Introduction of a process to automatically translate a coupled aero-thermal simulation into a structural simulation with fatigue analysis

Gabor Matulik

SIMULIA
Dassault Systemes Deutschland GmbH
Meitnerstraße 8, 70563 Stuttgart-Vaihingen

gabor.matulik@3ds.com

Abstract: The author introduces a process, that is based on an in-house development, which enables CFD Simulation Engineers to automatically convert their PowerFLOW®-PowerTHERM® coupled aero-thermal simulation into an Abaqus® structural simulation including a fatigue analysis with fe-safe®. Structural simulation and fatigue analysis are using the results of the thermal simulation as their boundary conditions. With this automation process the time of creating the mesh and parametrizing the model for structural simulations are greatly reduced and the possibility of user errors is also lowered.

1 Introduction

Simulating the behavior of exhaust systems under extreme thermal and mechanical loads is essential for avoiding failures like cracking and fatigue. Traditional workflows require running aerothermal simulations and then manually preparing new meshes for structural analysis, which can be slow and error-prone. The Power-2-Abaqus tool addresses these challenges by integrating PowerTHERM® with Abaqus®, enabling seamless data transfer and automating structural simulations for efficient and accurate analysis.

The vision of the tool is to empower users conducting aerothermal simulations by enabling them to seamlessly reuse their geometry and results for structural simulations, fatigue analysis, and geometry optimization. Utilizing the same mesh for two different modeling fields reduces the overall preparation time. Connecting the members of the fluid and structural teams beyond simple boundary condition exchange helps for the colleagues to gain a deeper understanding of a complex system.

2 Power-2-Abaqus Workflow

2.1 Workflow concept

Aerothermal simulations alone are often insufficient for the design and optimization of many components, as they do not account for the critical structural and fatigue factors that influence performance and durability. It is essential to integrate structural simulations and fatigue analysis into the design process to ensure comprehensive evaluation and optimization.

Figure 1 highlights the conventional workflow for a typical company, with a fluids (CFD) and a structures (FE) team. The CFD user prepares the geometry, makes the mesh, creates the simulation model and performs aero-thermal simulation. Then exports the necessary boundary conditions, which the FE user needs to import into the structural simulation model, which took similar effort to create. However, it is easy to note the following inefficiencies in the conventional process:

1. Generating multiple meshes for Powerflow®/PowerTHERM® and for Abaqus®
2. Data transfer between teams and/or departments
3. Manual generation of input file for Abaqus®, which can be time consuming for complex cases

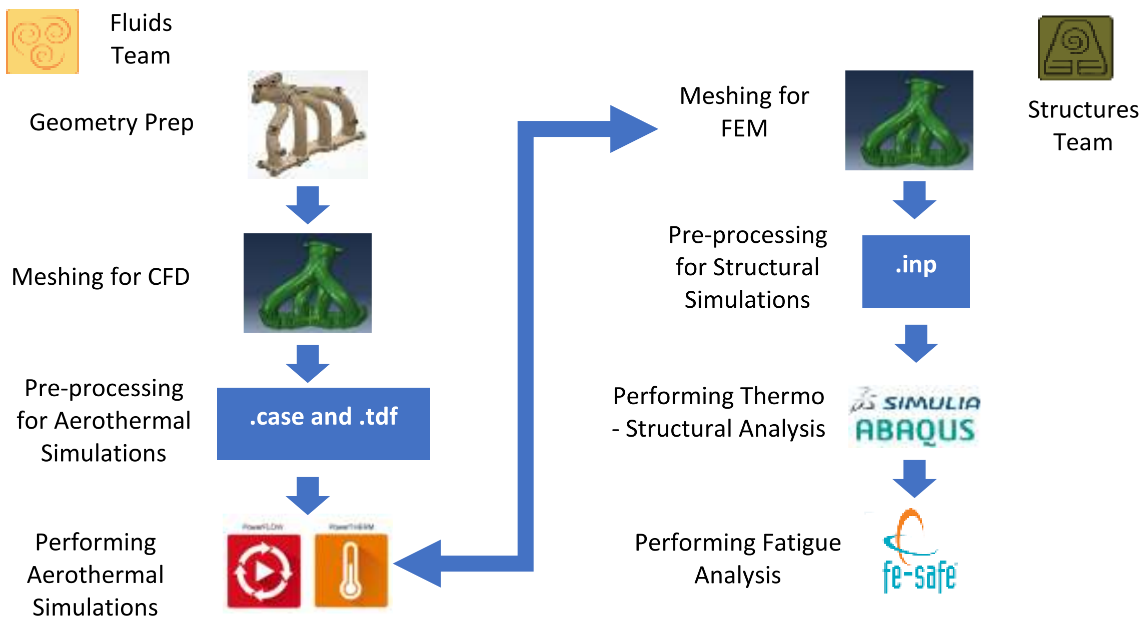

Figure 1: Conventional Workflow

The Power-2-Abaqus tool aims to improve the inefficiencies of the conventional workflow as shown in Figure 2. Utilizing the similar requirements for the structural mesh of the Abaqus® model and thermal mesh of the PowerFLOW®/PowerTHERM® simulation, the tool enables the reusability of the mesh with a conversion step to 2nd order elements in order to provide suitable structural results. It also allows for the automatic creation of input files for Abaqus® simulation, seamless data transfer and mapping of the temperature field from PowerTHERM® to Abaqus®. Moreover, lets the members of the fluids and structural teams to cooperate on a deeper level, shorten the iteration loops for the aero-thermal and fatigue analysis, contributing to an overall accelerated design process.

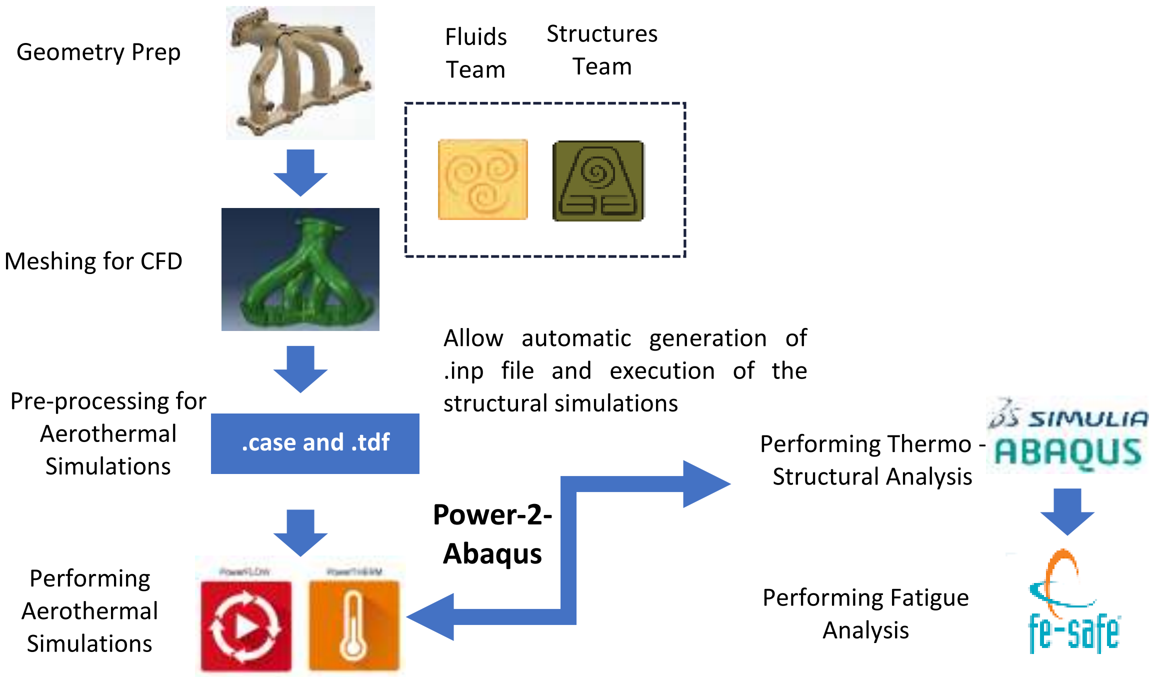

Figure 2: Workflow with Power-2Abaqus tool

2.2 Workflow steps

1. **Geometry export and analysis:** The tool extracts the mesh from the PowerTHERM® .tdf file and converts it into a NASTRAN format. It further processes the model and generates a geometry analysis report for the user.
2. **Geometry Processing:** After controlling the geometry analysis report from step 1, the users can modify the input file for the tool, which is then reprocessed during this second step.
3. **Temperature Export:** The tool exports the temperature fields from PowerTHERM® in .odb and .inp formats, ensuring compatibility with Abaqus®.

4. **Full Abaqus® Input File Creation:** The processed geometry and temperature data are used to generate the full Abaqus® input file, which is ready for simulation.
5. **Abaqus® Simulation Run:** The tool initiates the Abaqus® simulation, utilizing multiple CPU cores to accelerate the computation.

2.3 Capabilities and limitations of the tool

The process is controlled with an input file (JSON). This file contains all parameters for the simulation setup. The user has great control over the complete workflow: besides general settings like the path of SIMULIA® software executables or path to working directory, the configuration allows to setup assemblies or parts to be excluded, bodies to handle as rigid, surface-to-surface coupling, loads and pre-tensions and also to configure which steps to execute.

The current scope of the tool is limited to structural simulations of exhaust components, that can later be used for fatigue analysis. Therefore, the following analyses and capabilities are present:

- Elastic structural simulations accounting for thermal and mechanical loads using Abaqus® Standard Solver
- Capability to work with shell and solid elements
- Direct contact can be specified as surface-to-surface coupling, but the contact type is limited to simple friction
- Ability to automatically apply Pre-Tension values for bolts and screws

Despite these limitations, the tool is built on solid foundations and it is easy to extend the capabilities.

2.4 Material database

Abaqus® inherently does not have a predefined database. This is because it is a general-purpose tool, which can solve many types of complex problems, each requiring an individual and specialized set of material data.

However, to ease the user moving between PowerTHERM® and Abaqus®, a database is created which may be populated and used by the user to perform supported analysis types. The database stores the thermal properties such as density, specific heat and heat conductivity and the elastic properties, like Young’s modulus, Poisson’s ratio and expansion coefficient. This material database also contains a section for the friction coefficients between different materials.

In order to update and extend the material database a graphical interface was created, which lets the user to quickly put new materials, or change the existing ones.

2.5 Supported mesh elements

The Power-2-Abaqus tool only supports tetrahedral elements for solids and triangular elements for the surfaces. Those elements were chosen in order to minimize the numerical errors in mapping during the coupled PowerFLOW®/PowerTHERM® simulations and as the objective was to reuse the same mesh for aero-thermal and structural simulations, only support for these elements was programmed in the tool.

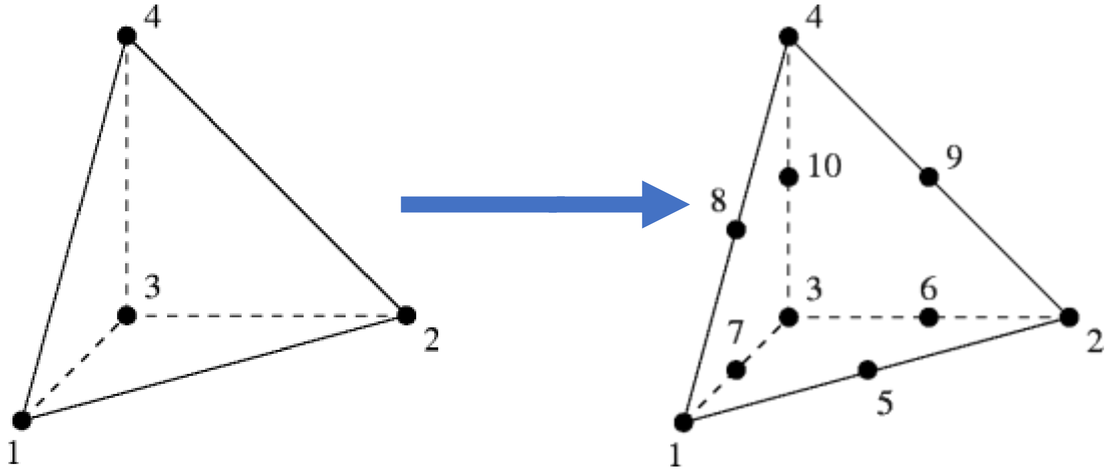

Figure 3: 1st Order tetrahedral volume elements converted to 2nd order elements

To improve the accuracy of structural simulations, the tool uses ANSA to convert the solid tetrahedral elements from 4 nodes to 10 nodes as seen in Figure 3, and the shell and surface elements from 3 node triangular elements to 6 node triangular elements as seen in Figure 4.

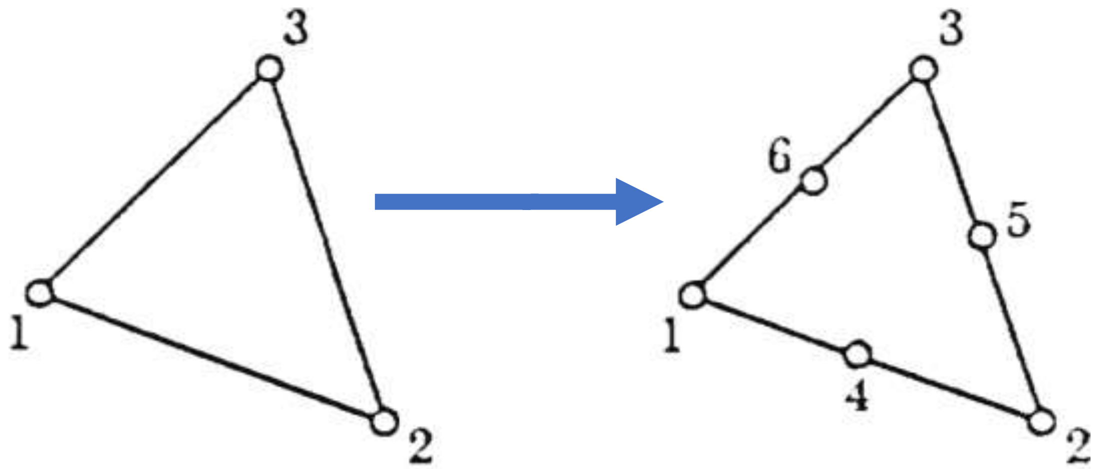

Figure 4: 1st Order triangle surface elements converted to 2nd order elements

3 Examples

In the following section two small IP-free examples will be presented from the tutorial section of the tool.

3.1 Exhaust manifold simulation

In this example the simulation of a simple exhaust manifold is shown. This part is subject to high thermal and mechanical loads. The boundary conditions are:

- Inlet Temperature: 900°C
- Flow rate 0.05 m^3/s

The tool automatically maps the temperature data from PowerTHERM® to Abaqus®, enabling accurate structural simulations without manual setup.

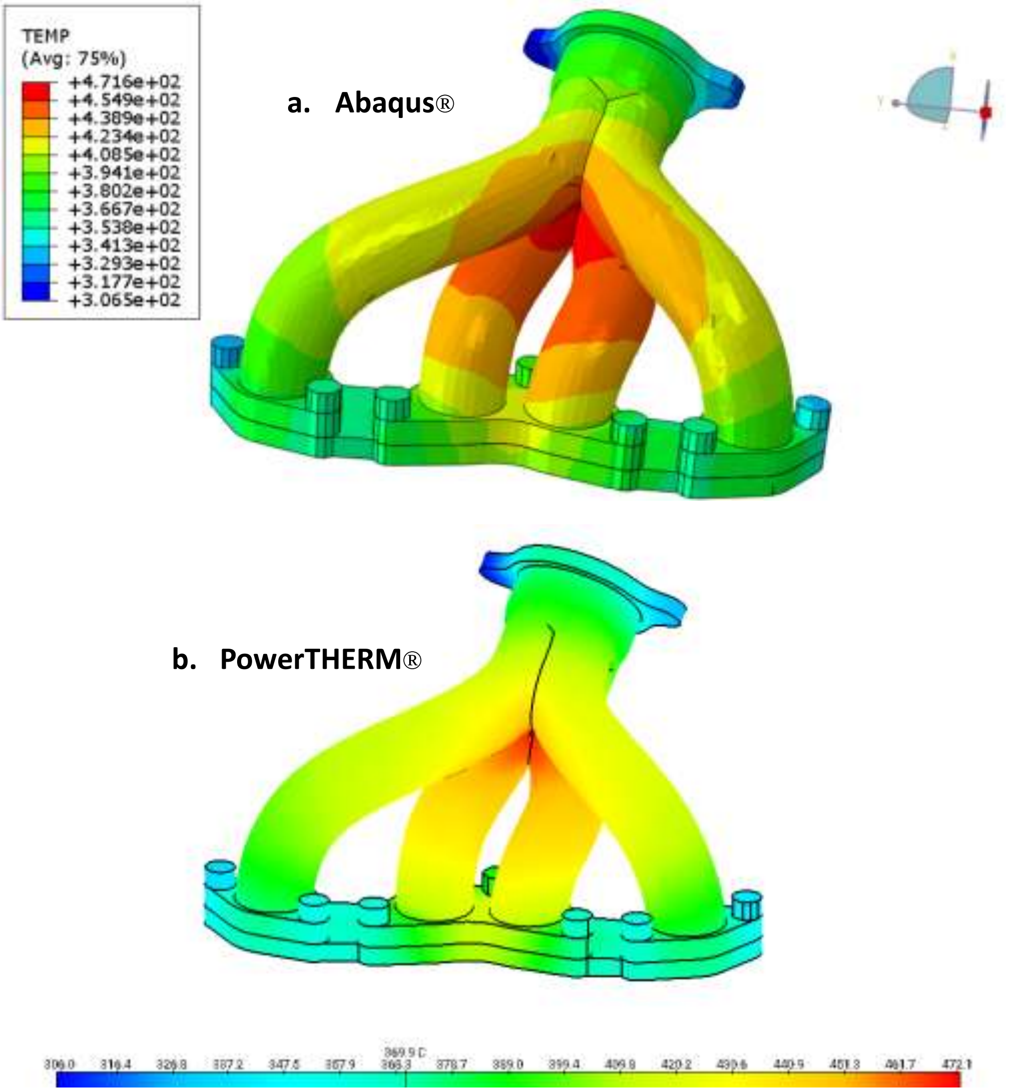

Figure 5: Mapping of the temperature field from PowerTHERM® to Abaqus®

This streamlined process allows engineers to quickly transition from thermal to structural analysis. The critical regions are (as expected) around the screws and the top part where the pipes are joined.

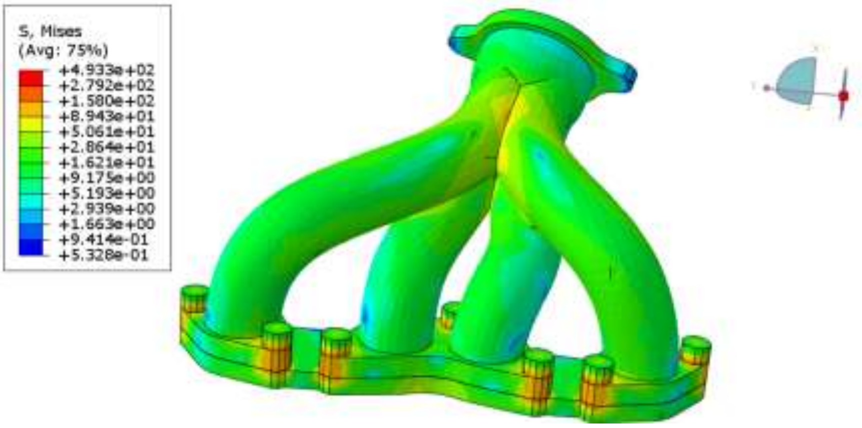

Figure 6: Von Mises stress distribution (Log scale) [MPa]

3.2 Motorcycle exhaust system

This example involves simulating a motorcycle exhaust system, which faces both thermal stress and mechanical impacts (such as the rider's foot). The exhaust system simulation includes:

- Inlet temperature: 900°C
- Foot impact load: 150N at the exhaust tip

By using shell elements for the simulation, Power-2-Abaqus allows the user to reduce model size while allowing them to model the complex interactions between thermal and mechanical forces. The mapping of the temperature is again perfect between PowerTHERM® and Abaqus® (as shown in Figure 7). The tool ensures accurate stress distribution across critical areas (as seen in Figure 8), enabling engineers to assess both thermal performance and structural durability.

This example highlights the tool’s versatility in handling different load conditions, component geometries and meshing strategies, allowing engineers to simulate real-world scenarios more effectively.

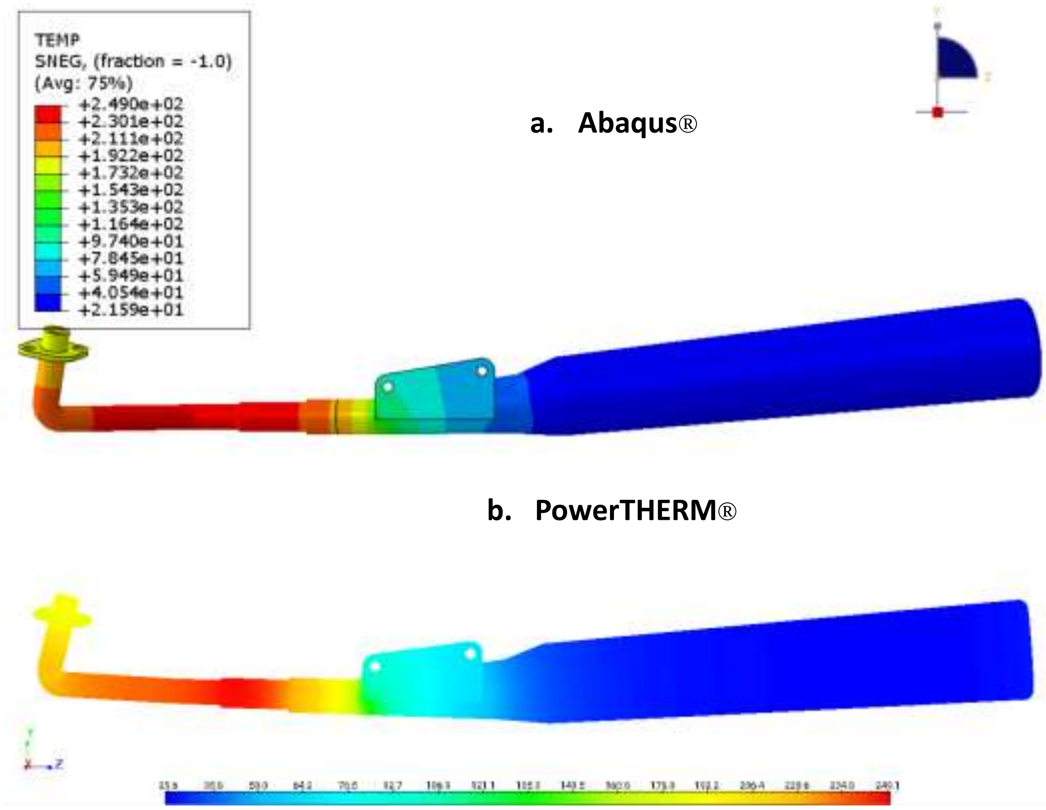

Figure 7: Mapping of the temperature field from PowerTHERM® to Abaqus®

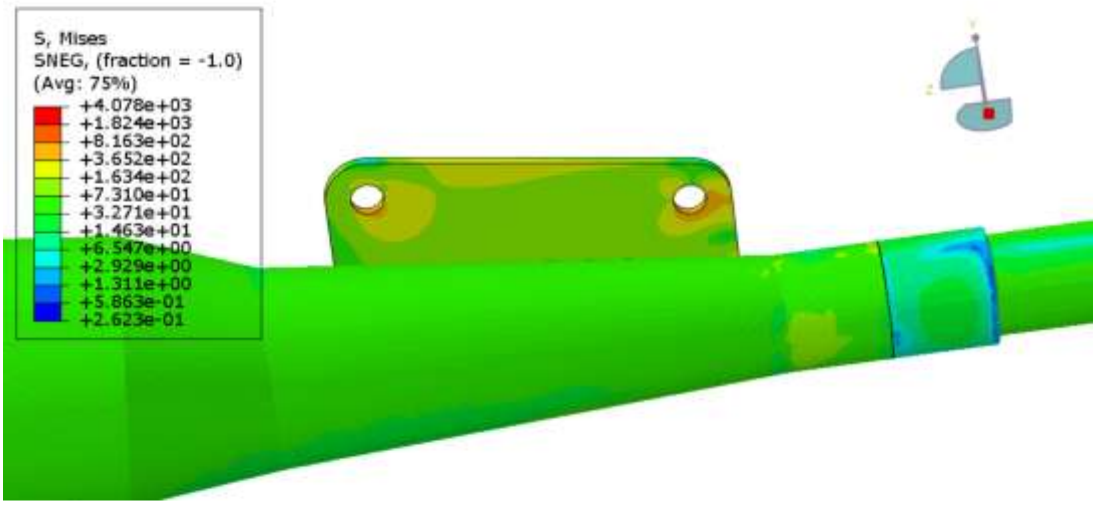

Figure 8: Von Mises stress distribution (Log scale) [MPa]

4 Future applications and Conclusion

Power-2-Abaqus is designed for flexibility. While it’s optimized for exhaust system simulations, the tool can easily be expanded to other applications. With its modular architecture, it is possible to add new features like more complex boundary conditions or other advanced simulation capabilities of Abaqus®.

The tool’s automated workflow and accurate data mapping lay the foundation for integrating aerothermal and structural simulations with fatigue analysis tool fe-safe® or with geometry optimization platforms like Tosca®, opening the door to even more comprehensive simulation capabilities.

Power-2-Abaqus transforms the simulation workflow for exhaust system design by bridging aerothermal and structural simulations, automating critical steps, and improving team collaboration. Its focus on mesh reusability, file automation, and precision makes it a valuable tool for engineers aiming to reduce time-to-market and optimize the performance of thermal-mechanical systems. Whether applied to the exhaust system or expanded to other application areas, Power-2-Abaqus ensures engineers can deliver durable, high-performing designs with greater efficiency.