Abaqus & OpenFOAM Interface for Thermal-Stress Analysis

Enric Aramburu, Jordi Arbiol, Xavier Ballesta, Pablo Cruz, Jordi Viñas

IDIADA Automotive Technology, S.A

The usage of CFD simulation has nowadays become an essential tool for automotive development centres. Current development programs rely on CFD tools to study the vehicle's aerodynamics, to assess the cooling system or to develop the HVAC systems amongst others. In recent years, the range of CFD applications and the accuracy requirements have grown.

Presently, the most common way to access CFD software is through commercial codes. However this approach is becoming unfeasible in certain markets and scenarios. This is especially true for the development of products in which CFD analysis is a required previous step, but in which design requires both thermal stress analysis as well as cost-effective development.

OpenFOAM is an open source CFD code developed by OPENCFD. The core technology of OpenFOAM is based on public known algorithms which are the same as the ones widely used by the well known commercial codes.

In this paper the interface between Abaqus and OpenFOAM is described in the context of a case study of a thermal stress analysis of a coolant tank. The generated interface extracts the pressure, wall temperature and convection coefficient distribution from the OpenFOAM analysis and maps the mechanical and thermal loads over the structural ABAQUS FE-model. This interface allows the engineering team to carry out the thermal stress structural development with an accurate thermal and mechanical load prediction based on CFD analysis without compromising the development cost

1. Introduction

The objective of this paper is to illustrate the approach followed to carry out a thermal stress analysis of a coolant tank. The study involves both fluid dynamics and mechanical analysis. OpenFOAM and abaqus are used for the CFD and the thermal stress analysis respectively.

For confidentiality reasons a simplified expansion tank has been created and used as a study case in this paper.

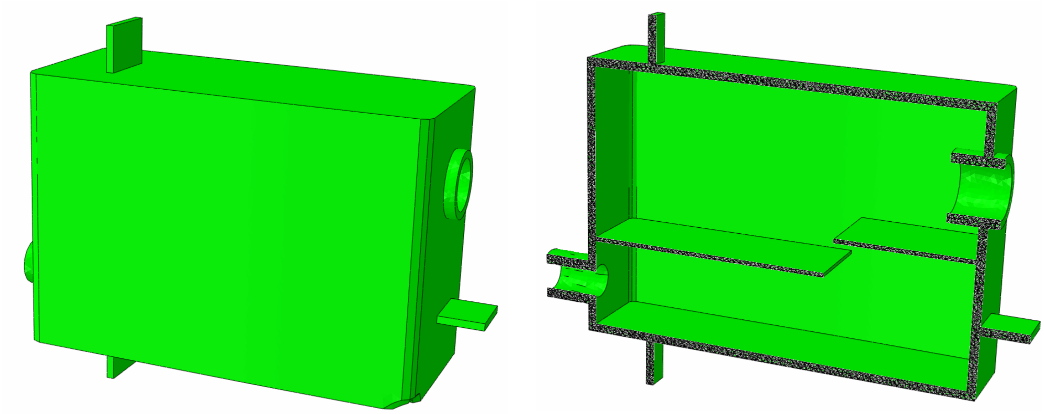


Figure 1. Model studied

The main objective of the study is to highlight the coolant tank regions where the stresses exceed a target threshold.

In order to be able to carry out the uncoupled thermal stress analysis an IDIADA tool was generated to map (1) pressure; (2) fluid wall temperature and (3) convection coefficient distributions from the OpenFOAM analysis to Abaqus FE model.

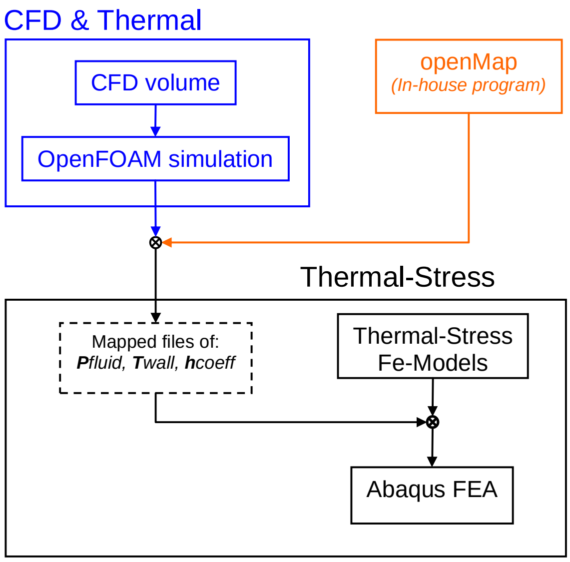
2. FEA Approach

The methodology is a sequence of multidisciplinary analyses chained one after the other.

First of all, the CFD analysis is carried out using OpenFOAM. This kind of analysis allows the study of the behaviour of a fluid under certain conditions of velocity, pressure and temperature.

CFD and thermal FE model had a boundary in common, that is the wet surface.

As mentioned above, the in-house program called openMap was developed. OpenMap maps (1) pressure; (2) fluid wall temperature and (3) convection coefficient distributions from the OpenFOAM analysis to Abaqus FE model. Once the mapping is performed, a thermal-stress analysis can be carried out.



OpenMap generates the following files:

- **Hcoefficient.csv**: Convection coefficient on each wet surface facet.
- **Pressure.csv**: Wall pressure on each wet surface facet.
- **Temperature.csv**: Wall temperature on each wet surface facet.
- **Velocity.csv**: Velocity on each wet surface facet.
- **Pressure.inc**: Pressures mapped over the Abaqus FE model.
- **T-Hcoefficient.inc**: Temperatures and convection coefficients of mapped over the Abaqus fe-model.

Once the three distributions (P, Twall and hcoefficient) are mapped, the steady-state thermal analysis is carried out using the temperature and convection coefficient distributions. The thermal load from the thermal analysis simulation results together with the mechanical load from the CFD pressure distribution feed the final stress analysis.

3. FE-Model

Some specific requirements have to be taken into account when preparing a multidisplicinary FE model.

3.1 OpenFOAM

The basic file structure of an OpenFoam case is shown in Figure 2 below. It contains the basic files required to run an application. Each case must have the following basic folders:

- Time directory which contains individual files of data for the particular fields. The data could be initial values and boundary conditions required to define the problem. OpenFOAM must be initialized even when the solution does not necessarily require it. The name of each time directory must be the simulated time at which the data is written. Since most of the applications are run at time zero, the time directory is usually named as 0.
- Constant directory which contains the mesh data in separate files for points, boundaries, cells, faces, etc. it also contains information about the physical properties required for the particular application. The physical properties file could be a porousZone file (used to define the porosity in a medium which has been defined as a porous medium) or transportProperties file (which includes the properties of the fluid which influence the flow of the fluid inside the control volume).
- System directory which contains data about the solution procedure. It has at least three folders called:
    - controlDict, which contains data run controls like star/end time, time step, parameters for data output.
    - fvSchemes, which contains the data regarding the discretisation schemes to be used during the run time.
    - fvSolution, which contains the solver equations, tolerances and other algorithm controls.

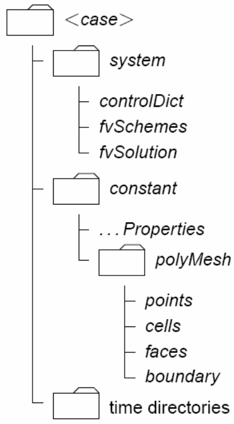


Figure 2. OpenFOAM model structure.

3.2 FE-Model for Abaqus/Standard, thermal analysis

As stated before, OpenFOAM allows us to obtain the pressure, fluid temperatures and convection coefficients in the wet surface. The temperature and convection coefficient distributions will be the boundary conditions for the thermal analysis.

A thermal analysis has to be done in order to obtain the component thermal stresses. Consequently, a proper model has to be defined. Such a model has its own guidelines and its particular model properties.

Regarding meshing guidelines, first order elements are desired for thermal analysis. This choice is due to the fact that first order elements are less likely to have spurious oscillatory results than second order elements.

We have to keep in mind backwards compatibility with the previous CFD analysis. In this scenario first order tetrahedrons are a good choice.

First of all, and most importantly, first order tetrahedrons use a triangular shell as a boundary; furthermore, full compatibility is possible between our CFD and thermal FE model. This way allows us to easily share CFD results with our thermal model due to the fact that the wet surface has exactly the same element shape.

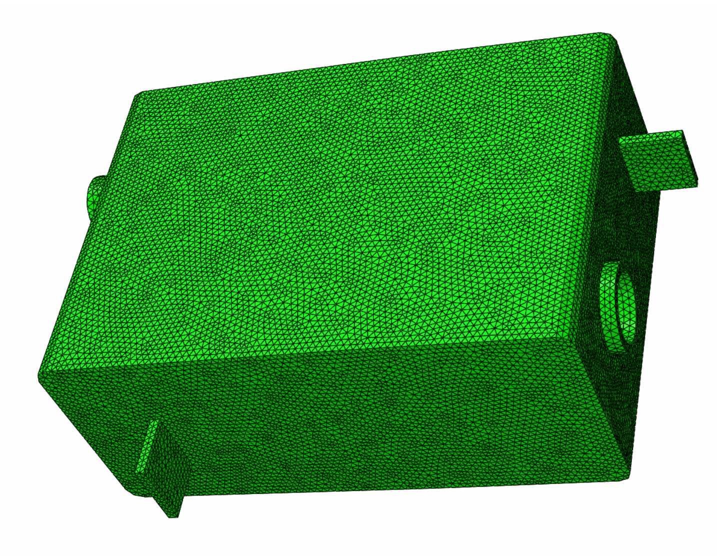


Figure 3. Example of mesh

Taking into account the reasons mentioned above, DC3D4 elements were chosen in this study for the thermal analysis.

3.3 FE-Model for Abaqus/Standard, mechanical analysis

Abaqus/Standard offers the possibility to perform coupled and uncoupled thermal-stress analyses. In the present case study, the uncoupled scheme is chosen as far as the thermal solution does not actually depend on the stress solution.

Thermal load is obtained from the previous Abaqus/Standard steady-state thermal analysis.

Temperatures are read and mapped automatically by Abaqus to the structural mesh.

The fluid pressure defines the mechanical load and is mapped from the CFD analysis results using IDIADA's tool OpenMap.

In a thermal-stress analysis capturing the bending behavior could be a key issue. For this reason, it is recommended to use second order element for this kind of analysis. Additionally, if part life has to be considered in the development, the element surface stress should be considered during the fatigue analysis.

Taking into account the reasons mentioned above, C3D10I elements were chosen in this study for the stress analysis.

4. CFD and Thermal-Stress Model Set Up

4.1 OpenFOAM

The CFD analysis was submitted considering the appropriate (1) fluid properties (water); (2) boundary conditions: inlet and outlet pressure and temperatures, wall conditions, etc; and (3) for this case study the turbulence model.

Figure 4. Plane cut of the wet surface used as boundary for CFD Mesh

4.2 Thermal analysis

The thermal loads outputted by the CFD analysis are related to the wet surface. Nevertheless the objectives in our study are the stresses and displacements in the part. Consequently, the temperature inside the part has to be obtained. This is done using an Abaqus thermal steady-state analysis where the temperature and the convection coefficient for the wet surface are obtained from the CFD analysis. Regarding the outer conditions room temperature and standard conditions for convection and radiation are fixed.

4.3 Stress analysis

The final step in order to obtain the stresses and displacements in the part is to carry out the stress analysis.

The nodal temperatures obtained from the thermal analysis are mapped over the mesh using the Abaqus, in this case the keyword **`*TEMPERATURE, FILE=`***`thermal_results.odb`***`, INTERPOLATE`** is used.

The pressure loads are read from the file output by OpenMap and applied directly to the wet surface, so all the necessary data is now available for the stress analysis.

5. Results

For each calculation specific results were obtained, as mentioned above for CFD analysis temperatures, pressure and coefficients are exported, although other results were supervised as for instance, the velocities.

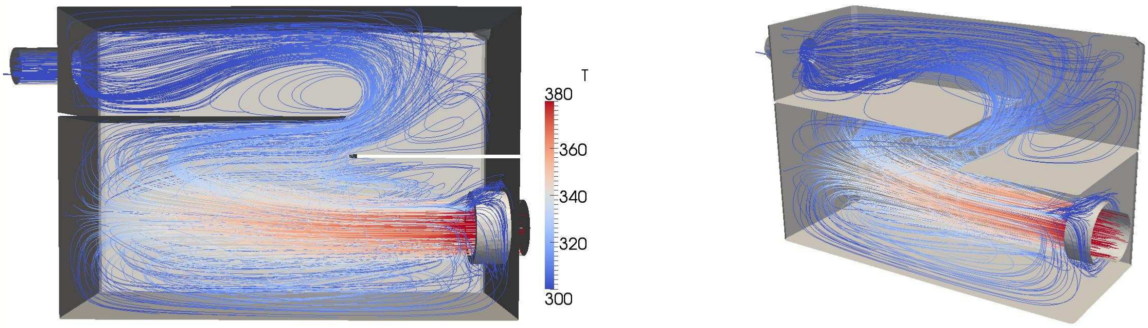


Figure 5. Fluid temperatures (Kelvin degrees)

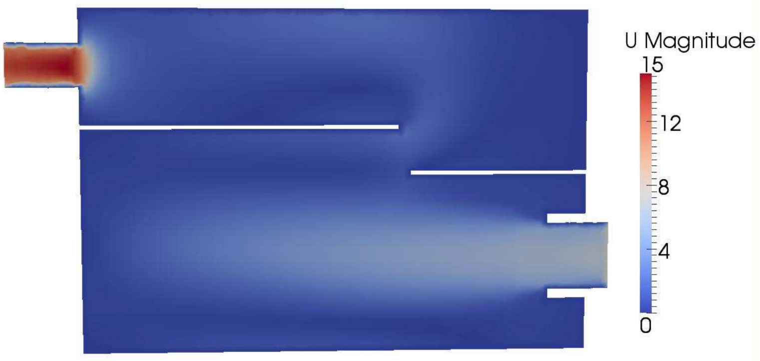


Figure 6. Fluid velocity (meters / second)

Regarding thermal analysis, nodal temperatures and heat fluxes were assessed in order to check the coherence of the thermal loads and boundary condition.

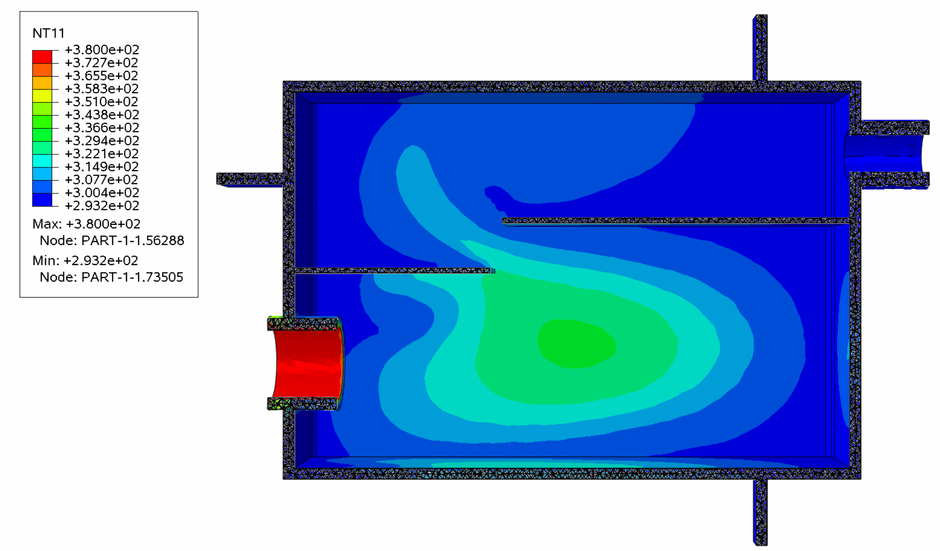


Figure 7. Nodal temperature at the end of the steady-state thermal analysis (Kelvin degrees)

Finally, the stresses and displacements in the stress analysis were evaluated according to the experience acquired in former similar tests.

For the kind of part studied in this example, maximum stresses are typically located near the fixings, and quite often the main cause is the expansion due to the thermal loads.

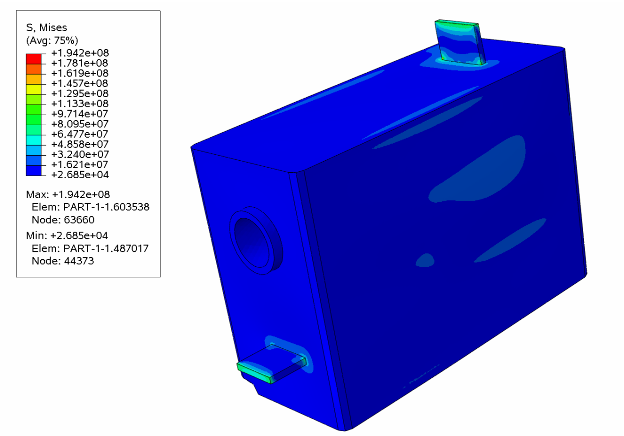


Figure 8. Stresses maximums are mainly located at near fixings (Von Mises stresses in Pascals)

On the other hand, displacements have their maximum values mainly in flat areas.

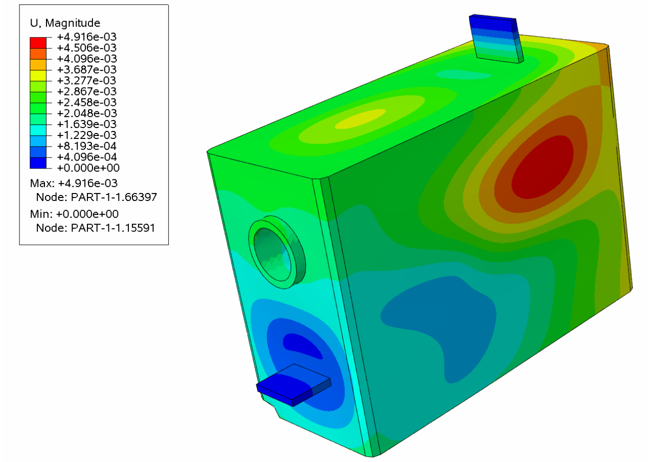


Figure 9. Resultant displacements (displacements are presented in meters)

6. Summary and conclusions

The aim of the presented approach was to obtain the necessary thermal and mechanical loads from a CFD calculation in order to carry out a thermal-stress analysis in Abaqus.

To do that, an in-house code was successfully developed that allowed the analyst to easily map the results obtained from the CFD analysis to Abaqus. This interface allowed the engineering team to carry out the thermal-stress structural development with an accurate thermal and mechanical loads prediction based on CFD analysis without compromising development cost.

This kind of approach allows IDIADA to perform development projects common to the automotive sector such as the present case study.

7. References

1. Simulia Abaqus 6.10 Documentation