# Coupled Electromagnetic and Fluid-Structure Interaction Analysis of a Solenoid Valve

## Abaqus Technology Brief

### Summary

Solenoid valves are electromechanical devices for controlling the flow of liquids and gases. Valves of this type are multiphysics systems with electromagnetic, fluid, and structural domains that couple in a complex manner.

An important characteristic from a design perspective is the time required to actuate the valve. Accurate evaluation of fluid and electromagnetic forces and an understanding of how they interact with the valve's structural components is essential to predicting the valve dynamics. The ability to computationally simulate the response of solenoid valves can significantly shorten the product development cycle and reduce testing costs.

In this Technology Brief, we describe a methodology for building a coupled electromagnetic and fluid-structure interaction analysis of a solenoid valve. A seamless integration between the applicable analysis techniques allows for the simulation of the complex multiphysics interactions.

### Key Abaqus Features and Benefits

- High Reynolds number incompressible viscous fluid flow in Abaqus/CFD
- Arbitrary Lagrangian-Eulerian framework with deforming meshes in Abaqus/CFD
- Boundary layer meshing in Abaqus/CAE
- Electromagnetic analysis to determine magnetic fields induced by current flow
- Co-simulation of valve dynamics using Abaqus/Standard and Abaqus/CFD
- Sequentially coupled electromagnetic/fluid-structure interaction analysis with user subroutine UAMP
- Unified pre- and post-processing in Abaqus/CAE

### Background

Solenoid valves are used in a wide variety of industries and applications. Actuation of solenoid valves is controlled by electric current, which energizes a magnetic circuit that applies the force necessary to allow or restrict flow.

Figure 1 shows a schematic diagram of the solenoid valve model under consideration. When the surrounding electrical coil is not energized, there is no electromagnetic field and thus no magnetic force on the plunger. The plunger is spring loaded which ensures that the valve remains closed in the absence of actuation. When an electric current is sent through the coil, a magnetic field is generated in the magnetically permeable circuit consisting of the core, the plunger and a small air gap. The resulting electromagnetic force moves the plunger upwards thus opening the valve and allowing fluid flow. During this motion, the spring is compressed. A higher force is required at start-up because of differential pressure; the force reduces during the holding (open) phase of the valve.

### Analysis Approach

The modeling approach is depicted in Figure 2. The complete analysis is carried out in two phases. In the first phase, electromagnetic (EM) forces on the plunger are calculated as a function of electric current and plunger location using a series of electromagnetic analyses in Abaqus/Standard.

In the second phase, a fluid-structure interaction (FSI) analysis is performed using Abaqus/Standard and Abaqus/CFD. The electromagnetic forces experienced by the plunger are incorporated into the Abaqus/Standard analysis through user subroutine UAMP. The current location of the plunger is passed in to UAMP through a sensor history output. The user subroutine uses the plunger position and the value of electric current to calculate the EM force. The EM force at a given time is obtained through multivariate interpolation using a set of force versus electric current/position data obtained from the electromagnetic analysis.

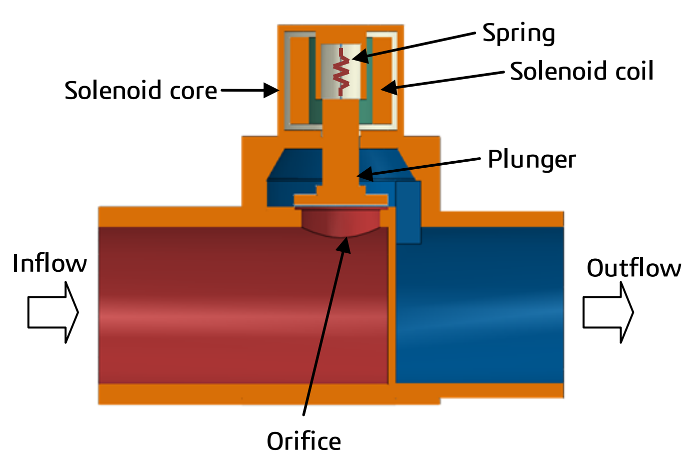


*Figure 1: Schematic diagram of the solenoid valve model*

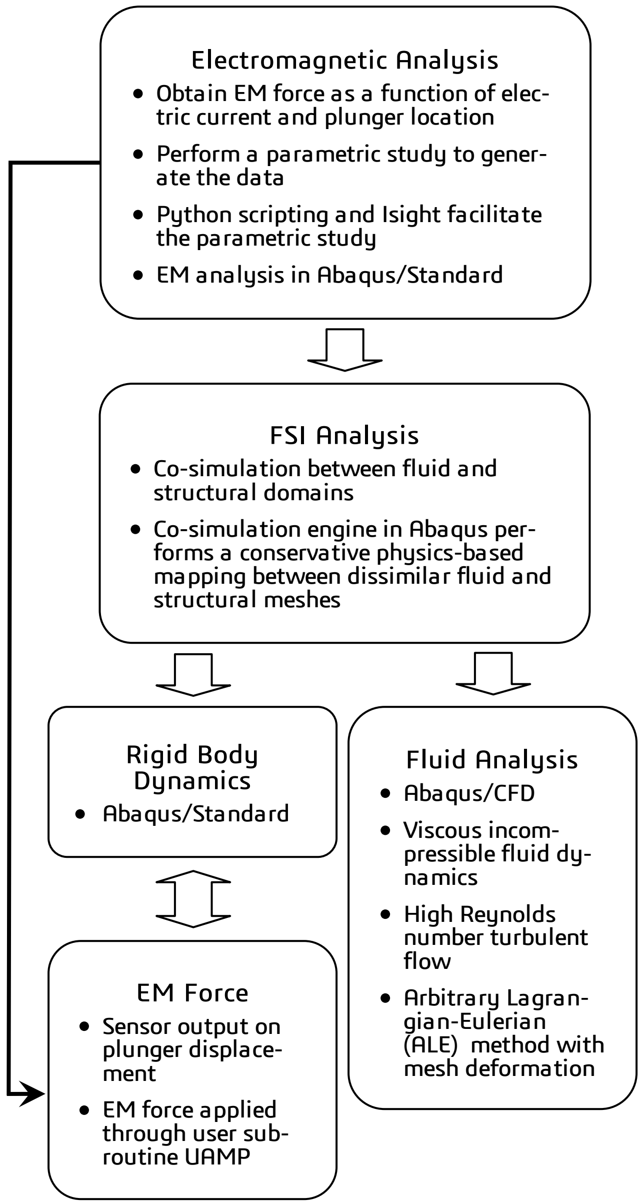


*Figure 2: Modeling approach*

### Electromagnetic Analysis

*Geometry and Model*

The cross-sectional view of the EM model geometry is depicted in Figure 3. The model consists of three parts: coil, plunger and core. The inner and outer diameters of the coil winding are 10.4 mm and 16.4 mm, respectively. The plunger shaft has a diameter of 6 mm. The core along with the plunger provides a path for the magnetic flux that is generated by the coil. The width of the small air gap between the plunger and the core measures 0.2 mm in the radial direction. The air surrounding the three parts is also included as a part of the model. The rest of the housing is assumed to be made of a non-magnetic material and is not modeled as part of the EM simulation. Due to symmetry, only one quarter of the geometry is modeled.

*Material Properties*

The plunger and core are modeled as soft magnetic materials with magnetic properties similar to that of an ASME

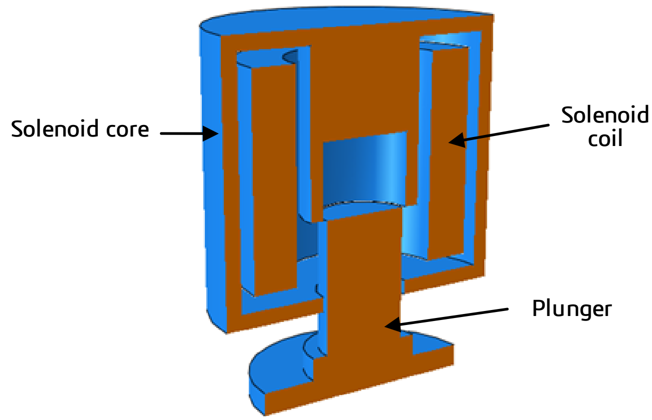


*Figure 3: Schematic diagram of electromagnetic model*

1010-grade steel. The nonlinear properties of the material are specified as a B-H curve (Figure 4) to account for the magnetic saturation at high field intensities. The rest of the domain is modeled as air.

*Loads and Boundary Conditions*

The direct current in the coil is assumed to be quasi-static and a magnetostatic response of the valve is computed based on the value of current at a given instant. The peak value of the direct current is taken to be 1 A. The number of turns in the coil winding is assumed to be 500. A magnetic vector potential-based formulation is used for solving the magnetostatic problem. Dirichlet boundary conditions are specified on both symmetry planes. Neumann boundary conditions are specified on all outer surfaces of the surrounding medium.

*Simulation Step*

A sequence of magnetostatic simulations are performed for various values of current ranging from 0-1 A and for various plunger displacements ranging from 0-6 mm. The simulation results are then postprocessed to produce tabular data of the total force on the plunger as a func-

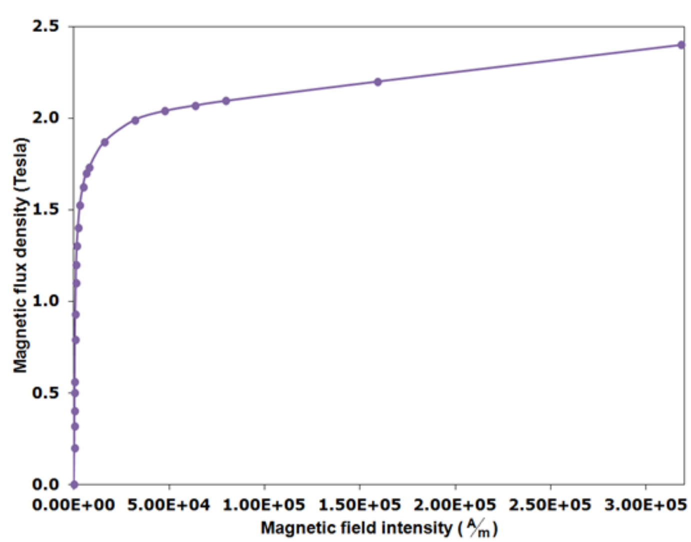


*Figure 4: Nonlinear B-H curve of the plunger and core*

tion of current and plunger displacement. The total force on the plunger is obtained by integrating Maxwell's stress tensor over the surface of the plunger. In the subsequent FSI analysis, the value of force at a given instance of time is obtained by performing a linear interpolation on this data. The analysis method assumes that the current is quasi-static and ignores eddy currents arising from the motion of the plunger in the magnetic field.

### Results

Figure 5 shows contour plots of magnetic flux density for three different positions of the plunger and a current of 1 A. The total upward force on the plunger for various values of current and plunger displacement is plotted in Figure 6. The large initial upward force is due to the effective passage of magnetic flux between the corners of the core and the plunger. Solenoid valve designs often feature slanted surfaces instead of sharp corners to produce even larger forces. Analyzing the data values of the plot for a given plunger displacement, one can conclude that the force scales almost linearly with the applied current.

### FSI Analysis

Co-simulation between the structural and fluid domains is carried out by coupling Abaqus/Standard and Abaqus/CFD through the Abaqus Co-Simulation Engine (CSE).

In general, the fluid and solid meshes are dissimilar in an FSI analysis and the CSE performs a physics-based conservative mapping of solution quantities (such as forces and heat fluxes) across the wetted interface.

The co-simulation coupling scheme is sequential, with Abaqus/CFD lagging Abaqus/Standard at every time increment. The coupling time increment size is the minimum determined by the automatic time incrementation schemes of the individual analyses.

#### *Fluid Model Definition*

*Geometry and Mesh*

The CFD model geometry is depicted in Figure 7. The CFD volume is extracted from the solenoid valve's CAD model in Abaqus/CAE through a set of geometric operations.

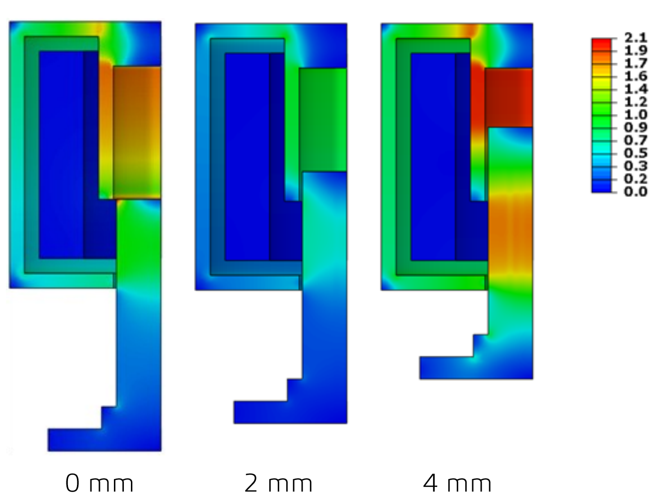


*Figure 5: Magnetic flux density for current of 1 A and plunger displacement of 0 mm, 2mm, and 4mm*

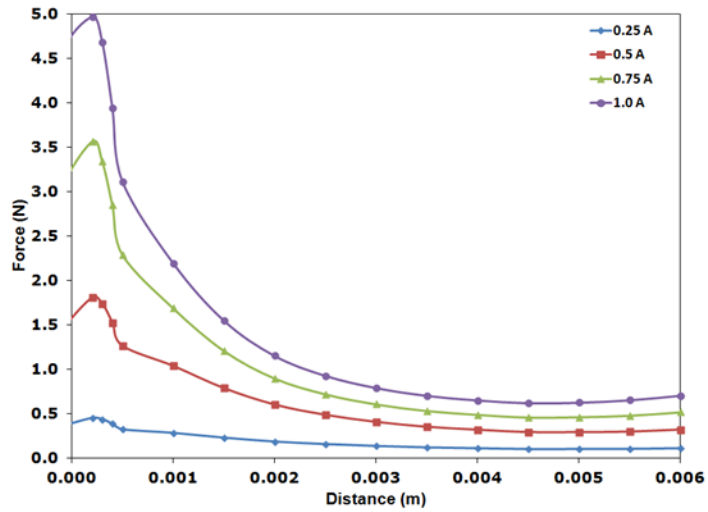


*Figure 6: Calculated force versus plunger displacement for various values of electric current*

The valve has an inlet diameter of *0.0254 m* (*1 inch*) and an orifice diameter of *0.0127 m (0.5 inch)*.

A boundary layer mesh in generated in the CFD volume. The first layer element thickness is 0.04 mm. Three layers of wedge elements are generated on the wall which is then transitioned to a tetrahedral mesh in the core of the flow domain. A total of 567602 elements are generated of which 125100 are wedge elements (FC3D6) at the wall and 442502 are tetrahedral elements (FC3D4).

*Fluid Properties*

Motor oil is the working fluid; since oil is nearly incompressible, it is approximated as an incompressible viscous fluid with a density of *980* $Kg/m^3$ and a viscosity of *0.15 Pa.s*.

*Simulation Step*

The flow procedure in Abaqus/CFD is used to solve the transient incompressible viscous Navier-Stokes equations. For the given operating conditions, the fluid flow is expected to have low turbulence due to the high viscosity of the fluid. We will, however, utilize the Reynolds averaged Navier-Stokes (RANS) approach with the one-equation Spalart-Allmaras turbulence model.

The Abaqus/CFD incompressible solver uses a hybrid discretization built on the integral conservation statements for an arbitrary deforming domain. A hybrid semi-implicit method with pressure-velocity decoupling and a node-centered finite-element discretization for the pressure is used. This hybrid approach guarantees accurate solutions

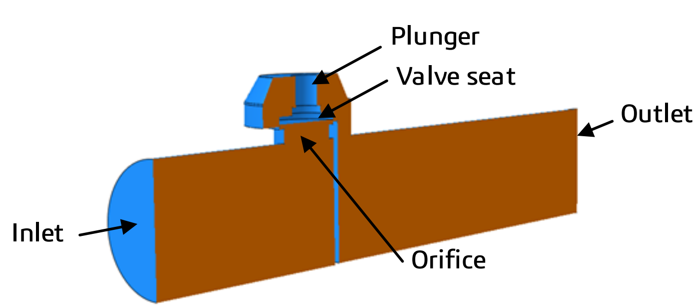


*Figure 7: Schematic diagram of CFD model (view-cut through the symmetry plane)*

and eliminates spurious pressure modes while retaining the local conservation properties associated with traditional finite volume methods. The solution methods for the momentum and transport equations (e.g., turbulence equations) in Abaqus/CFD rely on scalable parallel preconditioned Krylov solvers. The pressure, pressure-increment, and distance function equations are solved with Krylov solvers and a robust algebraic multigrid (AMG) preconditioner.

*Boundary Conditions*

The inlet fluid velocity is specified to be *1 m/sec*, corresponding to a mass flow rate of *0.5 Kg/Sec*. At the inlet section, a kinematic eddy viscosity of $5\mu/\rho$ is specified as the inlet turbulence.

Assuming that the outlet is open to the atmosphere, a reference pressure boundary condition of *0 Pa* is applied at the outlet section. A non-slip/no-penetration wall boundary condition is applied at the exterior of the fluid domain. Additional boundary conditions on the turbulent eddy viscosity and wall-normal distance function are automatically set at the wall.

The FSI simulation requires that the CFD analysis is solved in a fluid domain with moving/deforming boundaries. Abaqus/CFD uses an arbitrary Lagrangian-Eulerian (ALE) formulation and automated mesh deformation method that preserves element size in the boundary layers. The ALE and deforming mesh algorithms are activated automatically for problems that involve moving boundaries.

*Co-simulation Interaction*

A co-simulation interaction is defined at the plunger disk surface and serves as the necessary boundary condition at the interface between the fluid and the plunger. The fluid velocity and mesh displacement are dictated by the coupled solution at this interface.

### *Structural Model Definition*

*Model Geometry and Mesh*

The structural model geometry is depicted in Figure 8. The plunger is modeled as a rigid body, and the other components are set as display bodies. The plunger is attached to a spring and dashpot to model the installed spring and natural damping in the system. The valve spring is preloaded to ensure that the plunger can stop the flow in the absence of electric current.

*Material Properties*

Although modeled as rigid, the plunger is assumed to have the density of steel at *7800 kg/m*$^3$. The spring is modeled with an axial connector with spring constant of *3600 N/m* and a damping coefficient of *100.0 N.sec/m*. The spring is preloaded to supply an initial force of *3.6 N*.

*Simulation Steps*

An implicit dynamic analysis procedure is used in Abaqus/Standard to model the rigid body dynamics of the plunger and associated deformations of the axial connector (spring and dashpot).

*Boundary Conditions*

The axial connector is fixed at the top end while its other end is attached to the top end of the plunger. In addition, the plunger is constrained to move in only the vertical direction.

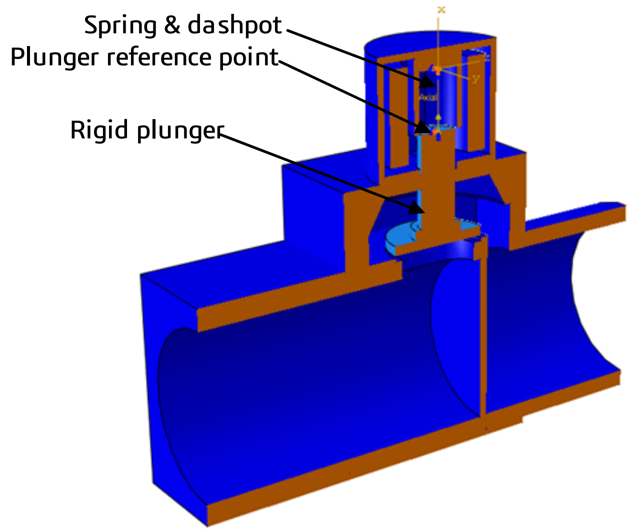


*Figure 8: Schematic diagram of structural model (view-cut through the symmetry plane)*

*Loading*

A point load along the vertical direction is applied at the plunger's rigid body reference point through an amplitude defined by user subroutine UAMP. This point load represents the electromagnetic force on the plunger. A sensor history output is enabled at the plunger's rigid body reference point, which enables tracking of the current plunger displacement.

The entire model is activated with a DC current (specified inside user subroutine UAMP). DC circuits typically consist of resistors and inductors and have a characteristic time constant $\tau$. A time constant of 10 ms is assumed in the current simulation. The electric current reaches its peak saturation value of 1.0 A at t = 5$\tau$ (see Figure 9).

User subroutine UAMP dynamically calculates the value of applied electric current and gets feedback of the plunger displacement. During the FSI analysis, both the current and displacement values are utilized to calculate the upward EM force on the plunger. The EM force is ob-

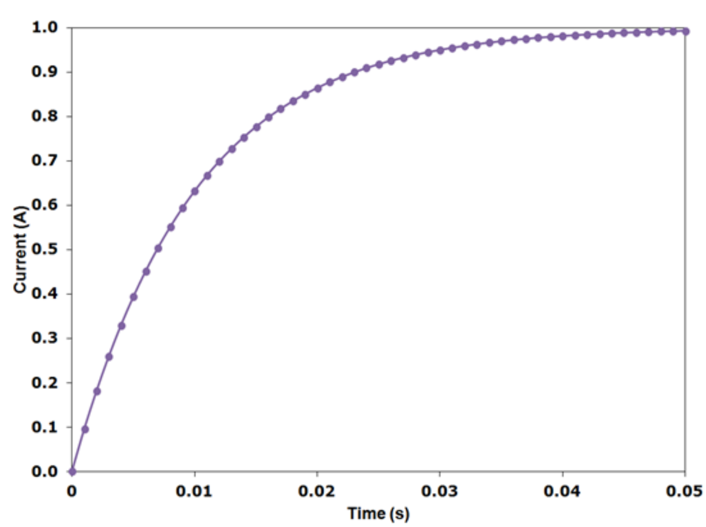


*Figure 9: Applied electric current v. time*

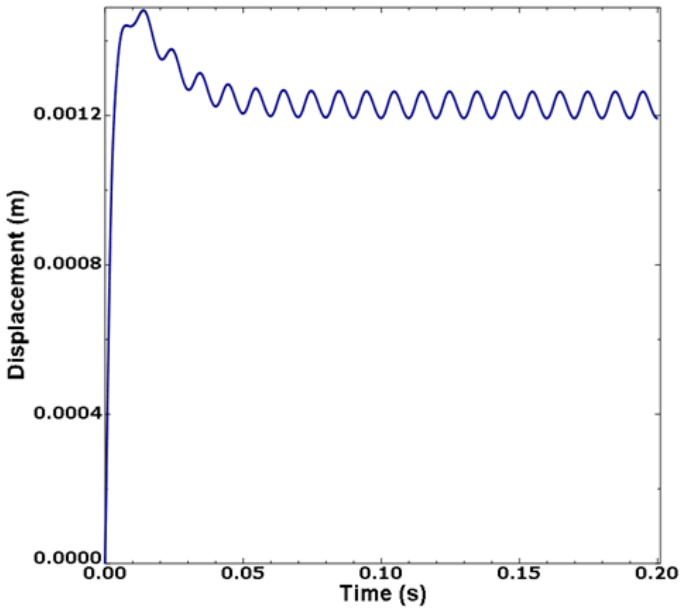


*Figure 10: Plunger displacement v. time*

tained by interpolating the force versus electric current and displacement data that was obtained through a series of electromagnetic analyses completed prior to the FSI analysis.

### Results and discussion of the FSI Analysis

The valve reaches a time periodic operating condition at *t = 60 ms* after the electric current is activated (Figure 10). The valve oscillates with a mean displacement of *1.23 mm* and amplitude of *0.072 mm*. The valve oscillations reflect the periodic oscillations seen in the fluid caused by vortex generation. Figure 11 depicts the kinetic energy of the fluid where these oscillations can also be noted.

Figure 12 displays fluid velocities and pressures at the beginning of the simulation. It can be seen that the simulation starts from a slightly open position of the valve to avoid mesh pinching. Figure 13 depicts the fluid pressure,

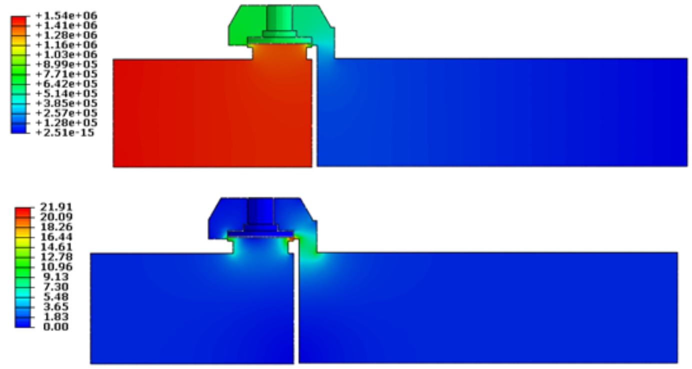


*Figure 12: Fluid pressure (top) and velocity (bottom) at t = 0*

velocity, and turbulent viscosity once periodic operating conditions have been achieved.

Figure 14 shows the streamlines at the mean plunger displacement, highlighting the fluid mixing that is occurring in the rear section. Figure 15 shows the ratio of kinematic turbulent viscosity to the kinematic viscosity of the fluid. It can be seen that dissipation due to turbulence is very low and is active only in the rear section where the fluid impinges and rushes outwards.

### Conclusions

This technology brief describes a methodology for simulating the coupled electromagnetic (EM) and fluid-structure interaction (FSI) analysis of a solenoid valve. The coupled simulation can predict the solenoid valve's response time, which is a very important design criterion. A number of design variables such as material properties, spring constants, spring damping, fluid operating conditions, coil properties, etc., can be studied to further understand their effect on the valve performance. Additional fidelity can be achieved by modeling the plunger as deformable. This approach can also be extended to calculated fatigue and failure calculations.

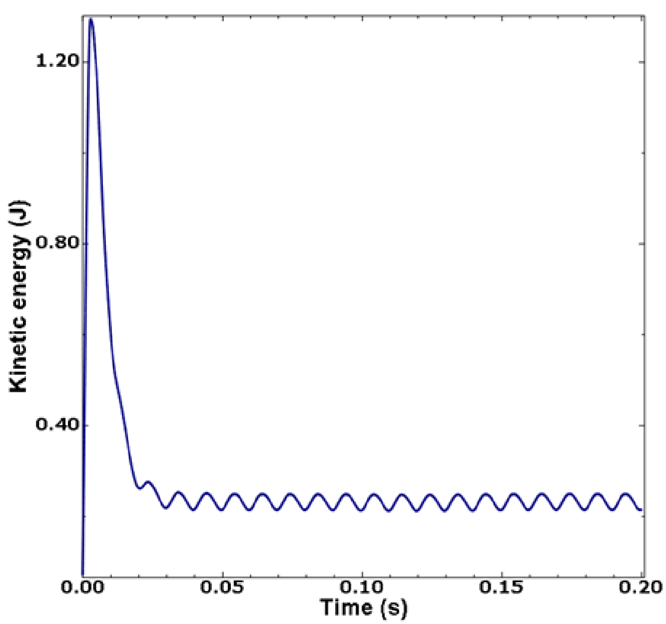


*Figure 11: Kinetic energy vs. time for the entire fluid domain*

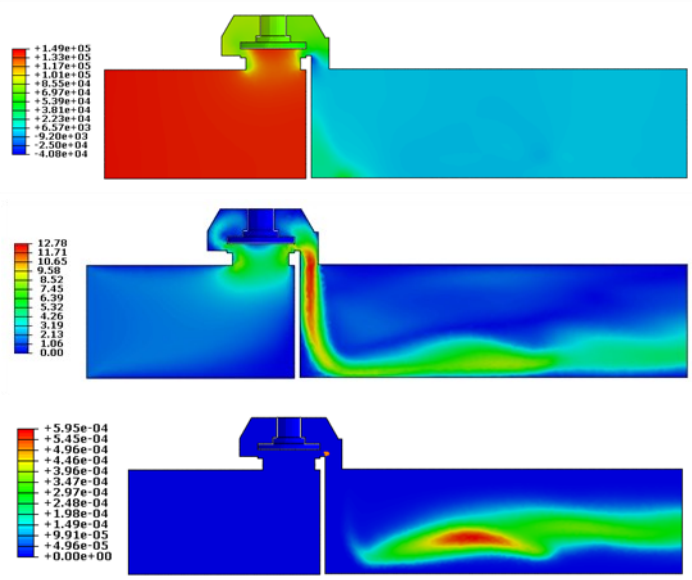


*Figure 13: Fluid pressure (top), velocity (middle), and kinematic turbulent viscosity (bottom) at mean plunger displacement*

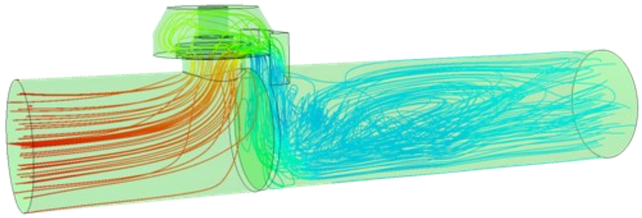

*Figure 14: Streamlines colored by pressure at mean plunger displacement*

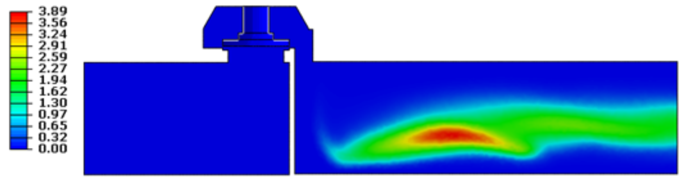


*Figure 15: Ratio of kinematic turbulent viscosity to the kinematic viscosity of the fluid at mean plunger displacement*

## SIMULIA References

For additional information on the Abaqus capabilities referred to in this document please see the following Abaqus 6.12 documentation references:

- Analysis User's Manual
  - 'Electromagnetic analysis,' Section 6.7
  - 'Fluid dynamic analysis,' Section 6.6
  - 'Abaqus/CFD to Abaqus/Standard or to Abaqus/Explicit co-simulation,' Section 17.3.2
  - 'Available user subroutines,' Section 18.1.2

Visit the Resource Center for more SIMULIA customer papers.

## 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.3ds.com/simulia

DASSAULT SYSTEMES

## Delivering Best-in-Class Products

CATIA
Virtual Product Design

SOLIDWORKS
3D for Professionals

SIMULIA
Realistic Simulation

DELMIA
Virtual Production

ENOVIA
Global Collaborative Lifecycle Management

GEOVIA
Virtual Planet

EXALEAD
Information Intelligence

NETVIBES
Dashboard Intelligence

3DSWYM
Social Innovation

3DVIA
Online 3D Lifelike Experiences

---

Dassault Systèmes, the **3D**EXPERIENCE Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 150,000 customers of all sizes in all industries in more than 80 countries. For more information, visit www.3ds.com.



---

**Europe/Middle East/Africa**
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

**Asia-Pacific**
Dassault Systèmes
Pier City Shibaura Bldg 10F
3-18-1 Kaigan, Minato-Ku
Tokyo 108-002
Japan

**Americas**
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

Visit us at
3DS.COM/SIMULIA