

Investigation into the distribution of contact pressure in roller bearings used for high speed automotive applications

Luca Furbatto, Stephen M King

Luca Furbatto – McLaren Racing, Stephen M King – ABAQUS UK Ltd

Abstract: Roller bearings are an integral part of the transmission system of a modern race car and their performance under arduous loading conditions needs to be assessed. The local distribution of contact pressure on the bearing surface can have a significant effect on the wear and durability of component parts, and simulation using finite element analysis methods form an important part of the design process. In this study, ABAQUS/Standard was used to examine the Hertzian contact stress in two-dimensions before extending the study to three-dimensional models. Further studies using a global model revealed the sensitivity of the design to the shape and strength of the bearing surface and to the number and size of rollers used. Detailed three-dimensional studies were also carried out using the sub-modelling technique in ABAQUS/Standard.

Keywords: ABAQUS/Standard, Roller Bearings, Contact Pressure, Hertzian Contact, Sub-Modelling

1. Introduction

The sizing of gearbox bearings has been traditionally approached by using empirical data and guidelines from bearing manufactures derived from Hertzian theory and specific tribology codes.

Given that F1 components have a quite limited life; this approach could produce over-designed solutions that are detrimental in terms of overall packaging and weight.

In this particular case a Finite Element approach has been used to quantify the contact stresses of a roller bearing positioned at one extremity of a transmission shaft.

Firstly a 2-D model has been produced to compare the results with Hertzian theory and quantify the effect of mesh on stress peaks using ABAQUS/Standard.

A fine mesh of 4-node, fully integrated plane strain elements (CPE4R) with particular refinement in the vicinity of the contact patch is used to accurately capture the contact stress. While this level

of mesh refinement is perfectly feasible in a 2-dimensional model, an extension into 3-dimensions rapidly results in impractical finite element models.

Firstly, because of the complexity and large number of contact pairs, a generalized parametric model approach has been used for investigations at 2-D levels. Once the optimum load distribution has been reached between rollers, a 3-D model has been generated in order to capture the contact stress distribution along each needle roller.

Secondly, a coarse 3D model has been generated in order to capture the stiffness of the assembly and the reaction load on each needle roller. A significant number of contact pairs (>200) was required prior to the final run. However this approach produces significantly lower contact stresses.

For analyzing the contact in the 3-D models, a coarse mesh of reduced integration, 8 node solid elements (C3D8R) was used in the first instance. However, although these elements capture the stiffness of the assembly reasonably well, the use of fully integrated, 8-node elements (C3D8) was found to produce better quality contact surface stresses.

Thirdly a detailed 3-D model has been produced using the sub-modelling technique available in ABAQUS: this approach proved significantly more accurate in terms of contact stresses. The sub-modelling technique in ABAQUS is used to study a local part of a model with a refined mesh based on interpolation of the solution from an initial, relatively coarse, global model. It is very useful when it is required to obtain an accurate detailed solution in a local region and the detailed modeling of that local region has negligible effect on the overall solution.

Finally, the effect of shaft bending on contact pressure has been quantified too. This edge effect could not be predicted by the simplistic empirical approach used in the recent past.

It should be noted that all analyses referred to in this paper were completed using ABAQUS version 6.6.

2. Hertzian contact stress on needle roller

The contact stress between two cylindrical bodies is:

$$\sigma_c = \sqrt{\frac{2}{\pi}} \sqrt{\frac{pE_c}{R_r}} \quad (1)$$

$$\text{Where the parameter } E_c = \frac{E_1}{1 - \nu_1^2} + \frac{E_2}{1 - \nu_2^2} \quad (2)$$

accounts for the materials of the contact bodies and R_r is the relative curvature

The relative curvature is calculated as:

$$R_r = \frac{2R_1R_2}{R_1 + R_2} \quad (3)$$

for the case of two cylinders in contact, and as

$$R_r = \frac{2R_1R_2}{R_2 - R_1} \quad R_2 > R_1 \quad (4)$$

for contact between a cylinder and its socket.

The contact pressure distribution for non-conforming contact bodies, that is, cylinder against cylinder is given by:

$$p = \frac{2P}{\pi a} \sqrt{1 - \frac{x^2}{a^2}} \quad (5)$$

where P is the compressive load per unit axial length and a is the radius of the contact area.

The maximum pressure stress on a steel needle roller in contact with a rigid surface is given by:

$$P_o = \left[\frac{PE^*}{\pi R} \right]^{0.5} \quad (6)$$

Where:

P is the force per unit length providing the contact.

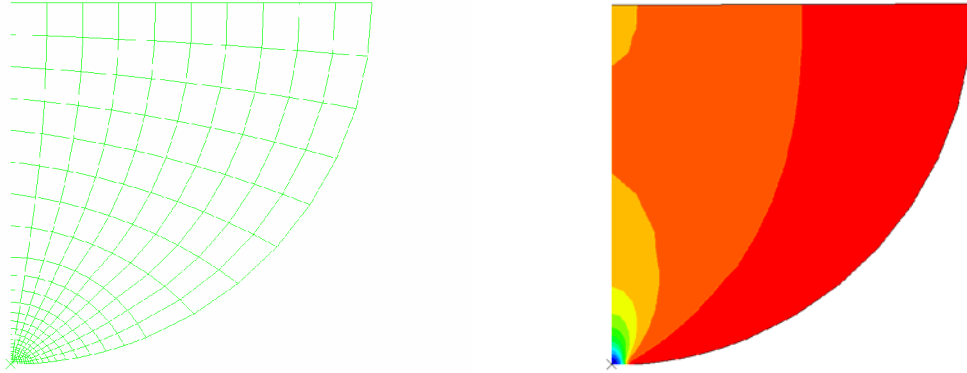
E^* is the effective Young's Modulus and is equal to $E(1-\nu^2)$ for a plane strain simulation.

R is the radius of the needle roller.

Figure 1a illustrates the kind of mesh density required to capture the contact stress in a 2-D, plane strain simulation using ABAQUS/Standard, and figure 1b shows the distribution of vertical direct stress resulting from a line force load of 308 N/mm.

If the radius of the roller is 1.0mm, and the Young's Modulus is 200000 N/mm² with Poisson's Ratio 0.3, then the contact pressure stress is equal to 4,642 N/mm².

An ABAQUS model using 4-node plane strain elements as shown in figure 1 gave a contact stress of 4,605 N/mm², which is in good agreement with the theoretical value.



Figures 1a and 1b - Mesh density required for 2D, plane strain Hertzian contact analysis and distribution of vertical stress

3. 2-D roller bearing analyses

The development of a full 3-dimensional to study the distribution of contact stress was precluded by a study of a typical needle roller bearing assembly in 2-dimensions.

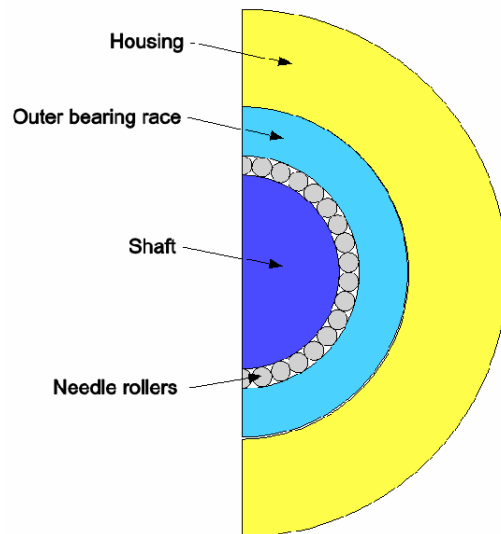


Figure 2 - General arrangement of 2-D bearing assembly

In particular the study concentrated on mesh refinement and mesh quality in order to ensure reliable results at the contact surface. The half symmetric model shown in figure 2 comprises a central, circular post surrounded by N needles, which in turn are enclosed within a cylindrical outer race. The housing, which has a slightly greater diameter than the race, is deflected downwards by a load which creates contact stress for the upper needles at the symmetry boundary and for several needles around the circumference. The mesh density used in this analysis is illustrated in figure 3 and the resulting Von Mises stress distribution is shown in figure 4.

3.1 Contact modelling considerations

The efficiency and quality of an analysis of this type depends on how the contact is modelled, and there are several ways in which this may be done – some more appropriate than others. Convergence and accuracy of results in the vicinity of the contact surface depend on details of how the contact is modelled and what assumptions have been made.

ABAQUS uses the concept of contact pairs to define contact between component parts of a structure. Each pair consists of a master and slave surface where slave nodes are in contact with a master surface. The general rule is that a slave node may not, as default, penetrate a master surface

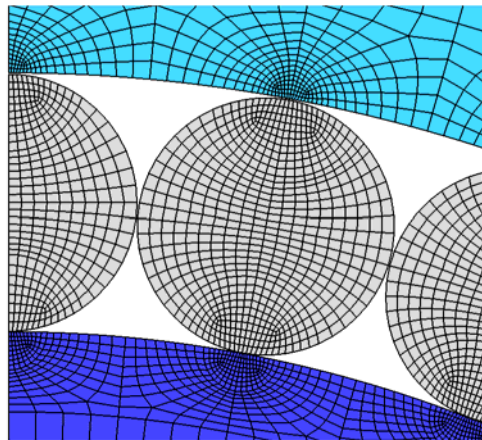


Figure 3 - Mesh density used in 2-D bearing analysis

whereas the opposite is true. This has ramifications on the mesh density used for master and slave surfaces. ABAQUS contact pairs can be defined as ‘small sliding’ or ‘finite sliding’. In the former, the contact direction is set at the start of the analysis and is not updated as the solution proceeds, whereas in finite sliding the contact direction is updated at each iteration. For many Hertzian contact type problems small sliding is appropriate and the solution is more efficient than for finite sliding conditions.

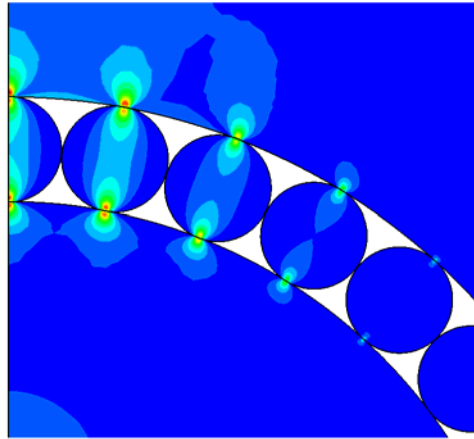


Figure 4 - Von Mises stress distribution around central needles

3.2 Mesh quality considerations

The quality of the mesh in the vicinity of the contact zone can have a profound affect on the smooth variation of the contact force around the circumference of the needle. This can be highlighted by applying a load sufficient to produce significant indentation in the bearing surface. This is illustrated in figure 5, where a mesh created external to ABAQUS, was converted and used in an ABAQUS analysis. Here the presence of interspersed triangular elements close to the contact zone is liable to create a disturbance in the local force field. This combined with the small-sliding approximation and the pure master-slave, node-to-surface contact definition, was the cause of considerable surface penetration for the needle adjacent to the central needle, and this is illustrated in figure 6. Rotation of the adjacent needle was possible whereas the central needle was held in place by the symmetry condition at the boundary.

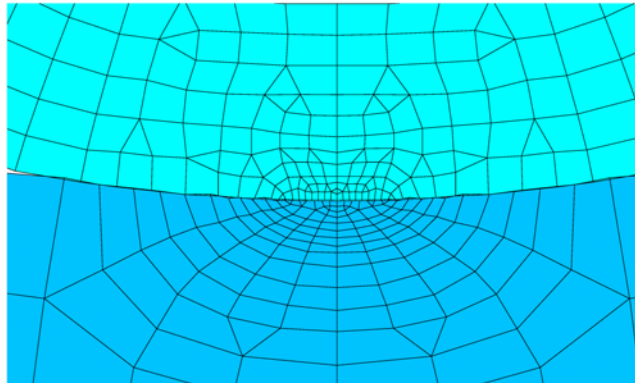


Figure 5 - Surface penetration at central needle

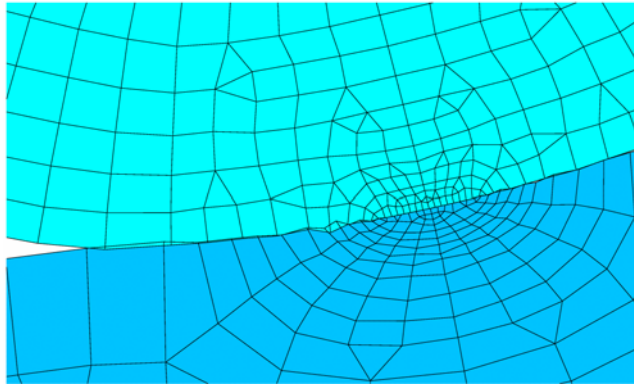


Figure 6 - Surface penetration at adjacent needle

The excessive needle rotation was limited by the use of stabilization forces activated via the ***STATIC, STABILIZE** keyword and the small-sliding approximation was changed to finite-sliding contact. These changes, together with the improved mesh shown in figure 2, produced a significantly smoother distribution of contact pressure at the needle post interface as illustrated in figure 7.

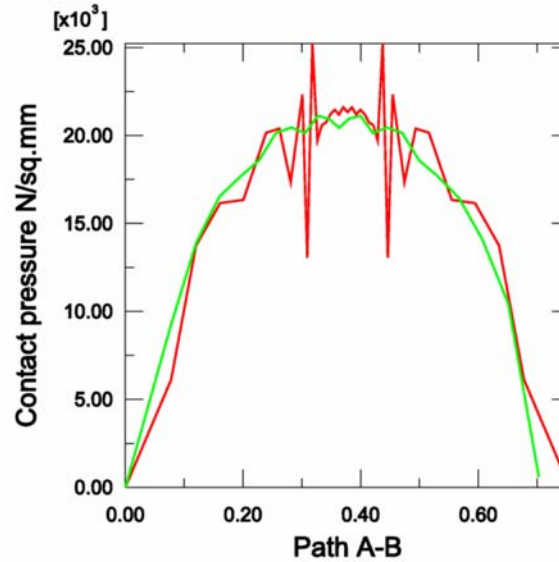


Figure 7 - Contact pressure distribution across contact surface

The jagged trace shown in figure 8 represents the contact pressure along the bearing contact face for the original mesh and the smooth trace represents the contact pressure from the re-modelled mesh using all 4-noded elements and finite sliding. The improvement in the quality of the result is evident with this modeling approach.

The 2-D bearing analysis, using the mesh illustrated in figure 2, was deemed to be producing good quality results based on the modeling principles established in the mesh study. Each needle contained 656 CPE4 plane strain elements and the total number of elements defined in the 2-D model was 25,850.

4. 3-D roller bearing analysis

In order to include 3-dimensional effects into the model, such as bending and shear, to investigate the effect on the distribution of contact pressure, a half-symmetric 3-dimensional model was constructed. However, to enable sufficient detail to be built into the model to retain the quality of the result obtained from the 2-D analysis, would have resulted in an extremely large finite element model of the order of several million elements. A more efficient analysis strategy was to first run a relatively coarse global 3-D model of the bearing assembly and then to utilize the sub-modelling facility to refine the mesh in the desired areas to better capture the distribution of contact stress. The general arrangement of the 3-D bearing assembly model is shown in figure 8.

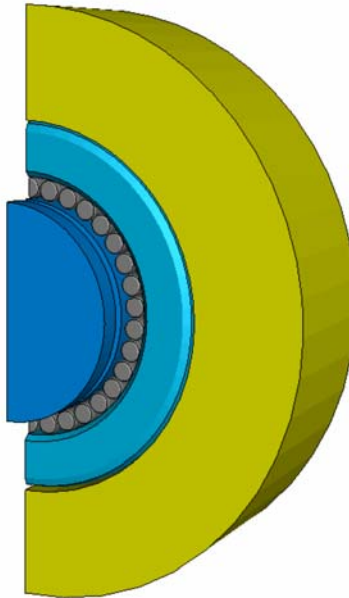


Figure 8 - Half-symmetric model of 3-D bearing assembly

The 3-D bearing model was modeled in a relatively coarse fashion. An example of the mesh around the uppermost central needle is shown in figure 9.

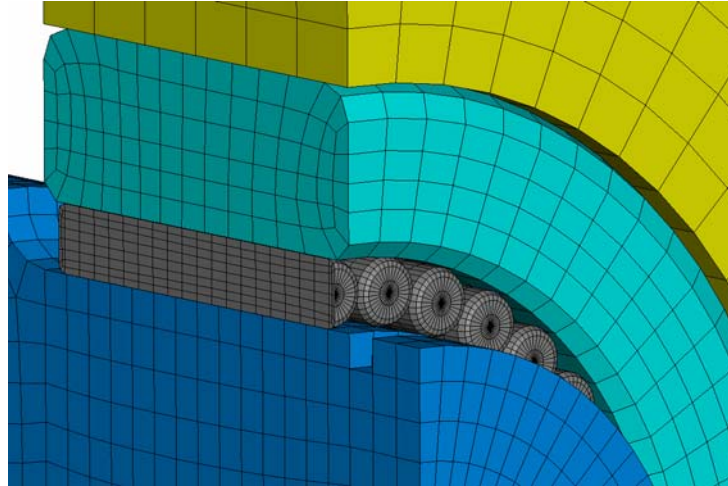


Figure 9 - Mesh detail around uppermost central needle

Sufficient detail was included to enable the use of boundary displacements in driving a sub-model of discrete needles. Even so, this 3-D model contained approximately 60,000 fully integrated C3D8 elements.

In order to make a comparison between the 2-D plane strain analysis and the 3-D analysis, the post was held fixed in the vertical sense at its centre line and an equivalent vertical load was applied to the housing.

The resulting direct stress in the vertical direction is shown in figure 10. This figure contours the results on a cutting plane orthogonal to the needles at approximately half way along the length of a needle. For comparison purposes, the result was taken at the mid point of the needle as the contour averaging process in the 3D model produced values that were twice the mid point value. This was due to only half the contact area being used at the ends of the needle whereas the nodal forces are more or less constant along the length of the needle.

Comparing the mean contact pressure obtained from the coarse 3-D analysis with that from the 2-D analysis, it was observed that it was approximately 60% of the 2-D value. Hence further mesh refinement was required and this could best be achieved by utilizing a sub-model.

The sub-modelling technique in ABAQUS enables a separate, more refined model of part a structure, to be driven by the application of boundary displacements obtained from a more coarse, global model. The application of the boundary displacements is achieved automatically through the use of the /CAE database. In this way, more detail in terms of stresses and strains can be obtained from the sub-model without incurring the penalty of running larger, over-refined global models.

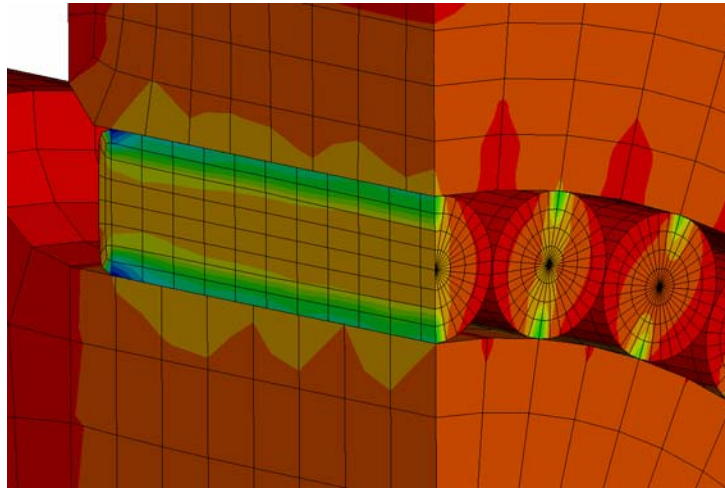


Figure 10 - Vertical direct stress at the central needle

5. 3D Sub-model roller bearing analysis

The sub-model for this analysis consisted of the uppermost central needle, its adjacent needle and half of the needle adjacent to that. This model is shown in figure 11.

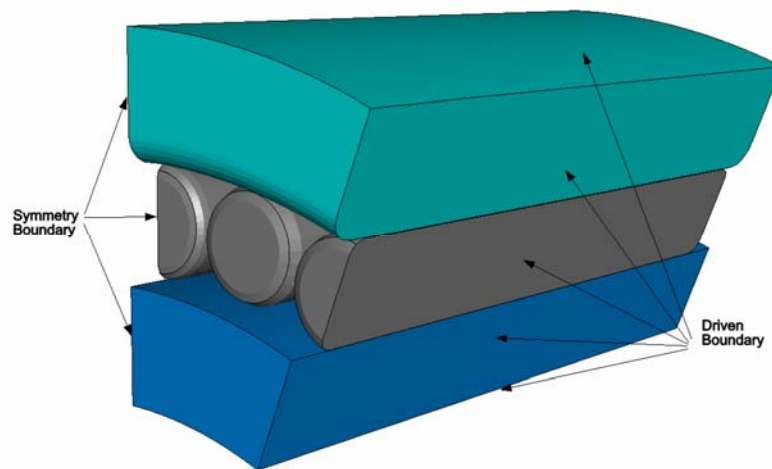


Figure 11 - General arrangement of bearing sub-model

The symmetry boundary and loading remain the same as in the coarse global model, but this time the driven boundary is governed by displacements that were calculated from the global model. The increased mesh refinement of the sub-model is shown in figure 12.

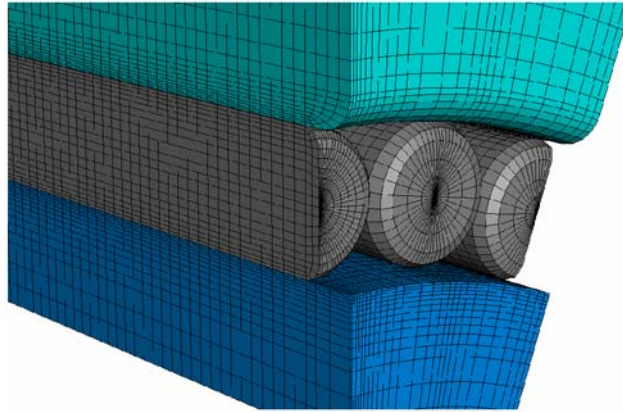


Figure 12 - Mesh refinement used for bearing sub-model

The mesh used for the needles was significantly refined, with a full needle comprising 12,864 elements. Mesh grading was also employed to increase the density in the vicinity of the contact patch. The total model contained approximately 70,000 C3D8 elements which was greater than that used for the complete 3-D model.

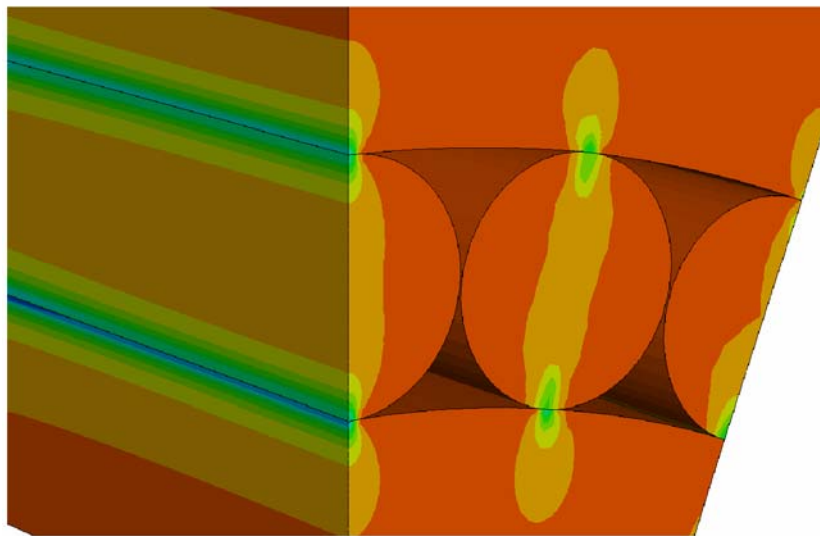


Figure 13 - Contour of vertical direct stress for refined sub-model

As a result of this refinement, the results obtained were more in-line with the 2-D analysis. The vertical direct stress contour shown via a cutting plane mid-way along the needle is shown in figure 13.

Here the general stress patterns in the needles are much closer to those in the 2-D model and the contact pressures recorded on the central needle are up to 80% of those recorded by the 2-D analysis. Although the model is of reasonable size, further mesh refinement could be incorporated without negatively impacting run times too much, and this would provide still more accurate results.

6. Differential contact pressure

The comparative work using the 2-D and 3-D models as described was done in order to enable 3-D analyses involving bending in the bearing to be investigated. In this case, as shown in figure 14, the load is tending to bend the post thereby creating uneven contact pressure on the needles. The remote end of the post is fully fixed and a shear load is applied to the housing. With the central vertical restraint removed on the post, the applied load creates a bending moment which results in a differential contact pressure along the length of the upper needles.

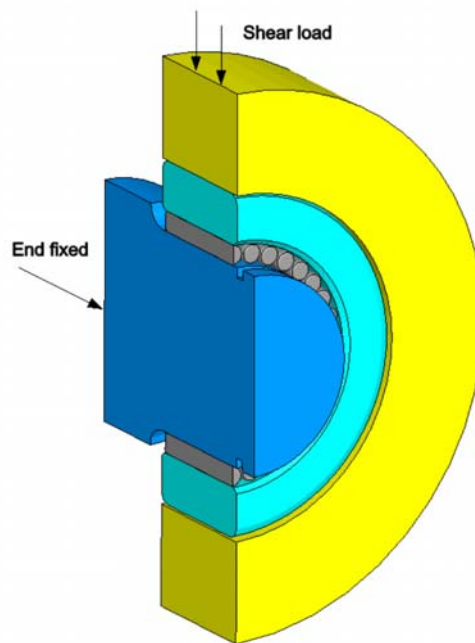


Figure 14 - 3D global model for differential contact pressure

The variation of Von Mises stress in the model under the applied load can be seen in figure 15, with the concentration of stress in the needles located at the shoulder end. Again, the sub-modelling technique can be used to obtain a more accurate distribution of contact pressure in this situation and figure 16 shows this distribution over the central and adjacent needles. Here the results mirror capability in /Viewer has been used to provide a symmetric result. The variation of contact pressure along the needle can clearly be seen with the refined sub-model predicting an increase in value over the coarse 3-D model of approximately 13%.

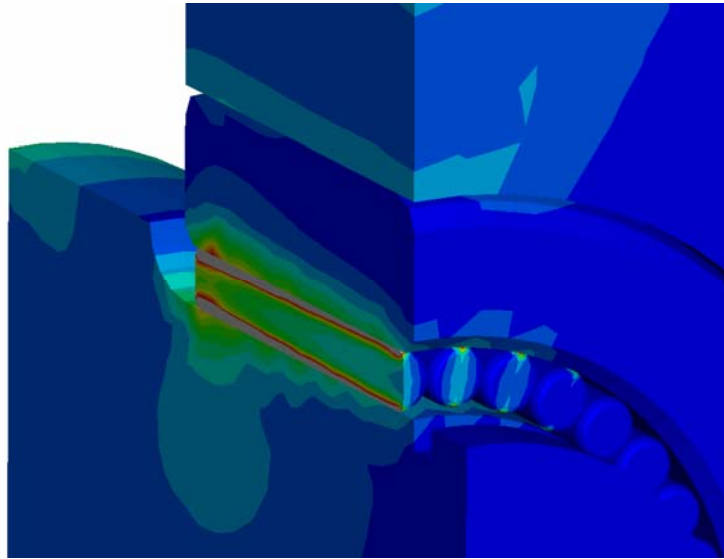


Figure 15 - Distribution of Von Mises stress in upper needle under bending loads

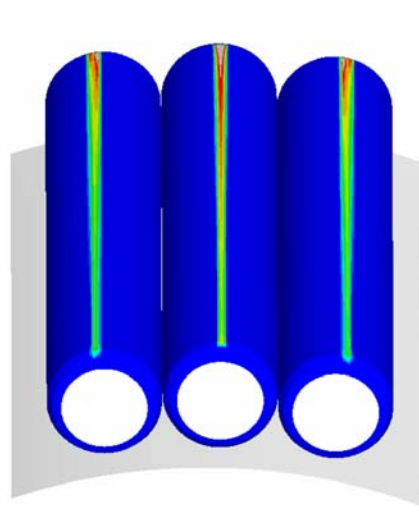


Figure 16 - Distribution of contact pressure in needles under bending loads

7. Conclusions

Contact stresses generated by ABAQUS/Standard have been validated at 2D level. The results showed a degree of mesh dependency and the meshing strategy needed to be refined in order to capture the small contact width of each roller ($<50\mu\text{m}$).

The coarse 3D approach is not really accurate in terms of contact stress recovery but a much finer sub-model proved effective in quantifying realistic contact stresses at 3D level.

Following this activity, the overall design envelop for this bearing was reduced with gains from weight and packaging point of view.

This methodology is now applied to a series of critical bearings, mainly for dyno use, where overload are applied in order to accelerate the fatigue life of the component.

8. Future enhancements

It has been shown that the construction and analysis of models of this type can be made efficient through the use of the sub-modelling technique. However, further improvements and enhancements in ABAQUS will serve to improve the accuracy and efficiency of this type of analysis.

Improvements have been made to the contact algorithm such that surface-to-surface contact with finite sliding is now possible in ABAQUS Version 6.7. This should improve the accuracy of the contact pressure calculation and make it less dependent on mesh refinement.

Improvements to the solver in terms of its processing speed, contact processing, multi-processing capability and a more intelligent implementation of stabilization forces should make obtaining a converged solution in a shorter time a real possibility.

On the model construction side, automatic detection of contact pairs with /CAE is now possible. For roller bearings of this type with a large number of needle rollers, the specification of surfaces and contact pairs was a labour intensive process. The automatic detection of contact surfaces and definition of contact pairs within /CAE has reduced this operation drastically.

9. Acknowledgements

The Authors would like to acknowledge the assistance of Dr. Matthew Knight (ABAQUS UK) in the production of some of the finite element models used in this study.