27/28th International Conference
ENGINEERING MECHANICS 2022
Milovy, Czech Republic, May 9 –12, 2022
Paper **#34**, pp. 349–352, doi: 10.21495/51-2-349

SIMULATION OF TIME-DEPENDENT PRESSURE FIELDS IN MODEL OF THE CONTROL RODS OF NUCLEAR REACTOR

Schuster M.*

Abstract: *The paper deals with the modelling of unsteady flow-field in computational domain caused by movement of the simple rigid body at fluid flow. It discusses the possibilities of CFD software to model movements and describes the new methodology of CFD simulations to solve this problem. The motion of control rods in the reactor core was chosen as a case study. The developed methodology is focused on investigating the dynamic response of the control rod model to seismic load. A number of the computational models of moving control rods with various setup of boundary conditions were created and tested. The simple database of model pressure pulses was created from the results of simulations. This database is prepared for next research use. The paper briefly describes the next steps in research of mathematical modelling and computer simulation of the seismic response of chosen control rod of reactor core cluster.*

Keywords: Fluid dynamics, Pressure pulse, Unsteady calculation, Seismics, Dynamic response.

1. Introduction

Changes in pressure fields caused by relative movement of components occur in various machines and in parts of power-plants equipment or transport means. Steam or gas turbines, compressors, nuclear reactors, pumps and combustion engines etc. include many moving parts. Computational fluid dynamics (CFD) can play important role in various studies of interactions between moving rigid bodies and flowing fluid. Simulations of time-dependent pressure changes in the computational domain are a significant challenge for CFD methods. From the point of view of CFD simulations, we can divide these cases into two groups. The first group, let us call it “single-body”, represents a situation when only one rigid body is moving in a fluid zone. For example, it is fuel rods and control clusters of nuclear reactor or valves and other regulating armatures of a steam turbine. The second group, let us call it “multi-body”, represents a situation when more bodies are moving (rotating) and a stator - rotor interaction type is generated. All axial blade machines, turbines, compressors and pumps belong here. The paper briefly discusses the possibilities of CFD software to solve movements from the first group, a simple rigid body in fluid flow.

Changes in the flow field caused by the movement of a rigid body in the computational domain can be solved by CFD methods and procedures. Standard commercial software Ansys/Fluent offers several approaches to modelling this "movement" (Fluent, 2021):

a/ the method “moving-reference-frame” allows simulating a time-dependent situation of movement by stationary computational domain with "unsteady" boundary conditions. The modelled motion is often rotational.

b/ the method “sliding-mesh” allows simulating of translational or rotational motion where two or more separate fluid zones of computational domain move relatively to each other. The relative motion of stationary and moving zones generates time-dependent flow-field changes.

c/ the method “dynamic-mesh” allows modelling the motion by shifting the boundaries of a fluid zone relatively to other boundaries of the zone and/or deforming the zone, if necessary. Then some remeshing processes are required. The dynamic-mesh method is not simple, has higher calculation requirements and therefore is less efficient.

* Dr. Ing. Milan Schuster: Research and Testing Institute Plzen Ltd., Tylova 1581/46; 301 00, Plzeň; CZ, schuster@vzuplzen.cz

The rest of the paper deals with the application of the “sliding-mesh” method to a selected technical problem. The motion of one control rod in the reactor core was chosen as a case study.

CFD simulations are applied to solve time-dependent changes of pressure field and pressure load of a rigid body in a flow. Various time-dependent flow pulsations play important role in this category of flow-structure interaction. The working regime of reactor control rods may be affected by flow-rate changes which can be caused by:

a/ by pressure pulsations generated by main circulation pumps in the coolant loops of the reactor primary circuit. Slightly different pumps revolutions generate periodic vibrations which can cause an amplification of a load of control rod components.

b/ by changing the position of the bodies when the control rod moves in pressurized water of reactor core,

c/ by flow-induced vibrations of rod and small movement of walls of the guide channel.

The simulation described in this paper is focused on the study of flow field changes generated by “body-motion” by changing the position in the computational domain.

2. Computational models of control elements

One possible approach how to simulate “motion in the domain” is the method of “sliding-mesh” which is often used in CFD simulation of pressure changes. As the engineering case to demonstrate the main possibilities of the CFD "sliding-mesh" method, simulations of the hydrodynamic situation around the model of moving control rods were chosen.

The basic principle of the “sliding-mesh” method is known: this method is based on the motion of one or more moving zones relatively to stationary non-moving zones. The 3D-case domain includes moving/stationary zones as volumes with “unchanging shape and size” and non-deforming computational mesh in the domain. An important part of the “sliding-mesh” setup model is the “interface-zone”. Each domain cell zone is bounded by at least one “interface zone”. These cell zones slide (rotate or translate) relatively to each other along the mesh interface in discrete steps during the calculation.

The computational model was created to help simulate the slow motion of a control rod in guide-channel:

a/ longitudinal movement of the rod during regulate process,

b/ very small movement of the rod and walls of guide-channel in the transverse direction as a result of a seismic event.

The basic setup of “sliding-mesh” computational models includes:

- the geometry of moving and stationary zones, moving zone contains “moving-body”, in our case it is the rod, see Fig. 1, which depicts a simple scheme of geometry and main dimensions of control rod model for simulations,

- computation mesh of domain, types of mesh elements, boundary layer requirements,

- suitably defined planes of interfaces (dimensions and location),

- types and parameters of boundary conditions, material properties of flowing media,

- time-dependent solvers and numerical algorithms: choice of time-step size, number of time steps or number of iterations per time-step, all this can significantly affect the process of convergence of the calculation.

Seismic event from a CFD point of view is considered as follows:

- computational model is created in such a way to describe pressure changes caused by small relative movement between rod and guide channel as a result of a seismic event,

- this chaotic seismic motion is modelled “quasi-dynamically”, walls of the guide channel cylinder narrows (step-changes in diameter), accordingly various models have various positions of walls whereby modelled various dimensions of the gap between rod and channel at seismic motion. Fig. 1 shows an example of one version of the “seismic” model and its main dimensions of step-changes (diameters D_v, D_s and length L_s, etc.),

- seismic motion for CFD is considered as a motion with low frequencies and with small amplitudes, approx. of frequency 2Hz and amplitude 1mm,

- and finally, the model of the rod is moving longitudinally around the "wall-step" and thus generate additional pressure changes in the flow-field of the domain.

By performing a series of separate simulations of individual versions of the computational models, results in the form of time-varying pressure pulses were obtained.

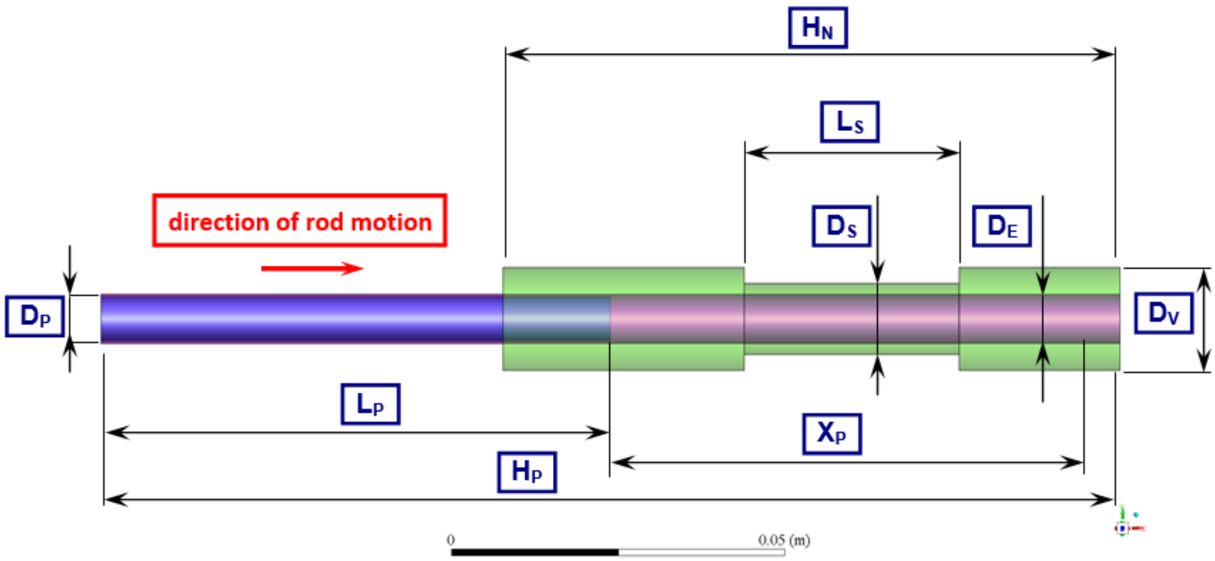

Fig. 1: Model of the control rod and main dimensions of channel bounce for seismic (D_v, D_s and L_s).

3. Testing and evaluation procedure of simulations

A number of test simulations were performed for different versions of computational models. The following quantities were changed in the computational model: rod movement speed, channel bounce dimensions (D_v, D_s and L_s), shift length of the rod. The aims of testing simulations were:

- to confirm the identification of small pressure pulses that occur when the front of the rod model passes around the model taper of the wall of the guide channel (see Fig. 2 which shows one chart of pressure pulse for illustrating),

- to check various possibilities of the setup of unsteady simulations of a seismic event,

- to obtain an overview of the pressure load values,

- to obtain an opinion on the influence of the input values of the models (D_v, D_s and L_s) and the resulting pressure pulse.

Fig. 2 shows the graph of a pressure pulse. Graph draws the time-dependent values of pressure on the front part of the control rod of the cluster model during its movement in the channel. The model describes the pressure pulse when passing the cluster through a simple constriction D_s/L_s (the time points of entry and exit of the cluster from the constriction are determined by the vertical lines t_1 and t_2).

CFD simulations helped to solve developing versions of computational models with various setups of boundary conditions (geometry and dimensions of D_s/L_s constriction, cluster speed). This created several computational models to verify the methodology of examining the pressure load of the rod of regulating cluster. The result of this research is the database of possible pressure pulses. This created database of the time-dependent pressure pulses can be used to investigate the effects on cluster dynamics during seismic events in the next stages of the project.

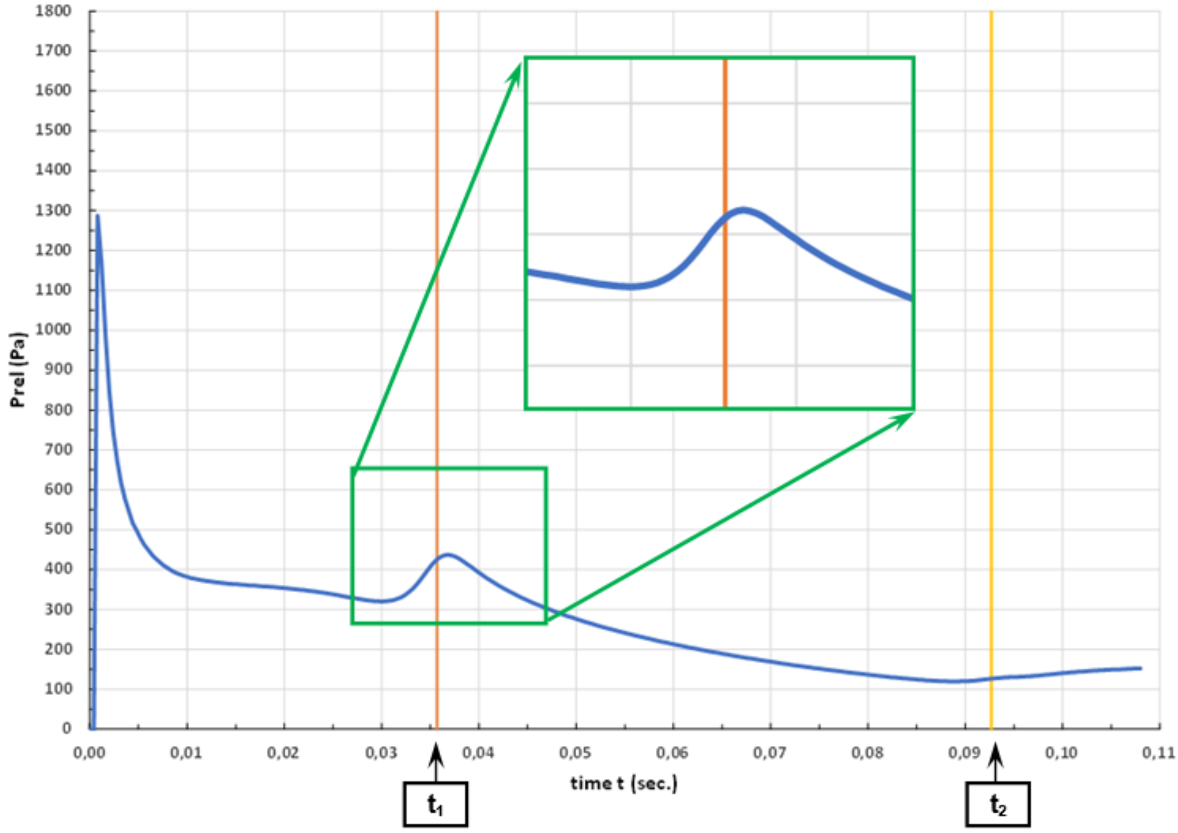

Fig. 2: Typical pressure pulse as a result of simulation (with detail of the pulse).

4. Conclusions

A relatively new approach to calculations and CFD simulations was briefly summarized. The possibility of applying CFD modelling as one of the tools for investigating the dynamic response of a nuclear reactor component to unexpected events was indicated. A change in pressure fields caused by seismic mainly, or by changes in the flow rate of coolant, respectively, is considered as an unexpected event. The developed methodology of CFD simulations and the creation of computational models of seismic load allow investigating additional pressure pulses and changes in pressure fields in the computational domain. Currently, additional simulations are still being performed to extend the created database of pressure pulses. The results correspond to the defined combinations of the dimensions of the guide channel bounce (constriction) and the movement of the control rod of the cluster.

The next steps in solving the problem are now being analyzed. In particular, how to use the database of pressure pulses to solve the dynamics of the control rods of the cluster, how to modify the results in the database of pressure pulses for substitution into equations of motion and how to edit them to the form of additional fluid coefficients. The dynamic response of the control rods of the cluster during a seismic event can be then solved by the equations of motion. A requirement to further modify the computational models may arise in the future.

Acknowledgement

The paper has originated in the framework of institutional support for the long-time conception development of the research institution provided by the Ministry of Industry and Trade of the Czech Republic to Research and Testing Institute Plzen.

References

Fluent (2021) *ANSYS/Fluent User's Guide, Release 2021/R1*. ANSYS, Inc., Canonsburg.