**20th International Conference on Structural Mechanics in Reactor Technology (SMiRT 20)**
**Espoo, Finland, August 9-14, 2009**
**SMiRT 20-Division 3, Paper 3141**

# Numerical Studies on Dynamic Behaviour of Pipelines

**Kim Calonius**[a]

[a]*Technical Research Centre of Finland, Espoo, Finland, e-mail: kim.calonius@vtt.fi*

**Keywords:** Numerical, modelling, dynamics, elbow, impact, nuclear power plant, pipes, pipelines, ovalization, pipe break, pipe whip, guillotine

## 1 ABSTRACT

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 in a nuclear power plant, different types of restraints and supports are designed for the pipelines.

Here, the usability of different types of elements provided by Abaqus (Abaqus, 2006), a commercial general-purpose finite element code, in modelling the dynamic behaviour of pipelines is tested. A relatively short pipe line section with one bend and one restraint is chosen as a test case. 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 pipe and/or elbow elements for the pipe and a spring element for the restraint.

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.

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.

## 2 INTRODUCTION

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.

This study 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 a simpler model using structural and special purpose elements. A pipe guillotine break is chosen as a dynamic test case that is simulated with different kinds of models. The main aim is not simulate the pipe break as realistic as possible, but to compare different model and analysis types with each other.

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 structural and special-purpose elements instead.

The pipes in power plants are long, slender structures with relatively thin walls. Therefore, it is convenient to model them with shell elements. Since the beam theory can be used to calculate their behaviour quite accurately, also pipe elements can be used. Beam, elbow, shell and solid volume elements are used in this study for modelling the pipe.

There are many types of supports and restraints in power plants. They 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. Solid volume and spring elements are used in this study for modelling the restraint.

## 3 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 (6 cm in size), 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. The lower right end of the pipe is rigidly fixed and the postulated pipe break takes place in the upper left end of the section.

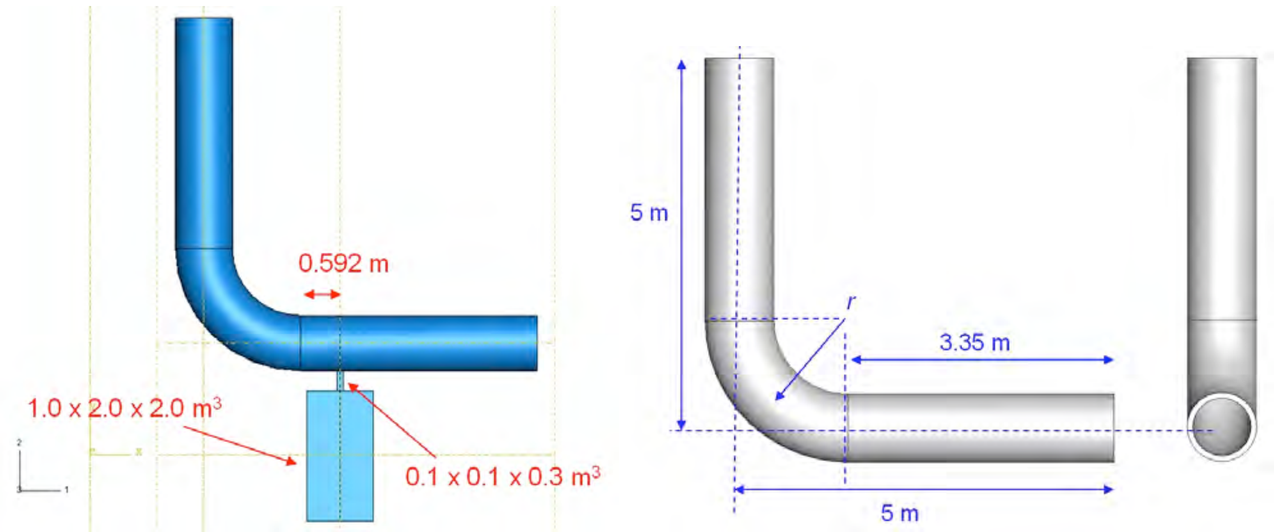


**Figure 1.** Dimensions of the pipeline section

The different finite element model types are shown in Figure 2. The concrete block and the restraint are modelled either with solid volume elements (referred as C3D8R in Abaqus) 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 6 864. The model with a coarse mesh has only 12 elements around the pipe (total number of elements 456).

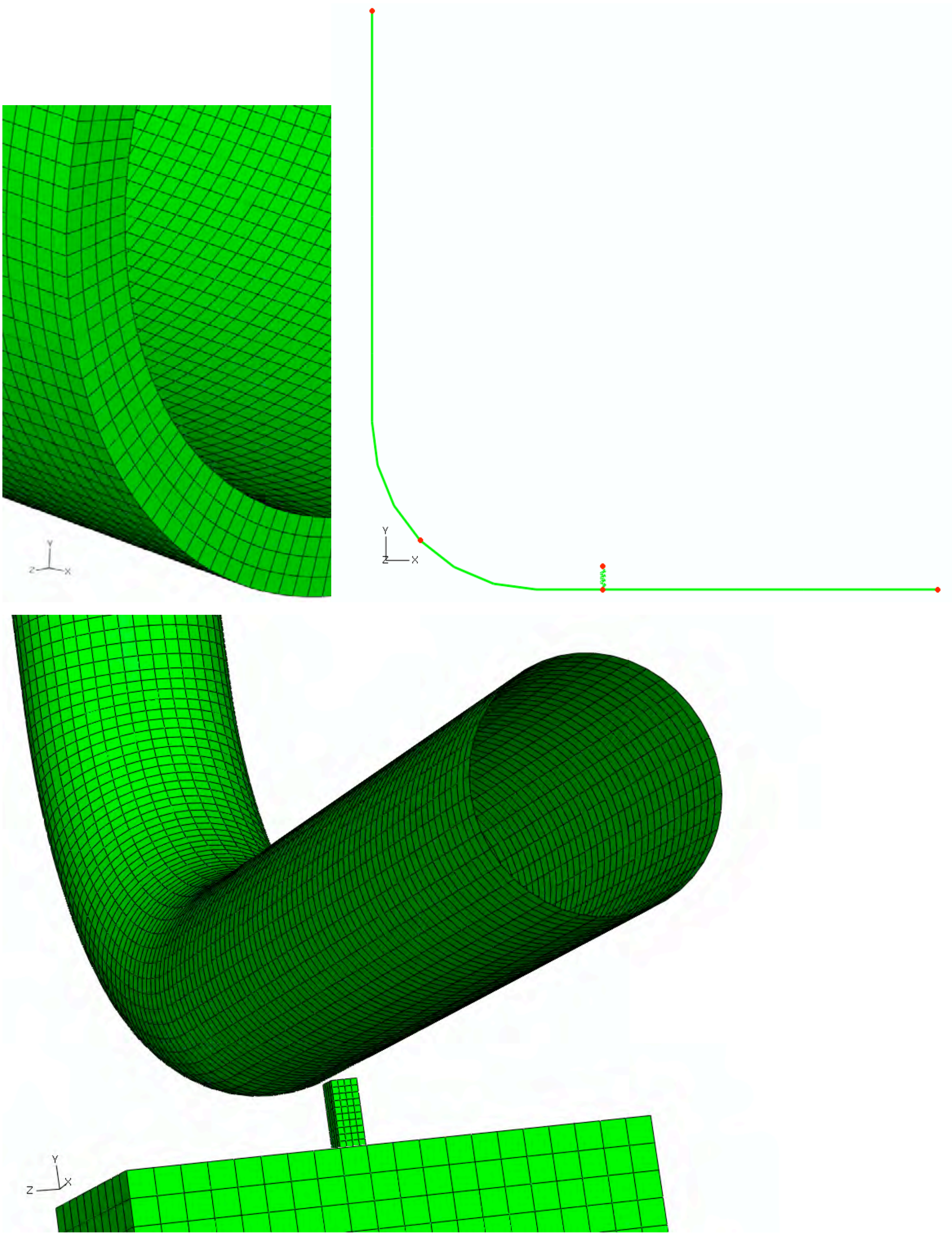

**Figure 2.** Solid volume (above left), pipe/elbow (above right) and shell element (below) models.

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 the nonlinear dynamic cases and in one of the many natural frequency extraction 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. Table 1 shows the material property values.

**Table 1.** Material properties.

| Material | Steel | Concrete |
|---|---|---|
| density, $\rho$ (kg/m$^3$) | 7850 | 2400 |
| Young's Modulus, $E$ (GPa) | 210 | 35 |
| Poisson Coefficient, $\nu$ | 0.3 | 0.2 |
| Yield point, $\sigma$ (MPa) | 210 | - |

## 4 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, but results of only seven of them are shown here. 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.

**Table 2.** Natural frequencies of different models.

| | C3D8R | S4R fine mesh | S4R fine mesh rotation free | S4R fine mesh pressure | S4R 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 |

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).

The deformed shapes of the 7th mode of different model types are shown in Figure 3 (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 nonlinear dynamic analyses: the model with a fine shell element mesh (the 2nd column) and the model with 18 linear elbow elements (the 7th column).

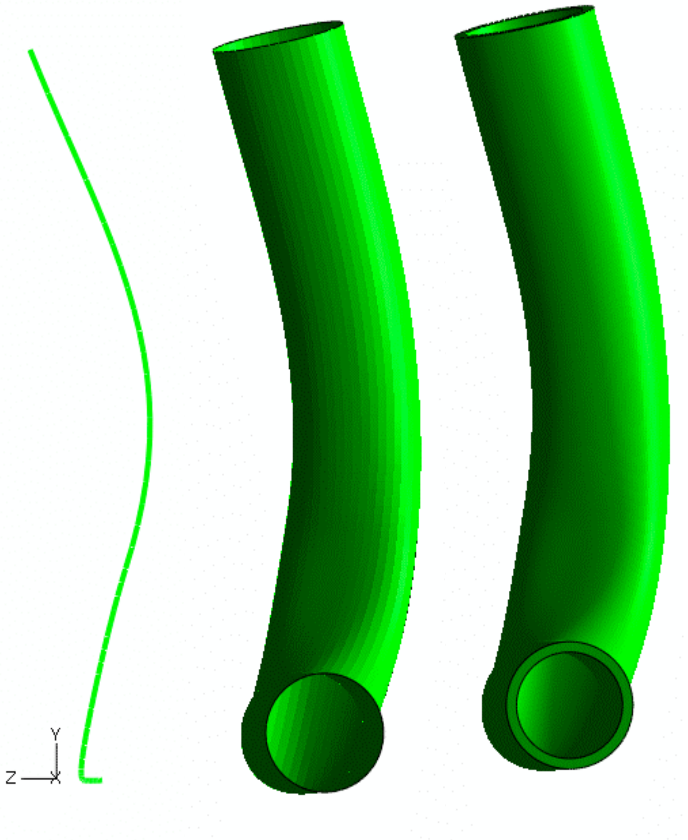


**Figure 3.** Mode 7 of different type of models.

## 5 DYNAMIC ANALYSIS OF A GUILLOTINE PIPE BREAK

A guillotine pipe break is chosen as a nonlinear 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 structural and special-purpose 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 can 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:

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

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 analyses in this study consist 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 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 instantaneously. 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. Thus, the analyses are conservative in many ways. Table 3 lists all the six nonlinear dynamic analysis cases. Element, material and analysis (time integration) types are varied.

**Table 3.** Dynamic analysis cases.

| Case code | Main element type | Main material type | Analysis type | Analysis time (dynamic step) | CPU time |
|---|---|---|---|---|---|
| SEI | Shell | Elastic | implicit | 1 s | 48 418 s |
| SEE | Shell | Elastic | explicit | 1 s | 25 491 s |
| SPI | Shell | Plastic | implicit | 0.30 s | 75 148 s |
| SPE | Shell | Plastic | explicit | 0.26 s | 22 887 s |
| EEI | Elbow | Elastic | implicit | 2 s | 565 s |
| EPI | Elbow | Plastic | implicit | 0.22 s | 274 s |

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. The 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.

In the elastic cases, the pipe vibrates dominantly according to its second mode and hits repeatedly the restraint near the highest point of its natural path (that would be unobstructed). Since the restraint or the pipe do not give in, there are only small displacements and the models are in good correlation with each other. Figure 4 shows the vertical reaction forces in the fixed end and in the restraint in cases SEI and EEI (elastic shell and elbow element models with implicit direct time integration). They correspond with each other very well.

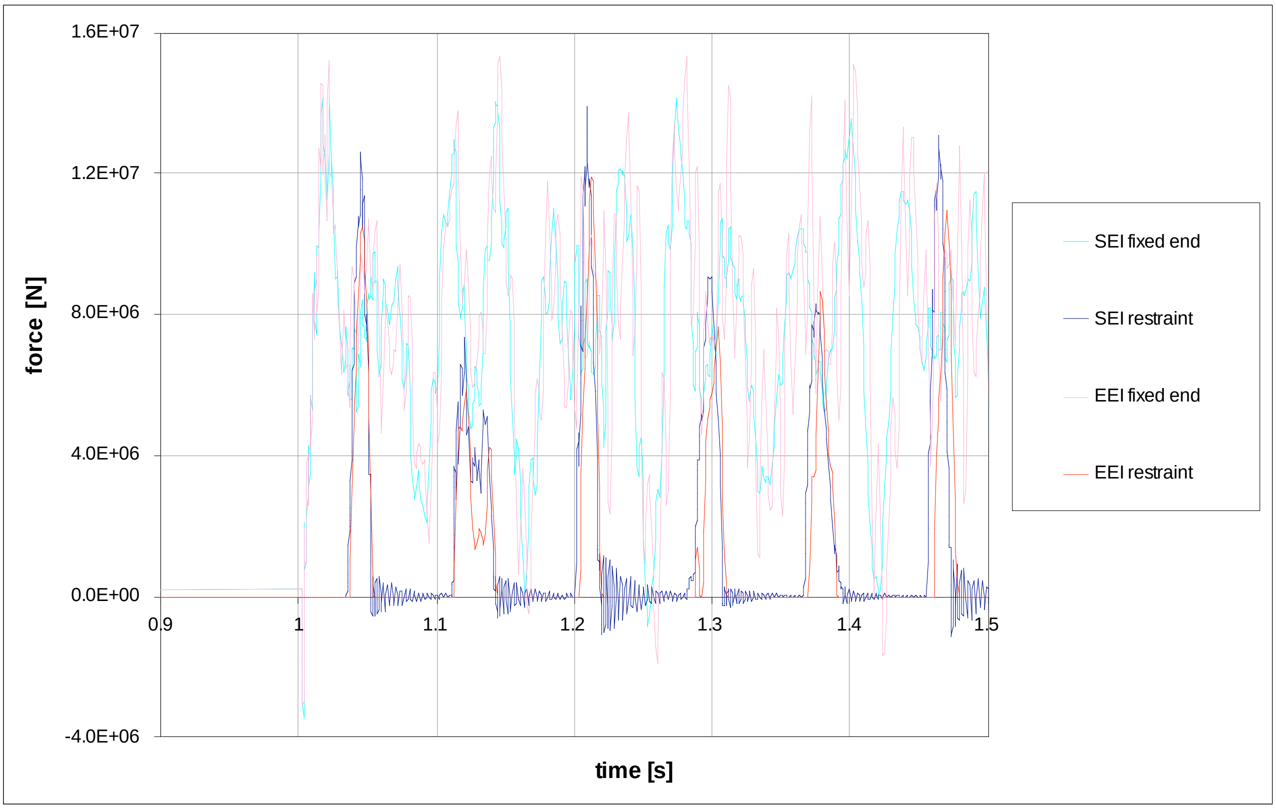


**Figure 4.** Vertical reaction forces in the fixed end and in the restraint in the elastic cases SEI and EEI.

In plastic cases, the stress wave reaches the fixed end 2 ms and the pipe hits the restraint 17 ms after the pipe break. In case SPI, the steel restraint bends and gets very high plastic deformations. The pipe hits the edge of the concrete block 48 ms after the pipe break. Figure 5 shows the deformed shapes in cases SPI and EPI 0.06 s after the pipe break. The von Mises stress distribution on the outer surface is also shown. In case EPI, the stresses are plotted from the side closest to the viewer. The plastic hinge has been formed first in the fixed end and then near the restraint in both cases.

The results with explicit time integration corresponded well to the ones with implicit integration.

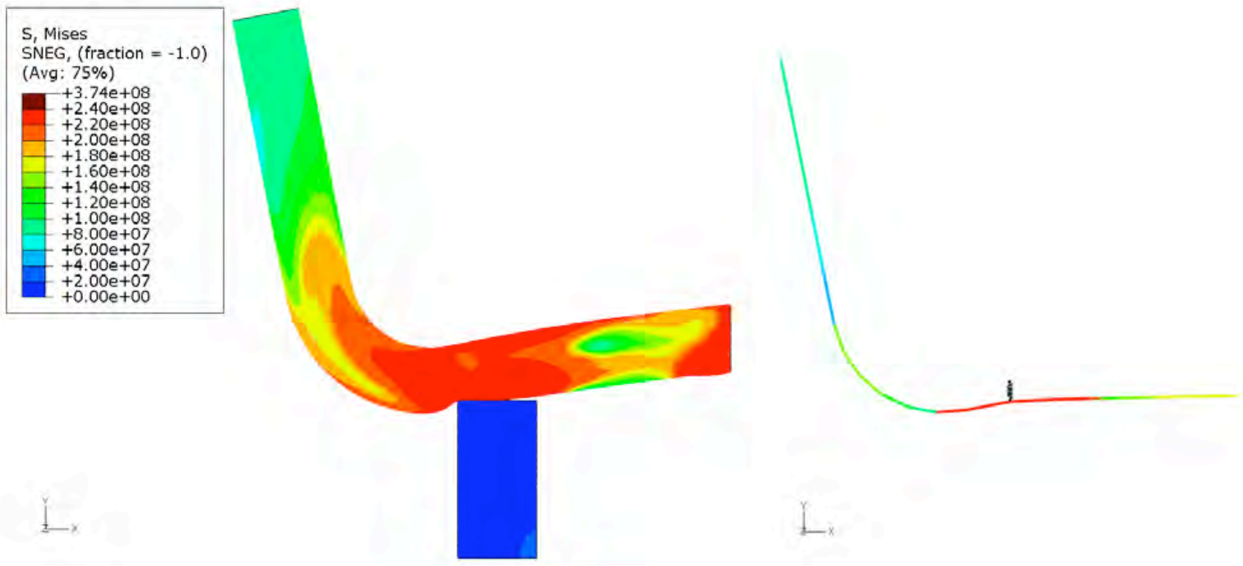


**Figure 5.** Deformed shapes with von Mises stress distributions on the outer surface 0.06 seconds after the pipe break in the plastic cases SPI (on the left) and EPI (on the right).

Figure 6 shows the cross sections viewed horizontally along the axis of the pipe. In case EPI, the values are from the middle of the elbow element lengthwise and from 20 points around the circumference of the pipe wall. Thus, it is impossible to get directly the deformed shape exactly at the location of the spring element. Also, the spring element cannot be fixed to any of those points around the pipe cross-section but only to an imaginary middle point of the cross-section. That is one of the reasons the deformation of the cross-section cannot be modelled very accurately with elbow elements near these kinds of restraints. In spite of that, the behaviour of the cross-section corresponds well to the one of the shell element model.

In Figures 7 and 8, the x-axis represents the radial angle in which 0 degrees is the bottom of the pipe and the extrados in the bend, 180 degrees is the top of the pipe and the intrados in the bend etc. The y-axis represents the stress value and the range is from 300 MPa compression to 300 MPa tension in each figure. Remember, that the yield stress is 210 MPa. Figure 7 shows the axial stress as a function of circumferential position on outer surface in the bend at 0 and 0.14 seconds after the pipe break in cases SPI and EPI. Figure 8 shows the axial stress as a function of circumferential position on outer surface 0.3 m right from the restraint at 0 and 0.14 seconds after the pipe break in cases SPI and EPI.

There are some clear differences in the stress distributions of shell and elbow element models, but in general level the elbow element describes reasonably well the stress distribution. After the static step, the stress distributions of shell and elbow element models are in very good agreement.

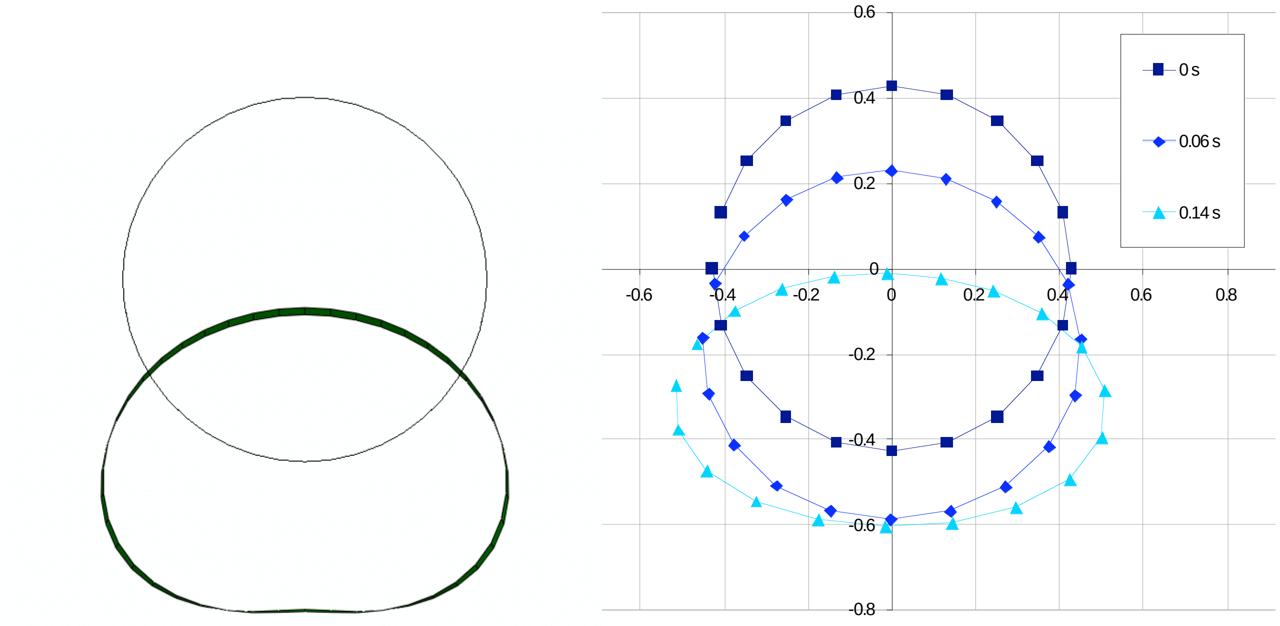


**Figure 6.** On the left: An undeformed cross-section and a deformed cross-section at 0.06 seconds after the pipe break in case SPI. On the right: Cross-sections at time instants 0, 0.06 and 0.14 seconds after the pipe break in case EPI.

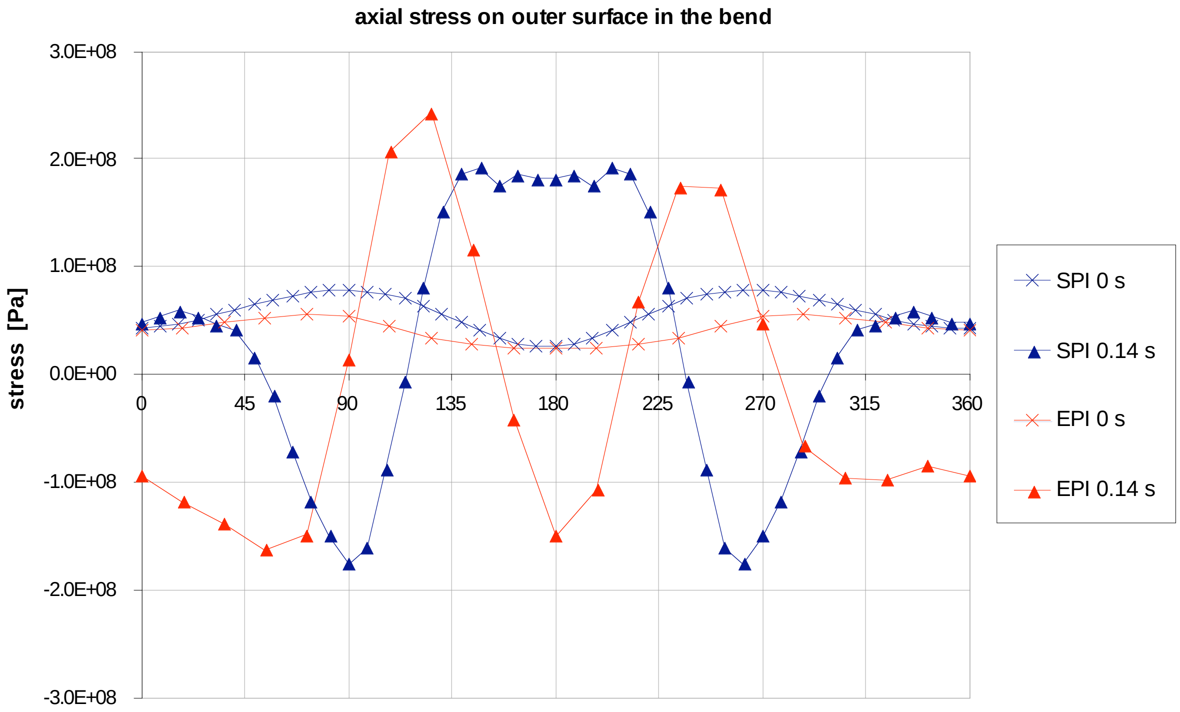


**Figure 7.** Axial stress as a function of circumferential position on outer surface in the bend at 0 and 0.14 seconds after the pipe break in cases SPI and EPI.

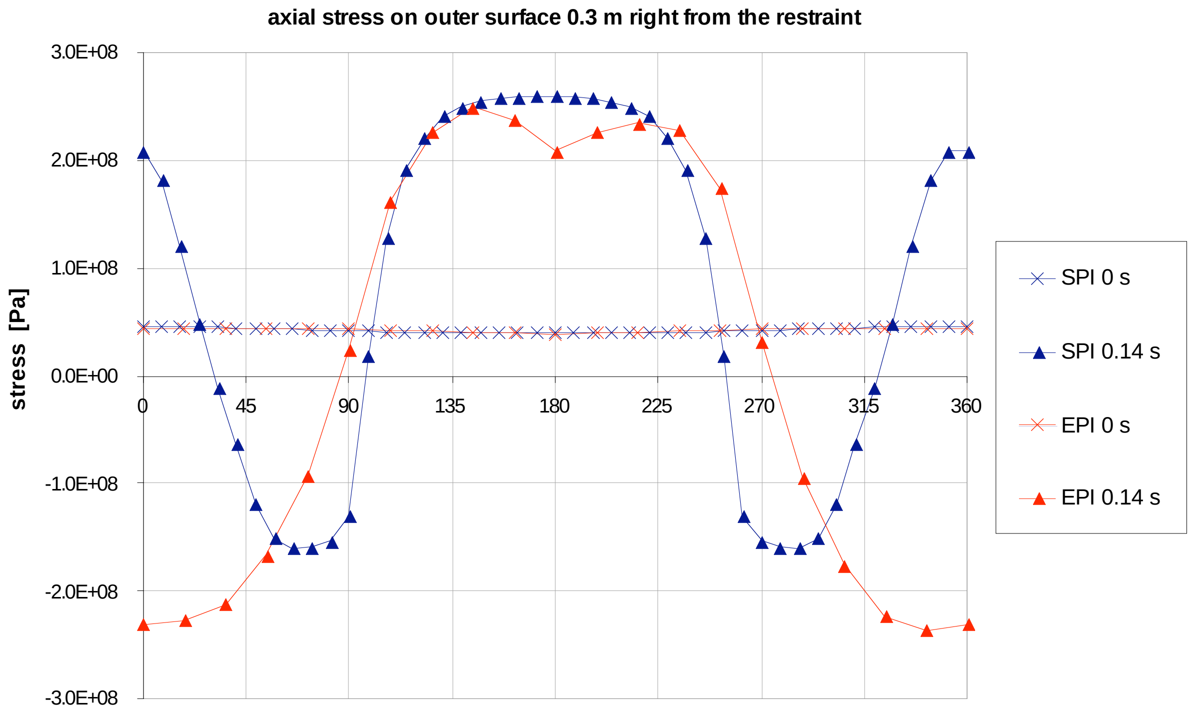


**Figure 8.** Axial stress as a function of circumferential position on outer surface 0.3 m right from the restraint at 0 and 0.14 seconds after the pipe break in cases SPI and EPI.

# 6 CONCLUSION

Different kinds of ways to numerically model with finite element code Abaqus the pipe runs and their supports are studied. A pipe guillotine break with whipping is chosen as a dynamic analysis case. Simple and typical nuclear power plant 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 or the mass of water 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 pipe or elbow and special-purpose elements. This is crucial when the restraints have to be simplified in extensive models with long pipe runs and many supports. The chosen restraint type is relatively stiff when having elastic material properties.

Dynamic analyses of a pipe break are conducted with different models that are confirmed as adequate. A shell element model with a relatively dense mesh and an elbow element model are chosen to be compared. Both linearly elastic and nearly ideal plastic material properties are assigned to the steel of pipe and restraint. 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. In general level, the models behave in a similar manner. The plastic hinges develop at the same time and the pipe deforms globally in the same way.

Nonlinear spring elements are found to act realistically and seem very suitable to model restraints for this kind of dynamic problem. The displacement and cross-section deformation results of the elbow element model corresponded reasonably well to the ones of the much larger shell element model. The situation is naturally different on a detail level near the restraint. There are some local discrepancies in the stress results.

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. First of all, it should be remembered that comparisons were made between several FE models instead of an actual structure and FE models. 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 simple structural elements such as pipe and elbow elements and special-purpose elements in order to save time and numerical errors due to overly large models. It is shown, how the stress distribution along the pipe axis, around the cross-section and through the thickness of the wall can be solved with elbow elements on a fairly detailed level.

A suggestion is to use for normal global vibration analyses approximately six ELBOW31 elements for the bends and PIPE31 elements of the same length for the straight parts. The same type of model with additional nonlinear spring elements is also adequate for simulations of pipe breaks.

Some future study suggestions can be made. The rate dependency of materials should be included. The plastic properties should be defined more carefully and they should correspond better the real nuclear power plant materials. Also, different types of supports and especially more spring-like and softer supports should be studied instead of the very rigid one (when elastic) considered here. The support structure was too complicated to be modelled with only a single spring element, if the nonlinear dynamic behaviour and the effects of the surrounding civil structures are to be taken into account. The spring elements are adequate when a longer section of a pipe line is examined and no extensive displacements and plastic hinges are anticipated. Especially the “tube support elements” are worth studying more in the future. In case of pipe or elbow elements, the pipe break load should also be modelled with point loads that have certain predetermined amplitude. A denser mesh would give more detailed local stress results.

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 even 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. To get more general conclusions, more extensive pipeline geometry with different types of supports and restraints should also be studied (Calonius, 2009).

***Acknowledgements.*** *This study has been conducted under SAFIR2010, which is a Finnish public research programme on nuclear power plant safety. The work was funded by the State Nuclear Waste Management Fund (VYR) and the Technical Research Centre of Finland (VTT).*

## REFERENCES

Calonius, K., 2009. Numerical Studies on Dynamic Behaviour of Pipelines. VTT Research Report VTT-R-01025-09.

Vörös, G., Zsidi, Z., 2002. "Analysis of the effects of postulated pipe rapture". Gépészet 2002, Budapest. 307-310.

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".

Abaqus Manuals, 2006. Version 6.6. Abaqus inc.