TECHNICAL TRANSACTIONS | CZASOPISMO TECHNICZNE
MECHANICS | MECHANIKA
2-M/2015

MARIUSZ DOMAGAŁA*

MODELLING OF DIRECT ACTING RELIEF VALVE USING CFD-FSI SIMULATION

MODELOWANIE ZAWORÓW PRZELEWOWYCH BEZPOŚREDNIEGO DZIAŁANIA PRZY WYKORZYSTANIU SYMULACJI CFD-FSI

Abstract

This paper presents an attempt of using new capabilities of CFD tools in field of fluid-structure interaction simulation, which is called "immersed solids". This techniques treats solids as objects immersed into fluid domain which moves along with the fluid. This paper presents comparison of selected CFD results of traditional approach and "immersed solids" technique on example of direct acting relief valve.

Keywords: *CFD, FSI, direct acting relief valve*

Streszczenie

W artykule przedstawiono wykorzystanie nowych możliwości modelowania CFD w zakresie oddziaływania cieczy na ciała stałe przy wykorzystaniu techniki *immersed solids*. Na przykładzie zaworu przelewowego bezpośredniego działania przedstawiono kilka wybranych wyników symulacji CFD i porównanie z tradycyjnym podejściem w modelowaniu CFD zaworów hydraulicznych.

Słowa kluczowe: *analiza CFD, analiza FSI, zawór przelewowy*

* Ph.D. Eng. Mariusz Domagała, Institute of Applied Informatics, Faculty of Mechanical Engineering, Cracow University of Technology.

1. Introduction

Modelling of hydraulic components is not straightforward, even using today's high performance computer CAE systems and associated CFD tools. For example, fluid flow through the channels introduces many nonlinearities and other complicated behaviors. Also, for the case of direct acting relief valves used in some hydraulic systems, there are challenges despite the ostensible simplicity of such problems. The main aim of such valves is to protect the hydraulic system against excessive increase of pressure in the hydraulic system. In the simplest case the direct acting relief valve consists of a lift component and a spring which determines the opening pressure and controls flow through the valve. But even in case of such simple a structure there is a necessity for modeling dynamics of the valve. One of the factors that should be included in the of relief valves is flow forces; and this not an easy task [1]. During last years significant progress occurred in CFD modeling which capabilities were extended with Fluid Structure Interaction simulation [2, 3, 5]. This allows for simulation of dynamic behavior of hydraulic valves and including also flow forces. Despite this advance however, there are still some disadvantageous such as long calculation times and other computer resources demands. Moreover, traditional FSI simulation requires preparing few simulation models to avoid problems with deforming mesh [1]. Recently, a new solution method of FSI appeared. This method allows one to simulate fluid-solids interaction without using deformable mesh. Such approaches are called "Immersed Solids" and are available in CFD code Ansys CFX.

This paper presents an attempt of using "Immersed Solid" technique in modeling of direct acting hydraulic valves.

2. Mathematical modeling of "immersed solids" simulation

An immersed solid is a rigid body which moves along with the fluid domain (Fig. 1) The solid object occupies the same volume as a part of the fluid domain which contain the solids.

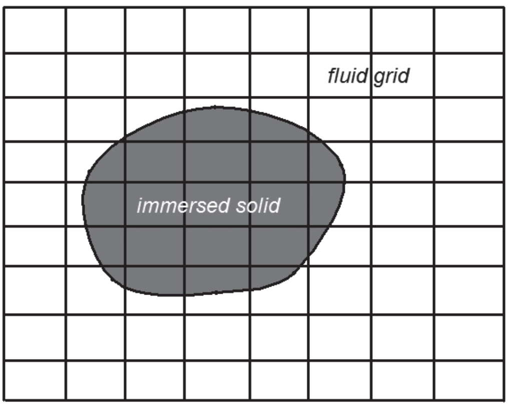

Fig. 1. Immersed solid inside the fluid domain

To model the interaction of fluid and immersed solid the momentum conservation equation needs to be considered as a source term.

For incompressible flows, the momentum conservation equation has the following form:

$$\frac{\partial U}{\partial t}\rho + \nabla(\mathbf{U}\times\mathbf{U})\rho = -\nabla p + \nabla\cdot\tau + \mathbf{S} \qquad (1)$$

where:

$\mathbf{S}$ – is source term,
$\mathbf{U}$ – is fluid velocity,
ρ – is fluid density,
p – is pressure,
$\tau = \mu\left(\nabla\cdot\mathbf{U} + (\nabla\cdot\mathbf{U})^T - \frac{2}{3}\delta\nabla\cdot\mathbf{U}\right)$ – is a stress tensor,
μ – is viscosity,
δ – is Kronecker symbol.

The source term **S** can be expressed as:

$$\mathbf{S} = -\,\alpha\beta C(\mathbf{U} - \mathbf{U}^S) \qquad (2)$$

where:

α – is the momentum scaling factor,
β – is the forcing function,
C – is momentum source coefficient,
$\mathbf{U}^S$ – is solid velocity.

2. Object of study

The object of study is the direct acting relief valve with a cone shape lift element presented schematically on Fig. 2.

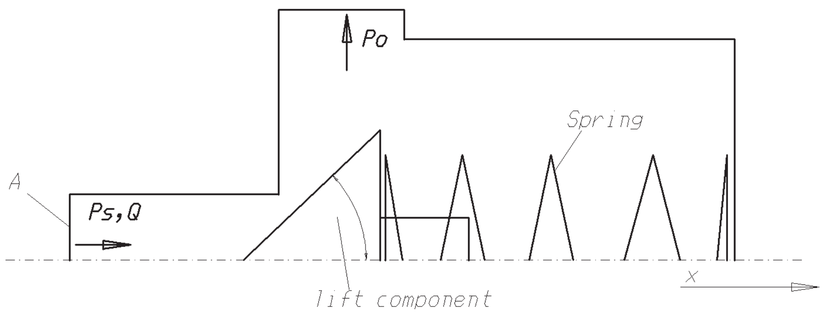

Fig. 2. A scheme of direct acting relief valve

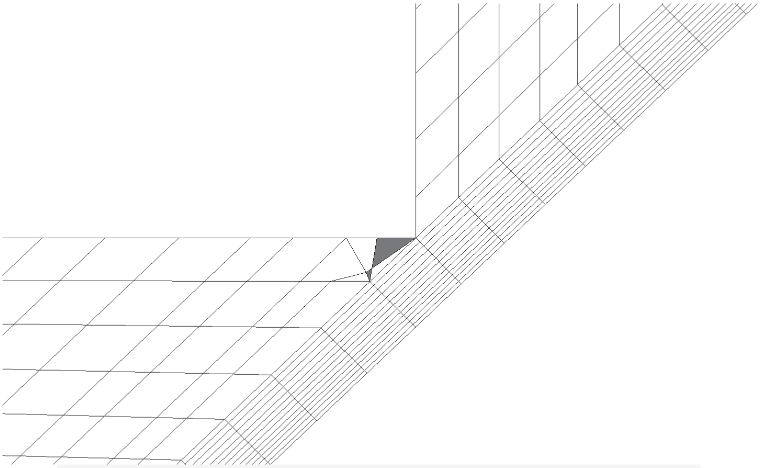

Fig. 3. A distorted cells which may appear during FSI simulation with grid deformation

When the pressure at the hydraulic circuit rise (port A) above the set value (p_s) the spring yields and the lift element opens the flow to the tank. A dynamic simulation of this valve requires information about all the forces that act on the lift element. Traditional FSI simulation requires a deformed mesh. However this may lead to relatively long calculation time and introduce other pathologies such as cell distortion. Due to the valve features (no damping element) during operation the lift element may close and open the flow. Therefore, the deformed mesh may cause the distorted cells which may appear during simulation (see Fig. 3). One solution to such a problem may be use few models with grid prepared for simulation of valve in defined opening or re-meshing, when distorted cells appears. However, both approaches are a time consuming process. The problem is solved by using "immersed solids"

3. FSI "Immersed Solid" simulation

Immersed solid simulation among many advantages has also some limitations. For example it does not consider viscous forces and the necessity of preparing very fine grid for estimated overlapping solid and fluid domain. For these reasons some preliminary tests were conducted in this work to check the pressure and velocity distribution as well as the value of flow forces for various valve opening. CFD simulations were conducted for steady state conditions for the same flow conditions for two models: in the first the lift element is a wall while in the second it is the immersed solid. Selected fluid velocity distribution is presented on below Figures.

Maximal fluid velocity for both models are very similar, however, the velocity distributions are not similar. This may lead to further consequences. Therefore, the next step is the investigation of flow forces which appear at the lift element during fluid flow. Results for various openings are shown below.

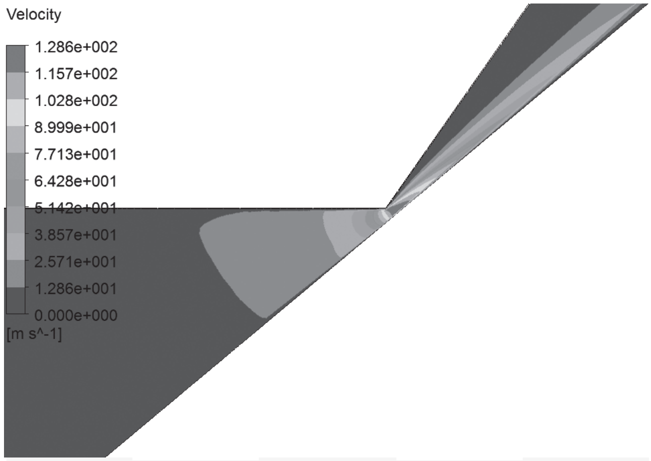

Fig. 4. Fluid velocity: lift element is a wall

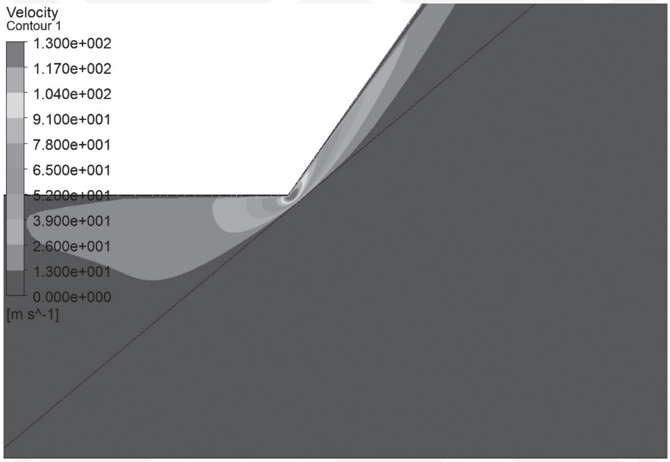

Fig. 5. Fluid velocity: lift element is the "immersed solid"

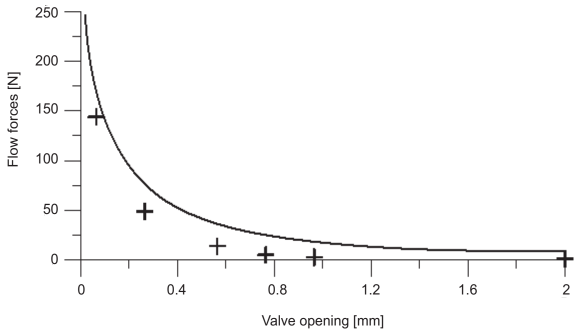

Fig. 6. Flow forces for both models: continuous line: the first models, symbols is the second one

Presented results shown what could be expected, that differences in flow forces even despite the fine grid are significant for both models.

4. Conclusions

This paper presents an attempt at using the "Immersed Solid" FSI technique for the simulation dynamics of direct acting relief valve. Preliminary CFD simulations show that the maximal value of velocity and pressure are very similar for the traditional approach and "Immersed Solid" technique, but velocity distribution for both cases are different. Additionally, CFD simulations provided the values of flow forces, which also confirmed that in case of "Immersed Solid" simulation flow forces have significant differences. Thus, it is concluded that despite the advantages of avoiding problems with deformed mesh, the "Immersed Solid" technique may be used in simulation of dynamics of relief valve in very limited way and rather for preliminary tests but not full value simulation method.

References

[1] Domagała M.J., *Metodyka modelowania zaworów maksymalnych bezpośredniego działania*, praca doktorska, Kraków 2007.
[2] Domagała M., *CFD analysis of pilot operated relief valve*, Czasopismo Techniczne 3-M/2008, Kraków 2008.
[3] Beune A., Kuerten J.G.M., Heumen van M.P.C., *CFD analysis with fluid–structure interaction of opening high-pressure safety valves*, Computers & Fluids, Vol. 64, 2012, 108-116.
[4] Ansys CFX Theory Guide.
[5] Huguet D., *Dynamic mesh modelling of a direct acting relief valve*, 3rd FPNI PhD Symposium, Terrasa, Spain 2004.