Fretting Simulation for Crankshaft-Counterweight Contact

A. Mäntylä and C. Lönnqvist

Department of Calculation and Simulation, Wärtsilä Finland – Engine Division

antti.mantyla@wartsila.com and christian.lonnqvist@wartsila.com

Abstract: The ability to accurately calculate the risk of fretting fatigue in the contact interfaces of clamped sub-assemblies in medium-speed diesel engines has become important as a consequence of, among other things, performance demands. An unfavorable combination of high clamping forces and oscillating dynamic forces may lead to a severe shear traction development in the contact interface of screw joints, which increases the fretting fatigue risk. The application of the finite element method to fretting fatigue risk evaluation is a demanding task because the contact has to be modeled very accurately with the help of, among other things, a very dense mesh to capture the steep stress gradient in a partial slip situation. A three-dimensional model of the contact between a counterweight and a crankshaft in a Wärtsilä diesel engine was made by using Abaqus. For this purpose, some Abaqus features, such as the contact interaction and adaptive remeshing tool, were utilized to their limits with the intention to obtain critical locations of the stress fields that potentially could nucleate a fatigue crack. The results show that the critical areas regarding fretting fatigue can be detected by using a three-dimensional model of real engine parts. The critical stress fields also show characteristics that have been detected, by analytical and two-dimensional models, to be present in the same kind of fretting situations. Redistribution of these stresses due to wear was taken into account by wear simulation. An in-house program allowed multiaxial fatigue criteria to be applied on Abaqus stress results in order to obtain the fretting fatigue cracking risk in a contact interface.

Keywords: Fretting, Wear, Partial slip, Contact, Multiaxial Fatigue, Adaptive Remeshing, Submodeling, Engine.

1. Introduction

Due to increased access to computational power and development of 'smart' modeling techniques, it is today possible to perform demanding non-linear finite element analysis on complex engine assemblies with software like Abaqus. Contact analysis involving friction certainly belongs to this category of analysis problems. The ability to evaluate the severity of contact stresses and interfacial slip in terms of risk for fatigue crack nucleation in contact interfaces of clamped sub-assemblies in medium-speed diesel engines has become increasingly important due to, among other things, performance demands. This very specific type of damage progress that involves crack nucleation and crack growth in the contact environment is frequently spoken of as fretting fatigue. The development of interfacial cyclic shear tractions and hence damage progress is often a consequence of an unfavorable combination of high clamping forces and oscillating dynamic forces.

Critical stresses typically rise at narrow bands in a stick-slip interface. Mesh refinement and contact formulation are hence of first order importance. The work that was undertaken revealed that the adaptive remeshing capabilities of Abacus/CAE may be very useful to allow adequate mesh refinement in the stick-slip interface. Abaqus provides several contact formulations, and of these the penalty and Lagrange multiplier formulation have been applied to the classic Hertzian contact in order to compare their influence on the contact behavior. Wear may be a key-contributor to re-distribution of contact stresses under cyclic loading. For this reason, an in-house Python script has been developed to allow Archard's wear law to be applied to both the Hertzian and the crankshaft-counterweight contact. The results show that the characteristics of the contact change completely due to high wear rates in the neighborhood of the stick-slip interface.

The experience obtained from the analysis of the Hertzian contact was required in order to obtain understanding and confidence in the application of the finite element method to the crankshaft-counterweight interface. The boundary conditions (loading) of the crankshaft-counterweight have been obtained from a multi-body simulation of the whole crank train. The analysis can hence be considered as a 'quasi-static' one in the sense that it includes 240 static load increments over one engine cycle. Except for the loading, quite similar modeling techniques have been utilized for the crankshaft-counterweight sub-assembly as for the Hertzian contact. In addition to adaptive remeshing, also the submodeling technique has been utilized in order to obtain mesh refinement. The results reveal that similar characteristic as those seen in the stick-slip interface of the Hertzian contact were also observed in the critical regions of the crankshaft-counterweight contact interface. An in-house multiaxial fatigue code was then applied to the obtained stress history. Wear simulations revealed that the safety factor against fatigue crack nucleation obtained from the multiaxial fatigue analysis decreased significantly as a consequence of the re-distribution of contact stresses.

2. Methods

An accurate contact model is required in order to obtain a realistic relative displacement field between the contact surfaces so that a realistic stick and slip behavior is captured in the contact interface. This requires accurate and numerically well behaving contact formulation.

Critical stress fields arise at contact edges and at stick-slip boundaries where stress gradients become very steep. In order to capture the critical contact stress field, the mesh has to be relatively dense in these interesting locations which may moreover often be unknown at the start of the analysis. Furthermore, cyclic slip in an un-lubricated contact interface leads to the evolution of the contact surfaces profile due to wear, causing the contact stresses to re-distribute. In order to overcome these challenges involved in the modeling of contact behavior and hence prediction of fretting fatigue risk submodeling technique, adaptive remeshing and wear simulation, as explained in the following chapter, had to be implemented.

2.1 Abaqus contact

Contact behavior, especially frictional, has a strong influence on fretting fatigue analysis results. This can be represented illustratively through investigation of a cylinder-against-a-plane (Hertzian) contact. This specific type of a contact is frequently utilized for fretting fatigue testing. Figure 1 represents a Hertzian contact case simulated by using Abaqus v.6.8-1 in order to investigate, among other things, the effect that tangential behavior has on contact stresses.

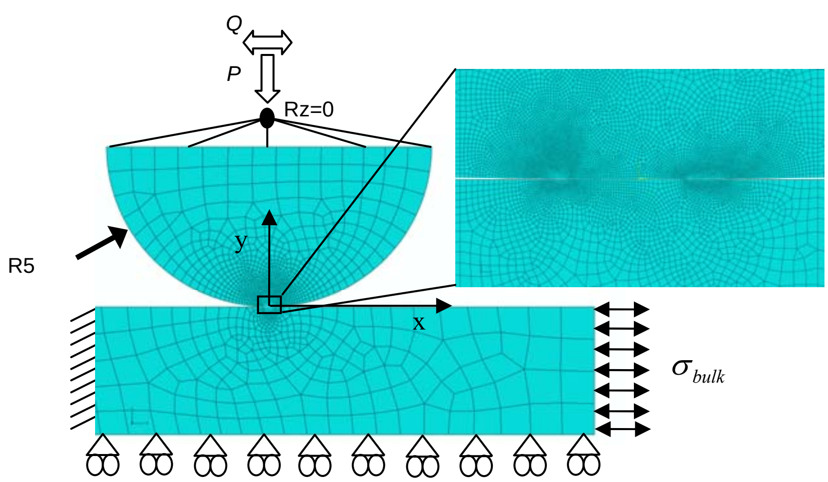

Figure 1. The model of a Hertzian contact case.

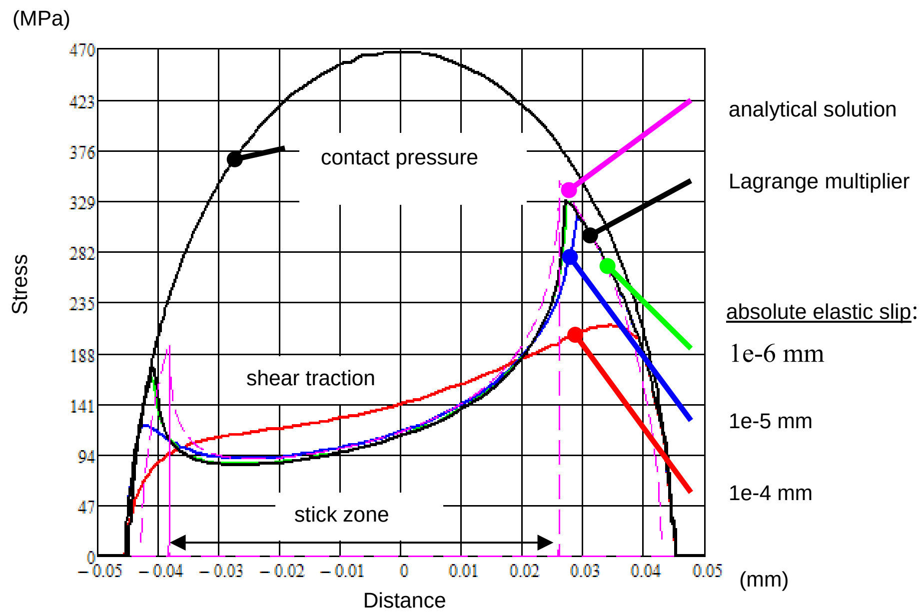

Figure 2. Surface stresses on the plane surface.

Contact interaction with surface to surface and finite sliding definition were used. Both normal and tangential behavior was defined. A "hard" contact was used to describe normal behavior together with either Lagrange multiplier or the penalty method for tangential behavior with the friction coefficient of 0.9. According to expectations, the application of the Lagrange multiplier formulation gives the most accurate shear traction distribution (Figure 2). The implementation of the penalty formulation requires that the limit magnitude of allowable elastic slip is given in the input along with the coefficient of friction. By decreasing the magnitude of allowable elastic slip, shear traction distribution and slip behavior approach the predictions obtained from the application of Lagrange multiplier and analytical solution. A characteristic of the penalty formulation is that slip occurs also on the stick areas and the stick-slip boundary becomes less accurately defined. The ultimate objective is to recognize stick and slip areas with accurate boundaries; it is hence favorable to apply either the Lagrange multiplier formulation or the penalty formulation with a very low value for the magnitude of allowable elastic slip. However, the fact that the elastic micro-slip exists in real contacts has to be remembered. (Abaqus Documentation)

2.2 Submodeling

The sub-modeling technique becomes very useful, especially if there are many contact surfaces between the contacting bodies. The use of the sub-modeling technique for models involving contact requires extreme care to be taken as the boundary conditions of the sub-model should match those of the global model. This is not satisfied in modeling of contact behavior with the help of the penalty formulation unless the magnitude of allowable elastic slip is equal for both models. The use of the same relative elastic slip value in both models leads too large displacement boundary conditions in the sub-model in which the mesh is much denser than in the global model. Lagrange multiplier method for tangential contact formulation of course prevents this error, but in many cases, especially subsequent to wear simulation, it may lead to severe convergence difficulties.

2.3 Adaptive remeshing

For simulation of the fretting phenomenon, a relatively dense mesh has to be utilized in order to be able to capture steep stress gradients accurately enough. The computational cost rises very fast when the element size is decreased manually to the required size in the contact interface. Therefore it is favorable to have a dense mesh only in the interesting areas, but these areas are often unknown before the first fretting analysis. In these situations, the application of adaptive remeshing becomes very useful for automated mesh control in the contact interface. Adaptive remeshing is based on error indicators that describe the error distribution of certain variables. The mesh can be focused on the stress concentration areas with dangerous partial slip case in the contact interface by using von Mises stress error indicator with carefully defined remeshing rules. Error indicators of surface shear stresses or principal stresses would be much more suitable for the fretting cases, but they are unfortunately not available in Abaqus/CAE by default and the use of them would require some scripting. Figure 3 shows the screen captures taken during an adaptive meshing process of the Hertzian contact case that was introduced in section 2.1.

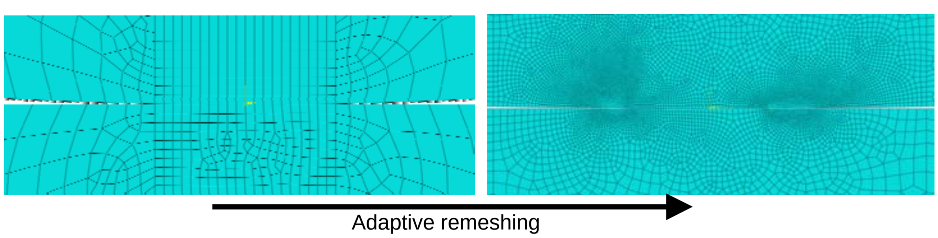

Figure 3. Focusing the mesh in the Hertzian contact by using adaptive remeshing.

It is important to notice that the use of the Lagrange multiplier method or the penalty method with a small elastic slip value is required in order to successfully utilize adaptive remeshing for the analysis of contacts. The steep stress gradients which follow from a partially slipping contact interface have to be captured in order to get a more distinct error in the interesting areas. It is slightly unfortunate that tetrahedral elements are required for three-dimensional analyses of contacts with the help of adaptive remeshing. Therefore it may in certain cases be more reasonable to use brick elements and control the mesh manually.

2.4 Wear simulation

Wear has a profound influence on contact behavior under cyclic loading conditions. Crack nucleation risk may hence change quite dramatically as wear damage builds up. In the partial slip case, the contact pressure may be relatively high in the slipping part of the interface, which favors the formation of wear debris. The wear mechanism can not be modeled accurately on a micro-scale level. Nevertheless, simulation of material removal due to wear is possible and may help to capture the key phenomena related to crack nucleation and propagation. In order to include wear effects into the fretting analysis, a Python scripting interface was utilized in the writing of a wear simulation code. The program iteratively simulates contact surface evolution by updating the mesh in accordance with the wear fields calculated by using Archard's equation. The underlying principle of the wear simulation is described in Figure 4.

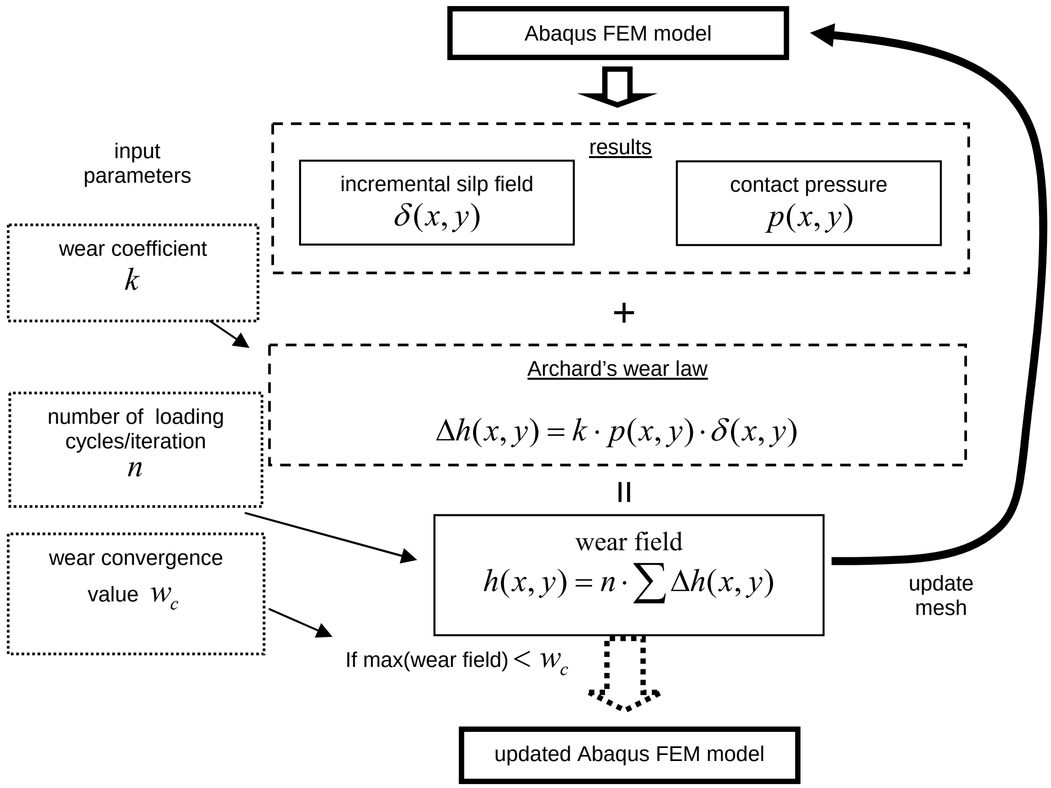

Figure 4. Iterative wear simulation by using Archard's wear law.

Abaqus calculates the orthogonal slip fields CSLIP1 and CSLIP2 which can be utilized in order to calculate the incremental slip field magnitude needed for the wear field calculation. The incremental slip magnitudes in increment *i* is given by

$$\delta(x,y)_i = \sqrt{\mathrm{CSLIP1}^2{}_{i+1} + \mathrm{CSLIP2}^2{}_{i+1}} - \sqrt{\mathrm{CSLIP1}^2{}_i + \mathrm{CSLIP2}^2{}_i} \qquad (1)$$

Contact pressure is calculated as the mean value of two subsequent increments *i* and *i* +1 and is hence given by

$$p(x,y)_i = \frac{\mathrm{CPRESS_i} + \mathrm{CPRESS_{i+1}}}{2} \tag{2}$$

One relevant concern is to ensure that the slip fields CSLIP1 and CSLIP2 are defined as zero in the stick areas. This requirement holds only for cases where the Lagrange multiplier formulation has been utilized for the definition of tangential behavior. For analysis cases involving convergence challenges, it may nevertheless be desirable to use the penalty method. Unfortunately, the use of the penalty formulation allows certain amount of elastic slip in the ‘stick zone’ which is interpreted as true slip and hence causes unintentional wear. This can be avoided by using a very small value for the allowable elastic slip and then eliminating it in the slip results. The elimination procedure is not described in detail within the scope of this paper. However, it can be considered that it is feasible within certain limits to remove the elastic slip without distorting the results. As slip results are only available on slave surfaces, the wear field obtained on this surface needs to be transferred to the master surface.

The code for wear simulation was first tested on the Hertzian case in order to ensure proper contact surface evolution. About a hundred iterations resulted in the surface evolution as illustrated in figure 5. In this case, the wear practically stopped after about a hundred iterations as the contact pressure in the slip areas wore away. This is an interesting finding due to the reason that it seems that, at least in this case, wear changes the contact characteristics towards a fully adhered type of contact where stress singularities arise at the contact edges. This could indicate that a fracture mechanics approach would be useful when trying to predict the cracking risk of a worn geometry. The authors have future plans to investigate methods that could be utilized for crack nucleation prediction in the neighborhood of the edge of a complete contact.

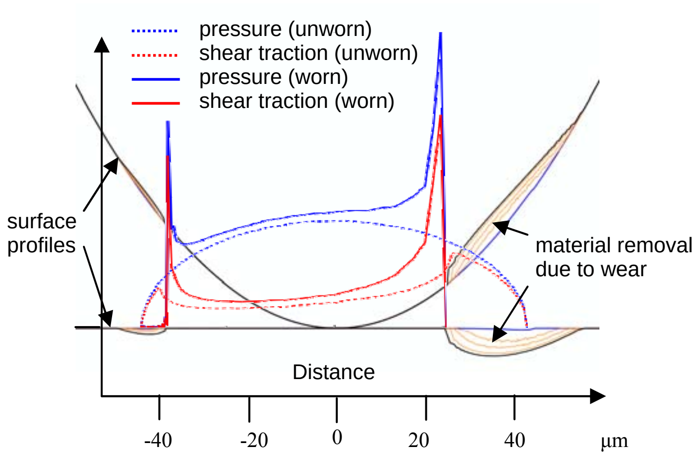

Figure 5. Evolution of the Hertzian contact due to wear.

Solving contact cases accurately enough with the goal to simulate fretting damage progress assisted by wear is a demanding task in terms of analysis convergence. The wear simulation may lead to severe discontinuity iterations as a consequence of the surface profile evolution and growth of wear regions. To solve these classes of problems, contact parameters have to be defined very carefully to obtain convergence and good results.

2.5 Multiaxial fatigue criterion

The use of a multiaxial fatigue criterion to define the fatigue crack nucleation risk is not always necessary. E.g. in the Hertzian case at the trailing edge of the contact, shear tractions superimpose on bulk stresses to form an almost uniaxial stress state. In the crankshaft-counterweight interface, the complexity of the stress state and how it develops throughout the engine cycle is less clear due to the non-proportional loading, i.e. it is challenging to define the critical time instances and directions of the principal stresses as the principal stress coordinate system rotates within one engine cycle. For this reason, an in-house multiaxial fatigue code Multi.V2 (Lönnqvist et al., 2007) was applied to the stress results obtained from the contact analysis. The Findley multiaxial failure criterion was chosen as it has been proven reliable (Rabb & Lassus, 2005).

In the Findley criterion, the damage parameter is defined as the sum of the shear stress range $\Delta\tau$ and a fraction of the normal stress σ_n on the critical plane

$$D = \left(\frac{\Delta\tau}{2} + k_f \sigma_n \right)_{\max} \leq f \ , \tag{3}$$

where k_f is a constant and f is the shear fatigue strength.

The shear fatigue strength f and constant k_f can be calculated when the fatigue strength from uniaxial testing is known at two load ratios (Rabb & Lassus, 2005). The mapping of the critical plane is essentially an extreme value problem where the critical combination of shear stress range has to bee searched among a set of plane candidates by transforming the stress tensor in a Cartesian coordinate system. The shear stress range can, on the other hand, be defined by enclosing the smallest possible circle around the history path that the shear stress resultant makes on the plane (Lönnqvist et al. 2007). The time instance of the normal stress σ_n needs not to coincide with any of those with which the shear stress range is defined.

According to the Findley criterion, a safety factor against fatigue crack nucleation may be defined as

$$S_{F,crit} = \frac{K_N K_R f}{K_{size} D} \quad \text{when} \quad A_{eff} \geq A_{ref} \quad \text{or} \tag{4}$$

$$S_{F,\,crit} = K_{size} K_N K_R \,/\, D \quad \text{when} \quad A_{eff} < A_{ref}\,, \tag{5}$$

where K_N is the life factor, K_R is the surface roughness factor, K_{size} is the statistical size factor, A_{ref} is the effective stress area of the reference specimen and A_{eff} is the effective stress area of the component.

The reader is turned to references (Rabb & Lassus, 2005) and (Lönnqvist et al., 2007) for additional reading on the definition of the correction factors. It is also possible to transform the damage parameter into an equivalent uniaxial stress case σ_a and σ_m, which allows the safety factor to be defined on the basis of a Haigh diagram (Rabb & Lassus, 2005).

3. Crankshaft-counterweight fretting analysis

3.1 Assembly

The contact interface between the crankshaft and counterweight is flat with sharp contact edges and can therefore be classified as a complete contact. High clamping forces and oscillating dynamic forces may lead to a severe shear traction to develop in the contact interface. This may moreover lead to wear and possible crack nucleation and growth. The analyzed crankshaft-counterweight assembly is illustrated in figure 6.

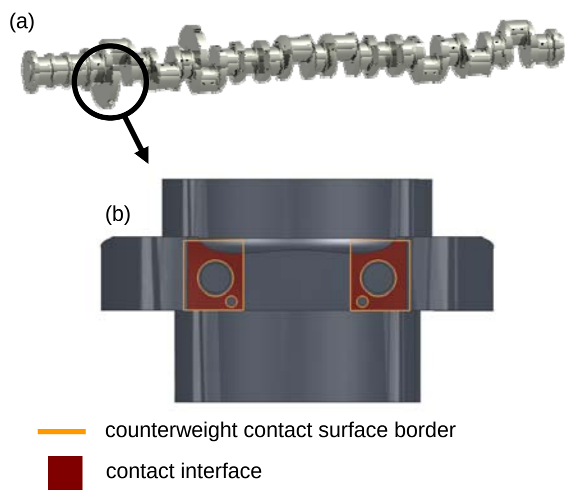

Figure 6. (a) Crankshaft with counterweights 1 and 5. (c) Crankshaft-counterweight contact interface.

3.2 Finite element models

Two finite element models (Figure 7) were used in the counterweight fretting analysis. The global model includes a part of the crankshaft and counterweight with two bolts. The boundary conditions (loading) of the global model have been obtained from a multibody simulation of the whole crank train. The analysis can hence be considered as a 'quasi-static' one in the sense that it includes 240 static load increments over one engine cycle. Of course the measured pre-tension forces of the bolts were included in the boundary conditions.

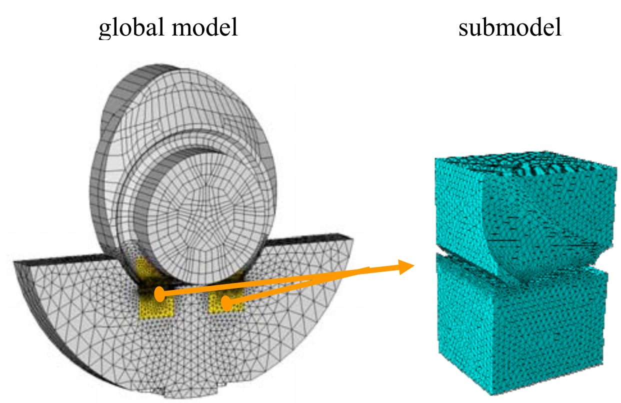

Figure 7. Finite element models for the fretting analysis.

The node-based submodeling technique was utilized so that the required mesh refinement was obtained in the contact region. In this case, the penalty method was applied for tangential behavior in order to obtain a converging solution over the wear simulation. Based on fretting test results for the same material pair, the friction coefficient was approximated to 0.9 (Aapo Pasanen et al.).

3.3 Analysis procedure

In this simulation, some contact opening occurs at certain moments and, at the same moments, also the most severe shear traction concentrations arise. Consequently, in line with what is expected in a partial slip case, shear stress concentration arises at the stick-slip boundary. This stress concentration, as in the Hertzian contact case, motivates the application of adaptive remeshing by the utilization of von Mises stress error indicator in order to focus the mesh-refinement to the interesting areas. However, the obtained mesh refinement is not fully satisfactory due to the unsuitable error indicator and a couple of areas with unnecessary mesh refinement.

Figure 8 illustrates how the partially slipping interface gives rise to a shear traction concentration on the contact surface at the boundary of the stick area and how mesh refinement can be focused on these interesting areas in order to capture steep stress gradients even more accurately. The remeshed model contains about 1.6 million degrees of freedom.

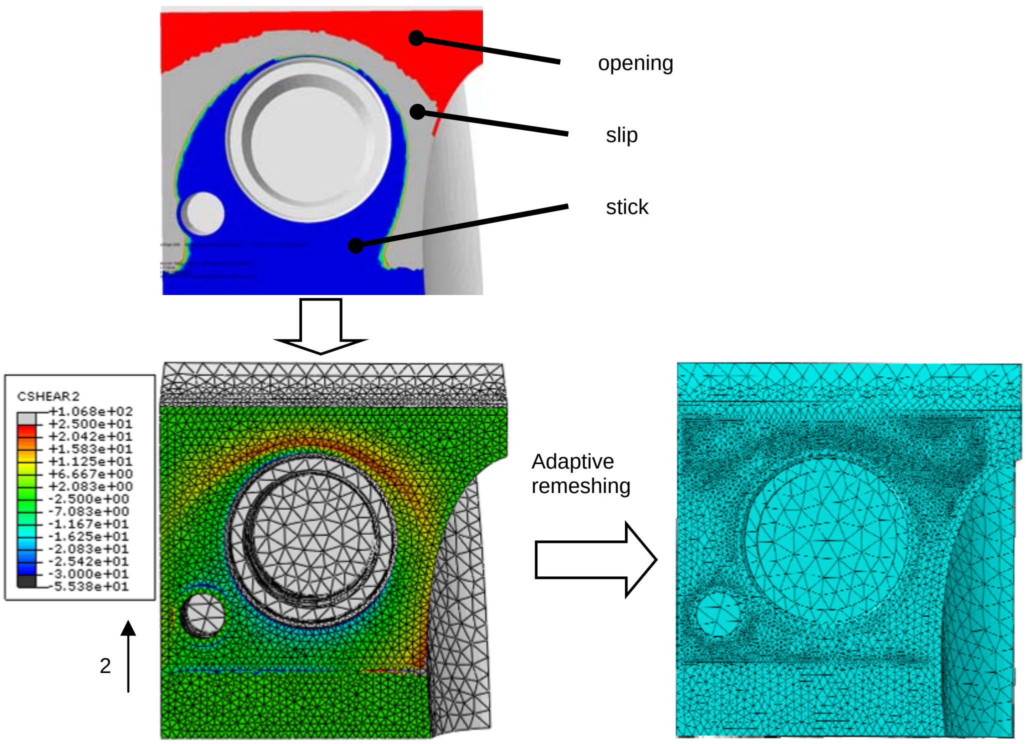

Figure 8. Mesh refinement by using adaptive remeshing.

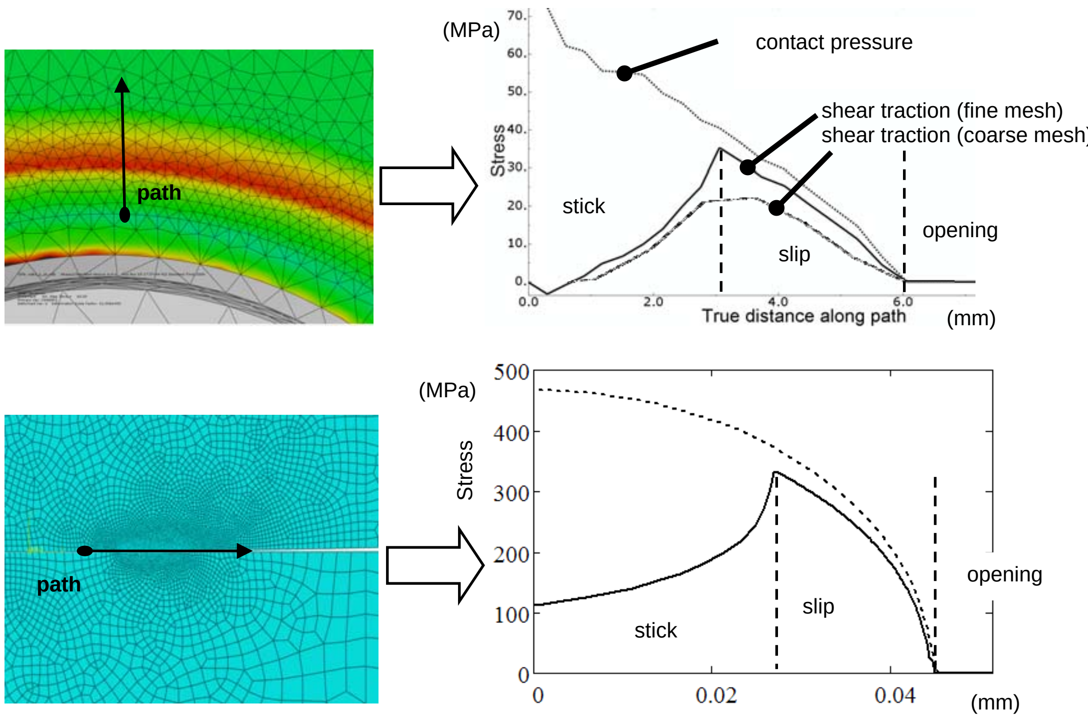

Figure 9. Critical surface stresses in the crankshaft case and in the Hertzian case.

Surface stresses can be plotted along the path (figure 9) and compared to the surface traction distributions of the Hertzian contact case. This shows how steep shear stress gradients can be captured with automatic mesh control in the contact interface. The plotted diagrams clearly illustrate the characteristics of surface stresses in the partially slipping/opening contact interface. These findings give rise to important questions: What are the conditions under which a partial slip case leads to cracking and further to crack propagation and how does wear affect this process in terms of evolution-critical stress fields?

In order to find answers to these questions, wear simulation and multiaxial fatigue analysis were performed on the crankshaft-counterweight contact interface. Unfortunately, the wear coefficient for the crankshaft-counterweight material pair has to be approximated, which practically means that the speed of wear could not be predicted. This was, however, not such a serious concern as the ultimate objective was to investigate the contact profile and stress evolution as a consequence of wear progress. In fact, a much more crucial issue was the choice of an applicable number of engine cycles per wear iteration to ensure that the contact surface evolution would become a history-dependent phenomenon as it clearly is.

With the application of the wear coefficient k=1^-7 MPa^-1, a wear simulation corresponding to 1.3 million engine cycles and thirteen iterations in total gave the wear results presented in figure 10 (a, b). The grey tone represents the unworn regions and the red tones correspond to wear depth (mm). After eleven iterations, the stick region practically stopped changing in shape and only the worn areas grew slowly in the normal direction. Wärtsilä has tested different counterweight shapes, and therefore experimental fretting results are available. The simulated wear fields correspond quite well to the experimental results (not presented in this paper). Fatigue analysis results of the more severe case can also be seen in figure 10 (c, d). As expected, wear leads to a re-distribution of critical stresses in the contact interface and has hence a dramatic effect on the calculated Findley safety factor, decreasing it from 5.9 to 1.41 in the more severe case.

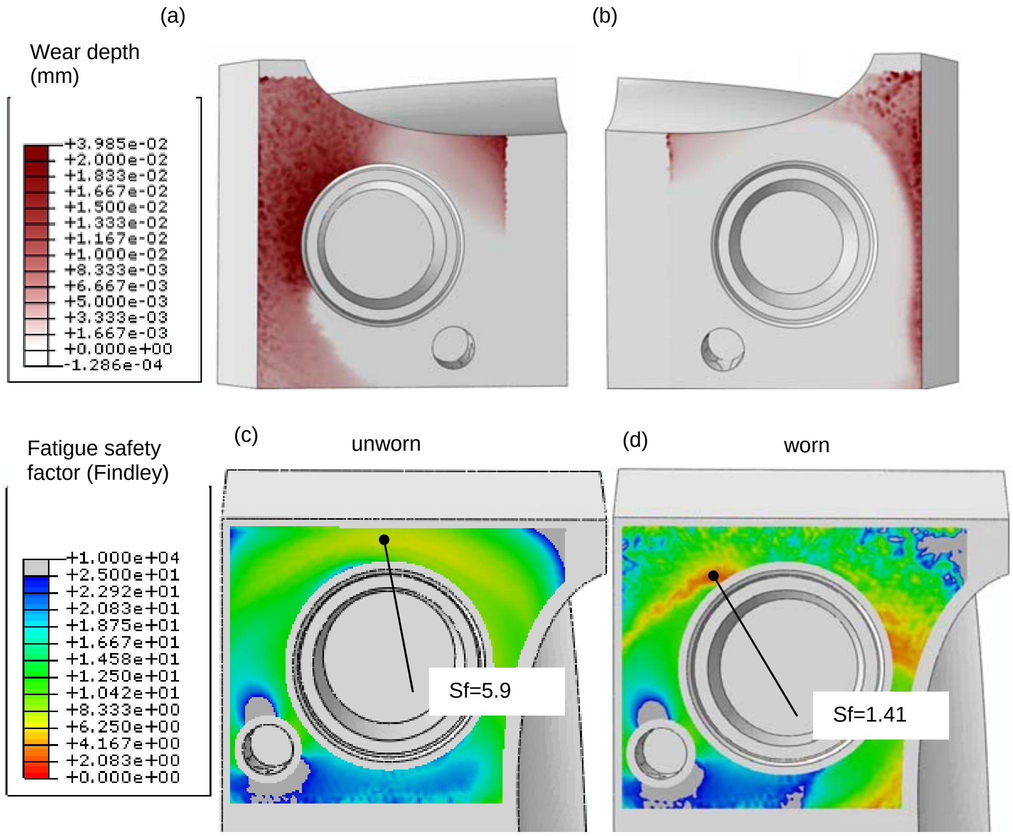

Figure 10. (a) Wear depth after the wear simulation in the first interface. (b) Wear simulation results of the second interface. (c) Fatigue safety factor of the unworn contact. (d) Fatigue safety factor of the worn contact.

3.4 Conclusion

The Hertzian case was first simulated in order to obtain understanding in the contact behavior under cyclic loading conditions. The Hertzian case also gave an indication of how mesh-refinement can be focused on stick-slip boundaries with the help of adaptive remeshing. In addition, the case helped to find a way to simulate wear within the Abaqus environment and it illustrated how wear influences the Hertzian contact under fretting conditions. An interesting finding is that, due to the wear progress, the Hertzian contact evolves towards a fully adhered or complete type of a contact where stress singularities arise at the contact edges.

The same methods were then applied to the crankshaft-counterweight contact interface, and the analysis results revealed that a partial slip situation develops at the interface. Also, the mesh can be refined adaptively to capture surface stresses in a similar way as for a simpler two-dimensional model. Although the loading history of the counterweight contact is very complex during an engine cycle due to gas and dynamic forces, the fretting analysis results in a fretting wear field that corresponds quite well to experimental results. It was also indicated that wear leads to a dramatic decrease of the Findley safety factor due to the re-distribution of critical stresses in the contact interface.

The methods used for the counterweight fretting analysis can be applied to any contact geometries with complex loading histories. The presented work has provided confidence in the methods developed from the fatigue analysis of free surfaces (plain fatigue) and their future applicability to ensure reliable design of high-performance engines that contain heavily loaded contact interfaces.

In the future, the authors will continue their work on the development of fretting analysis methodology with focus on the following topics:

- Verification of wear simulation for other contact geometries and those including complete contacts
- Prediction of crack nucleation at the edge of the complete contact
- Fracture mechanics approach for fretting fatigue prediction
- Investigation of the effect of elastic micro-slip

4. References

1. Aapo Pasanen,Arto Lehtovaara, Roger Rabb, Perttu Riihimäki. Friction behavior of quenched and tempered steel in partial and gross slip conditions in fretting point contact. 13th Nordic Symposium on Tribology. 10-13 June, Tampere, Finland.

2. Abaqus Analysis User's Manual. Abaqus Documentation. Version 6.8.

3. Lönnqvist, C., J. Kaas, and R. Rabb, "Comprehensive Multiaxial Fatigue Analysis with ABAQUS". ABAQUS Users' Conference Proceedings May 22-24, 2007, Paris, France.

4. Rabb, R., and M. Lassus, "Modern multiaxial fatigue analysis in ABAQUS environment. Part 1: Theory". ABAQUS Users' Conference Proceedings May 18-20, 2005, Stockholm, Sweden.

5. Acknowledgment

The first author wishes to thank Mr. Christian Lönnqvist for his great support during the Master's Thesis work and greatly acknowledge Mr. Roger Rabb and Mr. Magnus Lassus for support in the theoretical work and practical application of the finite element method.