Shape optimization with fatigue criterion on cylinder head water jacket, by means of Tosca, Abaqus and Fe-safe coupling

Maxime GUILLAUMY, Emmanuel COLLARD, Christophe BAROUX

Digital Product Simulation (DPS), F-78170 La Celle-Saint-Cloud, France

Franck SIMON, Caroline PASQUET

Groupe RENAULT, F-780840 Guyancourt, France

***Abstract**:*

In the hyper-competitive environment of the automotive industry, engineers are constantly looking for ways to improve the quality, durability and performances of their engines. Traditional shape optimization is a great solution for design improvements in later product development stages by minimizing stress or strain. Indeed, stress reduction often leads to a significant increase in durability, but stress peaks, as identified by static analysis, may differ largely from areas of maximum damage. Therefore, fatigue life simulation should always be included in the optimization loop, if the location of maximum damage and maximum stress are not matching.

The study that the **Groupe RENAULT** and **Digital Product Simulation** did was part of the numerical validation process of the thermal engines' cylinder heads. The objective of this project was to optimize the water jacket surfaces by increasing the safety factor's values, according to Dang Van's fatigue criterion.

For this purpose, we planned to use the whole potential of the **SIMULIA portfolio**, working simultaneously with **Tosca**, **Abaqus** and **Fe-safe**.

Abaqus was already used to perform the cylinder heads' thermomechanical calculations, followed by a fatigue analysis. Therefore, we decided to use Tosca Structure coupled with Fe-safe through a fatigue shape optimization, in order to modify the very complex and non-parametric surface geometry of the water jacket, and consequently, to avoid crack initiation in this highly stressed zone.

The method presented has the benefit to lead to a great improvement of critical zones, with a minimum optimization duration.

***Keywords**: Automotive, Coupled Analysis, Fatigue Life, Safety, Shape Optimization, Cylinder head, Thermomechanical analysis*

1. Introduction

The study presented, conducted by Digital Product Simulation in collaboration with the Groupe RENAULT had the objective to identify an optimization methodology for thermal engine cylinder head's water jacket, regarding critical areas' fatigue life.

Thanks to the SIMULIA portfolio that integrates software such as Tosca, Abaqus and Fe-safe, we were able to realize a shape optimization, taking into account the Dang Van fatigue criterion on a non-linear thermomechanical model (Figure 1).

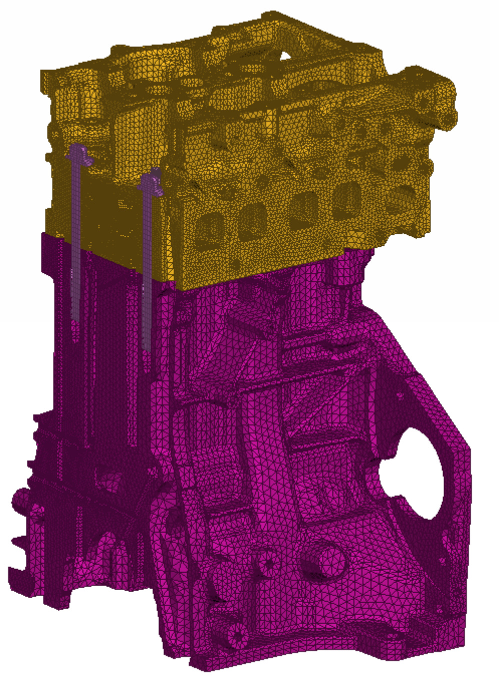

Figure 1. Thermal engine, half model, used for Abaqus analysis.

The initial phase of the project has consisted in ascertaining that the fatigue simulation results using Fe-safe matched RENAULT results as well as identifying the impact of the geometry on thermal simulation results.

After that, we needed to build a simplified model aiming to reduce the optimization time. Indeed, the initial Abaqus computation could took many hours, and the optimization could go until 20 iterations (so days of computation).

Next, we will describe and explain the different points of the optimization strategy we have chosen. In the end, we will review and discuss the main results of this study.

2. Preliminary work

We began by the validation process of the cylinder head's water jacket (Figure 2). At this stage, RENAULT already had stress & fatigue results available, respectively provided using Abaqus and an in-house fatigue code.

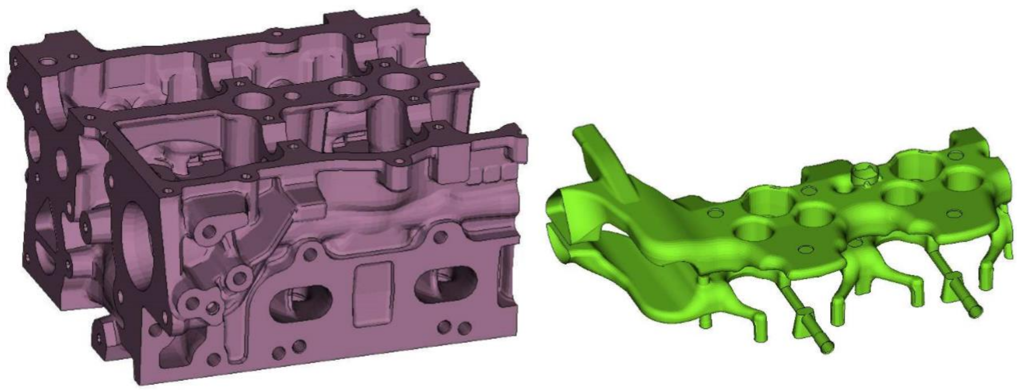

Figure 2. Cylinder head (left), and its water jacket (right).

The first step for DPS was to get familiar with the model; then relaunch the Abaqus simulation, as well as a Fe-safe simulation to get the same original values.

2.1 Abaqus simulation

The model used for the project was a half part of a thermal engine (see Figure 1)including the cylinder head, valve seats, valve guides, cylinder bolts, the cylinder gasket and the cylinder block. The overall size of the model is about 1 430 000 elements.

The simulation's input file has been separated in 8 steps to simulate the engine's standard process:

Step 1 – **Interference and bolt displacement**

Step 2 – **Bolt tension load**

Step 3 – **Injector load** (& bolt tension load)

Step 4 – **Thermal load** (& injector load & bolt tension load)

Step 5 – **Thermal and combustion load** (& injector load & bolt tension load)

Step 6 – **Cooling to ambient temperature** (& injector load & bolt tension load)

Step 7 – **Thermal load** (& injector load & bolt tension load)

Step 8 – **Thermal and combustion load** (& injector load & bolt tension load)

For the fatigue analysis, we only use the stress results of the last 2 steps.

2.2 Fatigue analysis

Once the Abaqus analysis has been done, we use step 7 & 8 stress results, contained in the *.odb output file, to compute the model's fatigue behavior.

Actually, the multiaxial Dang Van method, used for high cycle fatigue, does not calculate fatigue lifetime, but rather whether a component have infinite lifecycle or not.

The Dang Van's criterion defined the unlimited endurance on the maximal shear plane where, P(t) is the hydrostatic stress, and $\Delta\tau(t)$ is the variation of the local shear stress, given the following formula:

$$\max_t \left(\Delta\tau(t) - \alpha . P(t) \right) \leq \tau_0$$

where α and τ_0 are the two parameters of the damage line, specific to the material.

A graphic interpretation of Dang Van's criterion is illustrated in Figure 3:

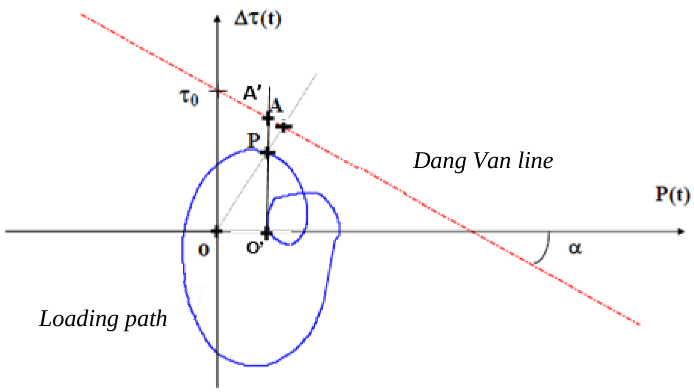

Figure 3. Dang Van diagram.

The distance between the loading path and the Dang Van line can be interpreted as a stress-based strength factor. In Figure 3, the ratio $FSF = \frac{OA}{OP}$ is the safety factor:

- If the safety factor computed on a node is up to 1 => the node have an infinite life.
- If the safety factor is inferior to 1 => the Dang Van's criterion is not verified for the node, meaning there is a risk of failure at this location.

RENAULT considers, taking into account a margin in specific areas, where the Fatigue Safety Factor has to be up to 1.x. They advised us to focus on three critical zones highlighted in Figure 4, where we get the same results for the Safety Factor using Fe-safe:

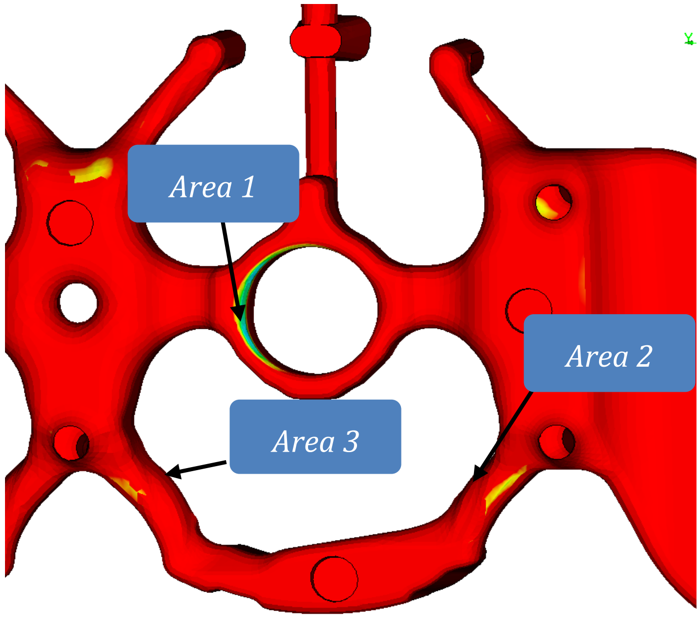

Figure 4. Fatigue Safety Factors, from Fe-safe, on critical areas of the water jacket.

2.3 Influence of geometry on thermal results

Shape optimization can involve important nodal displacements at optimization zones. Therefore, these modifications can influence thermal behavior at these locations. That is the reason why we needed to study the temperature gradients at these areas, to quantify the amount of authorized displacement.

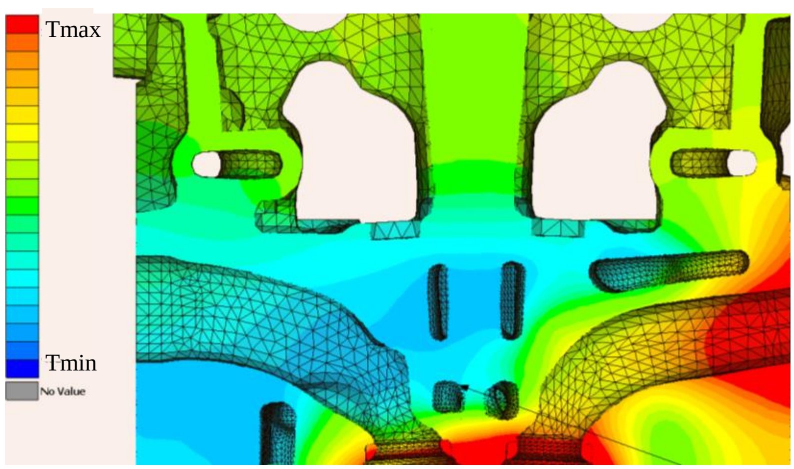

Figure 5. Temperature distribution - Cylinder head cutting plane, area 1.

As you can see on the cylinder head cutting plane, for the area 1 (Figure 5), the temperature gradient is lower at this location (2.5°C by banded) as well as at the 2 others critical areas. We had stated that these regions are thermally not problematic, so the maximum displacement allowed is 4 mm, according to other fabrication constrains from RENAULT.

At the end of the project, verification has been done to ensure that the thermal behavior of the fire deck was not impacted by the water jacket modifications.

3. Simplified model

An important part of the study consists in simplifying Renault's original model to perform optimization iterations within an acceptable computation time. Indeed, the original Abaqus computation time was very long, and a standard optimization loop could take until 20 iterations, which would have turned into months to get a Tosca's optimized set of parameters and to prove the strategy feasibility.

Numerous simplification technics have been considered, such as:

- Change the mesh order from quadratic (C3D10) to linear elements (C3D4)
- Increase the mesh size
- Contact management by tie

We tried each method in a test campaign, separately then combining them, with an objective of minimizing simulation time while maintaining accuracy of thermomechanical and fatigue results. You can see the main results of this campaign in Table 1 below:

Table 1. Font sizes and formats used in this document.

	Linear mesh	Increased mesh	Contact Tie	Linear + contact	Linear + increased	Linear + Contact + increased
Fatigue Safety Factor mean deviation	+17%	+639%	+9.5%	+31%	+321%	+552%
Computation Time	1h30	22h20	34h	2h	1h15	1h

The combination of all simplifications tremendously reduces the computation time, however the fatigue results are completely wrong. A substructuring technique was also considered to condense the cylinder block elements, but as they represent only 15% of the model elements, the gain in computation time would have been too low, and the method too complex in first approach.

According to Table 1, we have then decided to use the simplified model with linear elements, which fulfills both criteria: low time computation and low fatigue result deviation from the original model.

4. Optimization strategy definition

The overall optimization strategy in fatigue leans on the joint and sequential use of three software of the SIMULIA Portfolio: Abaqus, Fe-safe and Tosca.

Therefore, the optimization loop starts with the Abaqus thermomechanical simulation, to get stress distribution on the engine model. Then, the fatigue simulation can be realized with fe-safe to compute the Safety Factor from the Dang Van criterion on the water jacket.

Finally, the shape optimization simulation is performed using Tosca, modifying the part's contours by moving nodes of the mesh, to get a uniform distribution of the Safety Factor. A new Abaqus simulation can be run with this new mesh, and the whole process is to iterate until the optimization's objective is attained: we chose the "Controller Based" algorithm for the Tosca optimization, because it is suitable for standard problems such as stress or fatigue damage minimization within 15 to 20 iterations maximum (Figure6).

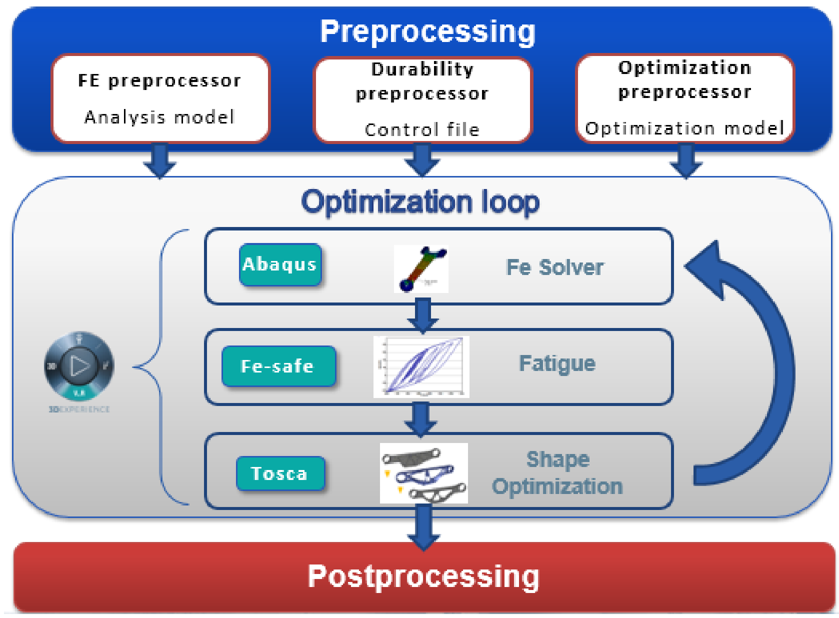

Figure 6. Shape optimization process.

Before running the optimization, the method requires a preprocessing step for each software. In order to define the design space for the Tosca optimization, we need to specify groups of nodes in Abaqus. Afterward, a first fatigue calculation with Fe-safe must necessarily be performed, with the purpose of recording a macro of it, to later replay it during each iteration of the optimization loop.

As a conclusion, the most difficult part was the Tosca preparation, in order to find the right parameters, and thus, correctly increase the safety factor in critical areas. This preparation steps for the Tosca optimization will be described in the following paragraphs.

4.1 Design spaces and objectives associate

Following the Renault's recommendations, the nodes' groups selected for the shape optimization correspond to nodes having a FSF<1.x in the three critical areas (Figure 7).

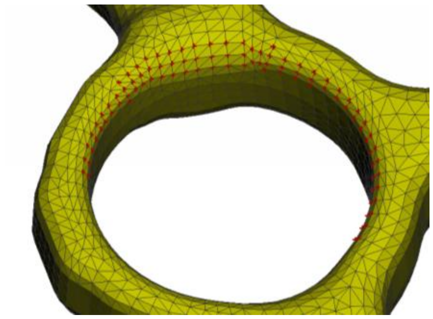

Figure 7. Nodes group selected on Abaqus for the area 1.

However, going from the RENAULT quadratic model to a simplified linear model generally increases the Safety Factor at these zones. Thus, a proportional readjustment of the optimization's target values is necessary.

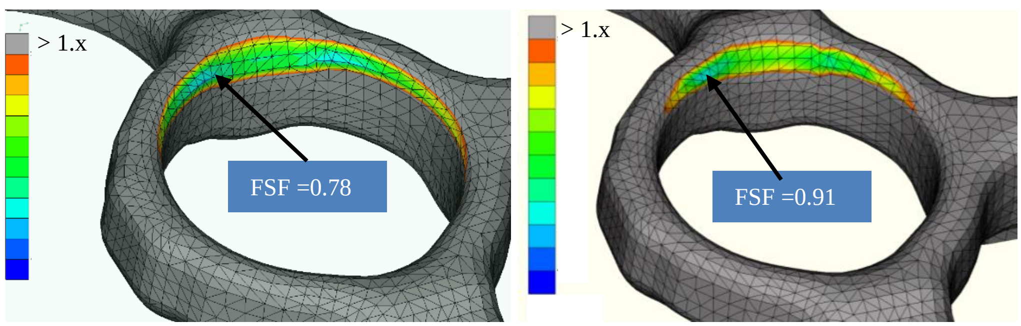

Figure 8. Fatigue result on initial model (left) vs simplified model (right).

For the area 1, the difference between the minimum values of the Fatigue Safety Factor of the initial calculation compared to the linear calculation is about 17% (see Figure 8). That is why we have to offset the objective value from 1.x to 1.y.

The optimization's objective is to maximize the minimum value of the Fatigue Safety Factor in all critical areas previously defined. Initial and optimization objective values for each areas to optimize are summarized in Table 2 below.

Table 2. Fatigue Safety Factor objectives

Area	Initial model	Initial Objective	Linear model	Optimization Objective
1	0.78	1.x	0.91	1.y
2	1.03	1.x	1.09	1.y
3	1.08	1.x	1.25	1.y

4.2 Constraints and fixed node's sets

Two different types of constraints are used in our optimization.

First, we want to minimize nodes' maximum displacement. To do so, we set the maximum displacement value to 4mm in accordance with geometry's influence upon thermal results and a shrinkage maximum value to 0 mm, to force Tosca to only "add material".

Secondly, we specified a non-design space especially bore holes and other machined ducts.

4.3 Mesh smooth

The Mesh_Smooth option allows us to define a domain, composed of elements adjacent to the nodes of the design space, smoothing the mesh modifications made by Tosca. Indeed, if only surface nodes are moved and inner nodes remain at their location, then surface layer elements will be strongly distorted. The rendering of the optimized surface will be more or less smooth, depending mostly on two parameters:

- The number of element layers around the design area, to absorb surface mesh modifications (Layer).
- The number of transition nodes on the free surface, which will be fixed at the domain's boundary or will remain free (Fix or Free).

Figure 9. Mesh Smoothing Layer=5, Fix=3 (left) vs Layer=6, Fix=6 (right) on area 1.

4.4 Stop conditions

At the beginning of the project, the optimization on the three areas was made by 1 optimization task, with three different objectives for each zone. This method worked out, but as you can see on the Figure 10, two areas exceeded the Safety Factor objective of 1.y, which means that too much "material" has been added. Moreover, the optimization took the maximum time allowed within 20 iterations before getting to an optimum.

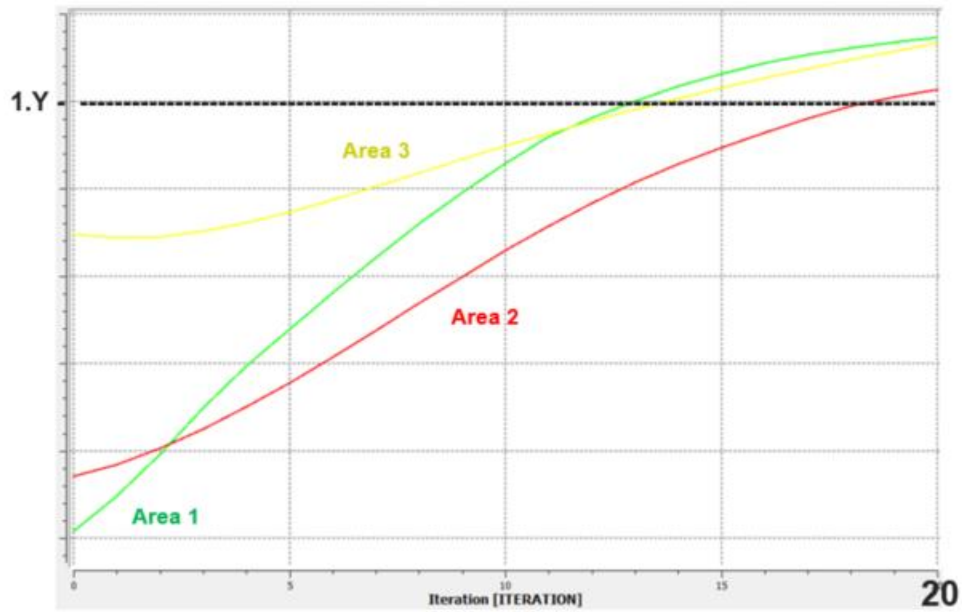

Figure 10. Safety Factors during the optimization without stop conditions.

Following this, we changed our strategy by separating the three areas in three different tasks. The most important effect brought up with this approach is the possibility to stop each area's optimization when its Safety Factor exceeds the objective. Furthermore, tasks' separation seem to help the algorithm to focus on each zone, making the whole optimization faster (16 iterations to converge) (See Figure 11).

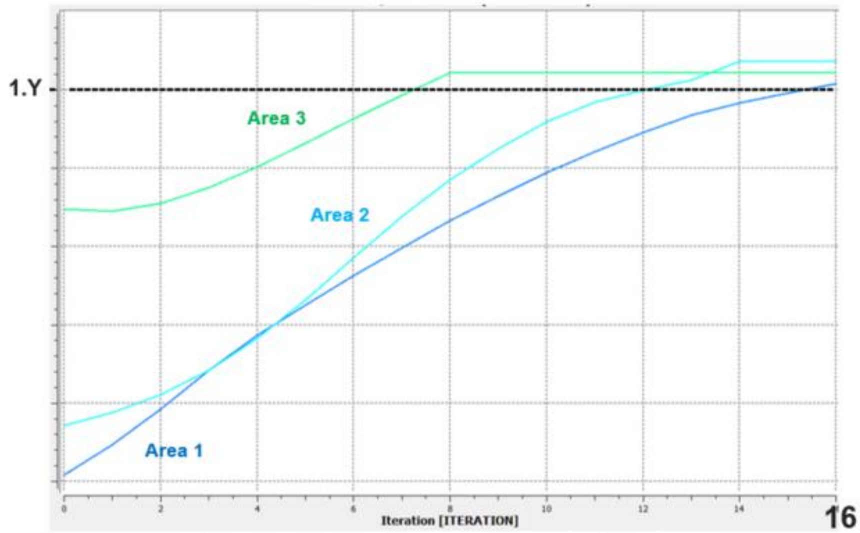

Figure 11. Safety Factors during the optimization with stop conditions.

5. Post-processing and main results

Two different methods have been used to perform the optimization's post-processing.

Firstly, we used Tosca's interface to verify results on the simplified model just after the optimization's end. The software generates a report enabling a graphic visualization of objectives' evolution (Figure 11), nodal displacements and Fatigue Safety Factors on the Mesh_Smooth domain, this for each iteration (Figure 12).

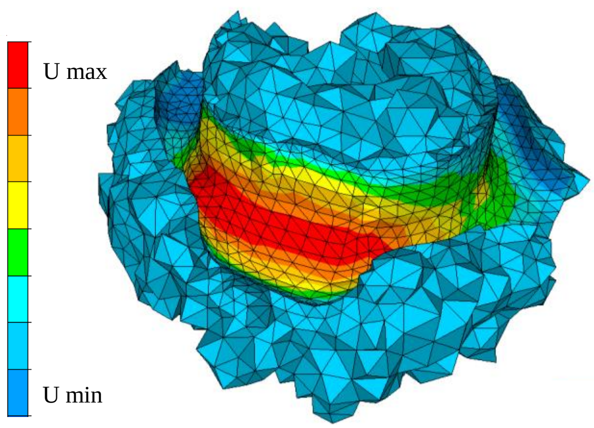

Figure 12. Nodal displacements on the mesh smooth area 1.

Secondly, as we did the optimization on a simplified model, we need to check again the results on a parabolic model, in order to compare these results with those from the Renault initial model. So we changed the linear elements of the optimized model into parabolic elements, then we relaunched the Abaqus calculation, and the Fe-safe analysis. After that, we were able to visualize the Fatigue Safety Factors on the cylinder head's water jacket (Figure 13).

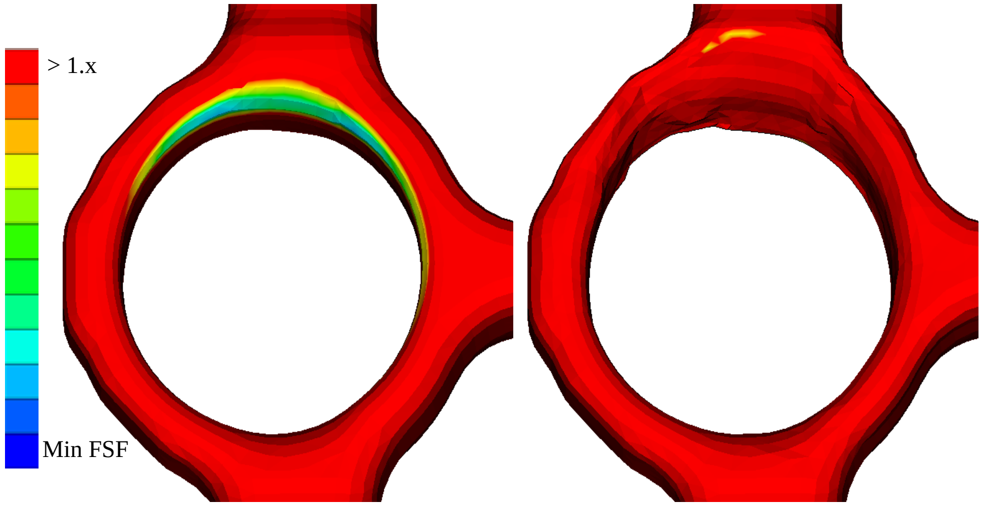

Figure 13. Fatigue Safety Factor on initial model (left) vs parabolic optimized model (right), area 1.

As you can see on Figure 13, the Fatigue Safety Factor has improved a lot comparing to the initial results. On the first area, where the Dang Van's criterion was the most critical, we succeeded to overtake 1.x for the most part of the zone. The water core size is reduced by a 3mm maximum mesh perturbation on this zone, which is acceptable according to RENAULT post-processing.

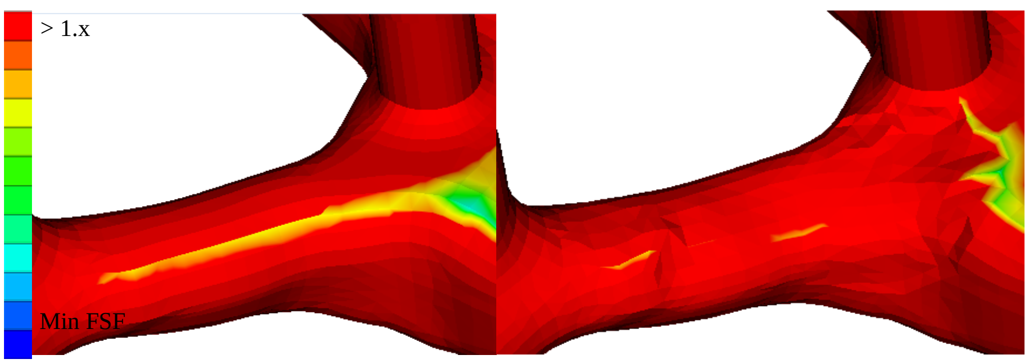

Figure 14. Fatigue Safety Factor on initial model (left) vs parabolic optimized model (right), area 2.

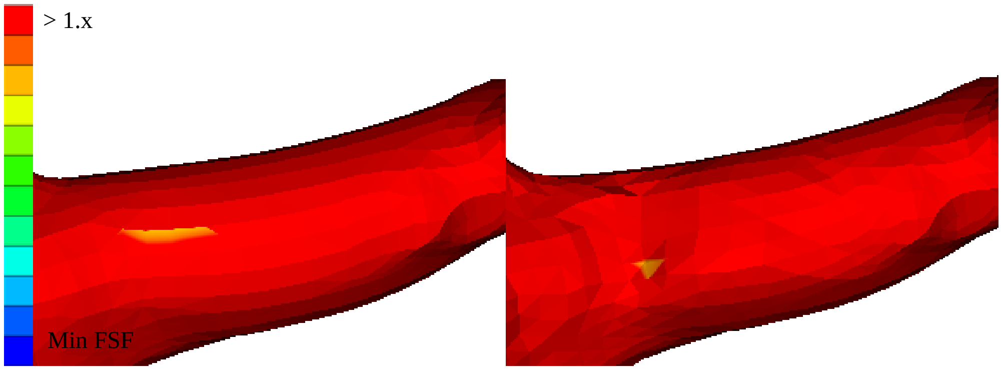

Figure 15. Fatigue Safety Factor on initial model (left) vs parabolic optimized model (right), area 3.

Even if we can observe that the optimization increase the Safety Factor on areas 2 and 3 (Figure 14 and 15), there are still areas under the RENAULT criterion of 1.x. The method which consist of changing the elements order, to reduce the optimization time, is therefore not optimal for zones where the Dang Van criterion is already greater than 1. Indeed, the error coming out of this simplification is too important compared with the criterion's small increase we expected.

On the other hand, this method is well suited to critical zones with an initial low Safety Factor, i.e. a finite life according to Dang Van.

Of course, as the optimization produced a modified mesh, the results are not perfectly smoothed. That is why these results are submitted to the design engineer's interpretation who needs to rebuild the cylinder head's CAD, smoothing further the results.

6. Conclusion

During this project, we developed a fatigue shape optimization methodology, using simultaneously Tosca, Abaqus and Fe-safe, to reduce cylinder head's water jacket crack initiation probability.

After some verification steps, as the thermomechanical and fatigue results, we ensured that geometry modifications had no impact on thermal results.

The retained strategy was a free shape optimization in Tosca, using a "Controller based" algorithm, enabling to reduce the necessary iteration number to less than twenty. Three separate optimization tasks have been implemented, to avoid dependencies between minimal Safety Factor on the three critical areas to optimize, therefore to accelerate the overall process.

We also had to simplify the model, changing the element order to linear with the purpose to reduce the optimization time to an acceptable duration; and then ultimately upgrade the element order of the model at the process' end. This method produces a lack of accuracy in the results precision, but a good improvement of the Dang Van Fatigue Safety Factor, especially for critical areas.

To go further, now that the methodology has been established, we are going to explore substructuring/submodelling techniques and other simplification improvement, or directly optimize the parabolic model, and so improving our results by getting rid of the model's linearization.

7. References

1. SIMULIA User's Manual, Version 2017, Dassault Systèmes Simulia Corp., Providence, RI.

2. fe-safe User's Guide, Version 2017, Dassault Systèmes Simulia Corp., Providence, RI.

3. Clément Roux. Dimensionnement en fatigue multiaxiale des toiles de roues ferroviaires sous sollicitations multi-paramètres à amplitude variable. Micro et nanotechnologies/Microélectronique. Université Paris-Saclay, 2016. Français. https://pastel.archives-ouvertes.fr/tel-01628281/document