PERFORMANCE-ORIENTED PARTNERS IN SIMULATION – ENGINE DEVELOPMENT AS AN EXAMPLE

[1]Dr.-Ing. Michael Klein*, [1]Dr.-Ing. Reinhard Helfrich
[1]INTES GmbH, Germany

CONTACT ANALYSIS, OPTIMIZATION, HIGH PERFORMANCE COMPUTING, ENGINE, PERMAS

Simulation and optimization of structures is an essential part of virtual engine development. In engine development, nonlinear structural analysis is a fundamental simulation method which now needs a coupled optimization process for high performance engines with lightweight design. The optimization process has a high demand for short run times of nonlinear engine analysis, because the size of today's typical nonlinear engine models with their long run times meets a certain number iterations in optimization. The resulting run times could easily reach weeks, which is often not acceptable for the engine development process.

First, only the best performance oriented tools will fulfill the needs of virtual engine development. A close collaboration of pre- and post-processor and solver is a prerequisite. ANSA and PERMAS are such tools, which both deliver state of the art features for today's simulation tasks in engine development. In particular, PERMAS provides an integrated nonlinear solution with an appropriate optimization process.

The presentation will show the typical model setup of an engine for PERMAS with ANSA. It also will discuss the various nonlinearities of the engine model followed by high performance aspects of solver run times including additional speed-ups for a sequence of only slightly modified models. This feature leads to the optimization process, where a freeform optimization is used to reduce the weight of an engine under stress and stiffness constraints.

TECHNICAL PAPER -

1. NONLINEAR STRUCTURAL ANALYSIS IN ENGINE DEVELOPMENT

The special challenge in nonlinear structural analysis in engine development is the combination of several aspects.
The first aspect is model size with its influence on run time and accuracy.
Second aspect is physical behaviour. The nonlinear behaviour should be represented in several aspects as good as possible. Essential nonlinearity are many contact areas, bolts with pretension, nonlinear elasticity and plasticity, nonlinear gasket behaviour and temperature dependent stiffness.
Third aspect is the nonlinear step history. For correct results realistic load history must be part of the simulation process.
The efficient analysis of such models is only possible with a preprocessor, solver and postprocessor combination that is able to support all these aspects for engine simulation.

Model Size

Model size always matters. It has influence on two very important aspects of engine analysis with opposite nature. More accuracy for displacements and stresses is possible with smaller elements, which means more elements. Shorter run times, with the same solver, are possible with less elements.
Today still more accuracy in terms of stiffness representation is required. This is only possible by more degree of freedom. The models have up to 60 million degrees of freedom and still the engineers ask for more accuracy. It seems that a convergence in model size is far away. All indicators predict further growing of model size.

So, we can't expect the reduction of run time by reduction of model size in the near future (figure 1). The only chance is the development of advanced solver and pre-processor technology.

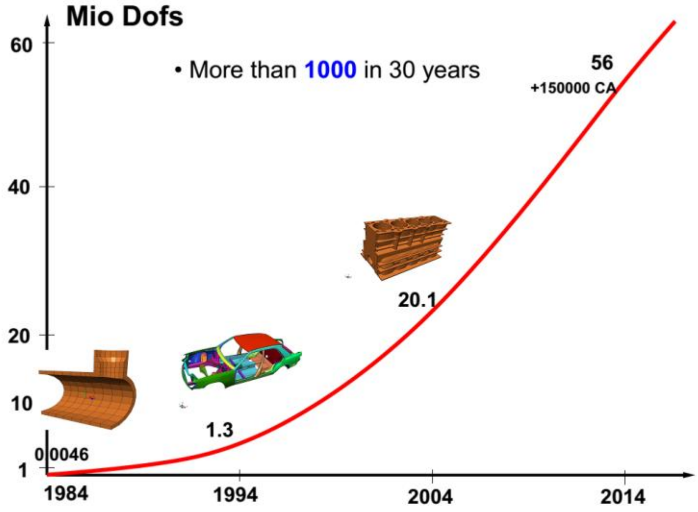

Figure 1 – Evolution of Model Size (1)

Physical Behaviour

Contact Analysis (with flexibility method)

In the past a big variety of solution schemas for the numerical treatment of contact boundary conditions have been developed, e.g. Lagrangian parameters, penalty functions or staggered u/p iterations (3, 4). PERMAS uses a slightly modified flexibility method which exactly simulates the discontinuity of the contact region. Furthermore, the method shows an excellent efficiency (5).

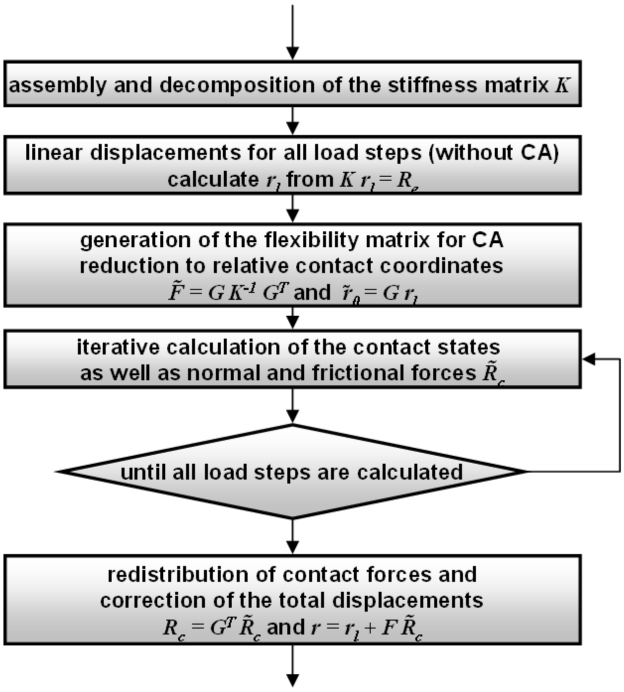

Figure 2 – Flexibility Method in PERMAS

Figure 2 shows the complete algorithm for a linear static analysis of several load steps with contact. Starting with the global stiffness matrix K and the applied forces R_e a linear-elastic solution r_l is calculated in a first step without consideration of the contact boundary conditions. This solution is then transformed to a significantly smaller system which contains only the relative displacements of the potential areas of contact. A condensed flexibility matrix $\tilde{F}$ is then built for the contact system. During subsequent iterations the contact is closed or opened at all potential locations, respectively, until penetration is compensated by reaction forces and a state of equilibrium is reached. Finally the contact forces are transformed back into the original displacement coordinate system and the global displacements are corrected by the relative displacements of the contact zones.

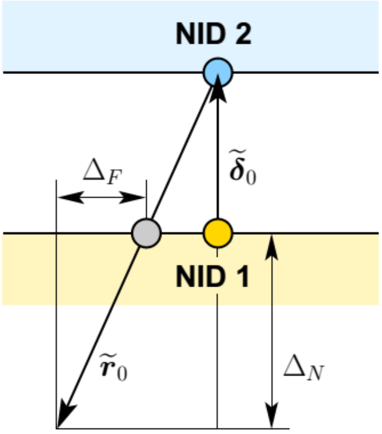

Figure 3 – Typical Gap Geometry

Figure 3 shows a typical contact condition between two nodal points NID1 and NID2 with an initial gap $\tilde{\delta}_0$. The applied loading results in the first instance in a relative displacement $\tilde{r}_0$, at which the two parts penetrate. During the contact iteration the piercing point of NID2 through the normal plane at NID1 is calculated. Successively, normal and frictional forces are introduced until the incompatible displacement components Δ_N and Δ_F are exactly compensated. If the frictional forces exceed the limit of the static friction $F_R \leq \mu_0 F_{norm}$, only the smaller forces of sliding friction $F_{slip} = \mu F_{norm}$ can be applied. In general, these will not fully compensate Δ_F-slipping takes place. The sliding motion leads to an additional load or to a relief at neighbored contact regions (redistribution of load), which may change their state from sticking to slipping or vice versa.

The efficiency of the solution scheme is primarily based on the reduction of the non-linear system of equations. Even models with several million degrees of freedom usually have only a few thousand contact pairs. So the size of the flexibility matrix is much smaller than the stiffness matrix. A modern iteration algorithm (6) with a convergence by far superior to the common load step methods (7) contributes to the excellent performance. In addition, the convergence improves with the complexity of the model. An additional advantage of the flexibility method is its numerical accuracy. Because of the strong influence of micro-slip on the frictional forces and consequent effects on the resultant calculations, it is essential to minimize numerical errors. Algorithms using penalty functions do in general lead to worse condition number of the stiffness matrix. The resulting numerical error in the displacements may be of the same amount as the relative movement of the contact pairs.

In contact analysis it is important to fulfill the boundary conditions as exactly as possible (8, 9). In PERMAS the discontinuity is exactly fulfilled, the normal contact has no respectively infinite stiffness. Also frictional contact is ideally simulated, even anisotropic friction is supported.

For contact definition and checking it is essential to use a preprocessor that has a dedicated visual feedback and a complete support of all specific settings. Only with this completeness the advantages of PERMAS can be used in a convenient and safe process. Figure 4 shows a typical contact definition between gasket and engine head in ANSA. Visual feedback of the contact area, a list of all contacts and a window with all settings is shown.

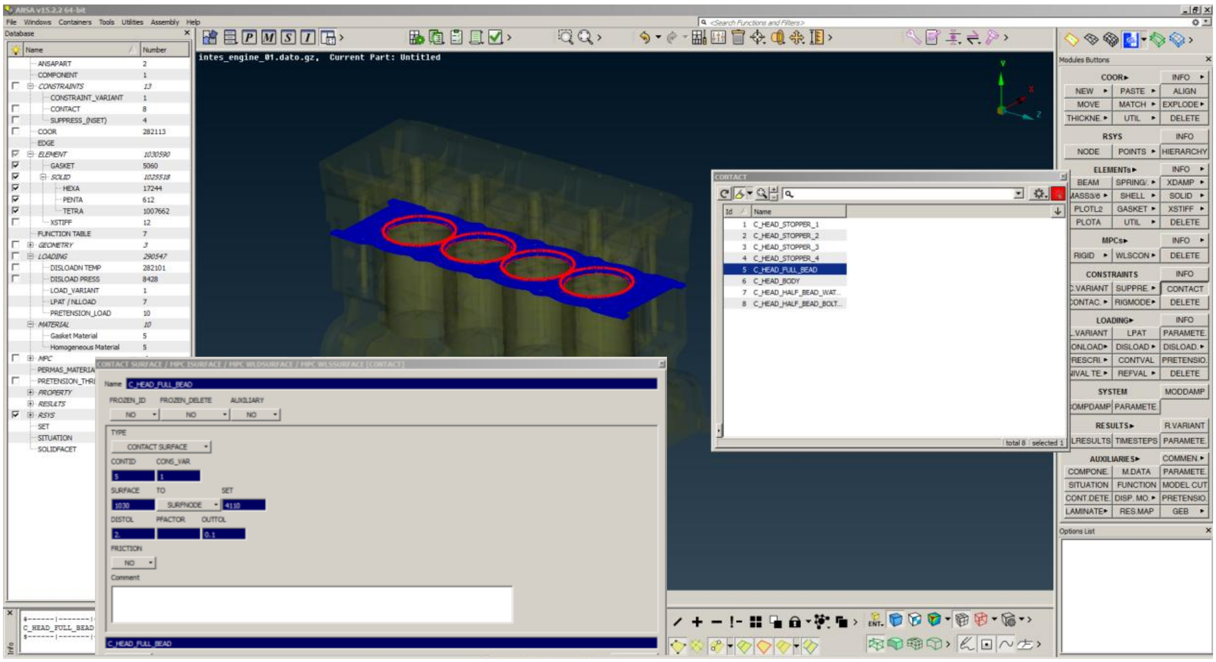

Figure 4 – Detailed Contact Behaviour Definition for PERMAS in ANSA

Bolt Pretension

Bolts are a standard unit in every engine. The pretension force is one of the essential loadings. Several states are required during simulation of assembly and work load, e.g. pretension force, lock of bolt and bolt force loss.
The objectives for modelling of bolts are: as simple/fast to create as possible, load path as realistic as possible and impact on run time as small as possible.
PERMAS has, in addition to the classical bolt pretension with pretension definition in the shaft of the bold, a new improved model with pretension definition in the thread area (figure 5). There are several advantages of this model (visualized by figure 5):

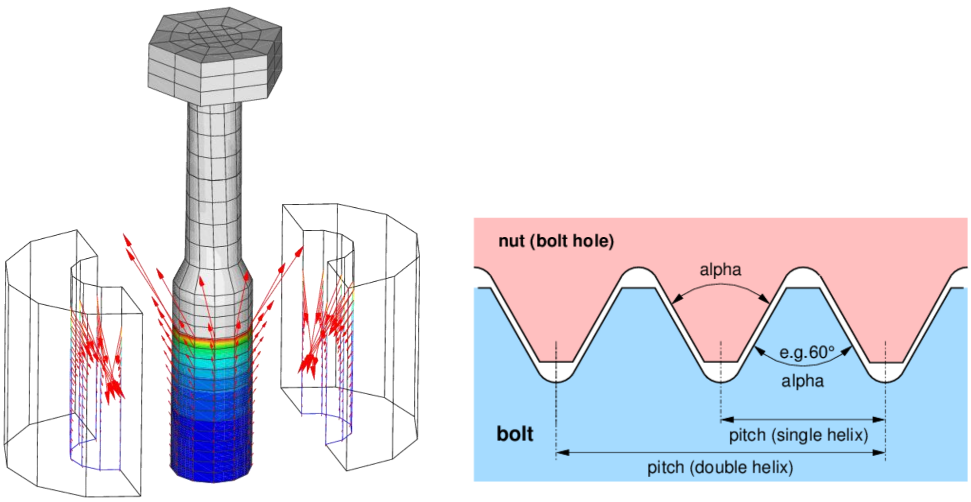

Figure 5 – Pretension Simple and Closer to Reality in PERMAS

- pretension area, the tread, is the natural part border between engine block and bolt
- exact definition of pretension force

- elongation of bolt, like in reality (other methods shorten the bolt)
- radial spreading, like in reality, by simple input of flank angle alpha, this leads to a load path like in reality, and
- torque in bolt, like in reality, by simple input of pitch

To get the complete benefit from such an advanced method it is very important that the complete process supports this method. In ANSA (see figure 6) the pretension thread method is completely supported with all parameters and visual feedback.

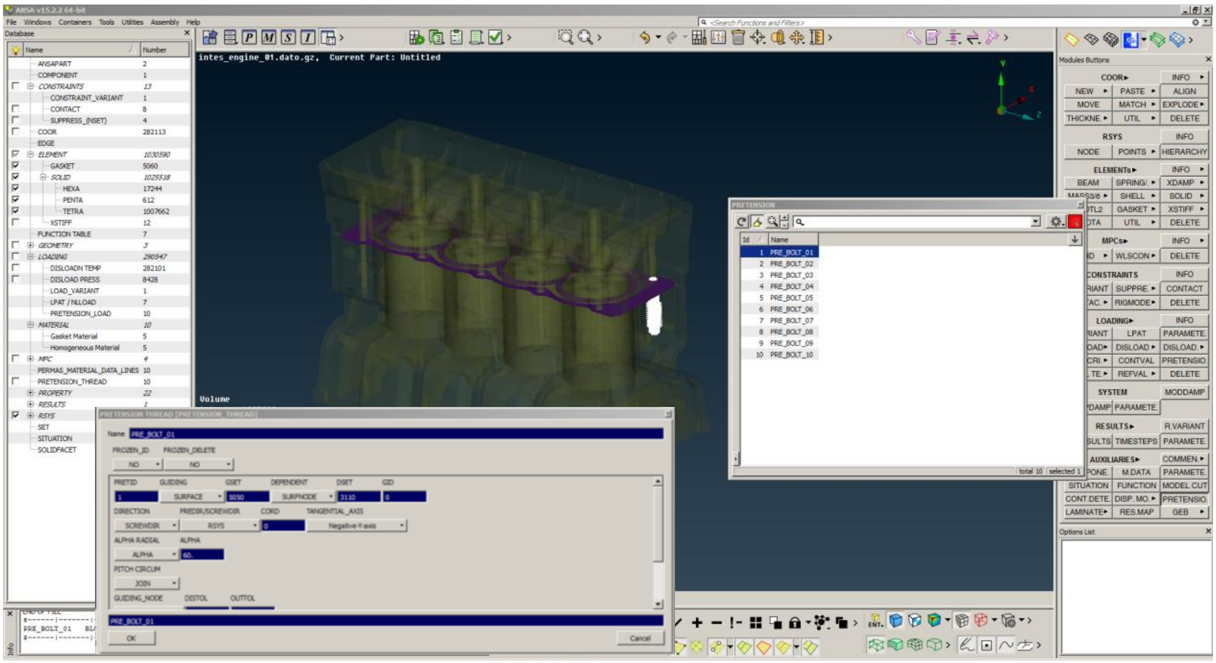

Figure 6 – Precise Pretension Definition for PERMAS in ANSA

Nonlinear Elasticity and Plasticity

The task of the solution process for non-linear materials is the handling of any material description. Typically used in engine analysis are non-linear material of cylinder head, crankcase or bolts. Besides classical elastic-plastic material properties, sometimes a cast iron material law is required where the non-linear material behavior in tension and compression domain is significantly different (see figure 7).

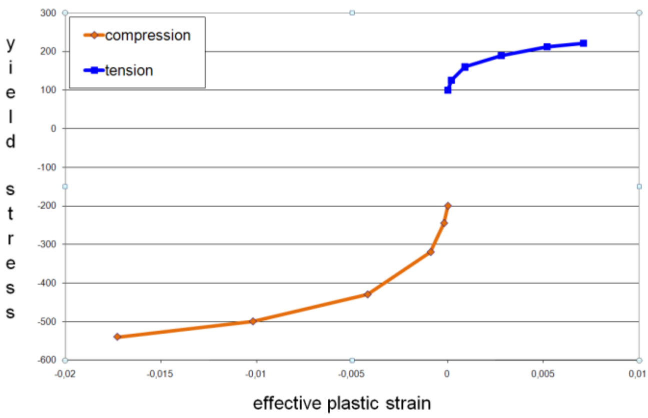

Figure 7 Cast Iron Material Data

Both material laws are handled by input of a strain stress relation. Also the characteristics of both are very similar, so that the algorithms for the solution process can be specialized to them for the most efficient overall solution process. If plastic or cast iron material non-linearity is taken into account for engine analysis a big fraction of the number of overall elements are affected.

Nonlinear Gasket Behaviour

For engine analysis the third material, gasket material, is of high importance, because it has a big influence on the overall behavior. Non-linear gasket material has a totally different characteristic. Described is the behavior orthogonal to the sheet plane through all sheets by a measured pressure/closure curve (figure 8).

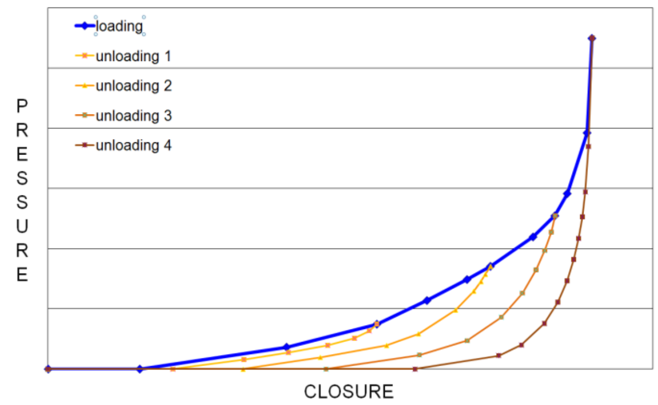

Figure 8 Pressure/Closure Material Behaviour of Gasket from Measurement

The material is elastic or plastic, has one loading curve and several unloading curves, which describe different unloading dependent on the load state. For different areas of the engine gasket, like bead, half bead or body, individual measurements are done. So, different properties are used at several regions of the gasket layer. Very typical is the ascending slope of the curve in opposite to the classic weakening material laws. The in plane material characteristic is linear elastic. The number of gasket elements is very small in comparison to the number of all other elastic elements.
The different behavior and the different function in comparison to the complete model are two big differences between other material non-linear elements and gasket elements. But up to now both are solved by one common iterative process. If the contact analysis with flexibility method, as described in the first section, is used, the non-linear material solution embraces with an iterative loop the contact iteration loop. This loop is very time consuming, because all operations are done with the whole stiffness matrix.

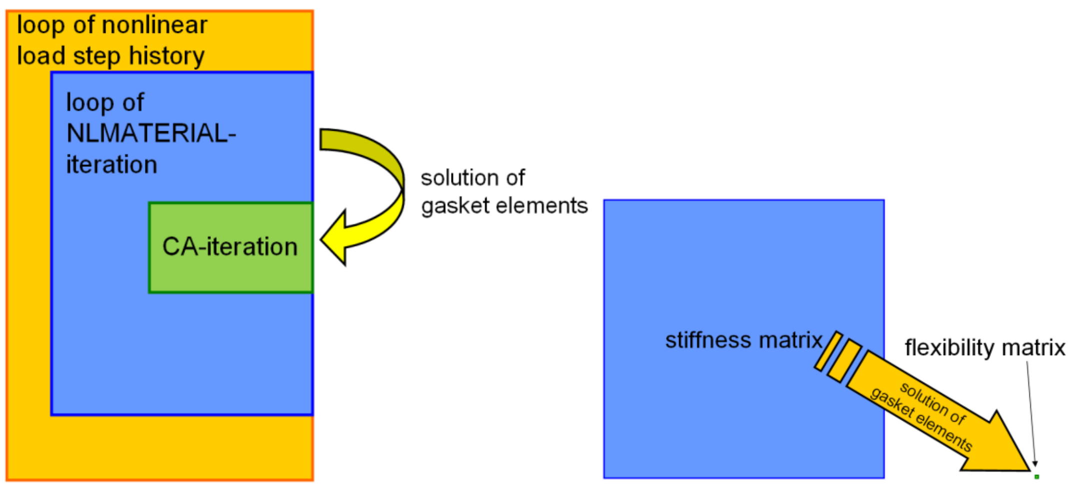

Figure 9 Shift of Gasket Solution from Non-Linear Material Iteration to CA-Iteration, at the Same Time the Non-Linearity is Shifted from Stiffness to Flexibility Matrix

In the new process the gasket element solution is shifted (**Error! Reference source not found.**) from the non-linear material loop to the CA-iteration. This is against the well-known rule for efficient algorithms, that it is more efficient to do the effort in the outer loops and not in the inner loops that are repeated very often.
But as shown in figure 9 the size of the flexibility matrix is for engine analysis typically by a factor greater than 100 smaller than the stiffness matrix. Perfectly fits in this in addition that the number of gasket elements (it is only one 2D-layer gasket elements in the 3D-model) is also very small.
As second factor the pressure/closure curve can be solved very similar to the contact by highly efficient algorithms. And the solution process for non-linear materials can be stronger focused on plastic and cast iron materials.

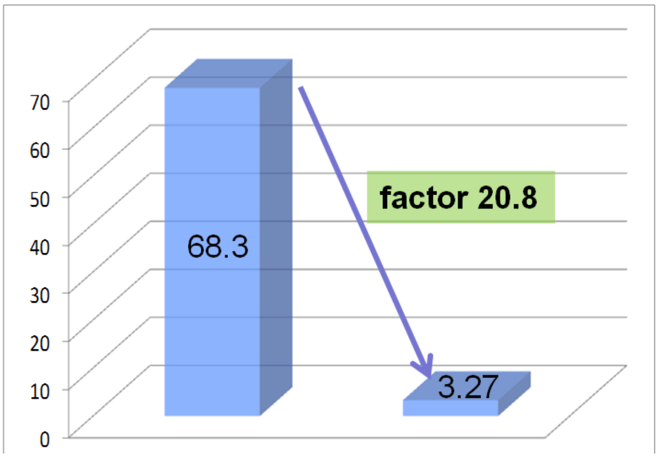

Figure 10 Run Time Reduction – Engine with Non-Linear Gasket, Old (left) and New Method

Temperature Dependent Material

A very typical requirement for engine analysis is the following: contact, pretension, nonlinear pressure-closure for gasket, linear material for all other parts and very important several temperature load states in combination with temperature dependent material behaviour. For this class of analysis PERMAS has a very special solver that reduces run time drastically. This special solver reduces typically the run time by a factor up to 4, which means that the run time is only 25% of the usual run time.
Figure 7 shows the algorithmic improvements of the new method regarding run time. The engine model has 56 million DOFs, 146,000 contact-DOFs and 30,000 gasket elements. The computer has 16 cores (2*E5-2680 with 2.7 GHz, 157 GB main memory and a NVIDIA Tesla K20c XPU).

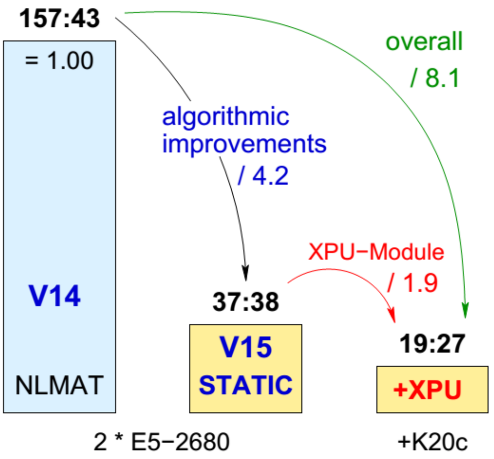

Figure 11 Engine Benchmark with Advanced Method for Temperature Dependent Material (1)

Nonlinear Load History

For the analysis control the specification of the load history is very important. Figure 12 shows a typical example for a load step control where the load steps are arranged from the bottom to the top and the abscissa represents the virtual time steps of the process during non-linear analysis. Pretension and locking of several bolts and loading/unloading of gas pressure take part in this example during the four load steps.

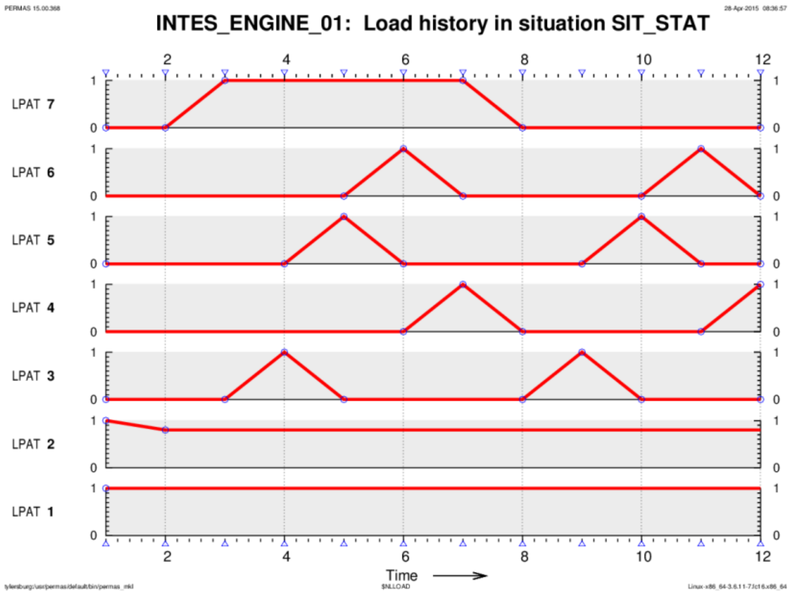

Figure 12 Plot of Non-Linear Load History for Engine Analysis

For an engine with a higher number of cylinders there should be also a higher number of load steps, because the gas pressure has to act on all cylinders separately in a specific sequence. Also the changes of temperature states (lpat 7) and the associated changes in stiffness can be investigated and results in a higher number of steps.

2. OPTIMIZATION

Non-Parametric Shape Optimization

In contrast to the parametric shape optimization with shape basis vectors, design element or morphing, in non-parametric or 'free' shape optimization another type of shape change discretization is used, where every node on the designable surface may be moved individually. The volume mesh inside the design space is relaxed automatically to follow the shape change at the surface with minimal impact on element quality. Necessary input data may be easily generated in VisPER with the FreeWizard by simply selecting a node set on the surface. The typical characteristic of non-parametric shape optimization is a huge number of design variable. PERMAS uses specific mathematical optimization method for this class of optimization.

Optimality Criteria

For reduction or limitation of stresses, so called optimality criteria provide an efficient and derivative-free method well suited for non-parametric shape optimization. This method is based on the simple principle of adding material at regions with high stresses and reduce material in areas with low stresses. Two approaches are possible:

– Reduction of maximum stress by *homogenization*, that means pure relocating of material (keeping approximately the same weight)

– Reduction of weight while keeping stresses below a user defined limit. In both cases additional design constraints like weight or stiffness may be specified.

Contact Status Files

Contact Analysis Status files (CAS files) are a tool to considerably accelerate the contact iteration of model or load variants (including e.g. the most trivial 'variant': a simple re-run of the same job). Using CAS files will not change the final results, but it can significantly reduce the number of iteration steps needed to obtain those results. I.e. CAS files are not relevant for the simulation as such, but they are an important issue for performance!

In the practice of the FE-modelling, it is general that almost the same contact problem is solved several times. For example, the static analysis for several modifications of a FE-model, optimization, nonlinear solution, and so no. As usual, solutions of such contact problems (contact states and contact forces) differ slightly from each other. Thus, we can take the first contact solution and use it as the initial guess for the iterative contact solver. In this manner, we reduce significantly the number of contact solver iterations. From our experience this approach can accelerate calculations in several times! The same success is possible for optimization loops. As usual, solution of the next iteration during optimization of such contact problems differ only slightly from the former iteration step. Thus we always can take the contact solution from the former iteration step and use it as initial guess for the iterative contact solver. Usually the run time will be reduced drastically.

The implementation of the described idea is based on CAS files, where contact state and contact forces of solution are saved (see the figure 13). The size of these file is very small in comparison to the database of a finite element analysis.

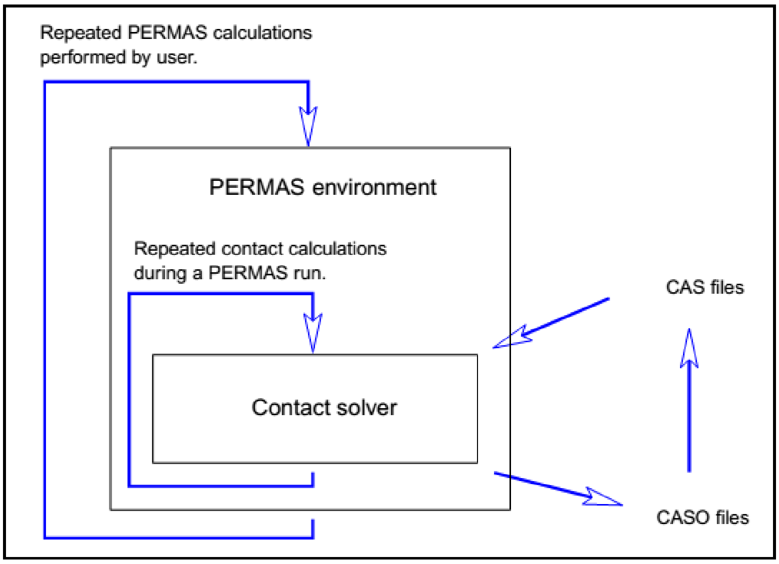

Figure 13 Process of CAS-File Usage

3. EXAMPLE

Initial Model

An artificial engine model serves as example. Basic characteristic of the model: 282,113 nodes, 1,030,590 elements, 5,060 gasket elements, 8 contact areas, 10 bolts with pretension, 792,088 DOFs and 12,920 contact-DOFs. The load history consists of 12 load steps. After the first assembly is done, then a bolt force lost is simulated followed by cylinder fire pressure load for each cylinder in hot and cold state.

The initial stress on the complete engine model is shown in figure 14. Step 4 (hot firing cylinder 1) is chosen for the figure, because the highest stress can be found there.

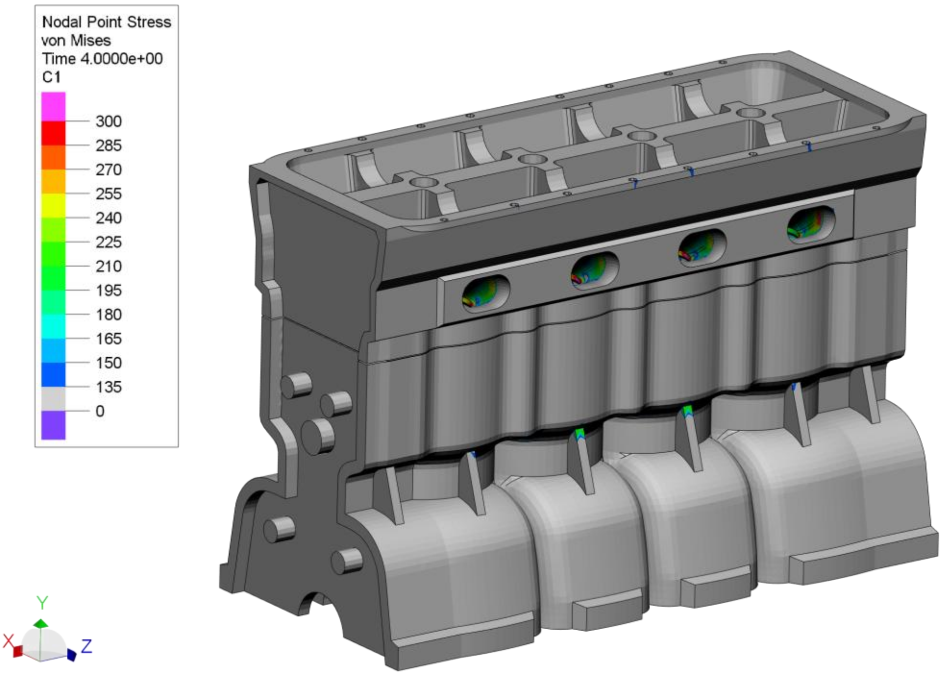

Figure 14 Engine Model with Initial Stress at T=4.0

Same perspective, but only the inner surface of intake and exhaust manifold is depicted in figure 15. Highest stress can be found at the hot (exhaust) manifold.

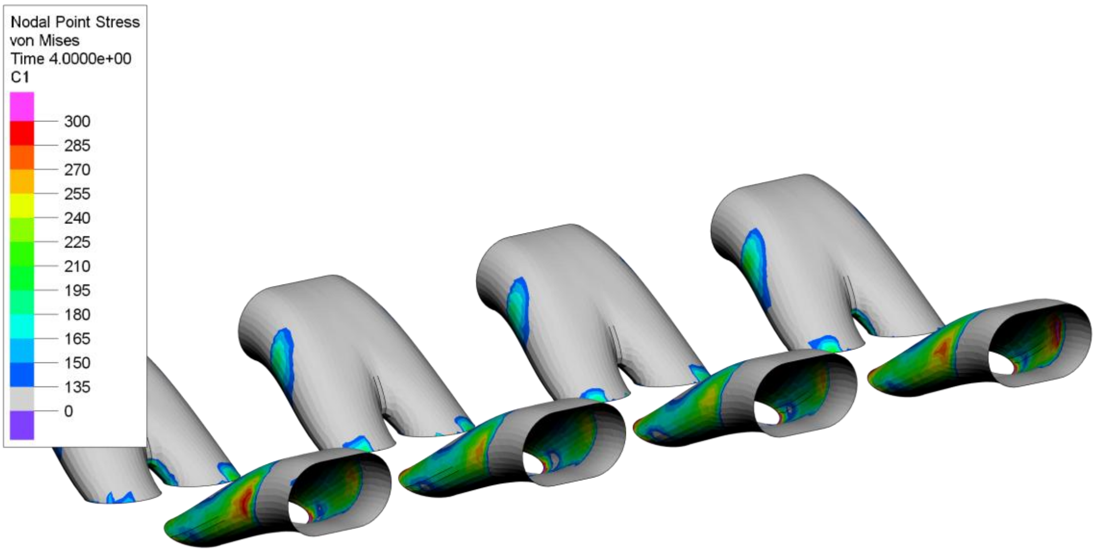

Figure 15 Intake and Exhaust Manifold with Initial Stress at T=4.0

<u>Optimization</u>

For the optimization the objective is chosen to reduce the stress at given weight of structure. A node set that contains all nodes at the inner surface of the manifolds defines the design nodes. The elements in the region around the design nodes for relaxing the elements during the shape changes is automatically created. This optimization contains 32,512 design nodes. All design nodes can be moved individually and take into account all 12 load steps for the optimization.

After 30 optimization iteration steps the stress is reduced for all load steps. Figure 16 shows the initial stress and the stress after shape optimization for the important manifold region. The results of the optimized shape are shown on the new node coordinates (magnification factor 1).

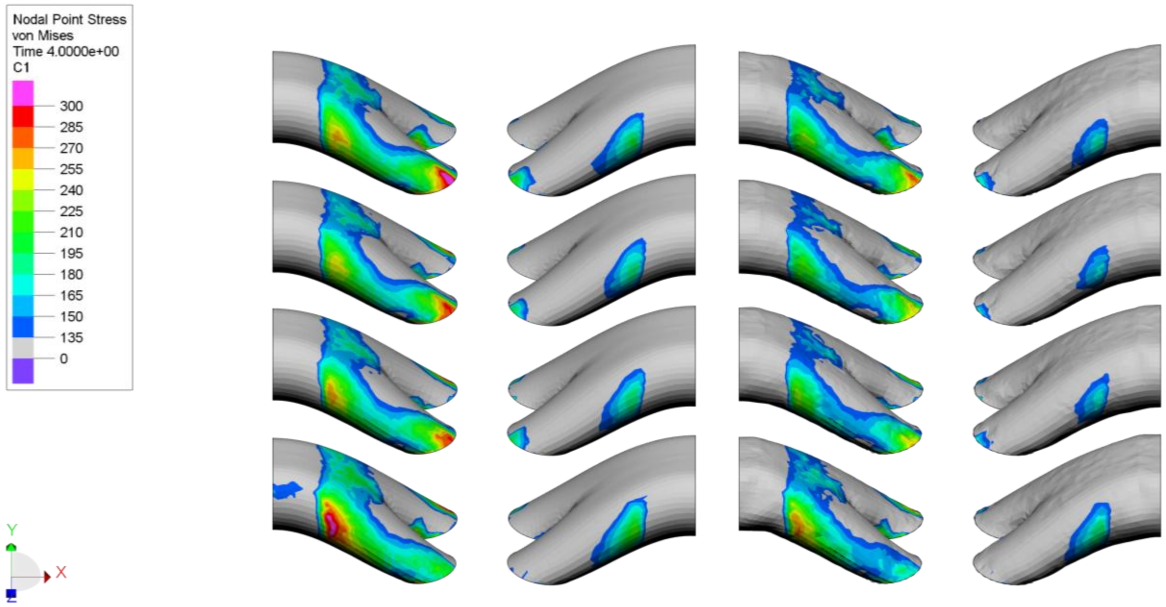

Figure 16 Intake and Exhaust Manifold with Initial Stress at T=4.0 (left) and the Reduced Final Stress (on New Shape), after Optimization (right)

Typically the changes are not easy to identify without a high magnification factor. But the result of normal change (figure 17) gives a very clear impression about the shape changes. In the blue regions material was removed and in the yellow to red areas material was added.

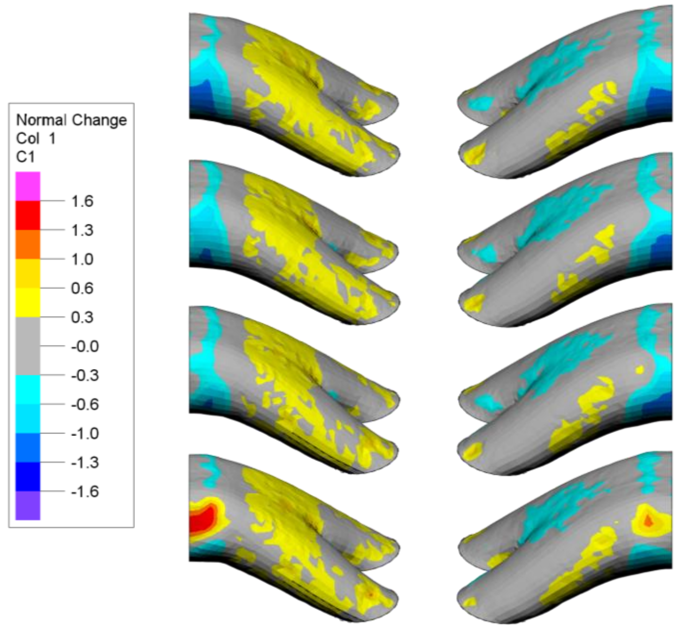

Figure 17 Intake and Exhaust Manifold Change in Normal Direction by Optimization

The non-parametric freeform optimization shows that it is possible with an easy definition and a huge number of design variable to reduce the stress considerable on complex geometries.

<u>Contact Status Files in Optimization</u>

As explained in the chapter “Contact Status Files” it is possible to reduce run time drastically with this feature. The run time of the given engine with 12 load steps and 30 optimization iterations is 14h 28 min. This run time is reduced by a factor of 3.7 to 3h 52min by simple switching this feature on (figure 18).

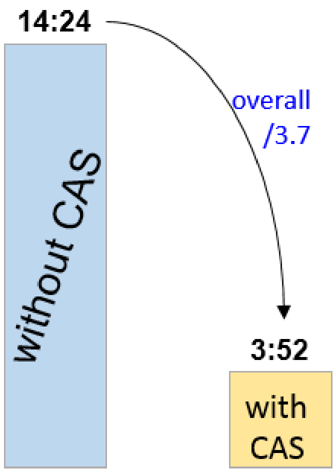

Figure 18 Reduction of Run Time by Contact Status Files

4. CONCLUSIONS

Two tools with performance orientation lead to reasonable advantages in simulation. For the pre-processor it is important to have the best in class meshing methods in combination with complete support of advanced features of a solver. The solver must contribute several features which improve the run time. ANSA and PERMAS are such tools. Together they show their full potential.

REFERENCES

(1) Insights to PERMAS Version 15, PERMAS Users’ Conference 2014, Rolf Fischer, INTES GmbH Stuttgart

(2) ANSA version 15.1.x User’s Guide, BETA CAE Systems S.A., June 2014

(3) PERMAS version 15, Users’ Reference Manual, INTES Publication No. 450, Stuttgart 2014

(4) Kikuchi, N., Oden, J. T., “Contact problems in elastostatics”, Finite Elements, Vol. 5, 1983

(5) Analysis of Linear Contact Problems, INTES Publication No. 229, Rev. C, Stuttgart 1985

(6) Leaky, J.G., Becker, A. A., “Benchmarks for Three-Dimensional Contact Problems”, Proceedings of NAFEMS World Congress 1997 (335-346), NAFEMS 1997

(7) Sellgren, U., Olofsson, U., “A frictional model for the micro-slip range”, Proceedings of NAFEMS World Congress 1997 (534-544), NAFEMS 1997

(8) Katz, K., Werner, H., “Implementation of nonlinear boundary conditions in Finite Element Analysis”, Computer & Structures Vol. 15, No. 3 (299-304), 1982

(9) M. Ast, S. Hüber, M. Klein, R. Helfrich, Performance Breakthrough in Engine Analysis, NAFEMS World Congress Boston, May 2011