An Overview of ABAQUS Use in Engine Engineering at Ford Motor Company

Michael DeJack[1], Alex Tang[1], Radwan Hazime[1], Xuming Su[2], and Chulho Yang[3]

[1] Analytical Powertrain, V-Engine Engineering, Ford Motor Company

[2] Materials Science Department, Ford Research Laboratories, Ford Motor Company

[3] Optimal CAE, Inc.

Abstract: The use of durability CAE in engine engineering at Ford Motor Company has grown substantially over the past few years. During this time, ABAQUS use has grown at a comparable rate, as the number of users and the skill of users has significantly increased. ABAQUS has seen a wide variety of applications in helping to understand and solve difficult engineering problems. Several analyses are presented as case studies to illustrate the role played by ABAQUS in engine design: Engine Assembly (Cylinder Bore Distortion and Head Gasket Sealing), Cylinder Head Stress and Fatigue, Engine Block Bulkhead and Main Bearing Cap Stress and Fatigue, Engine Block Assembly Crankshaft Bore Distortion, Exhaust Manifold Stress and Fatigue, and Cylinder Head Residual Stress from Casting Heat Treatment. Software used and critical software needs are briefly discussed with the goal of identifying enhancement opportunities.

1. Ford V-Engine Overview

V-Engine Engineering is the largest product development group within the Powertrain engineering organizations at Ford Motor Company. V-Engine has about 1300 employees located in Dearborn, Michigan and dedicated to design and development of V-Engines for Ford world-wide. V-Engine Engineering's mission is to be a highly motivated supplier of choice of world-class, innovative complete engine solutions that lead in quality, technology, and exceed customer expectations in satisfaction and value. Current products that were developed by V-Engine include the following engines:

- 2.5L & 3.0L 60 Degree V6 – 4-valve Duratec variants
- 3.0L 60 Degree V6 – 2-valve Vulcan
- 3.8L & 4.2L 90 Degree V6 – 2-valve Essex
- 4.0L 90 Degree V6 – 2-valve Cologne
- 4.6L 90 Degree V8 – 2-valve and 4-valve Romeo and Windsor variants
- 5.4L 90 Degree V8 – 2-valve and 4-valve Windsor variants
- 6.8L 90 Degree V10 2 – 2-valve Windsor

More information is available about Ford Motor Company and career opportunities at the following web site: http://ford.com/.

2. Use of CAE in Engine Design

Much of the CAE within V-Engine Engineering is done in the Analytical Powertain (AP) Department. CAE disciplines within Analytical Powertrain include Durability, NVH, Engine System Dynamics, and CFD. Durability related analysis has been a rapidly growing area. Ford Research Laboratories (FRL) also provide some support to AP and other activities within V-Engine through materials related research and CAE.

ABAQUS has been in use for engine engineering since the late 1980s. Since that time, ABAQUS use has grown from a tool used by a couple people to the primary Finite Element solver used by two sections (about 20 people) dedicated to durability related CAE. In addition to the growing number of users, there also has been a significant increase in skill level among the users over the past few years.

Ford has been increasing investment in CAE by increasing headcount and computer related expenses. There is a strong belief that up-front investment in CAE results in design improvements, reduction in late changes due to problems found during hardware testing, and reduction in warranty costs.

ABAQUS Applications include distortion, sealing, stress, and fatigue prediction for engine components and sub-systems. Most resources are devoted to up-front analysis with the purpose of driving design for new product development. Some resources also are applied to fire fighting problems observed during dynamometer tests. The vision is to avoid future fire fighting through effective use of up-front CAE.

3. Case Studies

Several case studies are presented as highlights of engine durability CAE projects at Ford.

3.1 Engine Assembly (Cylinder Bore Distortion and Head Gasket Sealing)

An engine assembly analysis procedure was developed and implemented in all V-engine program developments in Ford. The analysis is used to simulate engine durability test such as Deep Thermal Shock Test and Engine Durability Test with Thermal Overlay. The analysis results provide an overall picture of the engine thermal and structural behavior during the dyno test. Various information maybe extracted or processed from the analytical results including: cylinder head gasket sealing load, cylinder head and block deformation and stress, cylinder bore and valve seat distortion…etc. When more accurate stress calculation is required at critical area, a sub-model analysis may be conducted by using boundary conditions mapped from this analysis results.

Typically the analysis model includes cylinder head, head bolts, head gasket, and cylinder block. Depending on the analysis objective and timing requirements, additional components such as exhaust manifold or front cover may be added. An analysis model is presented in Figure 1. The model has about 230,000 solid elements and 300,000 nodes. Although first order hexahedral elements are preferred, second order tetrahedral elements maybe used to model engine components as illustrated in Figure 1. The cylinder head gasket is modeled by using GK3D8 elements. Contact surface at various locations are modeled which include interference fit between cylinder head and valve seat. Symmetric boundary conditions are used if a single bank of a V-engine is analyzed. The following loading steps are applied to simulate a dyno test:

- Assembly loading: press-fit of valve seat and clamp of bolt
- Thermal loading at each engine thermal overlay condition
- Combustion pressure loading at each firing cylinder

An engine thermal analysis is conducted separately to provide metal temperature distribution. Figure 2 & 3 shows the calculated temperature of cylinder head and block of a 3 - valve engine.

This engine assembly analysis has been conducted to help fix a gasket sealing problem discovered during dyno testing. Figure 4 shows calculated combustion seal line load at various loading steps. Cylinder bore distortion data are typically processed to avoid piston related failure and to minimize oil consumption and blow-by. Figure 5 presents the cylinder bore out-of-roundness data in polar plot form.

3.2 Cylinder Head Combustion Chamber Stress and Fatigue

Due to higher power output and operating temperature in modern engine design, there are increasing concerns on long term durability of cast aluminum cylinder head, especially in the combustion chamber area. AP has worked with FRL and U. of Illinois to develop a thermal-mechanical fatigue analysis process to predict combustion chamber fatigue life. The analysis uses a constitutive material model to represent cast aluminum property. The material model applies unified viscoplasticity theory and takes both temperature and strain rate effects into consideration. This material relation was written as an ABAQUS user subroutine and was included in "Engine Assembly Analysis" to provide more accurate stress/strain calculation.

Cylinder head stresses and strains are derived from the "Engine Assembly Analysis" and then processed by using a thermo-mechanical fatigue model (a damage accumulation model including fatigue, oxidation and fatigue damage) for combustion chamber fatigue life calculation. The predictions of thermal strains and fatigue life based on FEA analysis were compared with measurements from cylinder head Key-Life-Test. The predictions of TMF life are within 30% error of actual test data. Figure 6 shows stress/strain relationship of a cast aluminum calculated by using the unified material model. Figure 7 shows the calculated cylinder head fatigue life for a cylinder head Key-Life-Test with a certain thermal overlay condition.

3.3 Engine Block Bulkhead & Main Bearing Cap Stress and Fatigue

The cylinder block primary structure between the cylinder bores and the crankshaft main bearing is referred to as the bulkhead. The bulkhead is subjected to loads from assembly, hot operating conditions, and engine firing and inertial loads. These firing and inertial loads are transmitted through a crankshaft that is contained in the main bearing with oil lubrication. Fatigue under high cycle operating loads is a primary concern – and is evaluated with a high load / high cycle engine durability test and simulated with this analysis.

Typically, a 3d model is used, composed of a slice of the cylinder block with the main bearing cap, main bearings, all critical fasteners, and a simplified representation of the cylinder head – as illustrated in Figure 8. Either second order tetrahedral elements (C3D10 and C3D10M) or primarily first order hexahedral elements (C3D8I) are used in these models. Model sizes typically range from 100,000 nodes to 250,000 nodes – depending on the element type and whether or not sub-models are utilized in areas of stress concentration. Contact surfaces between the main bearing cap, bulkhead, and the main bearing are modeled. Interference fits with the main bearing and main bearing cap are simulated at these contact surfaces. A static stress analysis is performed with the following typical loads:

- Assembly loading sequence, including interference fits with bearing cap and main bearing.
- Thermal loading at operating condition.
- Crankshaft loading, including peak firing loads and other dynamic loading of interest.

Material behavior is usually modeled as elastic-plastic with an isotropic hardening model. Substantial plastic deformation occurs in the thread areas and sometimes at other stress concentrations. An example stress contour with maximum crankshaft loading is shown in Figure 9. A cross section, exposing the stresses in the thread areas is shown in Figure 10.

Higher stresses are observed in the main bearing cap than in the bulkhead, but modern designs employ higher strength cast or formed iron in bearing caps and often lower strength cast aluminum in the cylinder block. One must consider the material strength in addition to the operating stresses in order to evaluate the durability of the design. FEMFAT is used at Ford powertrain organizations to evaluate high cycle fatigue safety. Load cases which correspond to crankshaft induced loads are evaluated. FEMFAT considers multi-axial stresses, mean stress influence, and stress gradient influence (notch sensitivity effect) in the fatigue prediction. Fatigue safety factor contours are shown for the bulkhead in Figure 11 and the bearing cap in Figure 12.

Often, the thread engagement area in the bulkhead is a fatigue concern, as fatigue cracking has been experienced in certain cases on dynamometer tests. Interaction between the main bearing cap bolts and block threads are typically simulated with a simplified modeling approach involving constraint equations in the bulkhead model. In special circumstances, a more precise modeling approach has been used with sub-models, as shown in Figure 13. This modeling approach significantly increases complexity of the model, because the thread geometry is

ネジ山の塑性を考慮した
接触応力と疲労寿命解析

represented and a very fine mesh density is required to predict the local thread stresses. The analysis is further complicated by the large amount of contact along the length of the thread engagement area – causing difficulties in defining contact surfaces and convergence problems. Also, a significant amount of plasticity develops along the thread contact and root areas. An example stress contour in the thread area of a detailed sub-model is shown in Figure 14, and a fatigue safety contour is shown in Figure 15. Detailed thread models such as these are considered necessary to improve the accuracy of thread fatigue prediction, but common use of such models will not be realized until further advances are made to rapidly construct and solve such models.

3.4 Engine Block Assembly Crankshaft Bore Distortion

Crankshaft main bearing bore distortion is another area of interest where ABAQUS is the tool of choice. It is necessary to maintain a nearly perfect cylindrical bore with a well-controlled clearance between the crankshaft and main bearings. The manufacturing process is designed to control this distortion through the following process:

- Assemble the main bearing caps without main bearings and line bore the block.
- Disassemble the main bearing caps.
- Assemble main bearing caps with main beaings and crankshaft.

Some distortion occurs during the reassembly process due to the bearing interferences and other assembly loads. Additional distortion occurs during engine operating conditions due to thermal and firing loads. For aluminum blocks (in contrast with iron blocks), the distortion is greater under assembly conditions and increases substantially under thermal loading. At operating temperatures, bearing clearances may increase substantially due to differential thermal expansion -- potentially causing oil supply and aeration problems.

A bulkhead sliced model such as the one used for bulkhead stress and fatigue analysis may be employed to predict these distortions – as shown in Figure 8. The model content is the same as described in the previous section. Comparisons have been made with a full block model and with measurements to confirm that the boundary conditions used in the sliced bulkhead model are reasonable. If main bore alignment along the length of the block or bore-to-bore variation is of concern, then a full block model is required. The model shown in Figure 16 was used to characterize the primary distortion from assembly and operating loads. Two different analyses were run to determine the compensation from machining and the distortions from assembly and operating conditions.

Analysis 1: Simulate Machining Process

A perfectly round bore in the free condition is assumed as the initial shape in this model. The main bearing cap is assembled (without bearings) and the cap bolt loads are applied. The distortion from bearing cap assembly is measured as the change in displacement relative to initial condition. Rigid body motion is removed when calculating distortion. The distortion calculated here is removed during the boring operation and is considered as a correction to the results in the second analysis.

Analysis 2: Simulate Assembly Process and Operating Conditions

Again, a perfectly round bore in the free condition is assumed as the initial shape. The main bearing cap is assembled with the bearings in place this time.

The engine assembly process is simulated using the following loading sequence:

1. Main bearing cap assembly with bolt loads and bearing shell interference.
2. Head bolt loading.

Engine operating loads are applied:

3. Thermal loading.
4. Maximum crankshaft loading (firing load).

Distortion is evaluated after the main bearing cap assembly, head bolt loading, thermal loading, and maximum firing loading. Rigid body motion at the crankshaft bore is removed to quantify the distortion.

Mathematical description of crankshaft bore deformation process

A simple Matlab M-file was written to view deformations of the crankshaft bore. A rigid body motion is removed whenever distortion is calculated. Simple mathematical relations to calculate distortions are described as follows.

Step 1: Obtain distortion corrected by boring operation.

$$\mathbf{U}_{machining} = \mathbf{x^*} - \mathbf{x_0}$$

where, $\mathbf{x^*}$: coordinate of deformed bore at cap-assembly step w/o bearing shell

$\mathbf{x_0}$: coordinate of un-deformed bore

Step 2: Obtain distortion with bearing shell assembled from

Cap-assembly : $\mathbf{U}_{Cap\text{-}Assy} = \mathbf{x^*}_{Cap\text{-}Assy} - \mathbf{x_0}$

Head bolt assembly : $\mathbf{U}_{Headbolt\ Assy} = \mathbf{x^*}_{Headbolt\text{-}Assy} - \mathbf{x_0}$

Thermal loading : $\mathbf{U}_{Thermal} = \mathbf{x^*}_{Thermal} - \mathbf{x_0}$

Step 3: Calculate corrected distortion by $\mathbf{U}_{machining}$

$$\mathbf{U'}_{Cap\text{-}Assy} = \mathbf{U}_{Cap\text{-}Assy} - \mathbf{U}_{machining}, \quad \mathbf{U'}_{Headbolt\ Assy} = \mathbf{U}_{Headbolt\ Assy} - \mathbf{U}_{machining}, \quad \mathbf{U'}_{Thermal} = \mathbf{U}_{Thermal} - \mathbf{U}_{machining}$$

Step 4: Calculate shape of distorted bore by

$$\mathbf{X} = \mathbf{x_0} + \mathbf{U'}$$

where, $\mathbf{X}$: coordinate of distorted bore

Examples of the crankshaft bore distortion are shown in Figures 17 and 18.

3.5 Exhaust Manifold Stress & Fatigue

The exhaust manifold is one of the engine components that undergoes severe thermal cycling. The manifold is exposed to high temperature exhaust gases and it is only cooled thru conduction to the cylinder via thermal contact and natural convection to its surroundings. ABAQUS represents a great tool to model the thermal cycling that this components experiences during its key life testing and at the hands of the customer.

By perform transient heat transfer analysis using transient boundary conditions the nodal temperatures are used as thermal loading for subsequent nonlinear structural analysis. The film coefficients and sink temperatures are calculated using a internal Ford software. The heat transfer analysis involves forced convections on the internal surface of the manifold as well as natural convection on the outside surface. Cooling thru the cylinder head is modeled with the help of thermal contact surfaces. The exact value of the GAP CONDUCTANCE for a specific gasket material and thickness is determined experimentally. The cylinder head is approximated using a block of aluminum with specified temperature history at the bottom nodes to simulate cylinder head cooling. Fig. 19 shows the FEA model usually consists of first order hexahedral elements with approximately 20,000 nodes. The use of hexahedral elements provides reasonably accurate results with minimum CPU time. Fig. 20 shows the maximum temperature distribution of a V-6 high silicon molybdenum exhaust manifold simulating the end of a heat up cycle during key life testing.

ABAQUS built-in material models such as time hardening creep models and isotropic and kinematic hardening plasticity models allow for accurate computation of the inelastic strains that the manifold experiences. Contact is simulated between all mating surfaces (gasket/head, gasket/inlet flanges, studs/stud holes, nuts/flanges). Adding and removing contact surfaces and friction changes during the analysis allows for fast and efficient computation time. The transient structural analysis consists of several steps. First the assembly load is applied, then the nodal temperatures are imposed to simulate heat up and cool down of the manifold. Usually few cycles are sufficient to approximate the TMF life of a manifold. Fig. 21 shows the equivalent creep strain increment that is present in the manifold during the 3rd cycle of a key life test. The entire strain history is written to the .odb file which is later used by another program to perform low cycle fatigue calculations and get life estimates for the component.

In addition to the TMF life, the sealing contact pressure on the gasket is calculated to investigate possible leak locations during engine operation. The GK3D8 are used to model the gasket bead and body. Fig. 22 shows the sealing pressure on a stainless steel gasket used in the model of Fig. 19. It is interesting to notice that the clamping loads of the fasteners and the contact pressure on the gasket bead vary significantly during the thermal cycle.

In other applications, the time independent plastic strain becomes an important factor in computing the thermo-mechanical fatigue life. Fig. 23 shows the equivalent time independent plastic strain contours at the internal wall of a V-8 high silicon molybdenum exhaust manifold. Fig. 24 also shows the accumulated equivalent creep strain contours at the same location. Both creep and plastic strain contribute to failure in this instance.

3.6 Cylinder Head Residual Stress from Casting Heat Treatment

Cast aluminum engine blocks and heads are increasingly used at Ford Motor Company. To improve their mechanical properties, these components are usually heat-treated after the casting process. Ford Research Laboratories has developed material models and CAE methodologies to predict the residual stresses from the heat treatment. This analysis is mainly focused on residual stress caused by the transient temperature field during quench stage.

During quench, a cylinder head being solution treated at 495°C is plunged and submerged into water, which is heated to certain temperature less than 100°C. The transient temperature field of the cylinder head during quench is non-uniform, with large temperature gradients. The temperature gradients introduce non-uniform thermal expansion and thus plastic deformation. The plastic deformation is again non-uniform and causes inherent mismatch of local material across different regions of the cylinder head. Residual stresses are required in order to stabilize these internal stresses, and these residual stresses remain after the component cools and the temperature becomes uniform again.

Different heat transfer mechanisms, including film boiling, nucleate boiling, and convective heat transfer, occur very fast one after another during quench. The complicated geometry of a cylinder head and possible left over sand from casting makes the heat transfer analysis even more challenging. Our residual stress analysis consists of a sequential thermal mechanical analysis. The coupling of thermal and mechanical energy is neglected. An inverse modeling is used to get optimized heat transfer coefficients for different surfaces of the engine components. This inverse modeling is based on the temperature measurement at various points of the engine components. The heat transfer coefficients are temperature dependent. ABAQUS heat transfer process is used to get the transient temperature field with boundary conditions specified by using FILM. Figure 25 shows the temperature distribution from such an analysis. Apparently, the temperature is quite non-uniform and the valve-train side cools down much faster than the combustion chamber side. This would lead to tensile stress in the combustion chamber area.

Stress analysis is performed as a static analysis. Since material properties are both strain rate and temperature dependent, visco-plastic constitutive relation should be used. A UMAT was written to accurately describe the material based on a constitutive relation developed by joined effort of Ford Motor Company and University of Illinois [1]. Isotropic and kinematic hardening plasticity models have also been tried to speed up computation. To account for strain rate effects when isotropic or kinematic hardening models are used, a field variable is introduced to represent strain rate and a user subroutine used to define it.

Figure 26 and 27 shows predicted residual stresses throughout the cylinder head. Figure 26 shows the predicted residual stress on the combustion surface and Figure 27 is a cut away view of stress at the combustion side of the water jacket. Consistent with the temperature distribution as shown in Figure 25, tensile residual stress developed in the combustion chamber section of the cylinder head. For the example shown in Figure 25 - 27, the cylinder head was quenched with sand packed in the water jacket area to artificially increase the residual stress. Possible stress relaxation by aging was not included. Corresponding residual stress measurement has shown the predictions match reasonably well with experimental data. Efforts are being made in Ford Motor Company to increase the accuracy of predictions of residual stress to include aging and machining processes using ABAQUS.

4. Software Used and Critical Needs

Current commonly used CAD / CAE tools at V-Engine that are used for various ABAQUS analyses will be briefly described. These tools are commercial tools and in-house codes. Commercial tools include I-DEAS, Hypermesh, and FEMFAT. In-house codes are used for heat transfer analysis as well as to evaluate bore distortion and thermal mechanical fatigue.

Critical needs are described with the goal of identifying enhancement opportunities for ABAQUS. The following critical needs are considered necessary to realize efficiency improvements at Ford as well as for ABAQUS to remain competitive with other commercial Finite Element codes.

- Speed improvement for contact and metal plasticity problems.

  Most problems where ABAQUS is employed involve contact and/or plasticity with model sizes ranging from 50,000 to 500,000 nodes. Solution times vary between ½ day to several days on a multi processor supercomputer. Improvements in convergence rate and problem solution time are desired.

- Improvement on convergence rate for elastic-plastic type GASKET elements.

  Users have encountered convergence problems from time to time when using the elastic-plastic type of GASKET elements. These are typically very large engine assembly models, and the solution time varies between a few days to occasionally a couple weeks. Sometimes convergence cannot be achieved until the user changes to a damage type of GASKET element. Solution time and convergence robustness needs substantial improvement.

- Parallel solution in distributed computing environment.

  The capability to run moderate size to large jobs efficiently with unix type workstations in a large network environment is needed. The network is composed primarily of single cpu workstations, but typical model sizes prevent reasonable run times with the current version of ABAQUS. Efficient run times in this environment would require distributing single jobs across multiple workstations. Data transfer to solve such problems across a network environment needs to be minimized. This need has developed as a result of expanding work load and limited supercomputer resources.

- Automatic contact surface determination.

  This is necessary to realize efficiency gains and is a requirement in order to routinely undertake more complicated contact problems (e.g., thread contact in detailed thread models).

- Improved interfaces of Hypermesh and I-DEAS to ABAQUS.

  The capability is needed to read in and output equivalent ABAQUS .inp files to / from Hypermesh. While much progress has been made, various ABAQUS features still are not supported, and manual work (editing ABAQUS input files) is required to construct analysis decks when building new models or modifying models. Also, since Hypermesh is the most commonly used tool for pre and post processing CAE in V-Engine, more comprehensive support for post processing ABAQUS results in Hypermesh is needed.

- Improved interfaces of ABAQUS Viewer to ancillary software.

  The authors appreciate the increasing value of ABAQUS Viewer, as its capability has grown to complement and in some ways surpass that of other post processing tools. Some obstacles that reduce the use of Viewer include the slow speed with large models, the large file size associated with the .odb database, and the lack of support available from other software packages (e.g., FEMFAT). It would be helpful to be able to save settings in Viewer without having to run scripts. Additional math functions should be included for operating on field output. Also, read and write access to the .odb file is not straight forward, as the Python language is not widely used in the engineering community. It would be helpful if there were an easier interface with the .odb database. Several more detailed examples of scripts that read filed output variables and write user defined variables to the .odb file would be very helpful.

5. Conclusions

ABAQUS use has grown drastically over the past few years in V-engine engineering at Ford Motor Company. This trend is expected to continue as long as ABAQUS remains competitive with important features and solution speed. The primary use of CAE is to drive design decisions up front in the product development process. The growing demand for CAE has driven more use of analytical DOE type studies, further increasing the use of ABAQUS. Also, CAE has advanced to a level of performing "virtual testing" through increasingly accurate simulations of hardware tests. The virtual testing is an effort to avoid problems found much later in the product development process with conventional hardware testing, but such models are very large and require substantial computer resources to solve. The increasing demand for CAE has begun to overwhelm limited supercomputer availability – and improvements in ABAQUS solution time and smarter use of available network computer resources (i.e., solving ABAQUS jobs in a distributed computing environment) will be necessary to sustain the demand. Several enhancement areas for efficiency improvement are suggested for consideration in future software releases.

Figures

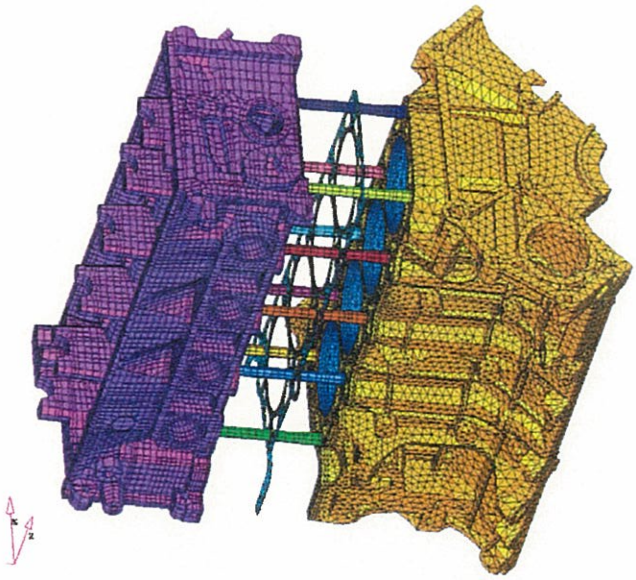


Figure 1: Engine assembly analysis model.

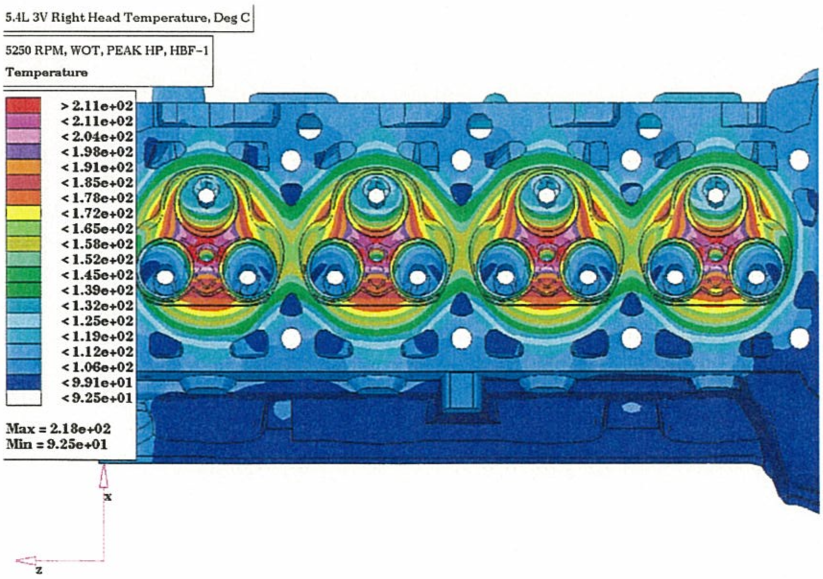


Figure 2: Cylinder head temperature distribution.

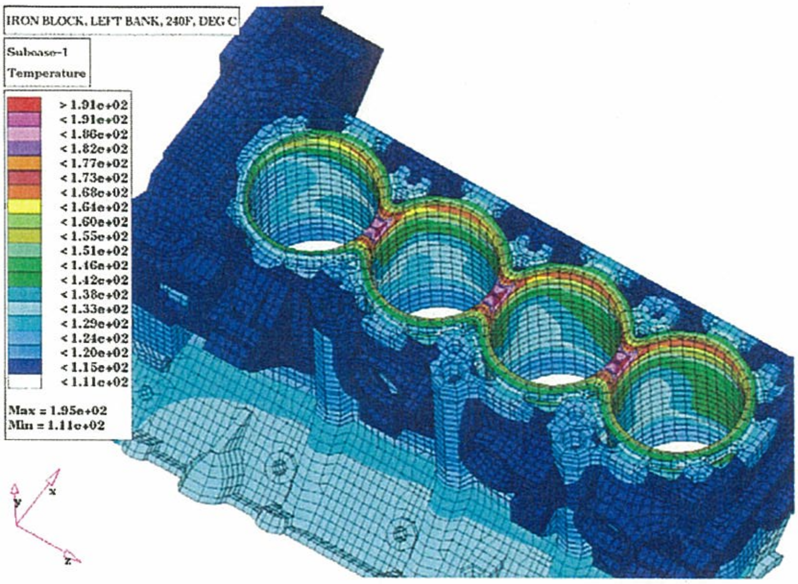


Figure 3: Cylinder block temperature distribution.

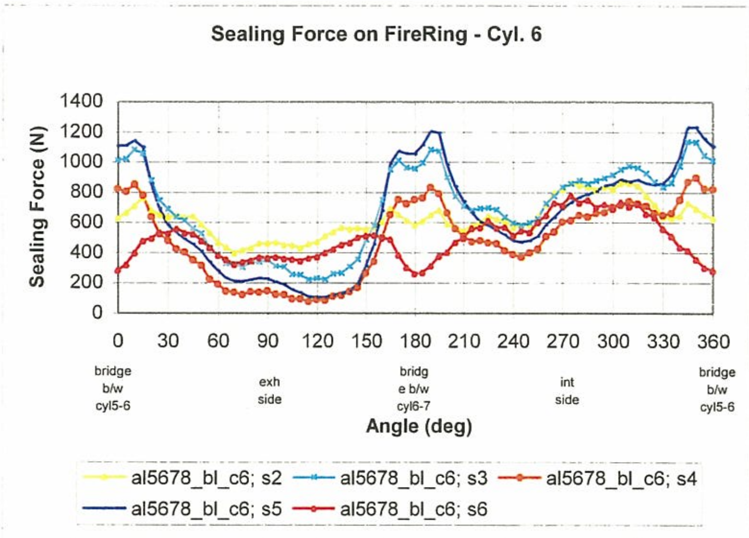


Figure 4: Head gasket combustion ring sealing load distribution.

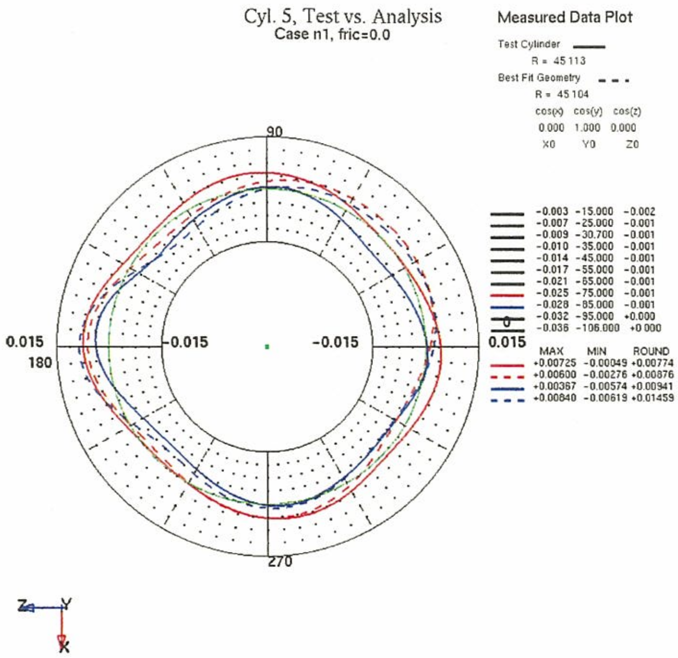


Figure 5: Cylinder bore distortion data.

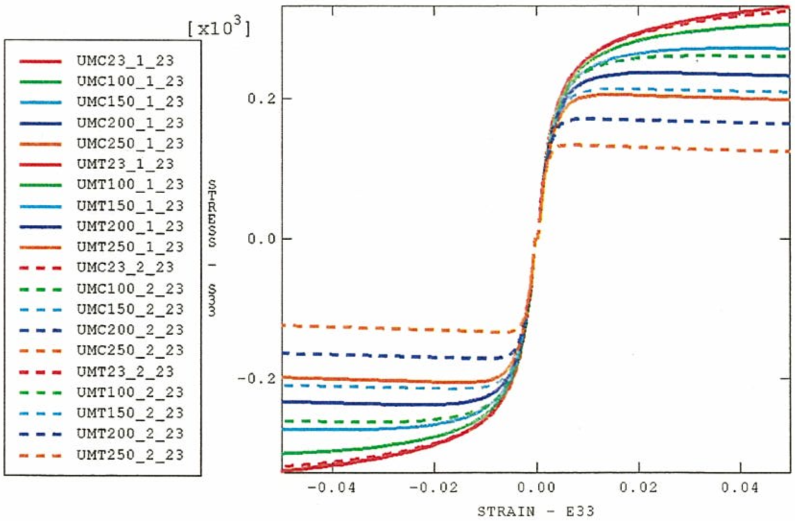


Figure 6: Calculated stress-strain relation for cast aluminum.

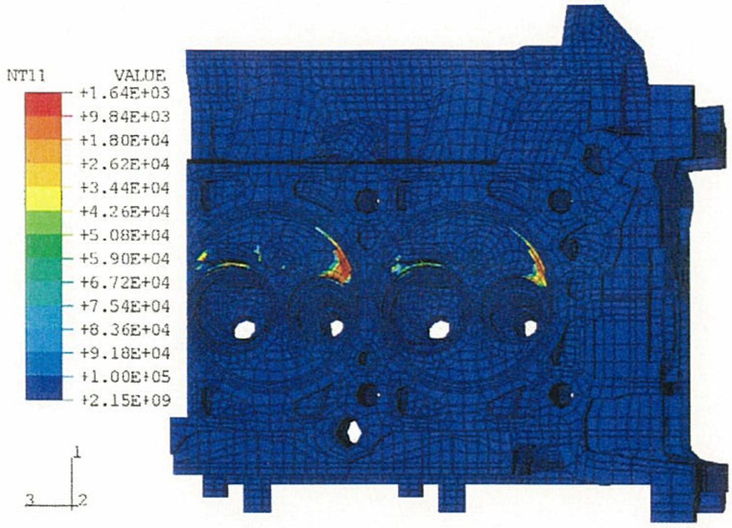


Figure 7. Fatigue life contour for TFT test at 230°C temperature profile.

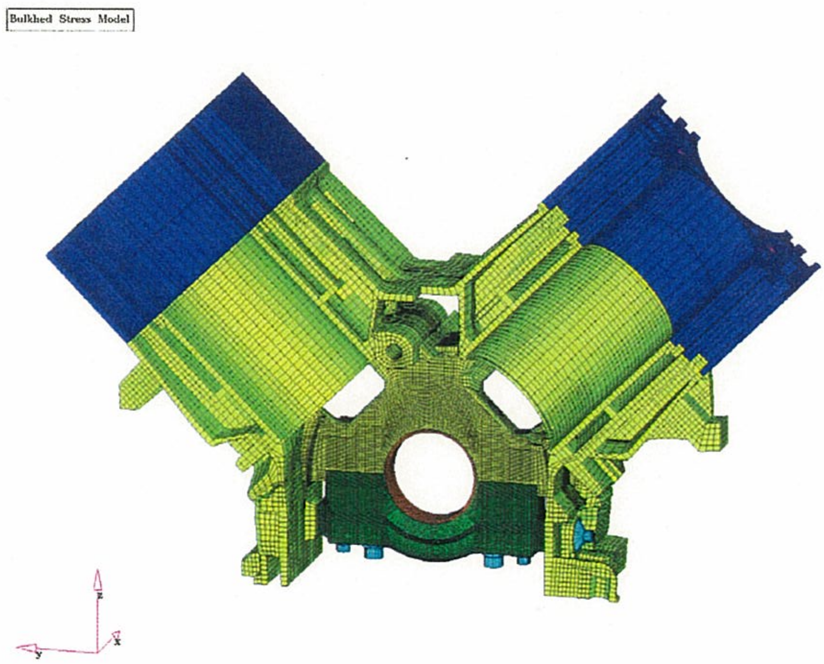


Figure 8. Bulkhead and bearing cap stress model.

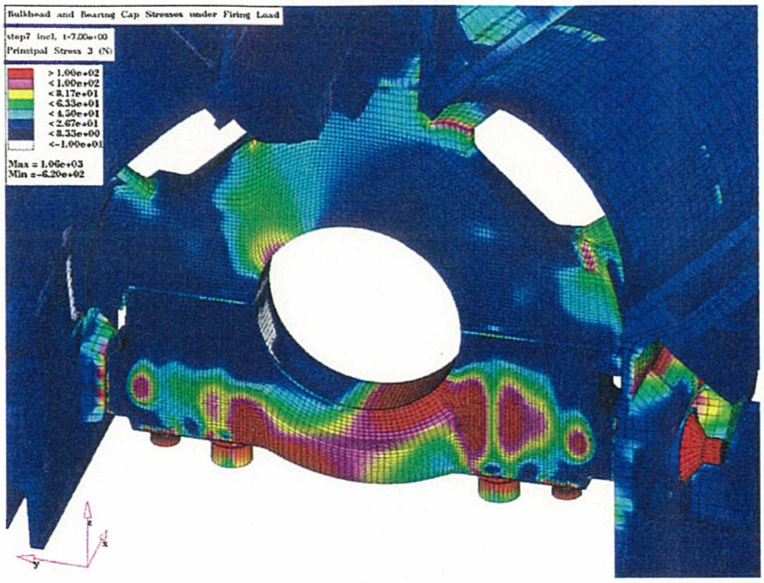


Figure 9. Stress contour on bulkhead and bearing cap from assembly, thermal, and firing loads.

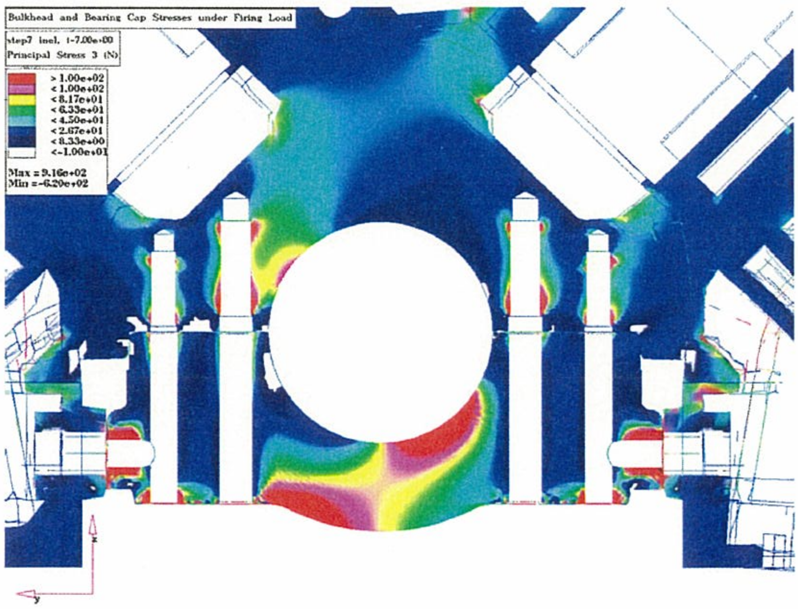


Figure 10. Stress contour of bulkhead cross section from assembly, thermal, and firing loads.

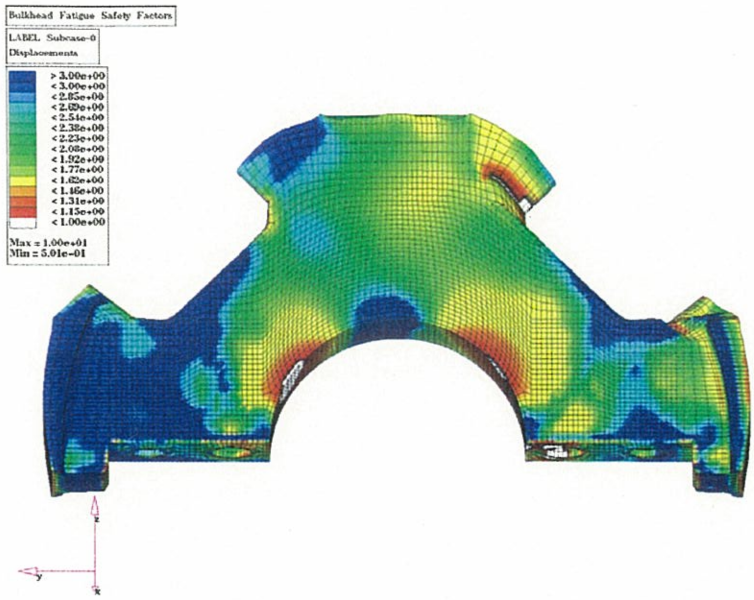


Figure 11. Fatigue safety factor contour on bulkhead from operating loads.

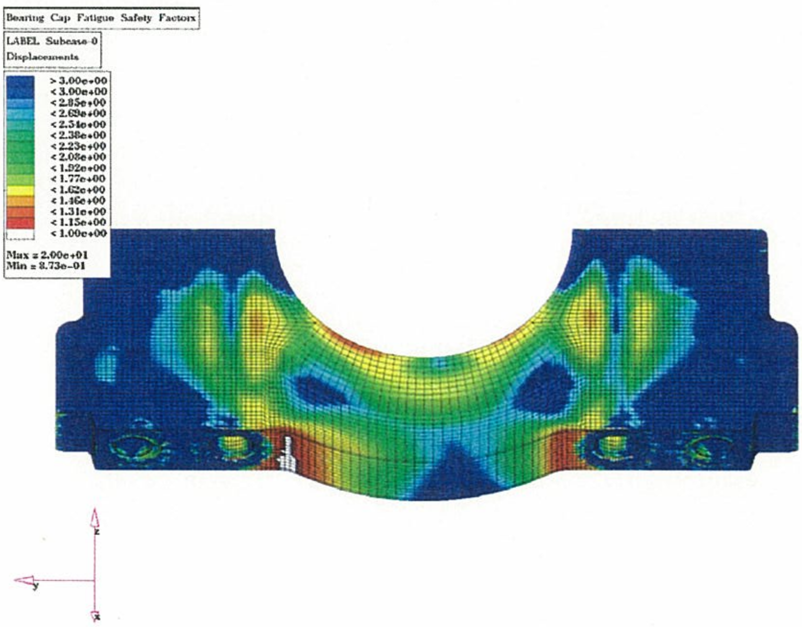


Figure 12. Fatigue safety factor contour on bearing cap from operating loads.

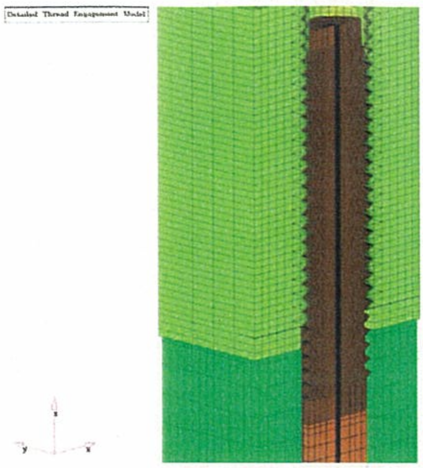


Figure 13. Detailed thread engagement sub model with a cross section view, exposing the bolt and thread contact surfaces.

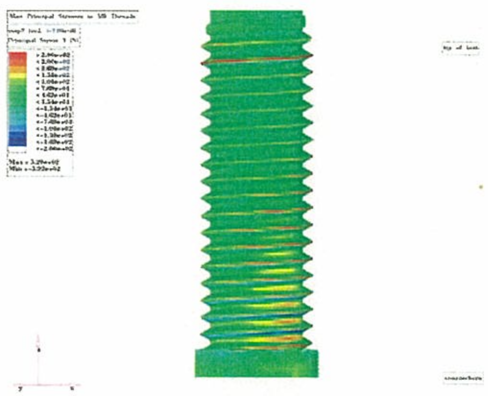


Figure 14. Stress contour on cylinder block cut threads.

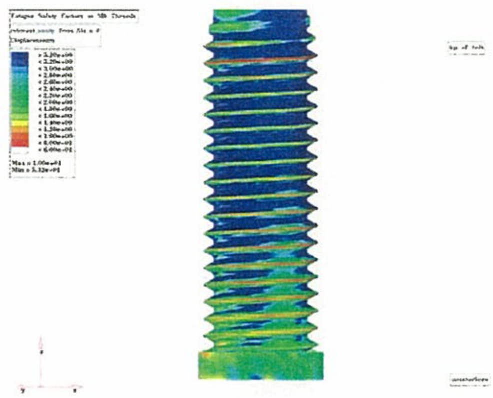


Figure 15. Fatigue safety factor contour on cylinder block cut threads.

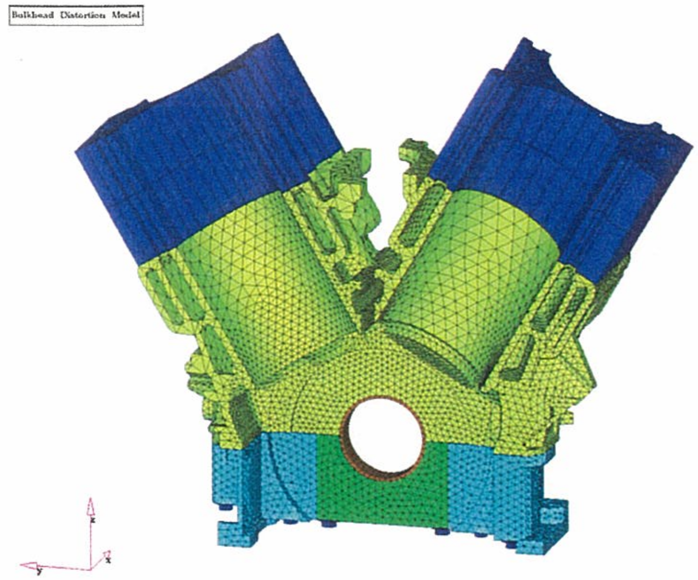
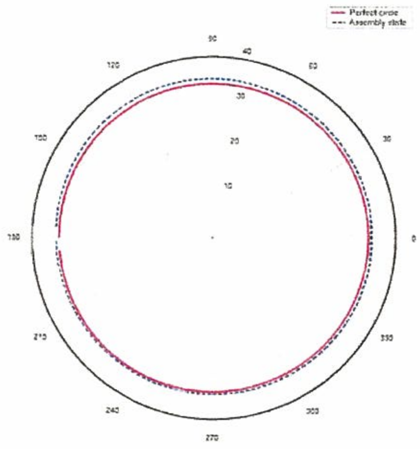


Figure 16. Bulkhead model used to predict crankshaft bore distortion.



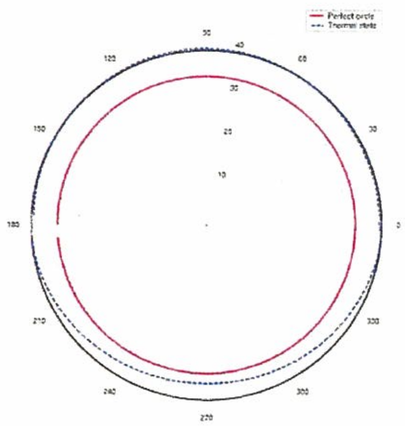


Figure 17. Distortion (blue dash) after main bearing cap assembly.

Figure 18. Distortion (blue dash) after head bolt and thermal loads.

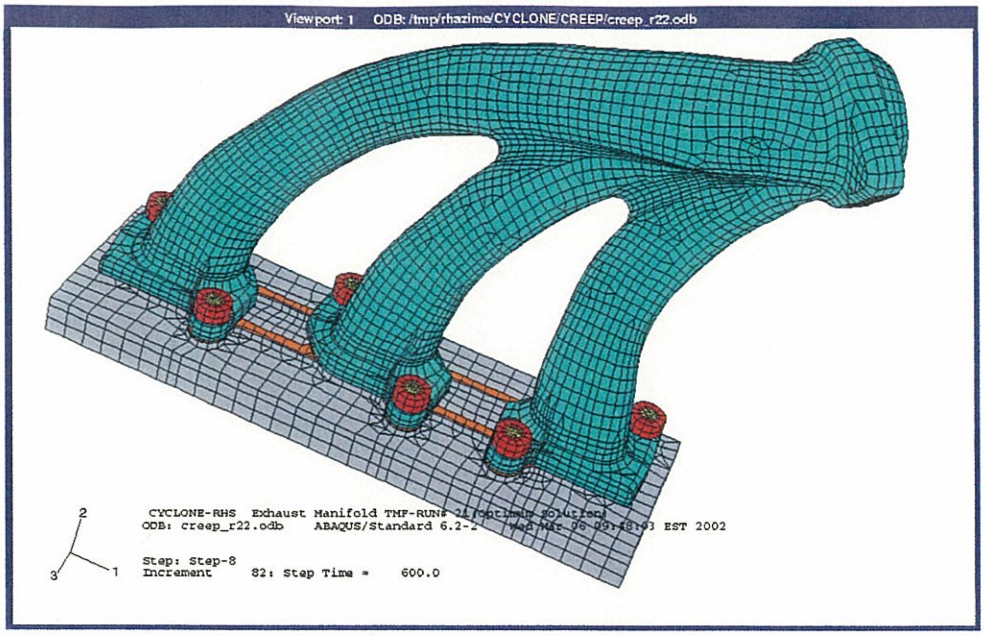


Figure 19. Finite element model showing the exhaust manifold along with the studs nuts and head

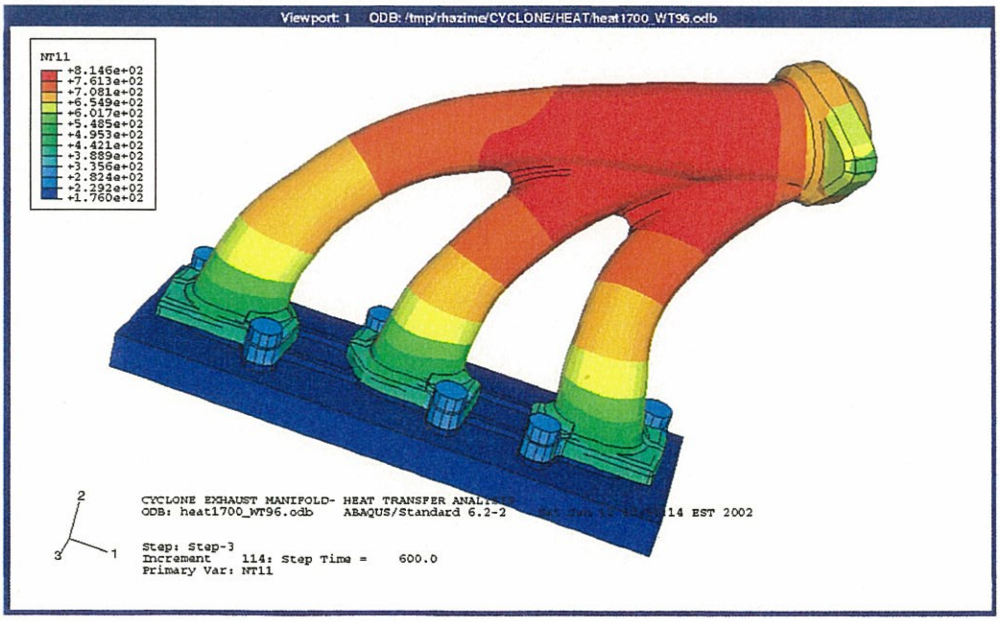


Fig. 20. Temperature contours showing the maximum temperature distribution at the end of a heat-up cycle.

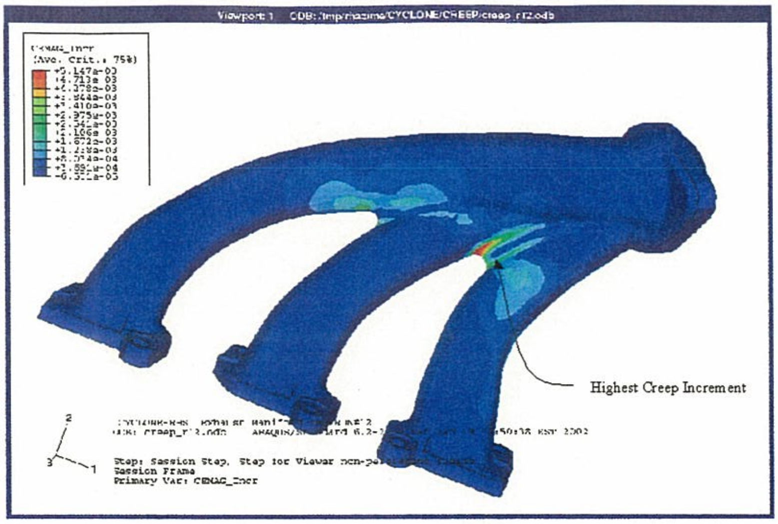


Fig. 21. Equivalent creep strain contours showing the creep strain increment during a complete heat-up and cool-down cycle.

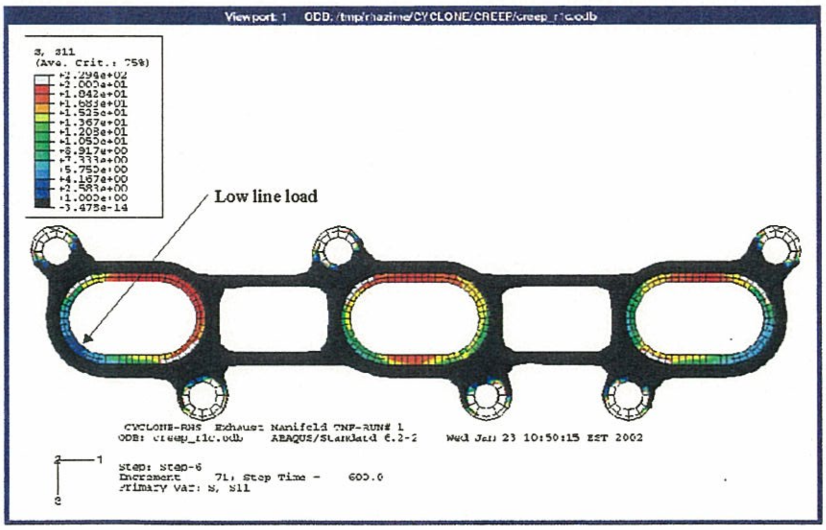


Fig. 22. Contact pressure distribution for the gasket bead and body showing a significant variation of the contact pressure along the bead length

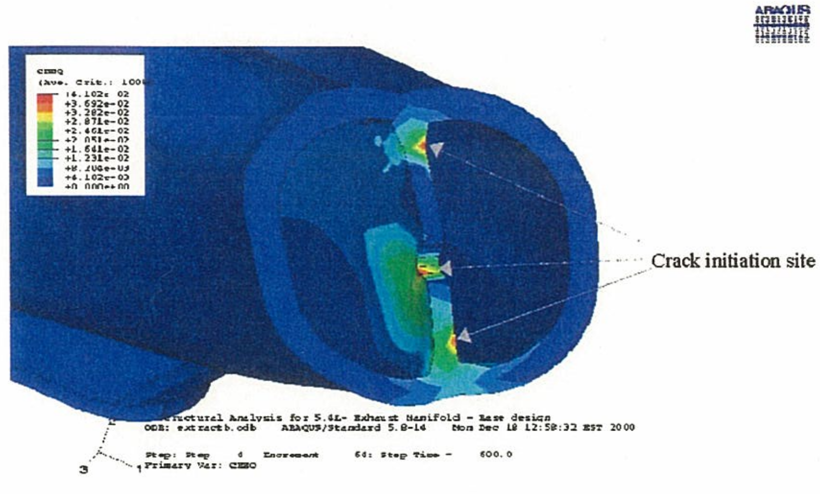


Fig. 23. Accumulated equivalent creep strain contours at failure location for a V-8

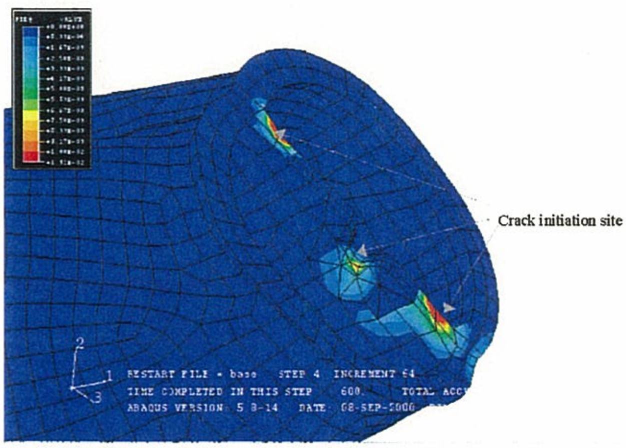


Fig. 24. Accumulated equivalent plastic stain contour at failure location V-8 manifold.

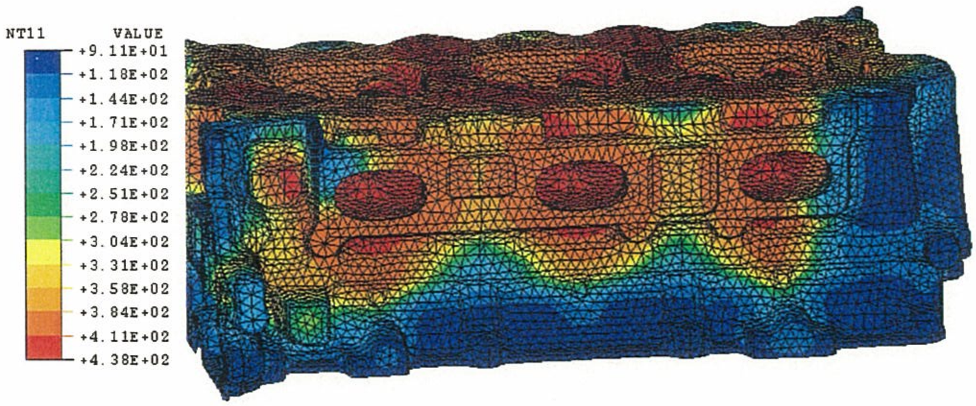


Figure 25. Temperature distribution of a cylinder head after 8 seconds into the quench.

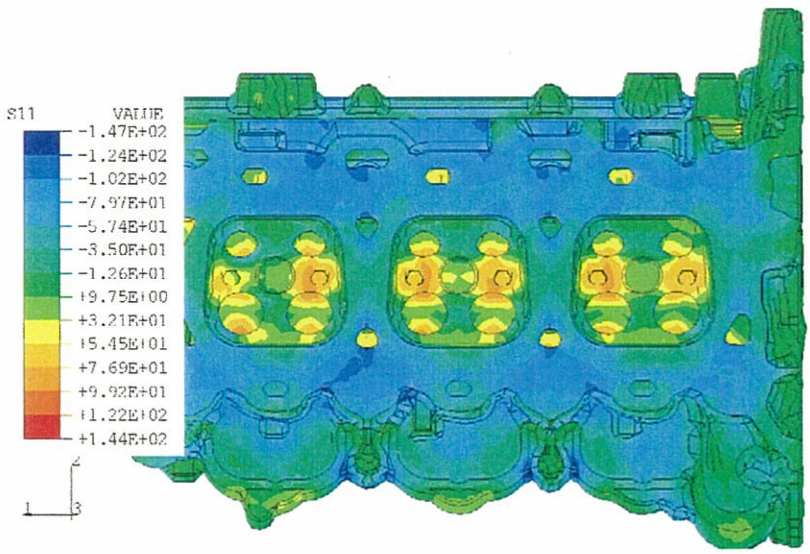


Figure 26. Predicted residual stresses of a cylinder head at the combustion chamber (after quenching).

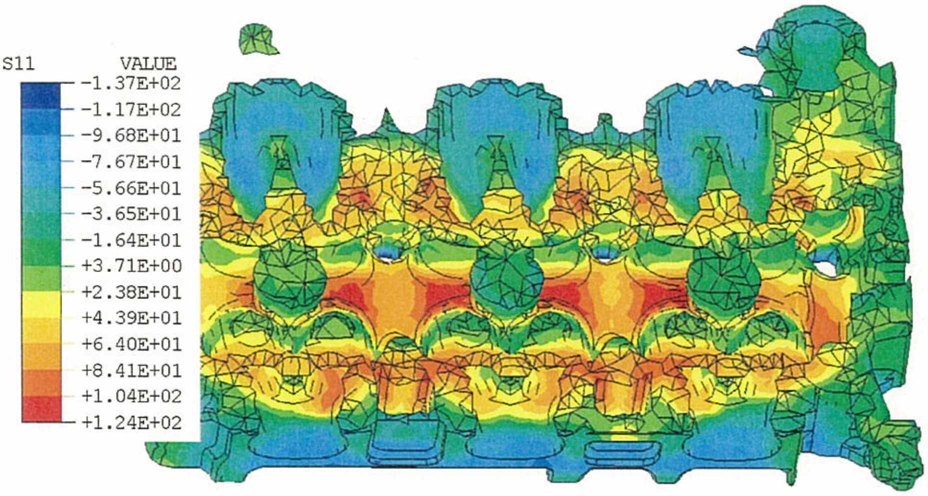


Figure 27. A cut away view of the predicted residual stresses at the combustion side of the water jacket area in a cylinder head (after quenching).

6. References

1. D. Slavik and H. Sehitoglu, "Constitutive Models Suitable for Thermal Loading", *Journal Engineering Material and Technology,* 1986, v. 108, p303-312, pp. 1885-1900, 1978.

7. Acknowledgements

The authors would like to acknowledge the many contributions to this body of work from within the V-6 and the V-8 / V-10 Durability CAE Sections in Analytical Powertrain and the Cast Metals Science Department within Ford Research laboratories.