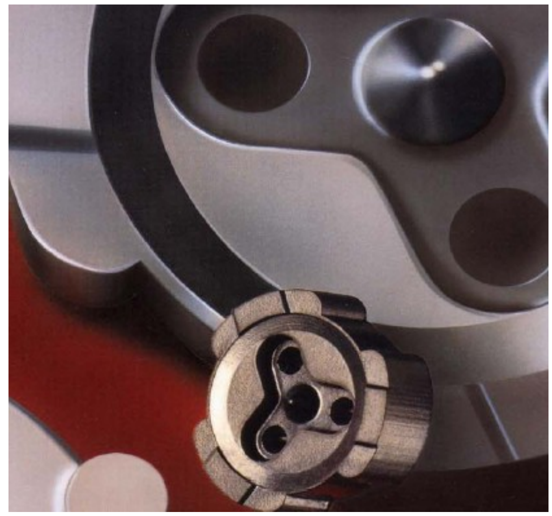
Fluid-Structure Interaction Analysis of a Flow Control Device

Summary

The Vernay VernaFlo® flow controls are custom-designed fluid flow management devices used in a wide range of applications and systems where consistent, reliable operation is essential. Elastomeric rubber components in these devices deform under the influence of upstream variations in fluid pressure. These deformations adjust the orifice diameter and help maintain a constant downstream flow rate. In this Technology Brief the performance of a custom VernaFlo® device is evaluated using the fully coupled fluid-structure interaction solution provided by the Abaqus co-simulation capability. The effects of cavitation on the flow are considered. The computational results compare favorably to available experimental data.



Background

Flow control devices are used in a wide range of fluid management applications and are commonly employed in the automotive, bio-medical, and consumer appliances industries. These devices are designed to maintain a constant bulk flow rate for varying inlet pressures; such pressure variations may result from pipe friction loss, downstream restrictions, distance from the water tower, elevation of the water tap, etc. Minimizing the impact of inlet pressure variation on the flow is essential for the reliable and consistent operation of the applications in question.

In this Technology Brief, the performance of a custom VernaFlo® flow control device from Vernay Labs, Yellow Springs, Ohio, will be studied, and the results will be compared to the experimental flow-rate data sheet of the device.

Figure 1 shows a schematic diagram of the cross-section of typical VernaFlo® device. An elastomeric rubber component is housed inside the flow path. This rubber insert rests on a rigid seat and deforms under the influence of incoming flow. At low operating pressures the rubber component undergoes very little deformation and allows the flow to develop. With increasing upstream pressure, the deformation increases, restricting the orifice diameter and thus limiting the fluid flow. Capturing the interaction between the fluid flow and the structural deformation is critical to accurately predicting the device shape and the subsequent flow behavior; a bi-directional fully coupled fluid structure interaction (FSI) analysis is thus required.

Key Abaqus Features and Benefits

- Fluid-structure interaction analysis capability using co-simulation with MpCCI
- Range of hyperelastic material models for the simulation of large deformation in elastomeric parts
- Deformable-to-rigid body contact capability

Analysis Approach

The effects of the fluid-structure interaction will be studied by coupling Abaqus and FLUENT using the co-simulation technique with MpCCI. With this method the fluid and the structural domains are modeled and solved separately, with solution information exchanged at the fluid-structure interface.

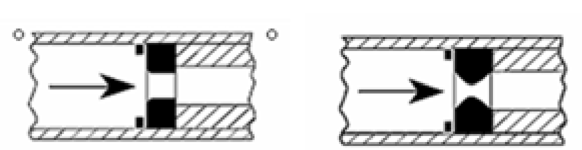


Figure 1: Schematic cross-section of a typical VernaFlo® device

Defining the sub-domain models

The inherent symmetry in the VernaFlo® device allows an axisymmetric analysis to be performed. The Abaqus sub-domain (Figure 2) includes the rubber component, which is modeled using reduced order hybrid axisymmetric elements.

The rubber component presses against a rigid seat which is modeled using a discrete rigid part. Penalty contact with a friction coefficient of 0.5 is defined between the rubber and the rigid seat. The simulation is completed in Abaqus/Standard and includes the effect of geometrical and material non-linearities.

The CFD domain shown in Figure 3 models the flow path, which includes a short upstream section, followed by the section around the rubber component, and ending a long downstream section. The upstream variation in pressure is accounted for using a pressure-inlet boundary condition while the outlet uses a pressure outlet boundary condition with zero gage pressure. To enable local remeshing, the flow path is modeled using triangular elements. The water is modeled as an incompressible fluid and turbulence and multi-phase fluid models are employed to enable the simulation of cavitation. The flow equations are solved using the steady-state implicit segregated solvers in FLUENT.

After the Abaqus and the FLUENT sub-domains have been created the fluid-structure interface is defined using MpCCI. A schematic diagram of the interaction definitions, which include the interaction surfaces and the desired solution quantities to be exchanged, is shown in Figure 4. Abaqus receives the fluid pressures from FLUENT and returns the resulting structural deformation to FLUENT.

In addition, MpCCI automatically handles the transformations necessary to account for the difference in axisymmetric conventions used by Abaqus and FLUENT.

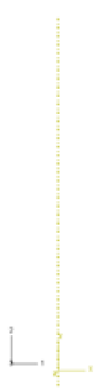


Figure 2: Axisymmetric model of the rubber insert

Figure 3: Axisymmetric model of the flow path

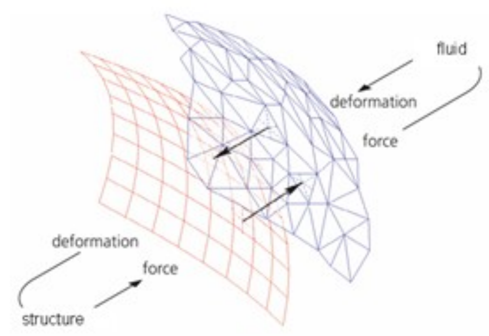


Figure 4: MpCCI interface definitions

Addressing Domain pinching issues

During the analysis, a constant fluid topology is required in FLUENT. As the rubber component deforms in response to the upstream fluid pressure, it will come into contact with its rigid seat, thus “pinching” the fluid domain completely. Since pinching terminates the coupled analysis, a special modeling technique has been used to prevent such an occurrence.

As shown in Figure 5, the rigid contact surface in Abaqus (dashed line) represents the true location of the rigid seat; however, the FLUENT wall zone that corresponds to the rigid seat has been slightly offset. The offset helps maintain a small, finite clearance at all times during contact between the rubber insert and the rigid seat. While this clearance leads to some localized flow behavior, the overall impact of this gap on the constriction path and the corresponding bulk flow behavior was found to be relatively small.

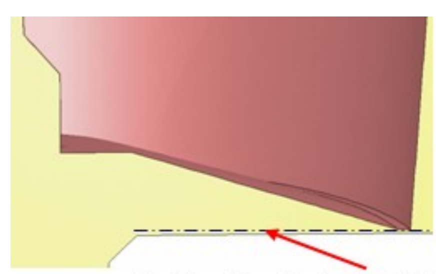


Figure 5: Contact offset to avoid domain pinching

Solution Methodology

The analysis is driven by an applied boundary condition on the fluid inlet pressure; specifically, this quantity is ramped up from 0 to 827 kPa (120 psi). During the simulation, the pressures acting on the rubber insert in the fluid sub-domain are mapped and transferred to the structural sub-domain in Abaqus via MpCCI. Abaqus then computes the deformations and the resulting stress state in the structure. The interface deformation quantities are then mapped and transferred from the structural sub-domain to the fluid sub-domain in FLUENT via MpCCI. This process of exchanging solution quantities continues incrementally until the analysis completes.

Results and Discussions

The main objective of the FSI analysis is to determine the effect of the variation of inlet pressure on the bulk fluid flow rate through the device. After presenting these results, the effect that each sub-domain has on the coupled results will be examined.

Global convergence

The fluid inlet pressure boundary condition is applied in an incremental fashion; specifically, the pressure is increased in fixed increments during the analysis. Global convergence between the two sub-domain solvers is obtained by iterating the exchange process several times at the given pressure level.

The nature of the convergence for this analysis is apparent from Figure 6, in which the displacement magnitude of the trailing edge of the rubber insert is plotted versus the global exchange number. This plot considers the increase in pressure from 13.8 kPa (2 psi) to 34.5 kPa (5 psi). During each pressure increment, the converged solution was obtained after five global exchanges.

Bulk flow rate vs. Inlet pressure

The computed flow rate as a function of inlet pressure is shown in Figure 7. Initially, as the pressure is ramped from zero the flow rate increases. As the inlet pressure approaches 138 kPa (20 psi) the flow is found to stabilize at approximately 2.1 liters/min. Further changes in the inlet pressure do not affect the flow rate significantly.

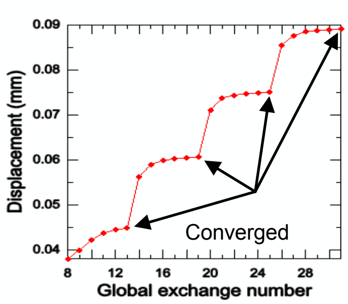


Figure 6: Displacement solution at trailing edge of rubber insert

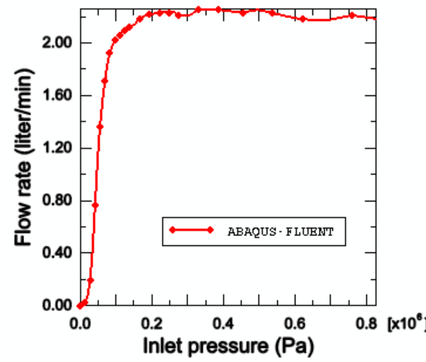


Figure 7: Computed bulk flow rate vs. inlet pressure

Effect of fluid flow on deformation

At any given inlet pressure level, the upstream surface of the rubber device is subject to a fairly uniform pressure distribution. These forces acting on the leading surface are largely responsible for compressing the device and forcing it down on the rigid seat. The pressure drop in the constriction region contributes to further narrowing of the flow path and results in a region of very high stresses as shown in Figure 8.

Effect of deformations on the fluid flow

At a given deformation state, as shown in Figure 9, the cross-section through the constriction path governs the flow behavior. As the flow quickens through the narrow constriction region the fluid pressure drops significantly, resulting in a dramatic drop in the absolute pressure of the liquid; this may lead to cavitation in the fluid.

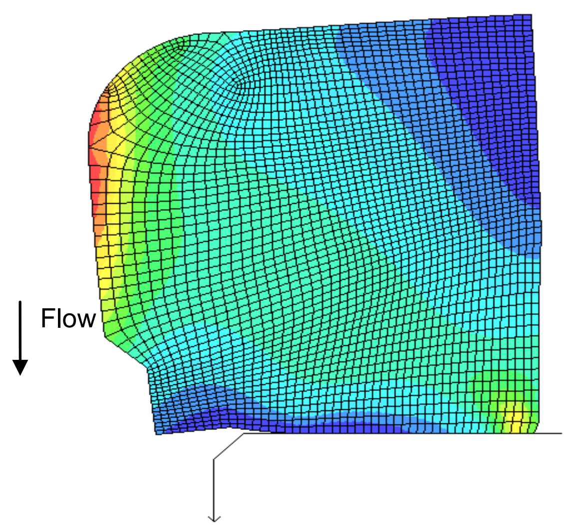


Figure 8: Mises stress in rubber insert at 276 kPa (40 psi) inlet pressure

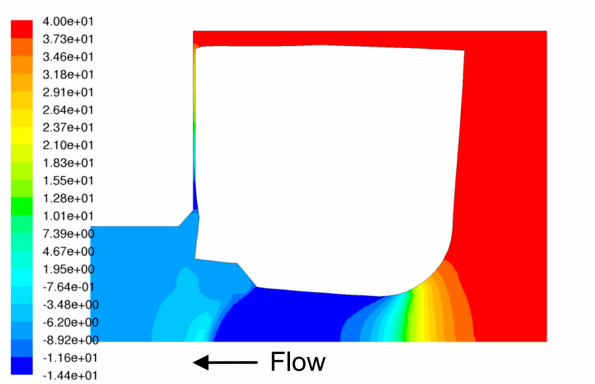


Figure 9: Static flow pressure at 276 kPa (40 psi) inlet pressure

Fluid-structure coupling effects on the flow rate

Figure 10 shows contour plots of the fluid pressures at inlet pressure levels of 138, 414, and 621 kPa (20, 60 and 90 psi). The corresponding deformed shapes are shown in Figure 11 with contour plots of Mises stress.

With an increase in the inlet pressure the rubber device deforms and experiences increased contact with the rigid seat. Partial contact is observed at 138 kPa, with full contact being established at the higher pressure of 621 kPa. The constriction path narrows during the inlet pressure rise resulting in an increased resistance to the fluid flow. However, the increased material stiffness helps maintain the bulk flow rate at a fairly constant level.

Cavitation

As discussed earlier, cavitation effects are observed at higher upstream pressures. Figure 12 shows the vapor region in the flow at an inlet pressure of 276 kPa (40 psi). The phase change is prominent in the downstream region right after the narrowest flow path. As the inlet pressure increases the vapor envelope stretches further downstream but the vapor region collapses before reaching the outlet.

Experimental Validation

Finally, the results from the analysis are compared to experimental results. Figure 13 shows a close match of the computational flow rate results to the experimental data. The flow rate increases from 0 to 2.1 liters/min during the initial pressure ramp-up from 0 to 138 kPa (20 psi) and has near constant flow in the operating pressure range of 138 to 827 kPa (20 to 120 psi).

Conclusion

In this Technology Brief, a fluid structure interaction analysis of the VernaFlo® flow control device using the Abaqus co-simulation technique for FSI is presented. The results obtained are in good agreement with experimental results. This study highlights the importance of FSI in this class of problems and successfully demonstrates the capability of Abaqus to perform such advanced coupled physics simulations.

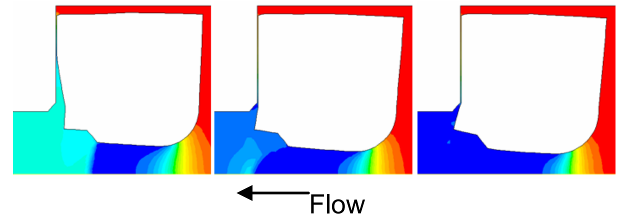


Figure 10: Static flow pressure at 138, 414, and 621 kPa

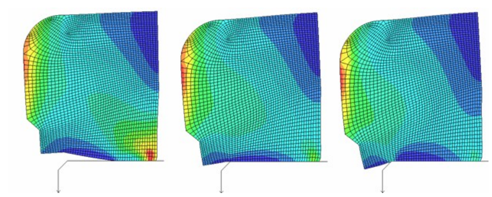


Figure 11: Mises stress at 138, 414, and 621 kPa inlet pressure (20, 60 and 90 psi)

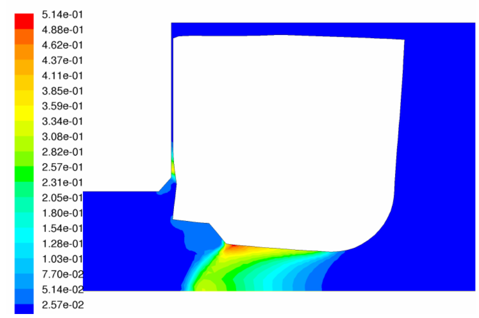


Figure 12: Volume fraction of vapor at 276 kPa (40 psi) inlet pressure

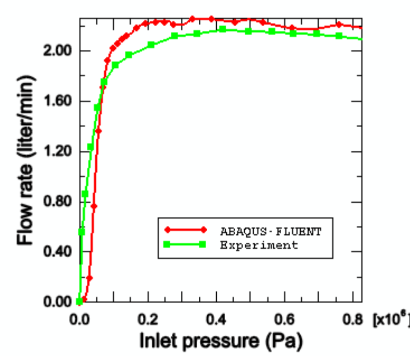


Figure 13: Experimental validation of computation results

Acknowledgments

Dassault Systèmes SIMULIA Corp. would like to thank Jim Bailey at Vernay Labs for providing the model and experimental data for the Vernay VernaFlo® device; and David Schowalter at Fluent, Inc. for assisting with the cavitation modeling.

References

1. Vernay VernaFlo® flow devices and related product information can be found at: www.vernay.com.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief please see the following Abaqus 6.11 documentation references:

- Abaqus Users' Guide on Fluid Structure Interaction Using Abaqus and FLUENT, available via Abaqus Answer 2420.
- Abaqus Analysis User's Manual
  - "Co-simulation: overview," Section 16.1.1

FLUENT References

1. For additional information on Fluent Inc. and the FLUENT capabilities shown in the brief see: http://www.fluent.com.

MpCCI References

1. For more information on MpCCI and Fraunhofer SCAI see: http://www.scai.fraunhofer.de

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com