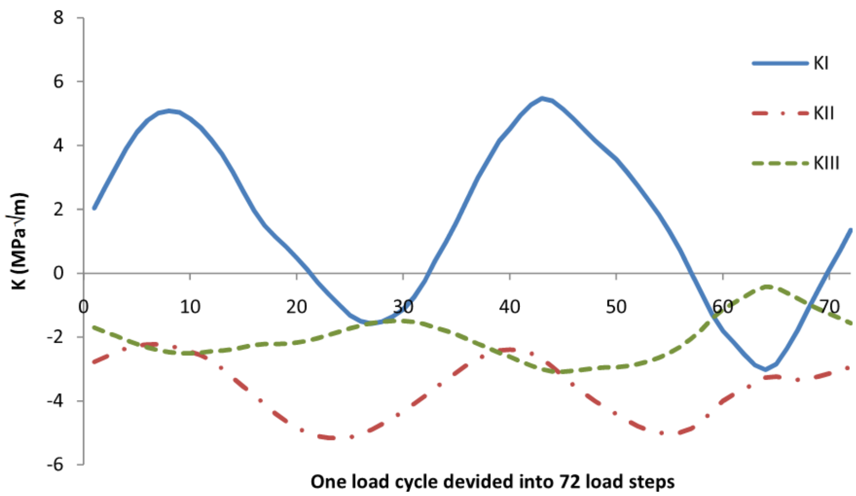


**Figure 10. Stress intensity factors calculated at a point on the crack front near the free end of the weld.**

Similar calculations as described above are performed on the most promising design alternative for repair of engines and new engines. The highest values of $\Delta K_{eff}$ are plotted in figure 11 against the limit curve as described in the recommendations by the International Institute of Welding. The figure clearly illustrates that the original design is outside the limit curve indicating an expected risk of failure. The two new designs are within the limit curve based on the calculated stress intensity factors. A question does however remain, whether large tensile stresses from the welding process will shift the mean stress level in the area of the crack front and reduce the distance to the limit curve. In order to investigate the residual stress field around the crack front it is chosen to perform simulation of the welding process.

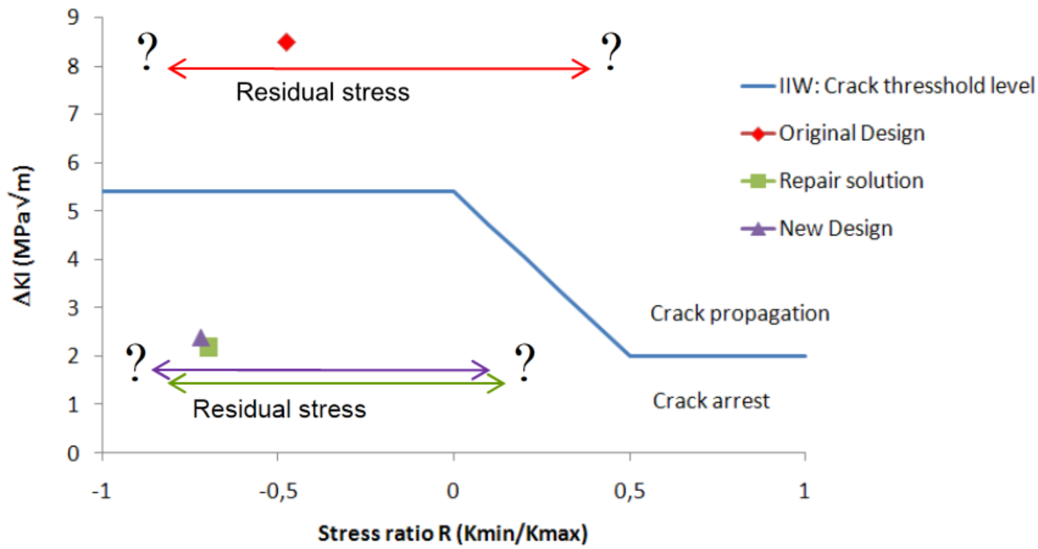


**Figure 11. Stress intensity factors calculated at a point on the crack front near the free end of the weld.**

## 7. Residual stress of the welding

The aim of the weld simulation is to illustrate the result of welding an additional supporting structure into an engine already in service. With this type of simulation it will be possible to estimate both the deformation of the existing engine structure and the residual stress field around the crack front. Several simplifications of the problem are introduced in the analysis presented in this paper. The analysis is less ambitious and the main concern is whether large residual stresses will build up along the crack front. It is assumed that filler material is welded in only three strings. In addition it is only chosen to simulate the welding along one side as illustrated in figure 12. The simulated welding is chosen as it will introduce residual stresses perpendicular to the crack face resulting in a opening or closing of the crack.

The heat transfer analysis is performed as described in a previous section. The results of each step in the heat transfer analysis are transferred to the static analysis and the stress level remaining in the direction perpendicular to the crack is shown in figure 12. In order to investigate the critical point between the added supporting structure and the original structure a cut is introduced in the model as illustrated in the figure 12. As expected large tensile residual stresses remain at the toes of the weld while the stress level is negative or close to zero at the root of the weld. Thus based on the current welding analysis it can now be concluded that no large tensile stresses are present at the root of the weld and thus the stress intensity factor evaluation of the two new designs will only

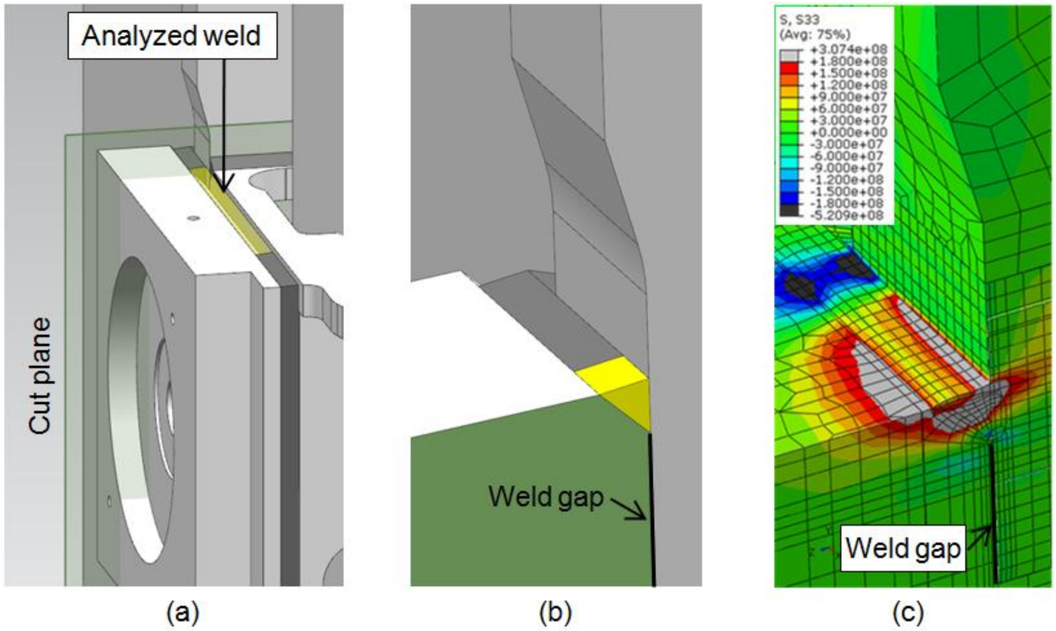


**Figure 12. Simulation of welding process a) analyzed weld b) cut view c) stress field perpendicular to weld gap.**

be marginally influenced by the residual stresses of the welding. At the same time it is also highlighted that it will be a good practice to take precautions like peening in order to limit the influence of the high tensile stresses on safety against fatigue failure.

## 8. Conclusion

Standard Abaqus calculations on a running engine have been used to evaluate the best design alternative to the second order moment compensator arrangement. In order to obtain an absolute evaluation of the original and improved designs it was chosen to use the Extended Finite Element Method (XFEM) in Abaqus. The calculated stress intensity factors show that the original design is expected to fail while the new design should be safe within the recommended limit curve. Large tensile stresses at the root of the weld could however remove the safety margin of the new design. Simulation of the welding process also performed in Abaqus did however show that no tensile stress was present at the root of the weld. Thus it can be concluded that the new design is safe within the limit curve. The three dimensional XFEM stress intensity factor calculations in Abaqus is considered as a strong tool. Improvements still remain even though the author has experienced improvements from the first implementation of XFEM to the recent implementation in Abaqus version 6.11. Improvements of the accuracy of the stress intensity factor calculations are important for the hexagonal elements. In addition much time could be saved in the modelling phase of the analysis if the accuracy of the second order tetrahedral elements could be brought to a level equivalent to the accuracy of the linear hexagonal elements.

## 9. References


1. Hansen A. V., Olesen J. F. and Agerskov H., "An investigation of the influence of root defects on the fatigue life of the welded structure of a large two-stroke diesel engine," Welding in the World, Journal of the International Institute of Welding, vol: 48, issue: 5/6, pages: 46-55, 2004
2. Hansen J. L., "Numerical Modelling of Welding Induced Stresses," PhD thesis, Technical University of Denmark, 2003
3. Hobbacher A., "Recommendations for fatigue design of welded joints and components," International Institute of Welding doc., XIII-2151r4- 07/XV-1254r1-07, 2007
4. Lucht T., “Analysis of cracks in large diesel engines," PhD thesis, Technical University of Denmark, 2008.
5. Lucht T., "Finite element analysis of three dimensional crack growth by the use of a boundary element sub model," Engineering Fracture Mechanics, Volume 76, Issue 14, Pages 2148–2162, 2009