Modeling and Simulation of Control Valves via COMSOL Multiphysics

S. Zhuang
CAEaid, Inc., Austin, TX, USA

Abstract: Control valves are widely used to control fluid flow in various engineering applications. However, because of control valves' challenging characteristics, it is still difficult for scientists and engineers to well understand all the necessary engineering phenomena of control valves, no matter via experiment or via simulation. This study aims to effectively model control valves and simulate the inherently multiphysics problem by use of COMSOL Multiphysics. A typical example is simulated to show that the modeling method has the ability of predicting complicated fully coupled fluid-structure interaction when the fluid pipeline shape is deformed, the valve sleeve material is hyperelastic and there is internal surface contact interaction.

Keywords: Control Valve, Fluid Flow, Fluid-Structure Interaction, Multiphysics, Fully Coupling Analysis, Hyperelastic Material, Surface Contact

Introduction

Control valves have been widely used for fluid flow control in various engineering applications. It's crucial to study the flow characteristics inside the control valves and the fluid-structure interaction between the fluid and the deformable valve sleeve for design optimization and improvement of control valves. However, because of complex fluid flow state, complicated contact status, large deformation, and nonlinear material characteristics, it is still a big challenge for scientists and engineers to well understand all the necessary mechanical phenomena no matter via experiment or via simulation.

Control valves have drawn great research interest from many scientists and engineers [1-2], where various theoretical methods were presented. Moreover, commercial CAE software products have also been applied to perform CFD simulation of control valves [3-5]. COMSOL Multiphysics is well because of its powerful Model Builder, user-friendly interface, and complete simulation capabilities. Usually 'Fluid-Structure Interaction' physics type is the first choice in COMSOL Multiphysics to perform a fully coupling analysis of the interaction between fluid and structures, because it is easy to set up and define. However, the control valves, which have deformable and hyperelsstic valve sleeves and internal surface contact interaction, have seldom been studied and simulated. The 'Fluid-Structure Interaction' physics type in COMSOL Multiphysics was also applied to simulate this kind of control valves, but it encounters difficulty to obtain a convergent solution.

To simulate control valves with deformable valve sleeves and internal surface contact interaction by use of COMSOL Multiphysics, this study presents one moving-mesh coupling method, which incorporates three physics types: 'Laminar Flow', 'Solid Mechanics' and 'Moving Mesh'. One typical 2D control valve model is simulated and discussed for purpose of verification.

Moving-Mesh Coupling Method

Figure 1 shows the control valve with deformable and hyperelastic valve sleeves (in dark blue) and internal contact interaction. The movement of the upper and lower adjusters controls the deformation of sleeves and then the flow of fluid via the contact interaction between adjusters and the sleeves.

The interaction between fluid and valve sleeves is two-way coupled: (1) the fluid pressure influences the deformation of the hyperelastic valve sleeves, and (2) the deformation/displacement of valve sleeves influences the fluid flow. Since the valve sleeve is hyperelastic, the 'Wall' boundary is deformable and move with the deformation of the value sleeves.

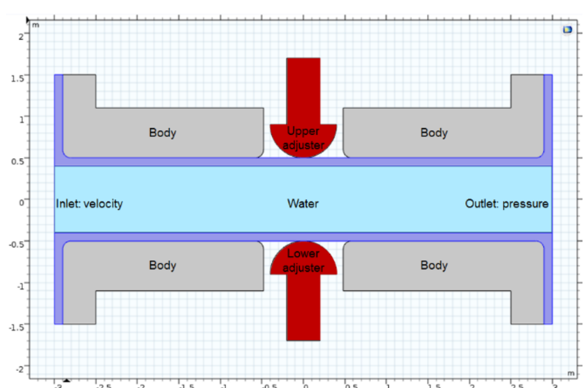

Figure 1. Control valve with deformable and hyperelastic valve sleeves and internal surface contact interaction between valve sleeve and valve body/adjusters.

To account for the deformable 'Wall' boundaries, the hyperelastic characteristic of sleeves, and the contact interaction between body/adjusters and sleeves, a Moving-Mesh Coupling Method is presented to incorporate three physics types: 'Laminar Flow', 'Solid Mechanics' and 'Moving Mesh'.

The 'Laminar Flow' physics type is applied to predict fluid flow. The 'Wall' boundary includes the surfaces between water and the sleeves.

The 'Solid Mechanics' physics type predicts the internal contact interaction between valve sleeve and body/adjusters, the hyperelastic deformation of valve sleeves, and the fluid pressure on valve sleeves. The fluid pressure on valve sleeves is defined by 'Boundary Load' on all the boundaries defined in 'Wall' boundary condition. The load type is defined as 'Pressure', and the pressure p is set as

$$p = p2$$

The 'Moving Mesh' physics type accounts for the influence of the deformable sleeves. Only the fluid is selected in 'Domain' definition. Two 'Prescribed Mesh Displacement' boundaries should be defined. One 'Prescribed Mesh Displacement' boundary includes the 'inlet' boundary and 'outlet' boundary, where the prescribed displacements are defined as:

$$\boldsymbol{d_x = 0}$$

$$\boldsymbol{d_y = 0}$$

The other 'Prescribed Mesh Displacement' boundary includes 'Wall' boundary and outlet boundary, where the prescribed displacements are defined as follows:

$$\boldsymbol{d_x = u_2}$$

$$\boldsymbol{d_y = v_2}$$

Numerical Model Set-up

Figure 2 shows that the domains of sleeves are partitioned to simplify the definitions of boundary conditions, contact pairs and mesh sizes.

In the 'Materials' section, the density and dynamic viscosity of 'Water' are defined from COMSOL Built-in Material Library. The valve body and adjusters are made of Steel AISI 4340, also defined from COMSOL Built-in Material Library. The sleeves are hyperelastic and have user-defined properties, as shown in Table 1.

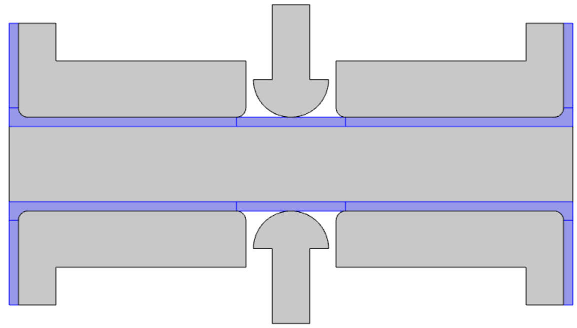

Figure 2. Partition domains of valve sleeves.

In 'Laminar Flow', all the initial velocity field and pressure are defined as 0, and the boundary conditions are defined below:

(1) Wall: 'No slip'
(2) Inlet: 'Normal inflow velocity' -- 0.05 m/s
(3) Outlet: 'Pressure' -- 0.5 psi

In 'Solid Mechanics', the hyperelastic material model is defined as 'Mooney-Rivlin, two parameters', with an initial bulk modulus of 1e4 MPa. Considering the stiffness of Steel AISI 4340 is much higher than the stiffness of valve sleeve, the valve body and adjusters are defined as rigid domains and the center of rotation is defined as center of mass:

(1) Rigid domain 1: defined for the valve body, with a fixed constraint since the valve body is motionless.
(2) Rigid domain 2: defined for the lower adjuster, with a constrained rotation and prescribed displacements as follows:
$$\boldsymbol{u_{0x} = 0}$$
$$\boldsymbol{u_{0y} = 0.35 * pw1(para)}$$
(3) Rigid domain 3: defined for the upper adjuster, with a constrained rotation and prescribed displacements as follows:
$$\boldsymbol{u_{0x} = 0}$$
$$\boldsymbol{u_{0y} = -0.35 * pw2(para)}$$

The two piecewise functions *pw1* and *pw2* are defined by Tables 2 and 3 respectively.

Figure 3 shows totally six contact pairs are defined between valve sleeves and valve body/adjusters. The 'Augmented Lagrangian' contact pressure method is applied. In each pair, the boundaries on body/adjusters are defined as 'Source Boundaries', while the boundaries on sleeves are defined as 'Destination Boundaries'.

The two sides of the sleeves are added with 'Fixed constraint". The 'Boundary Load' on all the boundaries defined in 'Wall' boundary condition are applied with forces defined in the previous section.

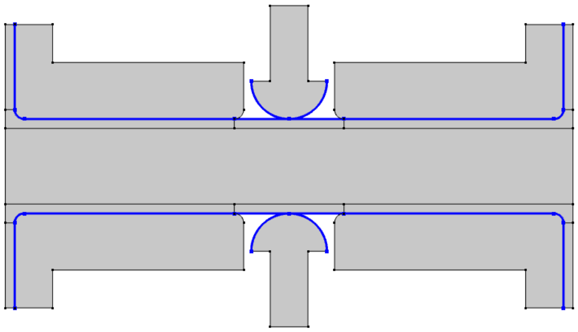

Figure 3. Six contact pairs.

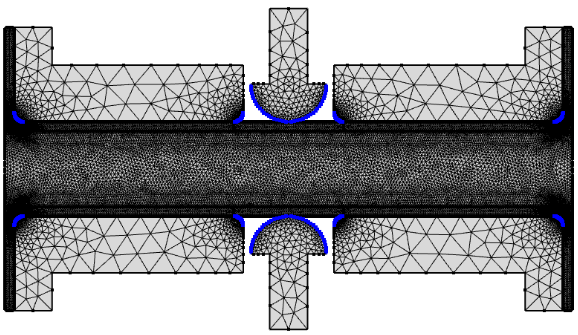

Figure 4. Meshed model with multiple mesh sizes.

In 'Moving Mesh', two 'Prescribed Mesh Displacement' boundaries are defined for inlet/outlet boundaries and wall boundaries as discussed in the previous section. The 'Free Deformation' is defined with zero initial mesh displacements:

$$dX_0 = 0$$

$$dY_0 = 0$$

Figure 4 shows that the 2D model is meshed with 'Triangular element' with three predefined mesh sizes with a 'Fluid dynamics' 'Calibrate for' option for different parts: 'Extra fine' mesh for the fluid, 'Coarser' mesh for valve body and adjusters, and "Extremely fine" mesh for sleeves. The curved lines shown in blue has a fixed number (20) of elements.

Results and Discussions

Simulated results at two typical times 2s and 4s are visualized. At t=2s, the lower adjuster moves to its highest position, while the upper one keeps motionless. To time t=4s, the upper adjuster moves to its lowest position.

The simulated results of flow velocity, flow pressure, and von Mises stress at t=2s are shown in Figures 5-7. The maximum flow velocity is about 2 times of the inlet velocity. The minimum pressure happens just after the closing area in the pipeline, only about 0.06% smaller than the outlet pressure. The higher pressure happens around the inlet area with a maximum pressure about 0.2% larger than the outlet pressure. Figure 7 shows that the closing area in the lower sleeve has the maximum von Mises stress, which means the largest deformation. The non-zero minimum von Mises stress tells that the fluid pressure results in deformation of the upper sleeve even though it has not moved.

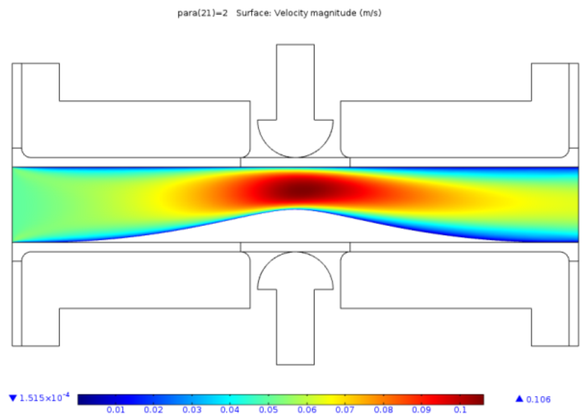

Figure 5. Simulated flow velocity at t=2s

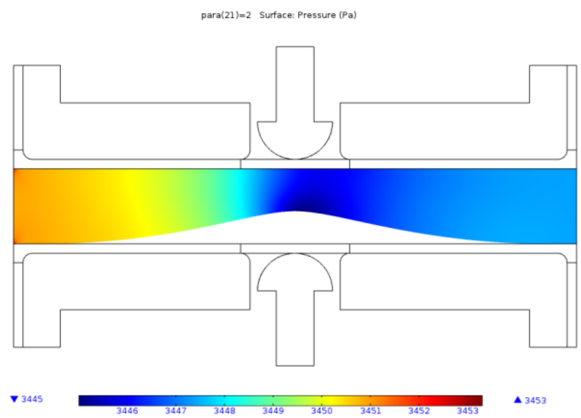

Figure 6. Simulated pressure at t=2s

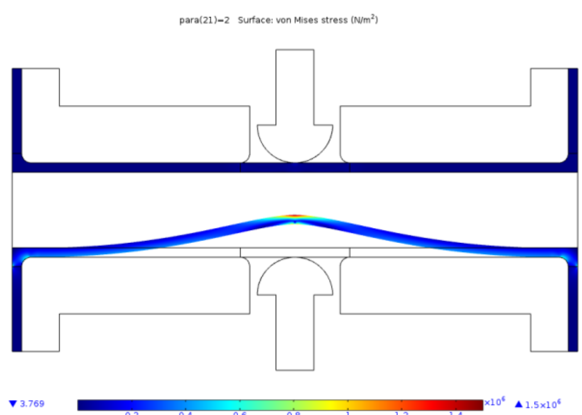

Figure 7. Simulated von Mises stress at t=2s

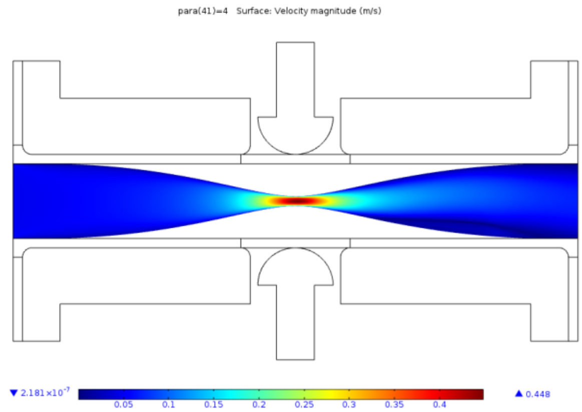

Figure 8. Simulated flow velocity at t=4s

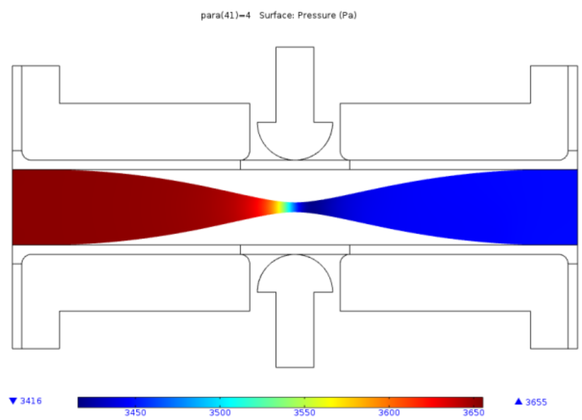

Figure 9. Simulated pressure at t=4s

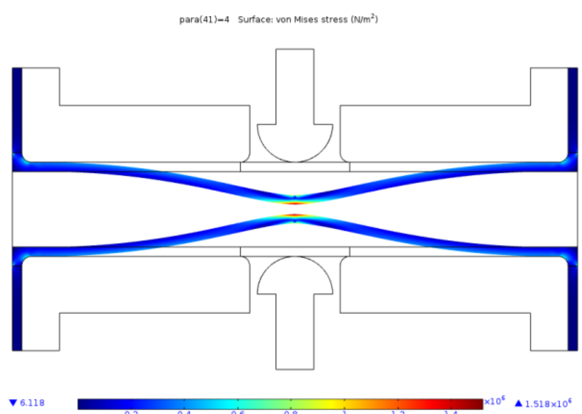

Figure 10. Simulated von Mises stress at t=4s

Figures 8-10 show the simulated results at t=4s. The maximum flow velocity is about 9 times of the inlet velocity. The minimum pressure happens just after the closing area, about 0.9% smaller than the outlet pressure. The inlet area has the maximum pressure, which is about 6.3% larger than the outlet pressure.

Table 4 compares the maximum velocity, minimum and maximum pressures, and maximum von Mises stress between t=2s and t=4s. It is shown that the minimum pressure becomes smaller from t=2s to t=4s with the increase of the closing percentage. The maximum von Mises stress at t=4s is about 1.2% higher than that at t=2s. Note that if without fluid pressure, the maximum von Mises stresses should be the same at t=2s and at 4s.

Conclusions

This paper presents the application of COMSOL Multiphysics to study the flow characteristics and the fluid-structure interaction between the fluid and the valve sleeve of control valves.

The moving-mesh coupling method can perform the fully coupling analysis of fluid flow in control valves with complex valve geometry, complicated contact status, large deformation, and nonlinear material characteristics.

For the control valves with inlet velocity and outlet pressure conditions, the minimum pressure decreases with the increase of the closing percentage. The inlet area has higher pressure, and the maximum pressure increases with the increase of the closing percentage.

References

1. V.G. Fester and P.T. Slatter, Dynamic similarity for non-Newtonian fluids in globe valves, *Chemical Engineering Research and Design*, **87**, p. 291–297 (2009)
2. C. Patrascioiu, C. Panaitescu, and N. Paraschiv, Control valves - modeling and simulation, *Proceedings of the 5th WSEAS Int. Conf. on Dynamical Systems and Control*, p. 63-68 (2012)
3. D. Wu, S. Li, and P. Wu, CFD simulation of flow-pressure characteristics of a pressure control valve for automotive fuel supply system, *Energy Conversion and Management*, **101**, p. 658-665 (2015)
4. C. Srikantha. and C. Bhasker, Flow analysis in valve with moving grids through CFD techniques, Advances in Engineering Software, **40**, p. 193-201 (2009)
5. V. Dossena, F. Marinoni, etc., Numerical and experimental investigation on the performance of safety valves operating with different gases, International Journal of Pressure Vessels and Piping, **104**, p. 21-29 (2013)

Acknowledgements

The author would like to thank David Kan and his team at COMSOL, Inc for valuable technical support.

Appendix

Table 1: Valve sleeve's material properties

Material Property	Value	Unit
Density	1000	kg/m^3
C10	3.05	MPa
C01	1.00	MPa

Table 2: Piece function 1

Start	End	Function
0	2	x/2
2	4	1

Table 3: Piece function 2

Start	End	Function
0	2	0
2	4	(x-2)/2

Table 4: Comparison of results at t=2s and t=4s

Simulated	t=2s	t=4s
Max. velocity (m/s)	0.106	0.448
Min. pressure (Pa)	3445	3416
Max. pressure (Pa)	3453	3655
Max. stress (N/m^2)	1.500E6	1.518E6