Consideration of manufacturing pre-history influence of fatigue life of sheet metal formed automotive parts

Dmitry V. Buzlaev[1], Viktor V.Kalaev[2], Dmitry Y. Zheleznyakov[2], Dr. A.Winkler[3]

[1]TESIS, Moscow, Russia

[2]GAZ Group United Engineering Center, Nizhny Novgorod, Russia

[3]Dassault Systemes

Abstract: *This contribution considers the effects of sheet metal forming processes on the fatigue performance of LCV wheel discs, manufactured from microalloyed steel. Traditionally, such parts had to pass standard cyclic load test in the process of experimental validation, but road tests in real life conditions sometimes show fatigue cracks in highly stressed areas, predominantly in bending loading mode.*

To estimate fatigue life and safety margins standard fatigue tests should be accompanied by numerical simulations based on FEA results. Unfortunately, in spite of the long history of fatigue analysis codes, based on FEA results, the industry still relies heavily on tests because of frequent disagreements between simulation and test results.

In most cases stress analyst does not take into account the manufacturing pre-history of the part, its thinning, plastic strains and residual stresses accumulated in a part as a result of sheet metal forming process used for its manufacturing. This study tries to fill the gap between sheet metal forming simulation tools, like Stampack, SIMULIA Abaqus as stress analysis tool and SIMULIA fe-safe as a fatigue analysis tool. It show significant difference between the original simplified model of wheel center disk, which used virgin material and did not take into account real disk geometry and residual stresses and hardening, and the model which used results from the sheet metal forming simulation as a starting point.

Keywords: *Bolt Loading, Wheel Disk, Fatigue Life, Deep Drawing, Metal Forming*

1. Introduction

Early implementation of automotive components simulation in the development cycle requires tight integration of various departments and organizations, including design, manufacturing, analysis and test departments. Reliability of results and reduction of development time can be achieved either through multi-disciplinary simulations based on unique model or the possibility of utilizing seamless model conversion between different codes. Unfortunately, at the moment there is a certain gap between metal forming simulation and strength and fatigue analysis. Meanwhile, the distinct influence of sheet metal forming processes on the fatigue cracks development at the areas experienced significant thinning and stretching during plastic deformation (typical for deep drawing processes) has been observed and documented among GAZ Group subsidiaries and in open sources [4,5]. This influence can often be rather negative, while in some cases positive or no effect is observed. As a positive influence, hardening caused by plastic work should be mentioned, which increases component durability in the case of low cycle fatigue. One particular case where the positive effect is obvious is shot peen forming, but this process is used in aerospace and not in the automotive industry. For the typical deep drawing processes used in production of car components, a stretching stress state is more characteristic which usually decreases fatigue strength of a part. Also, for thick sheet parts bending can also be a problem, causing early appearance of fatigue surface cracks in the stretched layers of components. In this work we evaluate the influence of a manufacturing pre-history on the durability of the central portion of light commercial vehicle wheel disk.

2. Simulation of fatigue testing

The United Engineering Center of GAZ Group set up a task of simulating the fatigue strength of a wheel disk in standard bending with a rotation test according to Russian Standard GOST R50511. The standard requires that the number of cycles before fatigue cracks should not be less than 600 000. The disk is fixed on the test rig using hub part and 6 studs. A rotating load equal to 2744 N is applied through a 1 m long console beam, attached to the hub (fig.1). The disk was made from the Russian steel alloy 07 GBU – a microalloyed hot rolled high strength steel produced by Severstal. Its closest European equivalent is S315MC. The sheet had a thickness of 6.5 mm and a yield strength of 360 MPa.

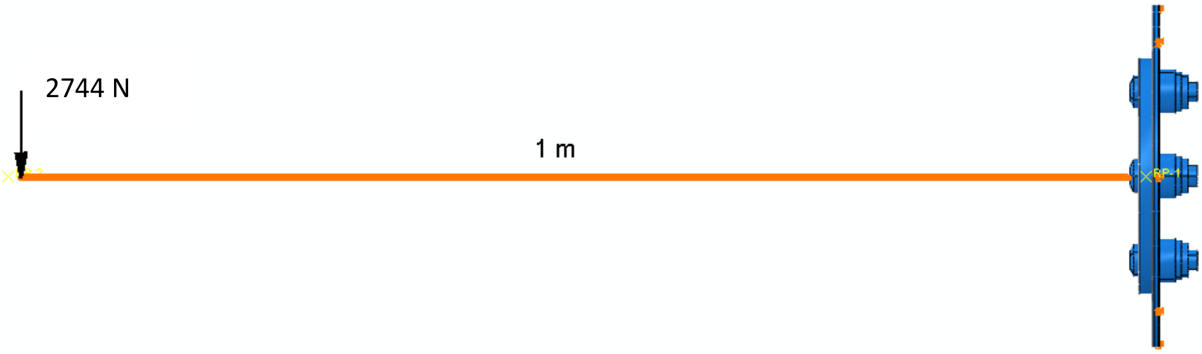


Figure 1. Model of wheel disk fatigue testing

The model was created in Abaqus/CAE with simulation execution using the Abaqus/Standard solver. General Static was used as a solution procedure. Initially a simplified FE-model was used, which included only the central flat section of the central disk (fig.2). The outer contour of the disk was fixed, the load applied through a console connector beam and its rotation was simulated by X and Y load components with associated amplitude curves. The disk itself was modelled with 5 layers of solid elements C3D8R. The material definition included elastic and plastic properties. The plastic properties included the hardening curve, which was defined up to 0.8 equivalent plastic strain. Studs, nuts and hub were also represented by C3D8R. The entire testing process was split up into three load steps: studs tightening, console load application, console load rotation. At the beginning of the studs tightening, only the disk component was fixed in space, the other components were loose, as the contact conditions had not yet been established. This caused some convergence problems in the first variants of the model. These issues were overcome by applying general contact with contact stabilization (using default parameters). Studs tightening was simulated using the standard procedure for bolt loads controlled by force magnitude (fig.3).

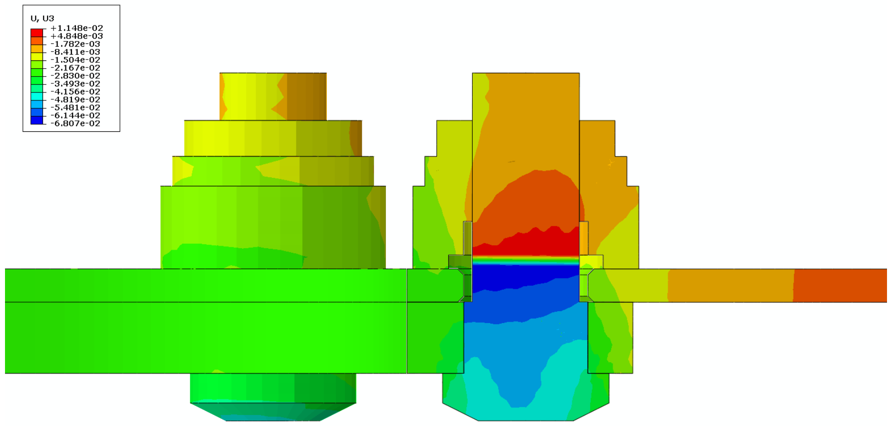


Figure 3. Studs vertical displacements after tightening

The use of solid elements for all components in the assembly was chosen to properly represent the stress field in the disk caused by bending with rotation taking the studs tightening into account. The stress field at each increment of rotation of the console load was used later for fatigue analysis. The load rotation step considered a 360 degrees full rotation cycle and was subdivided by 5 degree increments.

Abaqus results are presented in fig.4: maximum displacement in the disk is equal to 0.29 mm, the maximum principal stress over time is equal to 359 MPa. The area with the highest stress level is concentrated around the stud holes. It correlates well with real fatigue cracks in the disk observed in testing (fig.5). Fig. 6 represents the history of the maximum principal stress at the critical location.

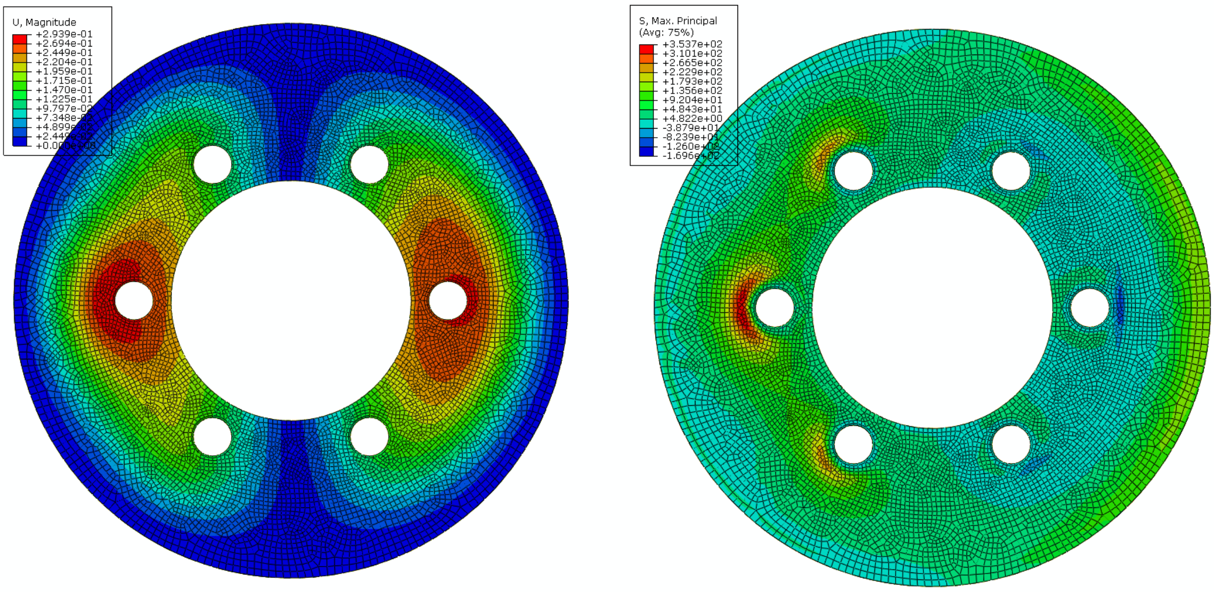


Figure 4. Displacements and Max. Principal Stresses at the end of the cycle

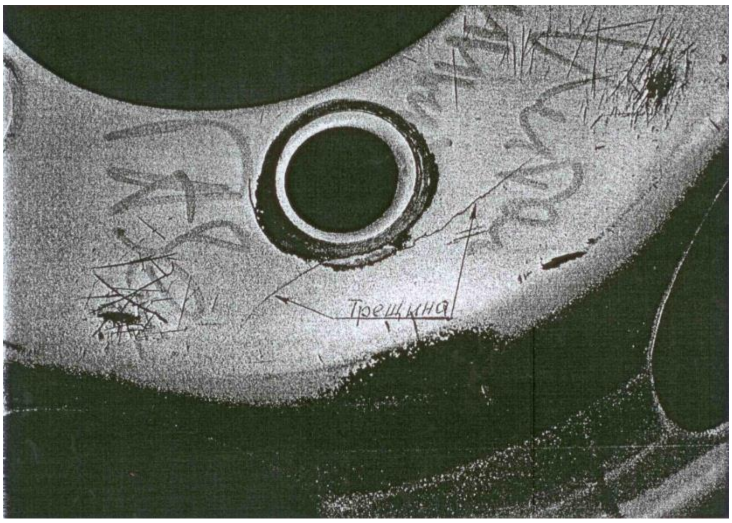


Figure 5. Fatigue crack after testing near stud hole

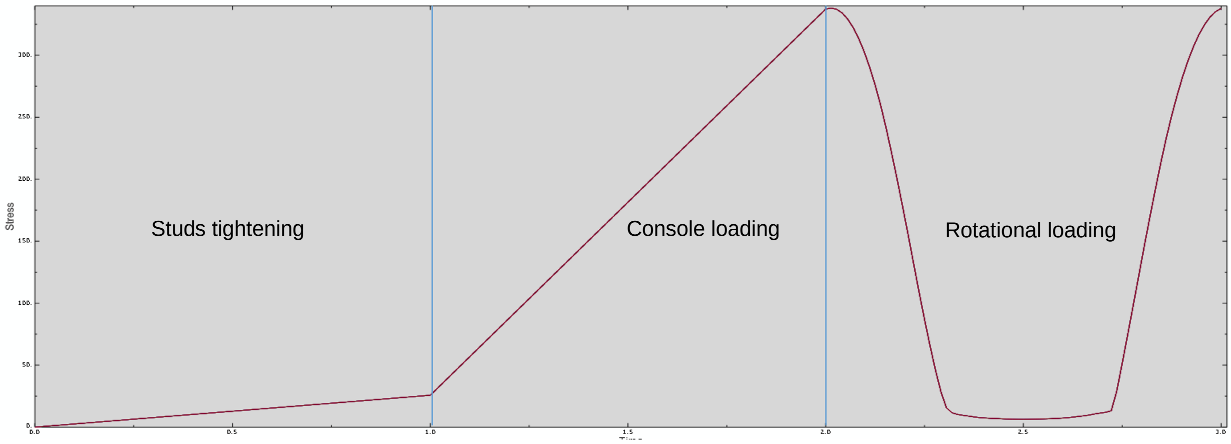


Figure 6. History of maximum principal stress in the critical point

Then, durability of the disk was analyzed in SIMULIA fe-safe. This system can directly use odb-file from Abaqus/Standard containing the stress field at each increment of the rotating load for a full cycle. This cycle was defined in fe-safe as a load block. In addition, a new material file for 07GBU was created, containing elastic-plastic properties and and endurance curves (fig.7).

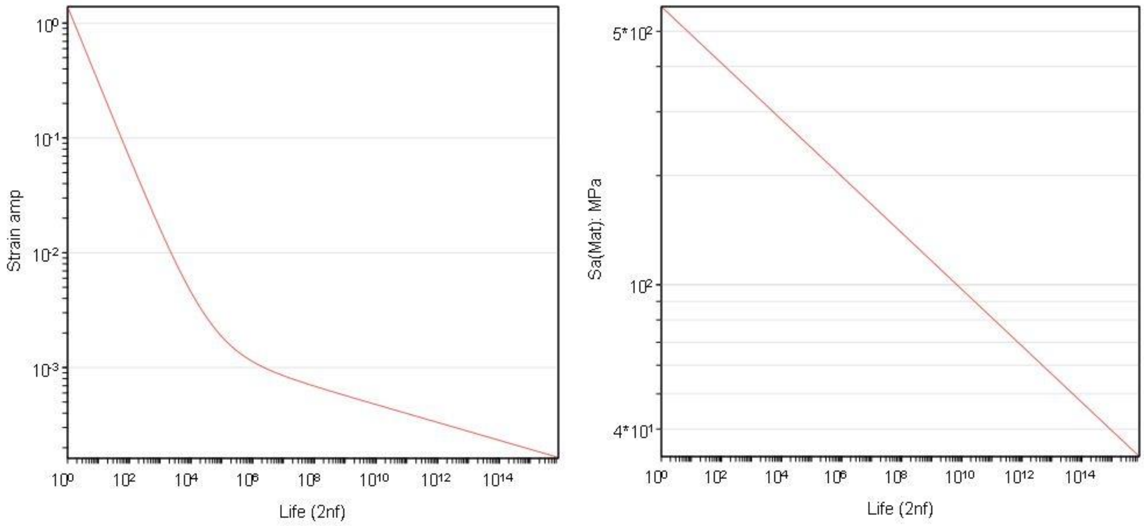


Figure 7. E-N and S-N curves

For durability assessment of this HSLA steel combined strain Brown-Miller criterion with Morrow mean stress correction was used. The analysis identified an element with a lowest life equal to 141 905 cycles, close to the stud hole (fig.8). The factor of strength (FOS) for 600 000 cycles was equal to 0.8. FOS is the factor which, when applied to either the loading, or to the elastic stresses in the finite element model, will produce the required target life at the node [1].

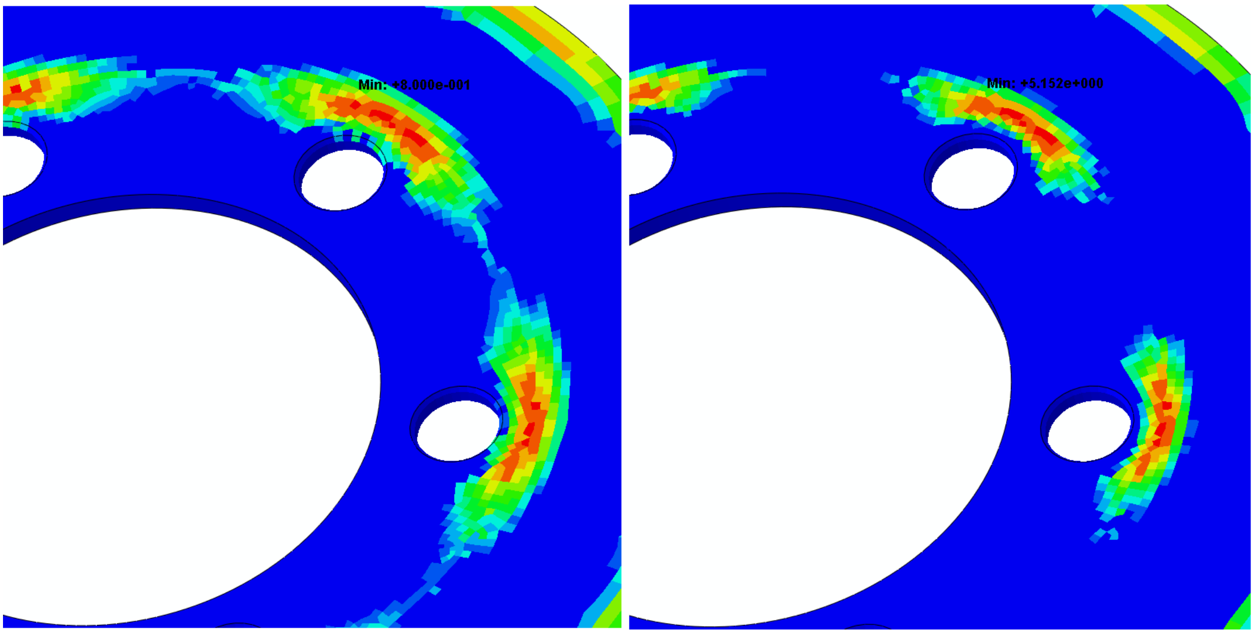


Figure 8. Factor of strength for 600000 cycles (left) and durability (right)

The critical location corresponds well to the area of typical fatigue cracks seen on this disk. The disks made from Severstal steel usually passed standard fatigue test, with cracks appearing only after road tests (fig.5 shows failed disk made from steel from another supplier). Therefore, simulation results based on the simplified disk FE-model were deemed to be unrealistic as they showed too low fatigue strength of the part. A decision was taken to simulate the complete central disk portion and take into account its manufacturing pre-history, including the thinning distribution, hardening and residual stresses, estimating their influence on durability. In spite of the generally negative effect of cold sheet metal forming on durability, some redistribution of stresses over the part was expected due to lower stiffness of the whole component in comparison with merely its central flat area.

3. Simulation of stamping process

The GAZ Group uses AutoForm software for sheet metal forming simulation. Nevertheless, it was not directly used in this work. AutoForm uses shell element for the simulation, while Abaqus model used hexahedral solid element. This element was used, in particular, to adequately take into account through-thickness stresses, caused by studs tightening and further rotating bending load. Unfortunately, the stamping process could not be directly simulated in Abaqus/Explicit due to trimming and piercing operations between two forming operations. Such operation types requires not only deletion of elements in scrap and holes areas, but also FE-mesh remeshing to guarantee smooth hole and blank boundaries. As Abaqus does not have this functionality, Stampack (by Quantech ATZ, Spain) was used (fig.9).

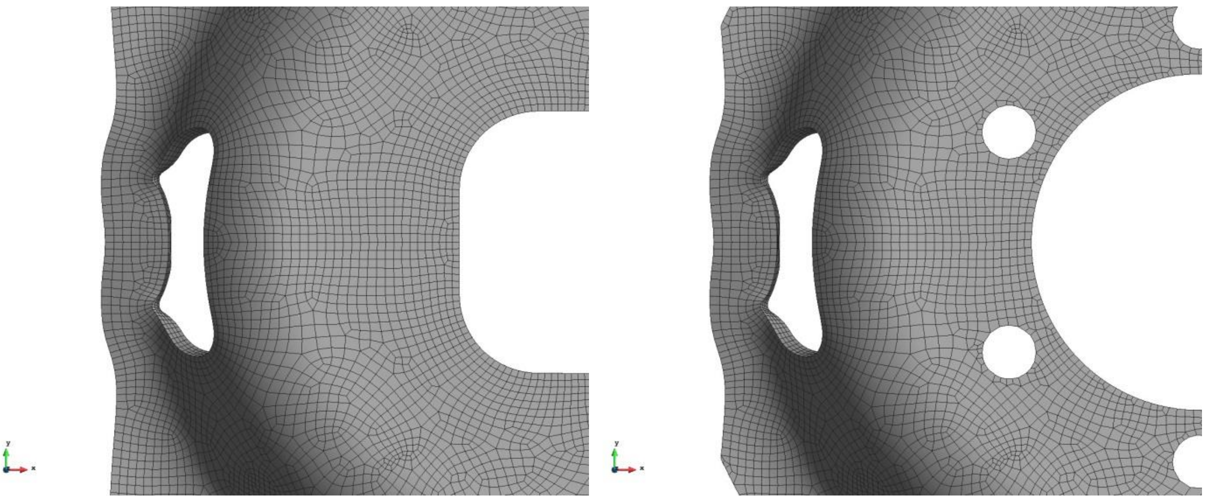


Figure 9. Disk mesh in Stampack before trimming/piercing (left) and after (right).

Stampack is a specialized software for simulation of cold sheet metal forming, using an explicit method of solving. The newest Stampack 7.0.1 has a possibility to use hexahedral solid element for thick sheets and simulate trimming and piercing operations with particular remeshing of hexahedral mesh in the areas of cut lines. The disk stamping process can be divided into three main steps: drawing, trimming/piercing and restrike/forming. In the second step, the central hole and stud holes are pierced, and scrap is trimmed out. The third step is needed to deform the part to its final shape. After that step a springback of the part should be simulated to obtain residual stresses and the true final geometry of the central disk, with consideration of all local thinning and thickening of areas. The initial blank and the process sequence are shown in fig.10.

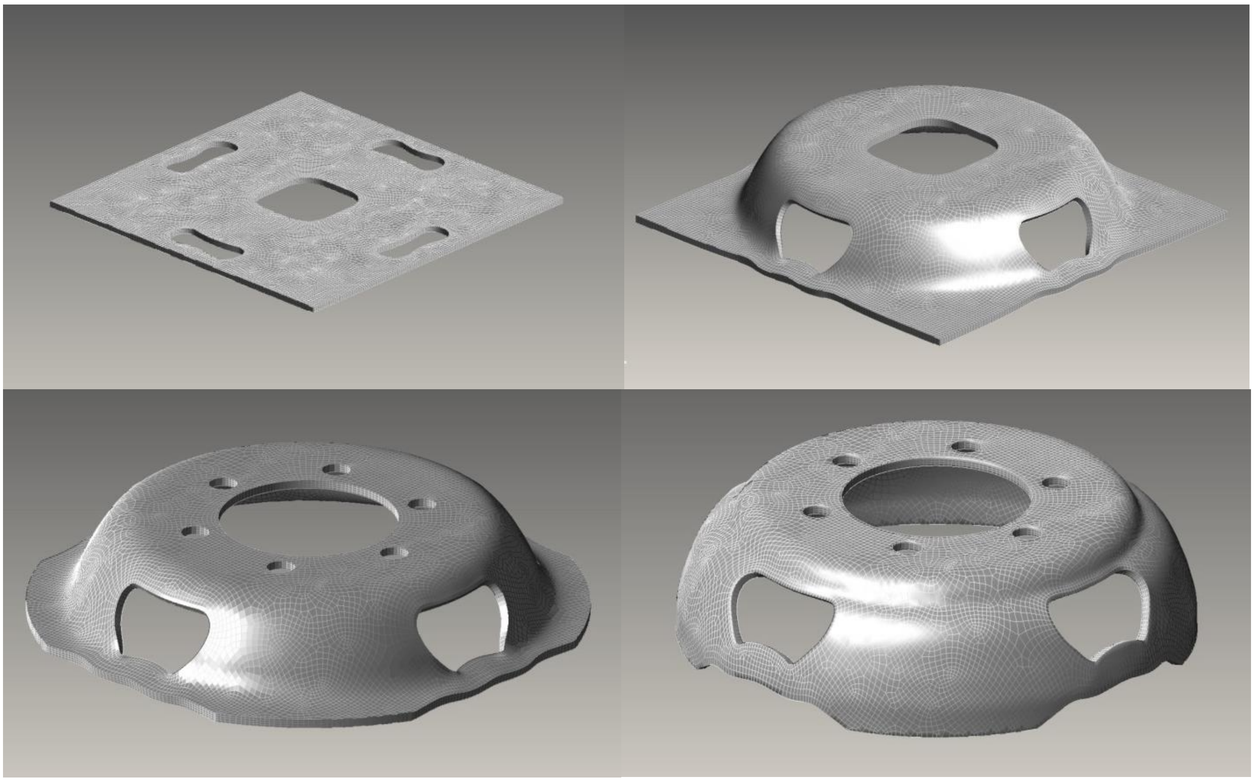
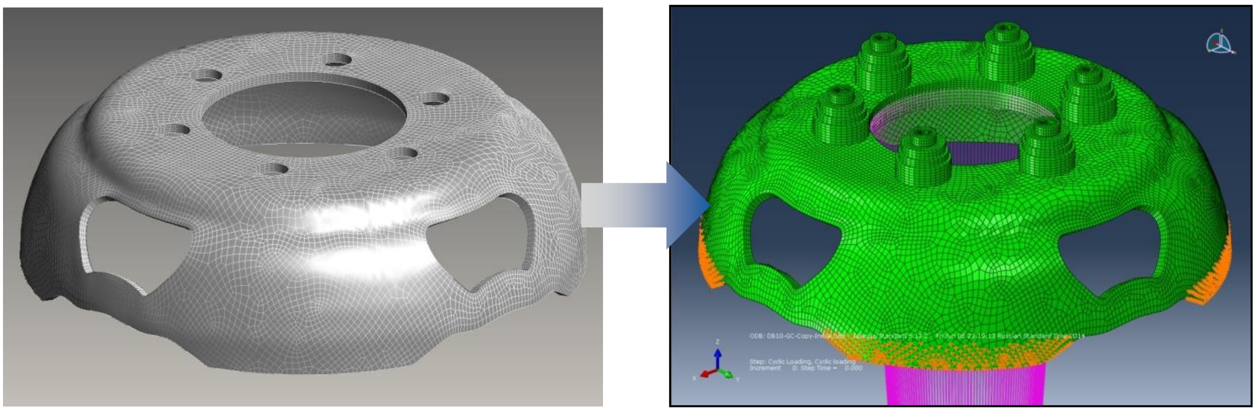


Figure 10. Process sequence in Stampack

The final mesh has 4 element layers through the thickness and reasonably good elements shape quality which can be used in Abaqus simulations. Very simple text editing of the Stampack export file with results, including coordinates and topology of the deformed mesh helped to create an input file for the disk part to be instantiated in the Abaqus



assembly (fig.11).

Figure 11. Export of deformed disk mesh from Stampack to Abaqus model

Export of the hardening and residual stress tensor required some programming efforts, because Stampack can export only stresses and strains as nodal results, while Abaqus allows definition of initial state variables at centroids or integration points. An extra program, created as a result, converted the element by element nodal results to elemental values using averaging at centroids. The program generated input files for Abaqus with equivalent plastic strains and residual stress fields, which were used as initial states of HARDENING and STRESS types (fig.12).

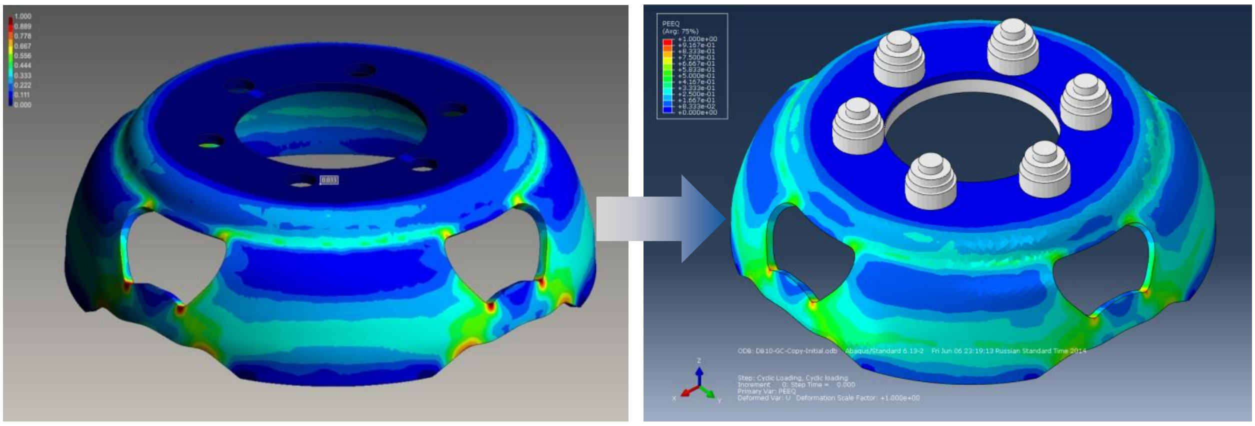


Figure 12. Mapping of hardening from Stampack on Abaqus model

Such conversion and averaging of stresses can potentially cause unbalanced stress condition in the Abaqus model. To resolve this issue an additional balancing process step had to be added. Introducing this step lead to another problem: the step had to be the first in the model, as it deals with the initial model state. On the other hand, the studs tightening step would also need to come first, at least when the model is defined in Abaqus/CAE. An attempt to combine residual stresses with stud tightening in a single first step resulted in unpredictable behavior of the model. Therefore, the problem had to be resolved on the level of manual modification of the input file in the following way: the first load step was balancing and the second was stud tightening (it's possible to do it in an input file, but not in Abaqus/CAE). The third step was the console load application and the fourth – console load rotation, at which stresses were exported in the odb-file for the later fe-safe usage.

3 different cases were simulated:

1. The disk was defined by the deformed mesh from Stampack and considered true part geometry after metal forming process with thinning and thickening. No INITIAL STATE condition was used.
2. The disk was defined by the deformed mesh from Stampack. Hardening was used as INITIAL STATE condition.
3. The disk was defined by the deformed mesh from Stampack. Hardening and residual stresses were used as INITIAL STATE conditions.

The results from the Abaqus simulations were processed in fe-safe in the same way like it was done for the simplified model. All three cases are summarized in Table 1.

Table 1. Results of fatigue analysis considering manufacturing pre-history.

Case	Max value of Maximum Principal Stress at the end of rotation cycle (MPa)	Cycles to failure	Factor of strength for 600000 cycles
1	255	1 359 000	1.08
2	313	1 264 000	1.07
3	362	1 021 000	1.06

The results show the expected tendency of reduction in the number of cycles to failure with consideration of the hardening and also, residual stresses. On the other hand, the influence of residual stresses is relatively low. Evidently, this can be related to the fact that residual stresses and plastic strains in critical areas, caused by metal forming process, are relatively low. The center of the disk is flat and during the drawing operation makes contact with the corresponding flat surface of the punch from the beginning. Frictional forces between the blank and the punch restricted stretching of the central flat portion. It also did not experience any bending. Further, one can observe significant increase of the number of cycles to failure in comparison with the initial simplified model. This was caused by the redistribution of stresses from the center of the part, as a result of increased flexibility.

4. Conclusions

A multidisciplinary simulation of an automotive stamped part was considered in this work from its stamping simulation through to fatigue test simulation. Results demonstrate the importance of detailed modeling and consideration of all factors for the reliability of the results. An integrated environment for such a simulation chain is important and unfortunately not yet implemented well in commercial codes. In spite of the fact that manufacturing pre-history analyzed in this work was not of major significance for the fatigue life of the wheel disk, the GAZ Group has other parts, where such analysis is both important and crucial. This work constitutes a good start for the development of standardized methods and their implementation into company processes. In addition, a further potential continuation of this project is a joint effort with a steel maker to identify an optimal alloy composition for the steel, combining extended durability and good formability, which can be contradictory requirements. Shell models should be regarded as a perspective which allows improved utilization of AutoForm results, as a standard code in the company.

5. References

1. fe-safe 6 User Manual, Dassault Systemes Simulia Corp., Providence, RI
2. Abaqus 6.14 Analysis User’s Guide, Dassault Systemes Simulia Corp., Providence, RI
3. Simulation Guide Stampack_Incremental 7.0.1, Quantech ATZ, Barcelona, Spain
4. P.J. Mc Grath, D.G. Hattingh, M.N. James and A. Els-Botes, “Effects of forming process on fatigue performance of wheel centre disks”, 2013
5. Guschin A.N., Pachurin G.V., “Performance properties of deformed mild steel sheet”, Fundamental Research, Vol.11, 2013
6. P.J. Mc Grath, “An investigation of residual stresses induced by forming process on the fatigue resistance of automotive wheels”, 2001