Simulation of flow induced vibrations in pipes using the LS-DYNA ICFD solver

Marcus Timgren[1]

[1]DYNAmore Nordic AB, Linköping, Sweden

1 Introduction

Flow-induced vibrations, (FIV), is a terminology that was first introduced by Robert Blevins in 1977 according to the paper by S. Miwa et al. [1]. For subsea piping systems, it is important to understand how the FIV affects the system so that failure can be avoided. The cause of the FIV is the transient behavior of the fluid/fluids inside the pipe, in most cases a mixture of liquids and gases are present. the transient behavior of the liquid and gases lead to fluctuations in the hydrodynamic forces and produces FIV. To understand this complicated multi-phase flow, fluid structure simulations (FSI) can be used. If only analytical models would be used to simulate this complicated phenomenon, it would lead to too conservative designs since the analytical models are often overly conservative. If an analytical method is to be used it should be calibrated to experimental data, or results from computational fluid dynamics, (CFD), simulation, and results from FSI simulation. In the offshore industry, it is often very expensive to get experimental data since the piping systems are located at the seafloor. Therefore, CFD and FSI simulation can play a big role in the design process of piping systems for offshore applications.

This paper will focus on an M-shaped jumper that has a circular cross-section, the same geometry has been analyzed in [2] but with Star-CCM+ and Abaqus FEA. The simulation in this paper will use LS-DYNA R9.1 [3], and since the CFD solver in LS-DYNA currently lacks support for multi-phase flow only single-phase simulations will be performed. The results from this paper are compared with the results from the multi-phase simulation in the paper by Voronkov et al. [4] and [2]. The effect of the damping from the surrounding water will also be looked at and a comparison between using shell elements versus solid elements for the pipe is made.

2 Model description

A jumper with the shape of an M is often used to connect a tree with a manifold in subsea applications. One reason for the M-shape is that the pipe should be able to flex due to thermal loads, pressure loads, and external fluid loads, see e.g. [5] and [6].

2.1 Geometry description

The structural part of the jumper is shown in purple in Fig. 1, the outlet and inlet have been elongated outside of the deformable structural part to avoid that the boundary conditions affect the flow in the pipe.

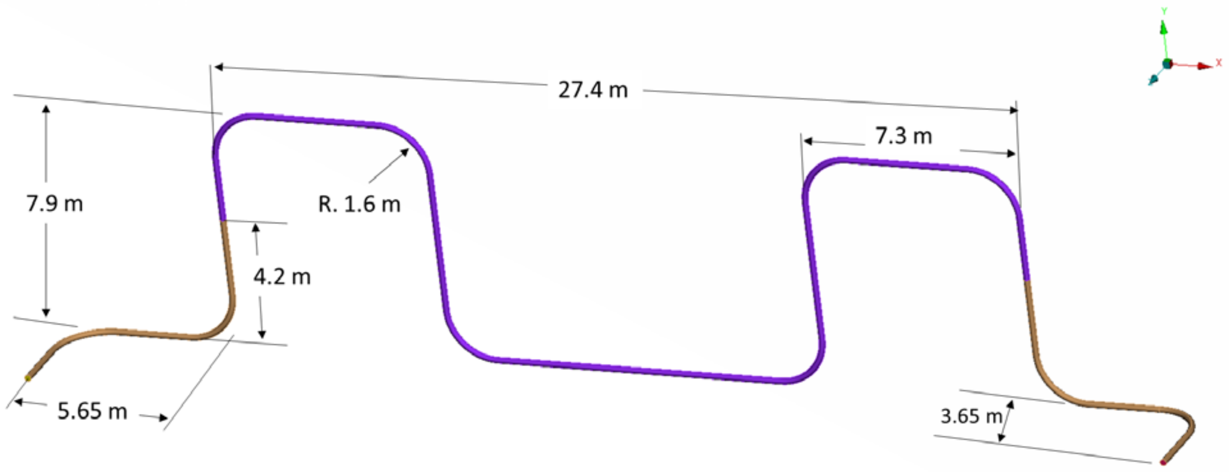

Fig.1: Geometry of the M shaped jumper.

2.1.1 Properties of the pipe

The pipe is 0.063 m thick and has an inner diameter of 0.210 m. The material model that will be used is an elastic model with the following parameters: Young's modulus = 205 GPa, density = 7800 kg/m³, and Poison's ratio = 0.29. The pipe is modelled with quadrilateral shell elements, element formulation - 16, a fully integrated shell element for higher accuracy, which is set on the keyword **`*SECTION_SHELL`**. Since the geometry of the pipe wall is located at the inside of the pipe and not at the midsurface, the option to move the reference surface was tested in an eigenvalue simulation, the parameter **`NLOC`** is set on the keyword **`*SECTION_SHELL`**. The mesh that is used contains 25344 elements with 32 elements around the circumference of the pipe. Only one load is applied, the gravitational load of 9.81 m/s² in the y-direction. For the nodes at the ends of the pipe a clamped boundary condition is used.

2.1.2 Properties of the fluid domain

The ICFD solver in LS-DYNA has an automatic volume mesher that creates the volume mesh based on the surface mesh that is provided by the user. The pipe is meshed with 239756 elements and three boundary elements were used in the simulation, which yielded a fluid volume mesh of 5.8 million tetrahedral elements. No mesh convergence study was performed since a similar mesh as in the study by Mueller and Voronkov [2] was used. The fluid is water with a density of 1000 kg/m³ and a dynamic viscosity of 1 mPas and for the inlet a prescribed velocity of 3.048 m/s is used and for the outlet a zero-pressure condition is used. For the pipe wall a no-slip condition is used as boundary condition.

2.2 Settings for the CFD solver

The time step was set to 0.0059 s which corresponds to 1/60th of the period for the seventh eigenvalue mode, see [2]. The end time was set to 10 s to capture the deformation movement over a couple of cycles. Due to limited time and computer resources a longer end time was not possible since the current FSI simulation took approximately 100 h on 64 cores. The turbulence model used is the default variational multiscale approach, which is based on a scale decomposition performed in the discretization process [7].

2.2.1 FSI settings

A strong two-way coupling was used which means that both displacement and pressure is transferred between the solvers. This means that the implicit structural solver must be used. LS-DYNA will use the smallest time step between the two solvers, which in this case was the fluid time step. To define the FSI coupling only two keywords are needed namely: **`*ICFD_CONTROL_FSI`** and **`*ICFD_BOUNDARY_FSI.`**

2.3 The structural model

To account for the surrounding water, the hydrodynamic added mass was added to the structure. The density of the structure was scaled based on the following equations:

$$M_E = M_S + M_{IM} + M_H L \tag{1}$$

$$M_H = \pi \cdot r^2 \rho \tag{2}$$

$$M_{IM} = V\rho \tag{3}$$

$$F = \frac{M_E}{M_S} \tag{4}$$

Here V is the fluid volume inside the structural pipe, r is the outer radius of the pipe, and L is the length of the structural pipe. M_H is the added mass per unit length, M_{IM} is the mass of the fluid inside the pipe and it is zero for the FSI simulation since then the fluid is modelled with the ICFD solver. M_S is the structural mass for the pipe, in this case it is 8664 kg. The scaling factor F is used to scale the original density of the structure [8]. Two scaling factors are calculated one for a pipe with water and one for a pipe with 50% water and 50% air. The densities used were 1000 kg/m³ for the water and 1.2 kg/m³ for the air. This gives two scaling factors of 1.5 and 1.41 for the water and the mixed system respectively.

These two factors were used in the eigenvalue analysis where only the structure is simulated, in the FSI simulation a different factor is used, 1.31, since then the fluid in the pipe is accounted for in the simulation.

2.3.1 Implicit solver settings

The nonlinear solver number 12 with BFGS updates was used with an initial time step of 1/60th of the period for the seventh eigenvalue mode, i.e. 0.0059 s. On the keyword **`*CONTROL_IMPLICIT_SOLUTION`**, the abstol value was changed to 1E-20 and for the rest of the parameters the default values were used. The dynamics was activated and some numerical damping was used by setting gamma to 0.6 and beta to 0.38.

2.4 Run configurations

The mpp/LS-DYNA R9.1 in double precision was used for all simulations. For the eigenvalue analysis 2 cores were used and for the FSI simulations 64 cores were used. The run time for the eigenvalue simulations are approximately 10 s and the run time for the FSI simulation with shells was approximately four days. The run time for the FSI simulation with solid elements was approximately seven days.

3 Results from the eigenvalue analysis

First an eigenvalue analysis was performed of the structural pipe only, without the fluid. To understand the effect of the added mass, simulations using both the regular mass and with the added mass were performed. Two different added masses were used, one for a system with water and one with a system with 50% water and 50% air, with added mass 1.5 and added mass 1.41 respectively.

Table 1: Frequencies in Hz of the first eight eigenvalues.

Case	Mode 1	2	3	4	5	6	7	8
No added mass nloc=-1	0.426	1.253	1.768	1.775	2.229	2.411	2.793	8.181
No added mass	0.619	1.379	1.781	1.874	2.168	2.174	2.969	6.407
Added mass 1.41	0.521	1.161	1.50	1.578	1.826	1.831	2.500	5.396
Added mass 1.5	0.505	1.126	1.454	1.530	1.770	1.775	2.424	5.232
Solid	0.710	1.562	2.039	2.122	2.452	2.478	3.392	7.288
Ref. [2]	0.59	1.30	1.70	1.77	2.06	2.07	2.82	6.07

Using nloc set equal to -1 did not give the desired result when comparing with the model with solid elements, this may be fixed in a later version of LS-DYNA. It was determined that nloc equals to zero gave the best result so that was used for all simulations with shell elements. The results for the case when no added mass was used agreed well with the results from the study performed by Mueller A. and Voronkov O. [2].

3.1 Eigenmodes 1-4

The first eigenmode is a movement of the lower bend in the horizontal direction compared to the second that is a vertical motion of the pipe. Eigenmode three is a rotational motion in the horizontal plane around the middle point of the lower u-bend. The last eigenmode is also a rotational motion around the midpoint of the lower u-bend but in the vertical direction.

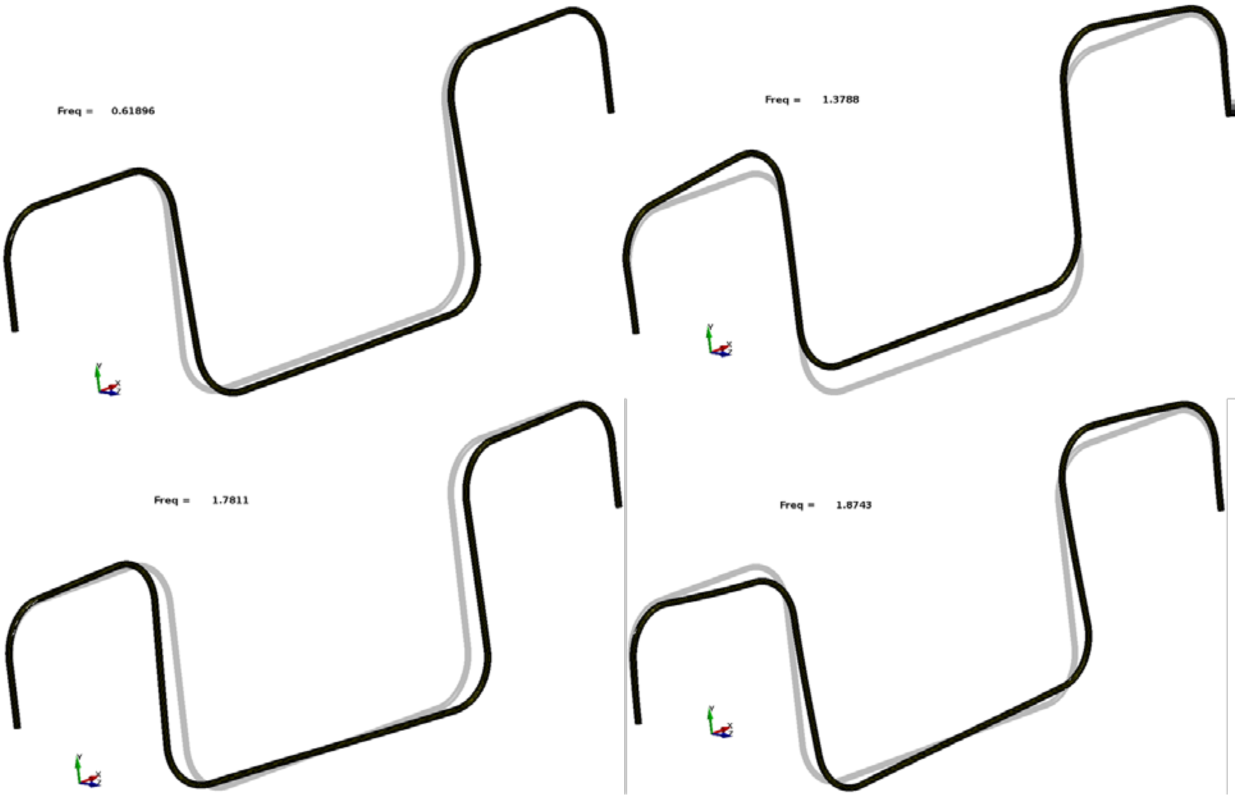

Fig.2: The first four eigenmodes with the original geometry in a shaded grey color, eigenmode 1 in the top left corner.

4 Results from the FSI simulation

In this section are the results from the FSI simulations presented and some results for the fluid part.

4.1 Fluid results

All results in this section are from the simulation with solid elements since that simulation represents the reality the best.

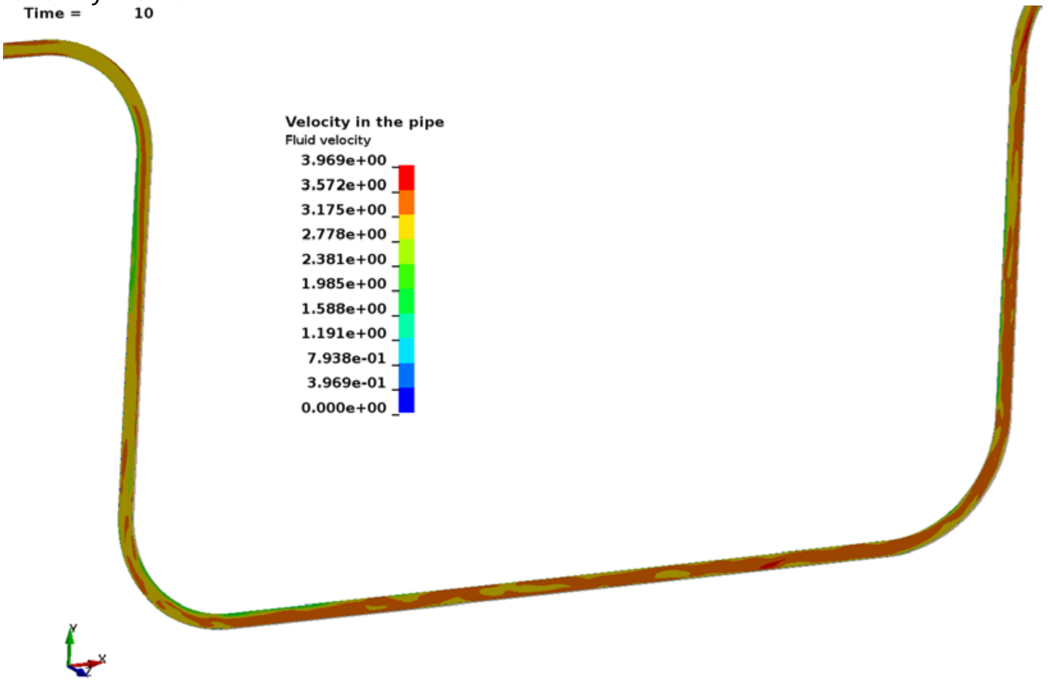

Fig.3: Fluid velocity in the pipe at the end time, t=10 s.

The bends of the pipe introduces turbulence and leads to the turbulent velocity field shown in Fig. 3. At the inner wall after a bend the velocity is lower since most of the flow is pushed against the outer wall of the bend.

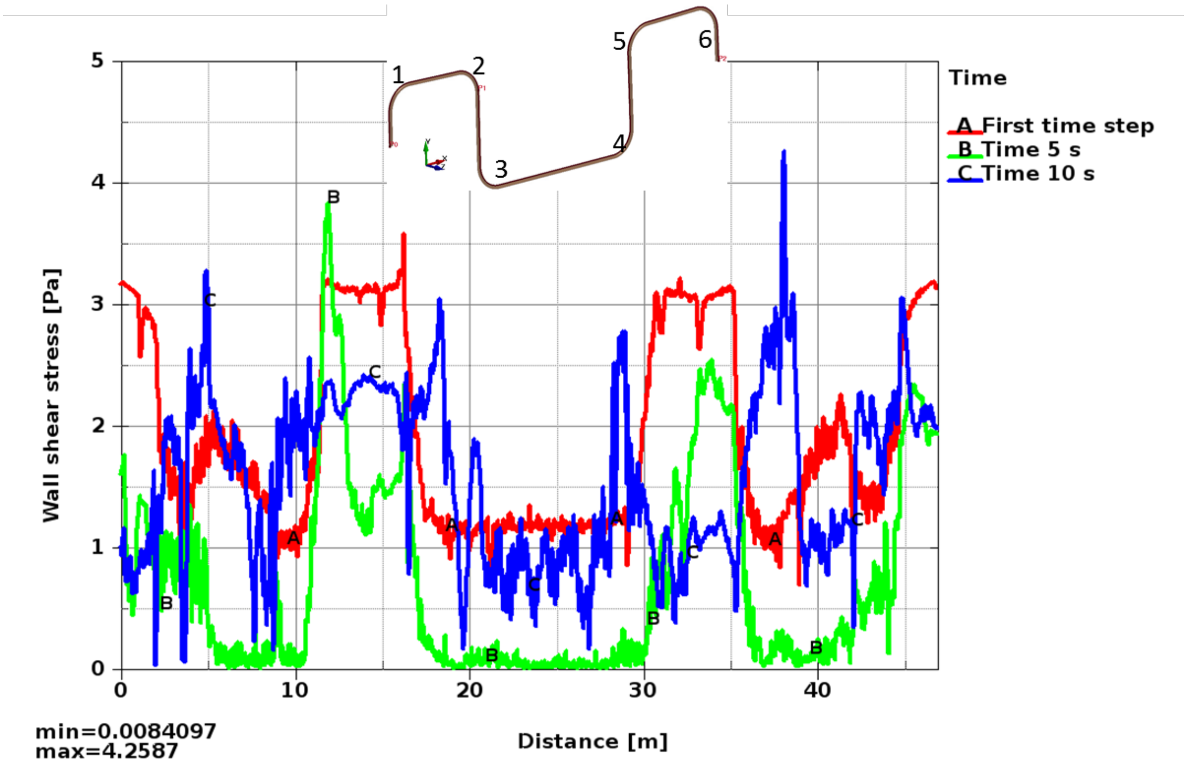

Fig.4: Wall shear stress (WSS) along the upper side of the structural pipe at Z coordinate zero.

The vertical sections of the pipe are between 11.5-18.6 m and 30.5-37.9 m for the U in the middle. The peaks for the WSS occurs in the vertical sections of the pipe. One exception is for time 10 s where the second peak has been shifted to the end of the fifth bend for the structural pipe.

4.2 Results for the shell element simulation

The results for the added mass do not differ that much from the simulation without added mass. Oscillations for the added mass case gets slightly higher and the period for the oscillations is also slightly different.

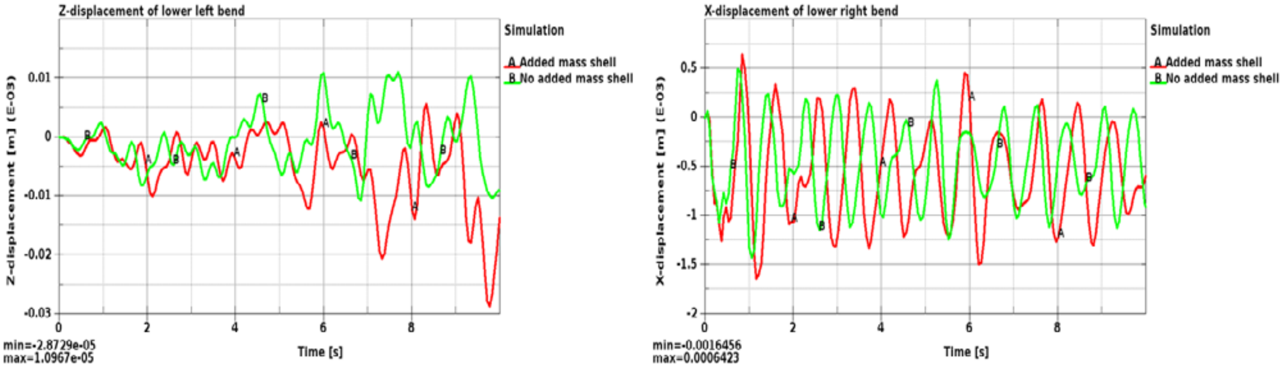

Fig.5: Displacement of the pipe at the middle of the lower pipe bends.

4.3 Results for the solid element simulation

A surprise is that the oscillations for the solid simulation were much bigger than for the shell cases. The z-displacement has a max displacement of 33 mm at approximately 5.8 seconds. This is a factor of 1000 bigger than in the shell simulations. The x-displacement only differs with a factor of 10, this difference could be explained by the different geometries of the two cases.

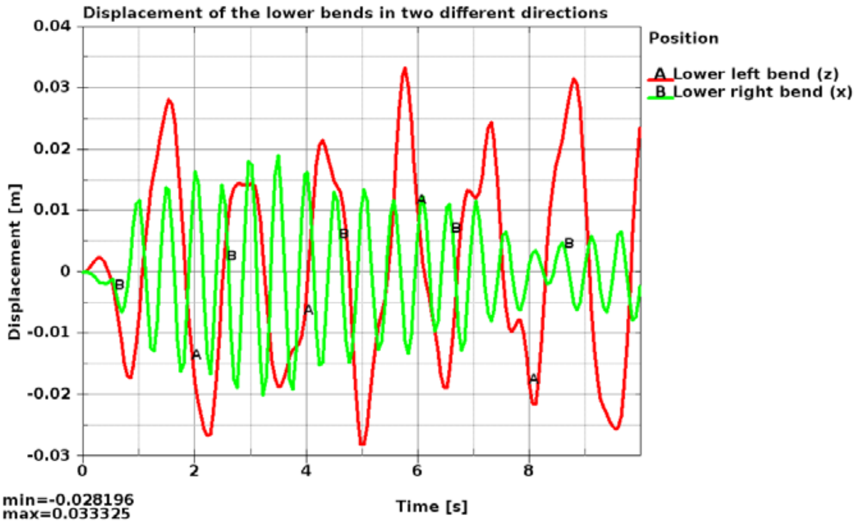

Fig.6: X-displacement of the lower right bend and Z-displacement of the lower left bend.

4.4 Difference between single-phase and two-phase flow

In the paper by Chica L. [9], the maximum displacement is 1.8 mm but for a two-bend model and not a whole m-shaped jumper. This was for a two-phase flow and for the structural pipe shell elements were used. The maximum displacement for the shell simulation without added mass aggress well, it has a max of 1.4 mm for the X-displacement. With added mass the displacement is even closer with a max displacement of 1.6 mm. For the solid model the displacements do not match at all. No good reference was found for the von Mises stress of a similar simulation and was therefore not included in the comparison.

5 Discussion

Eigenvalues for nloc option -1 did not give satisfying results, the expected results would be that would give frequencies between nloc set to zero and the solid simulation. Nloc set to zero gave a good agreement with the reference [2], which was the reason why nloc equal to zero was used in the FSI simulation. The added mass load cases gave lower eigenvalues as expected, since one adds mass to the system and keeps the structural properties constant. Based on the difference between the solid case and the reference it looks like the reference [2] also used a midsurface that is located at the inside of the pipe, i.e. nloc set to zero. This gives an error compared to the real geometry of the pipe, i.e. an approximation of the pipe is used both in the shell simulations and the reference [2].

In the lower section of the u-bend the flow is quite turbulent with velocity fluctuations throughout the pipe, see Fig. 3. This leads to fluctuations in the pressure and to vibrations of the pipe, in cases where the frequency is close to an eigenvalue the pressure fluctuations will lead to severe structural vibrations.

For the WSS it can be seen that the vertical pipe sections have the highest values, see Fig. 4. In the vertical sections the flow is accelerated/decelerated due to gravity. This leads to a high shear stress on the pipe wall before/after the lower pipe bends due to that most of the flow impacts the pipe wall here.

The displacement values do not match at all for the shell and solid simulations. There should be a small difference since the geometry is not the same. One cause of the large difference can be that the eigenvalues have a part in the difference, since a “lock in” may have occurred in the solid simulation: The fluid introduces vibrations with a frequency that is close to the natural frequencies of the pipe and that does probably not occur for the shell simulations.

To get more reliable results the simulation time would have to be extended so that the flow passes through the fluid domain at least 2 times. To have one fluid particle go through the pipe two times, the simulation time would have to be increased to approximately 23 seconds.

For the structural model the gravitational load is applied directly with the dynamics activated instead of performing a few time steps in static mode and then turning on the dynamics. This resulted in oscillations in the y-direction and can be an explanation of the difference in results between the shell and the solid element simulation. Another explanation for the difference in results could be that the shell simulation uses a different geometry since the nloc option -1 did not work as it should. One can also see a difference between the models when looking at the eigenvalues. For the solid model an eigenmode in both the x and z direction has been triggered so that the displacement is amplified. The same thing does not occur for the shell models, which explains the difference in results.

The damping of the water around the M-shaped jumper must be accounted for since a jumper is always surrounded with water when it operates. The added mass from the water does not account for the damping of the movement due to pressure forces from the water. The easiest way would be to apply a damping based on experimental values or analytical solutions. Another way would be to model the flow around the pipe as well but it will lead to long computational times and one might not gain that much more accuracy from such a simulation.

6 Literature

[1] Miwa S., Mori M., and Hibiki T, Two-phase flow induced vibration in piping systems, Progress in Nuclear Energy, 78, 2015, p. 270-284.

[2] Mueller A. and Voronkov O., A study of Jumper FIV due to multiphase internal flow: understanding life-cycle fatigue, Star Global Conference, 2015.

[3] LS-DYNA Keyword User’s Manual, Volume I-III, LS-DYNA R9.0, Livermore Software Technology Corporation (LSTC), 2016.

[4] Voronkov O., Mueller A., Read A, and Goodwin S., Simulation helps assess fatigue life of subsea jumpers, Offshore, 2014, p. 94-98.

[5] Pontaza J. P. and Menon R. G., Flow-induced vibrations of subsea jumpers due to internal multi-phase flow, ASME 30th International Conference on Ocean, Offshore and Artic Engineering, 2011.

[6] Nair A., Chauvet C., Whooley A., Eltaher A., and Jukes P., Flow induced forces on multi-planar rigid jumper systems, ASME 30th International Conference on Ocean, Offshore and Artic Engineering, 2011.

[7] Guasch O. and Codina R., A heuristic argument for the sole use of numerical stabilization with no physical LES modeling in the simulation of incompressible turbulent flows, 2007.

[8] DNV-RP-H103, Modelling and analysis of marine operations, 2011, p. 139 Appendix A.

[9] Chica L., Fluid Structure Interaction Analysis of Two-Phase Flow in an M-shaped Jumper, Star Global Conference, 2012.