Numerical Studies on Dynamic Behaviour of Pipelines

K. Calonius

Technical Research Centre of Finland (VTT)

Dynamic excitation due to a pipe break can cause pipe to abruptly displace and hit the components, instrumentation and equipment nearby. In order to minimize the extensive damage caused by such pipe whips, different types of restraints and supports are designed for the pipelines.

Here, structural dynamic behaviour of pipelines is studied with finite element method. A relatively short pipe line section with one bend and one restraint and rigidly fixed from its other end is chosen as a test case. The usability of different types of elements provided by Abaqus code (mainly version 6.7) in modelling the pipe, restraint and adjacent civil structure is tested.

First, the stiffness of the restraint as well as the flexural stiffness of the pipe cross-section are solved with static analyses with a detailed model using three-dimensional solid and shell elements. After that, the model is substituted with couple of simpler models using special purpose structural elements such as pipe, elbow, spring and pipe support elements having the corresponding stiffness properties.

The eigenmodes of models are calculated and compared with each other. The pipe whip is simulated with nonlinear dynamic analyses. The displacement and stress results of different models are compared with each other and the reliability and adequacy of different element types are discussed. Sensitivity study is made by varying the analysis type, material properties, mesh density and element properties (interpolation, number of integration points and ovalization modes).

The results of the most adequate simple models with the right combinations of special-purpose elements provided by Abaqus corresponded well to the ones of the much larger three-dimensional solid and shell element models. Long pipe runs should preferably be modelled with special-purpose elements in order to save time and prevent errors due to overly large and complicated models.

Keywords: Dynamics, Elbow, Impact, Mechanisms, Pipes, Pipeline, Plasticity, Tube Bending

1. Introduction

The paper concentrates on simulating the dynamic behaviour of a certain relatively short pipe section with one restraint. Its stiffness is first calculated with a relatively detailed finite element model, which is then substituted with simpler-special purpose elements. A pipe guillotine break is chosen as a test case that is simulated with different kinds of models. The results are compared with each other and their reliabilities are evaluated.

The study is part of the Finnish public research programme on nuclear power plant safety. The results will be applied to assess structural safety of nuclear reactor components, as well as to evaluate their remaining lifetime during service, by coupling between advanced numerical modelling tools and material's characterisation data, in order to cover the entire chain of structural safety related aspects coupling the interaction of effects of loads, existing flaws, material's relevant damage mechanisms and degradation of properties during service.

Every continuum structure can be modelled and solved with general-purpose solid continuum volume elements. However, that can be very ineffective and time-consuming, since usually very large amount of volume elements are needed to catch the right behaviour. That is why it is useful to have special-purpose structural elements instead. Beam, elbow, shell and solid volume elements are used in this study for modelling the pipeline.

There are many types of supports and restraints in power plants. The supports support components and pipelines mainly against the gravitational forces in vertical direction. Either the components rest on the support or they are suspended by the so-called hangers. The function of restraints is to take the dynamic loads in case of accidents such as pipe breaks or earthquakes when the pipes can have exceptionally large movements and velocities. The thermal expansion is taken into account by leaving sufficiently large gaps between the pipe and restraints.

If special-purpose elements are not used, contact behaviour has to be formulated between the pipe and restraint surfaces. That is why it is also useful to have special-purpose elements instead, which can include the gap, contact, stiffness and even possible viscous properties. Those elements in Abaqus (Abaqus, 2006) include for instance:

- Spring elements
- Dashpot elements
- Tube support elements
- Gap elements
- Tube-to-tube elements
- Frame elements
- Connector elements

Solid volume and spring elements are used in this study for modelling the restraint.

2. Model description

Since all kinds of sensitivity study is made and adequacy of different model types is assessed, it is practical that the structure is not too complex with many variables. A relatively short pipe section with one restraint and one bend is studied. The whole combined model consists of three parts: the pipe, the restraint with a small gap (approximately 6 cm), and a concrete block which represents the civil works or containment wall near the pipeline. Figure 1 shows the example pipeline geometry simulated in this study with main dimensions. In the following analyses, the lower right end of the pipe is rigidly fixed, unless otherwise mentioned. The postulated pipe break takes place in the upper left end of the section.

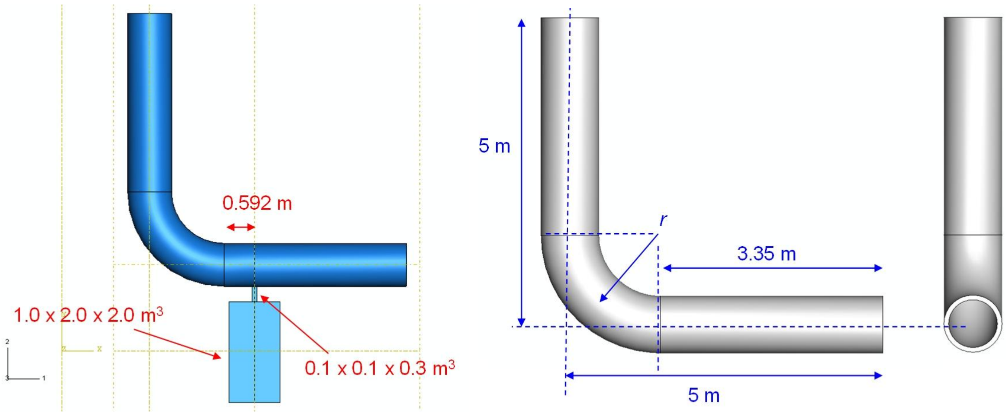

Figure 1. Dimensions of the pipeline section.

The different finite element model types are shown in Figure 2. The concrete block and the restraint are modelled either with solid volume elements (C3D8R) or with a nonlinear spring element (SPRINGA). The pipe is modelled with solid volume, shell, pipe or elbow elements. The detail of the solid volume element model of the pipe is shown in the upper left hand corner. It has four linear 8-noded elements through the wall thickness and altogether 272 136 elements (C3D8R). The pipe (or elbow) element model is shown in the upper right hand corner. Depending on the mesh density, it has either 18 or 36 elements (6 or 12 elements along the bend) (PIPE31, PIPE32, ELBOW31, ELBOW31B, ELBOW31C or ELBOW32). There are three different cases elementwise: 1) only pipe elements are used, 2) only elbow elements are used or 3) pipe elements are used for the straight sections and elbow elements for the bend section. The spring element can also be seen near the bend in the horizontal section. The shell element model with a fine mesh is shown below the other two models. It has 44 shell elements (S4R) around the pipe circumference and the total number of elements is 6864. The model with a coarse mesh has only 12 elements around the pipe (total number of elements 456).

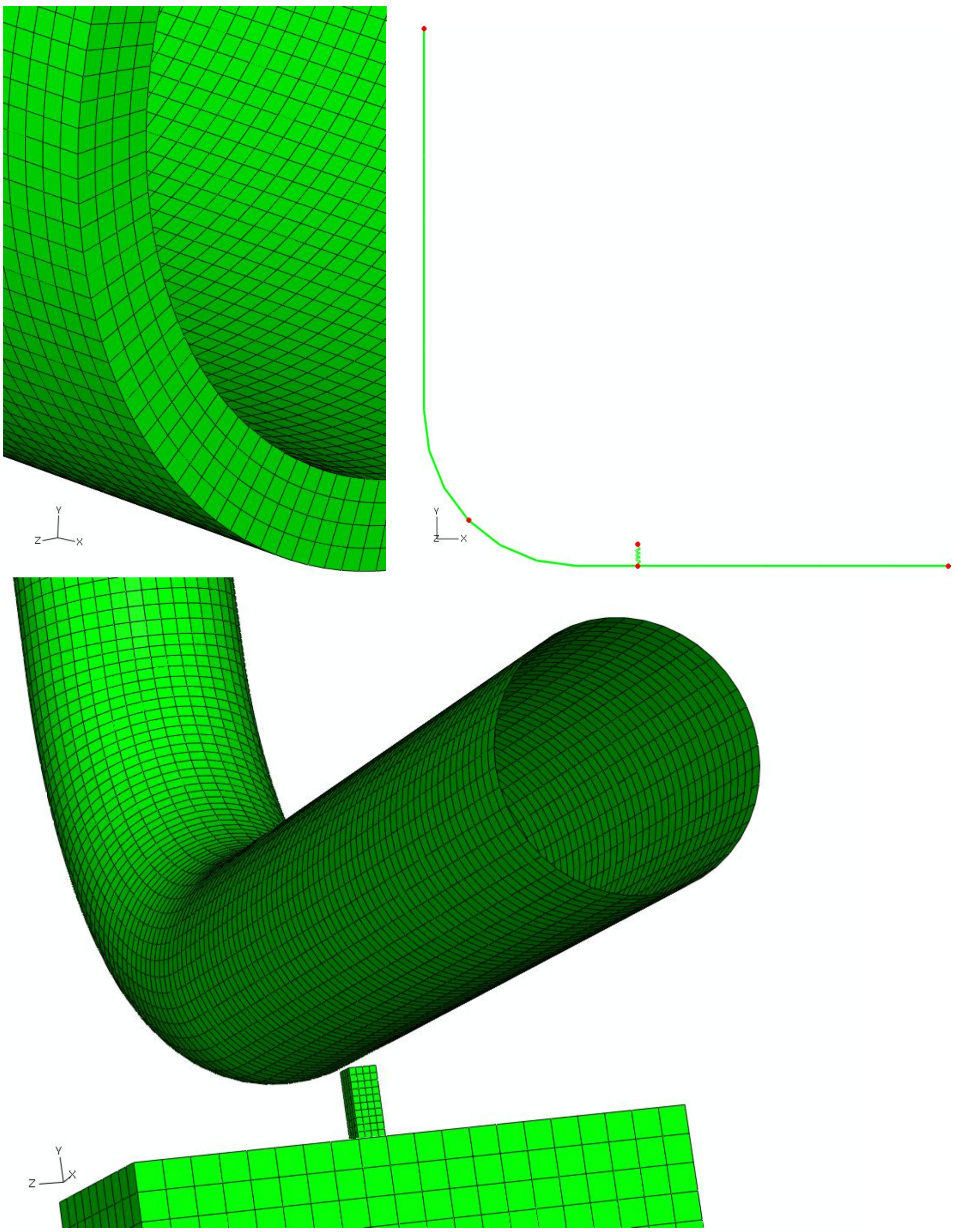

Figure 2. Solid volume, pipe and shell element models.

Two materials are used, steel and concrete. The same steel is used both for the pipe and for the simple restraint with a square cross-section. Water is not included and its mass is not taken into account, but the high inner pressure is applied in some cases. The material properties for the linear elastic materials needed for the analyses are the density, the Young's modulus and the Poisson coefficient. In some cases also plastic properties are included by giving the stress strain curve as data pairs. The yield point (at 0.2 % plastic deformation) is 210 MPa. Properties in room temperature are used. Most probably again in the case of an accident, the material would be in operational temperature and thus substantially softer. Table 1 shows the material property values.

Table 1. Material properties.

material	steel	concrete
density, ρ	7850 kg/m^3	2400 kg/m^3
Young's modulus, E	210 GPa	35 GPa
Poisson coefficient, ν	0.3	0.2
Yield point, σ	210 MPa	-

3. Natural frequency extraction of a pipeline section

In order to find out the adequacy of different element types and the sensitivity of the models to different parameters many different cases are studied by extracting their natural frequencies and comparing them with each other. The models are naturally linear. There is no restraint. The lower end of the pipe is fixed and the upper end is free.

Altogether 19 different cases were solved. Following characteristics were varied: Element type and interpolation, mesh density, number of integration points and ovalization modes, whether the rotations are free in the fixed end of the shell element model, the inner pressure type with elbow elements (open or closed) and the inner pressure with shell elements. First ten eigenfrequencies were solved with Lanczos method and they are shown together with total mass of the model in Table 2.

The default parameter values for elbow elements are used unless otherwise mentioned. There are five integration points through the pipe wall thickness and twenty around the pipe. The maximum available six Fourier ovalization modes are used. Element types ELBOW31B and ELBOW31C use a simplified version of the formulation, in which ovalization only is considered (no warping) and axial gradients of the ovalization are neglected. Also quadratic interpolation (instead of linear) along the pipe is used in some cases (for example PIPE32).

Table 2. Natural frequencies of different models.

	C3D8R	S4R shell fine mesh	S4R shell fine mesh rotation free	S4R shell fine mesh pressure	S4R shell coarse mesh	PIPE31	ELBOW31
mass (kg)	15 040	15 032	15 032	15 032	14 862	15 035	15 045
mode 1 (Hz)	10.847	10.834	10.821	11.223	10.585	11.311	10.839
mode 2 (Hz)	11.147	11.131	11.116	11.559	10.925	12.018	11.143
mode 3 (Hz)	28.181	28.202	28.136	28.580	28.139	33.330	28.226
mode 4 (Hz)	32.314	32.250	32.181	32.535	31.517	34.089	32.690
mode 5 (Hz)	105.86	106.03	105.78	106.38	105.93	115.35	118.99
mode 6 (Hz)	127.24	127.01	126.75	127.40	125.29	127.87	124.92
mode 7 (Hz)	150.85	151.11	151.00	151.56	150.69	160.35	181.86
mode 8 (Hz)	186.55	186.46	185.76	186.41	185.85	189.28	186.44
mode 9 (Hz)	215.11	214.90	214.11	214.97	213.02	216.63	220.52
mode 10 (Hz)	258.43	258.53	258.31	258.81	258.28	261.36	252.34

	PIPE31 + ELBOW31	PIPE31 + ELBOW31 fine mesh	ELBOW31 fine mesh	PIPE32 fine mesh	ELBOW32 fine mesh
mass (kg)	15 045	15 045	15 045	15 045	15 045
mode 1 (Hz)	10.674	10.696	10.854	11.320	10.858
mode 2 (Hz)	10.877	10.902	11.165	12.036	11.172
mode 3 (Hz)	27.086	27.107	28.160	33.565	28.138
mode 4 (Hz)	32.058	32.247	32.831	34.135	32.878
mode 5 (Hz)	102.95	103.17	118.72	115.59	118.63
mode 6 (Hz)	119.94	120.41	124.92	128.94	124.92
mode 7 (Hz)	149.63	150.16	182.53	161.45	182.75
mode 8 (Hz)	184.83	185.36	185.61	189.89	185.34
mode 9 (Hz)	214.21	216.36	220.92	218.99	221.06
mode 10 (Hz)	251.76	254.04	254.23	264.75	254.86

	ELBOW31B	ELBOW31C	ELBOW31 3 ovalization modes	ELBOW32 fine mesh pressure open	ELBOW32 fine mesh pressure closed
mass (kg)	15 045	15 045	15 045	15 045	15 045
mode 1 (Hz)	10.745	10.760	10.842	11.001	10.880
mode 2 (Hz)	10.939	10.962	11.147	11.192	11.200
mode 3 (Hz)	27.371	27.472	28.245	28.425	28.269
mode 4 (Hz)	32.235	32.265	32.694	32.613	32.933
mode 5 (Hz)	112.22	112.50	119.02	118.81	118.88
mode 6 (Hz)	116.67	116.94	124.94	125.67	125.12
mode 7 (Hz)	178.84	179.30	181.95	183.10	183.37
mode 8 (Hz)	183.35	183.52	186.47	184.16	185.58
mode 9 (Hz)	214.71	215.09	220.58	221.37	221.54
mode 10 (Hz)	249.92	250.10	252.36	255.32	255.08

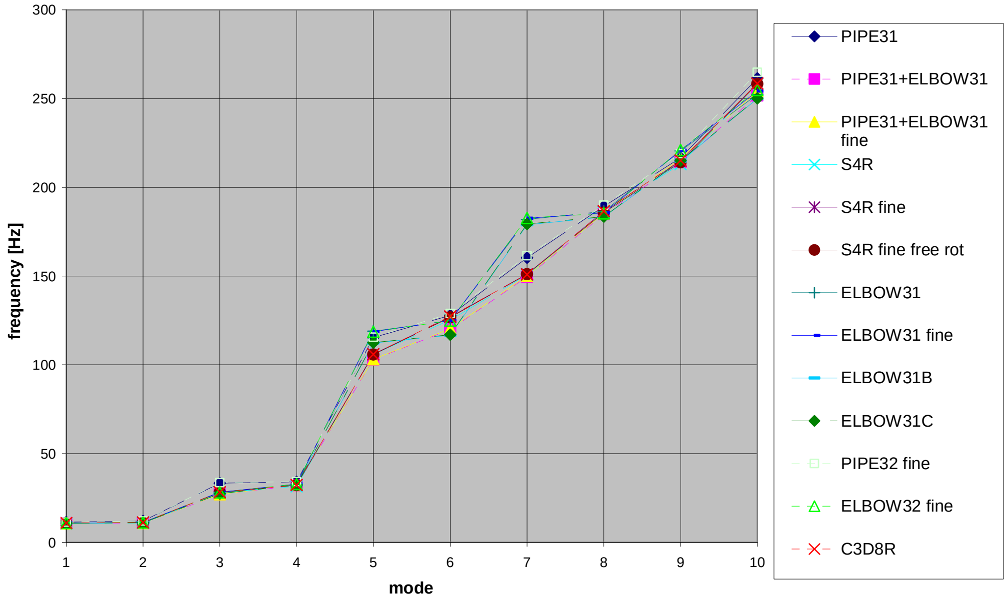

Figure 3. The eigenfrequencies [Hz] of different models.

The masses are really close to each other in every case, which partly verifies the models. The most accurate model can be assumed to be the one with solid volume elements (the first result column in Table 2) or perhaps the one with a very fine shell element mesh (the second column in Table 2). Their eigenfrequencies are very similar. The model with only linear pipe elements (the 6th column) has some discrepancies and is not accurate enough. The ones with also elbow elements correspond better with the detailed models. Only the 7th mode has got some major discrepancies (elbow element models being clearly stiffer). Figure 3 shows the eigenfrequencies of most of the models. They are all relatively close to each other.

The deformed shapes of the 7th mode of different model types are shown in Figure 4 (the leftmost model has only pipe elements). Whether the rotational degrees of freedom of the shell element model end are fixed or left free does not make much difference. The inner pressure makes the models slightly stiffer. The model that used both pipe and elbow elements is really close to that when it comes to eigenfrequencies, even closer than the coarse shell model. Mesh refinement slightly enhances its accuracy in a global sense. However, in a local sense a fine mesh would give more detailed stress results. Two pipe model types are chosen for the dynamic analyses: the model with a fine shell element mesh (the 2nd column) and the model with 18 linear elbow elements (the 7th column).

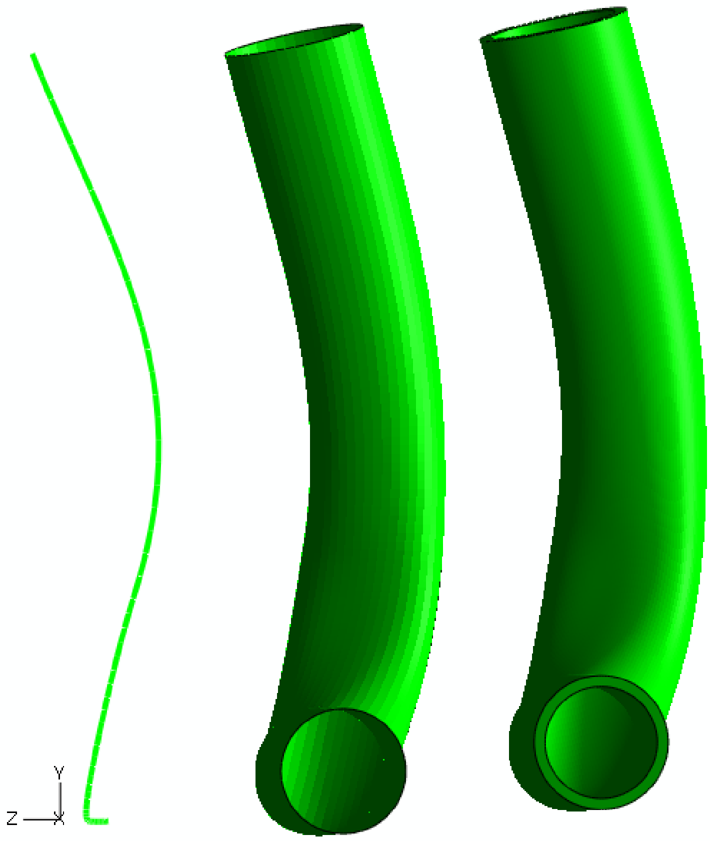

Figure 4. Mode 7 of different type of models.

4. Dynamic analysis of a guillotine pipe break

A guillotine pipe break is chosen as a dynamic analysis case. It is first analysed using the combined model shown lowermost in Figure 2. That is a relatively large model using shell elements for the pipe (chosen based on the eigenfrequency results above) and solid volume elements for the concrete and the steel restraint. After that as a comparison, the same case is analyzed with a model using only special-purpose structural elements; elbow elements for the pipeline and a nonlinear spring element for the support. The properties such as the stiffness of the spring element are determined by preliminary static analyses with the large combined model. Nonlinear spring element (SPRINGA) with a stiffness of 1 GN/m and a gap of 6 cm is used.

The release of fluid from a break in high-energy piping could result in significant changes in flow characteristic within the piping system, creating reaction forces, which dynamically excite the piping. If these forces are sufficient to cause pipe whip, nearby required systems must be protected or designed to withstand the result of pipe whip. In reactor primary circuit, it is also important to keep the break flow area and thus the outflow of coolant limited in case of pipe break.

The fluid forces acting on the ruptured pipe are a function of time and space, and depend on the fluid state within the pipe prior to rupture, the break flow area, frictional losses, and plant system characteristics. Break flow area is often assumed to develop within one millisecond after break initiation. Fluid forces can be divided into two parts: initial thrust force and steady state thrust force. The former is:

$$F_{in} = P_0 A_e \qquad \text{(Equation 1)}$$

where P_0 is the initial pressure in the pipe and A_e the break plane area. The steady state thrust coefficient is dependent on the fluid state and the frictional effects.

Normally, a dynamic time history analysis shall be conducted of the ruptured piping to determine its response to the exciting forces. The pipe whip constraints and other objects that can modify the pipe whip motion shall be considered in the analysis. Absorption of energy via plastic pipe deformation is typically the most desirable method in the design of piping and restraints for protection against pipe whip effects. The elastic and plastic behaviour of the pipe material shall be taken into account in the analysis (Vörös, 2002) (Micheli, 2003).

The analysis consists of two steps. The first step is static during which an inner pressure is applied to the pipe. It is linearly ramped up in one second time. The upper end (called “free end” in this report) and the lower right end of the pipe are fixed during this step. The second step is dynamic with automatic time incrementation. Right in the beginning of it the free end is released, which initiates the pipe break condition. The same inner pressure remains throughout the second step even though in reality the pressure decreases at some rate when the water bursts out through the free end. Table 3 shows all the six dynamic analysis cases. Element, material and analysis types are varied.

Table 3. Dynamic analysis cases.

Case No.	Case Code	Main element type	Main material type	Analysis type
1	SEI	shell	elastic	implicit
2	SEE	shell	elastic	explicit
3	SPI	shell	plastic	implicit
4	SPE	shell	plastic	explicit
5	EEI	elbow	elastic	implicit
6	EPI	elbow	plastic	implicit

Only some of the results are shown here. All the analyses were successful. The energy balance was maintained and the results of shell and elbow models had a good correlation, with both plastic and elastic material properties. With plastic steel properties the plastic hinges formed in both models in a similar manner and in a realistic way. Also stress results were in good agreement. In elbow elements, stresses can be defined around the circumference of the pipe and through its wall thickness. Also, the deformed shape of the cross-section is possible to be plotted. Figure 5 shows the vertical displacement of the pipe at the restraint for cases SEI and EEI (elastic shell and elbow models with implicit direct time integration). They correspond with each other very well. The restrain forces were also similar. Figure 6 shows the deformed shapes in cases SPI and EPI 0.1 s after the pipe break. The plastic hinge has been formed in both cases. Maximum displacements are 3.18 m and 2.38 m, respectively. The results with explicit time integration corresponded to the ones with implicit integration, maximum displacement in the plastic analysis (SPE) after 0.1 s being 3.07 m.

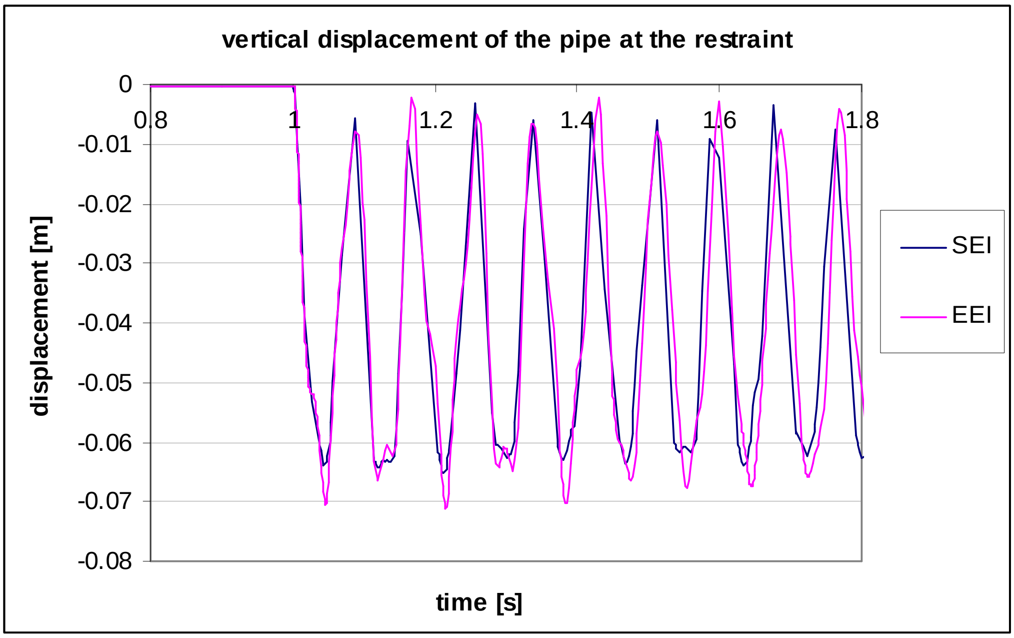

Figure 5. Vertical displacement of the pipe at the restraint.

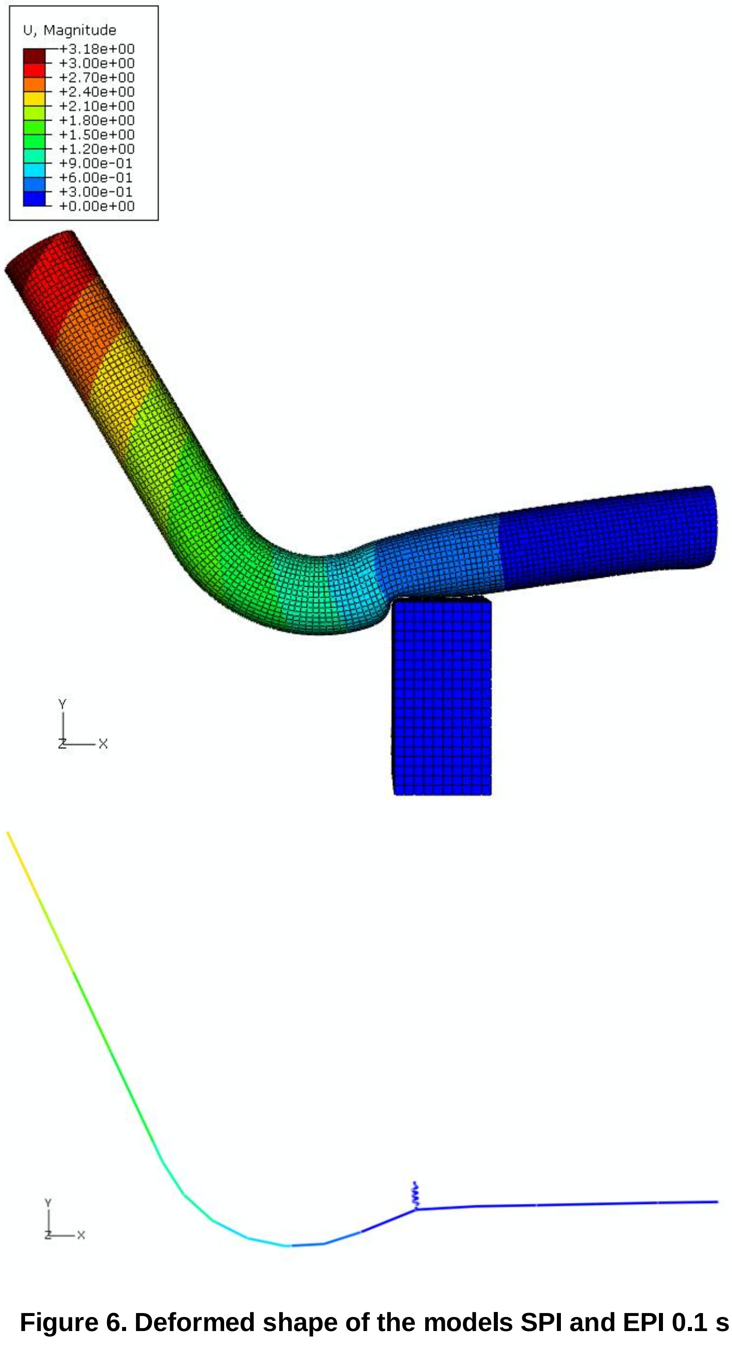

Figure 6. Deformed shape of the models SPI and EPI 0.1 s after the pipe break.

5. Conclusions

In this paper different kinds of ways to numerically model with finite element code Abaqus the power plant pipe runs and their supports are studied. A pipe guillotine break with whipping is chosen as a dynamic analysis case. Simple and typical pipeline geometry and materials are chosen. The test pipe line is modelled with different ways and the eigenfrequencies are compared with each other. The hydrodynamics of water or its mass are not included.

It is seen that in respect to the eigenmodes the pipe element model is adequate and almost as accurate as a shell element model with a fine mesh, especially when the pipe bend is modelled with elbow elements.

The stiffness of different components and the gap in a pipe impact to a restraint are evaluated and modelled with special-purpose elements. This is crucial when the restraints have to be simplified in extensive models with long pipe runs and many supports.

Finally dynamic analyses of a pipe break are conducted with different models that are confirmed as adequate. The chosen restraint type is relatively stiff. The calculated forces and stresses correspond to theory and the ones found in literature. The dynamic model behaviour also corresponds to the static model behaviour and the eigenmodes that are solved beforehand. Stresses would cause extensive yielding of the material.

Nonlinear spring elements are found to act realistically and seem very suitable to model restraints for this kind of dynamic problem. The results of the pipe element model corresponded well to the ones of the much larger shell element model.

This is not an extensive study but gives a good base for the future study and for making the decision how to solve similar kinds of problems. All the different ways and combinations to model these cases were not studied, but a few adequate ways were found. Long pipe runs with many supports and restraints should preferably be modelled with special-purpose elements in order to save time and numerical errors due to overly large models.

Some future study suggestions can be made. The rate dependency of materials should be included. Also different types of supports and especially more spring-like and softer supports should be studied instead of the very rigid one considered here. In case of pipe elements, the pipe break load should also be modelled with point loads that have certain predetermined amplitude. At least the mass of water should be included. The sudden rapid decrease of the inner pressure after the pipe break should be taken into account. That can be done fairly easy by assigning different pressure amplitudes to different sections of the pipe. To be more accurate, hydrodynamic effects of water in a more detailed manner should be taken into account. That would require computational fluid dynamics and even bidirectional fluid-structure interaction. More extensive pipeline geometry should also be studied. More about this study is discussed in a public VTT Research report (Calonius, 2009).

6. References

1. Abaqus Manuals, 2006. Version 6.6. Abaqus inc.
2. Vörös, G., Zsidi, Z., 2002. "Analysis of the effects of postulated pipe rapture". Gépészet 2002, Budapest. 307-310.
3. Micheli, I., Zanaboni, P., 2003. Transactions of the SMiRT Conference, 2003. Paper #J05-4. "An Analytical Validation of Simplified Methods for the Assessment of Pipe Whip characteristics".
4. Calonius, K., 2009. Numerical Studies on Dynamic Behaviour of Pipelines. VTT Research Report VTT-R-01025-09.

7. Acknowledgement

This study is part of SAFIR2010, which is a national nuclear energy research programme. The project work was funded by the Finnish State Nuclear Waste Management Fund (VYR) and the Technical Research Centre of Finland (VTT).