Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 213 (2018) 762–775

Procedia Engineering

www.elsevier.com/locate/procedia

7th International Conference on Fatigue Design, Fatigue Design 2017, 29-30 November 2017, Senlis, France

Innovative numerical fatigue methodology for piping systems: qualifying Acoustic Induced Vibration in the Oil&Gas industry

Anne Coulon[a], Rémi Salanon[a], Loïc Ancian[a]

[a]Vibratec,28 chemin du petit bois, Ecully Cedex BP36-69131, France

Abstract

Acoustic Induced Vibration (AIV) refers to the high acoustic energy generated by pressure-reducing devices that excite pipe shell vibration modes, producing excessive dynamic stress. Analysis of this risk is an important part of Asset Integrity Management systems as AIV can cause catastrophic piping failure. Existing guidelines address this risk through an analytical assessment. However, these methodologies are not fully known and input parameters are limited. Some limits to the guidelines are pointed out with recommendations to improve them.

The approach presented for identifying AIV damage is based on a dynamic stress evaluation at pipe discontinuities (welded connections and supports). This evaluation is performed through a fluid-structure coupling Finite Element Analysis. Pressure fluctuations inside the pipe are predicted and coupled with a pipe structural analysis. This methodology is provided with its validation through measurement on an actual AIV field case, corresponding to a crack initiation due to AIV on an FPSO flare network tail pipe.

To conclude the paper, the method is then applied to quantitatively assess the mitigation actions' efficiency on an actual case. Different solutions have been individually tested to end up with a final solution that reduces the damage to acceptable levels in the most cost-effective manner.

Peer-review under responsibility of the scientific committee of the 7th International Conference on Fatigue Design.

Keywords: AIV ; Acoustic Induced Vibrations ; Oil&Gas Piping ; Piping fatigue ; Acoustic Fatigue

1877-7058
Peer-review under responsibility of the scientific committee of the 7th International Conference on Fatigue Design.
10.1016/j.proeng.2018.02.072

1. Introduction

Acoustic Induced Vibration (AIV) in piping systems refers to high acoustic energy that excite pipe shell vibration modes, producing excessive dynamic stress that could lead to pipe failure.

The source of this high acoustic energy is a pressure-reducing device (valves, restricted orifices…) with a high pressure drop and important mass flow rate. In such devices, the amount of energy dissipated is quite high and even if most of the energy is converted to heat, a significant part is converted to sound or pressure waves that will excite the pipe wall. This broadband and high-frequency excitation propagates through the pipe, amplified by transverse acoustic pipe modes, and comes to excite the pipe's shell vibration mode. While running along straight pipes, the impact of vibration is limited due to axisymmetry of the pipe shell mode shape. However, when the excitation comes to a non-axisymmetrical discontinuity (branch, small bore, support…), vibrations are amplified, leading to high dynamic stress that can cause fatigue failure of the pipe. As these vibrations occur at high frequencies, i.e. with a high fatigue cycle rate, fatigue failure occurs within a few minutes to a few hours.

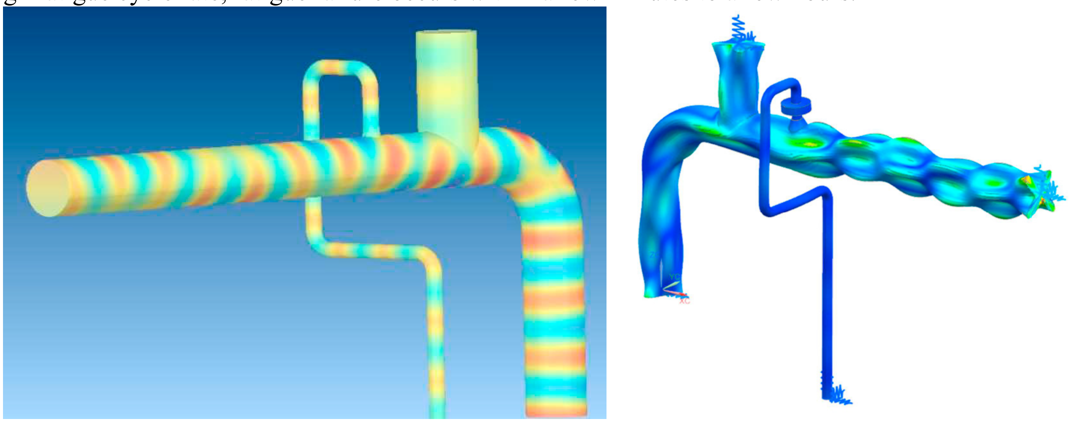


Fig. 1. (a) Fluid Acoustic mode; (b) Pipe shell mode.

The major risk associated with this phenomenon for offshore plants is related to flare systems. Blowdown valves, restricted orifices and pressure safety valves encountered in these systems are usually with large pressure drops and important mass flow rates. As the acoustic energy generated by these devices propagates downstream with small attenuation, the whole flare network is impacted by the risk of AIV failure. As flare systems are gas associated systems and safety-related, pipe failure would have catastrophic consequences. Therefore, assessing and controlling the AIV risk is an essential part of Asset Integrity Management.

AIV has been an on-going research subject since initial publications in the late 70s and methodologies have been developed to help engineers assess this risk. The dominant methodology for the Oil & Gas industry is that published by the Energy Institute. In its guidelines, the Energy Institute addresses this risk through an analytical assessment methodology. This tool is very efficient in performing a quick screening of large numbers of pipes. However, when it comes to mitigation measures, the limited number of input parameters used to quantify the Likelihood of Failure (LOF) reduces the range of possible mitigation measures, since the efficiency of certain mitigation measures are not LOF calculation parameters and therefore cannot be assessed. The LOF is calculated with the following formula:

$$LOF = -0.1303\ln(N) + 3.1 \text{, N representing the number of cycles before failure} \quad (1)$$

To overcome this limitation, a new detailed Finite Element methodology has been developed using coupling between fluid and structure, making it possible to predict dynamic stress for complex piping models. This methodology is introduced in the next chapter and includes validation through measurements on an actual AIV field case.

Using this methodology, different AIV mitigation actions (such as: use of sweepolets, forged tees, full encirclement supports, full encirclement wrap branch reinforcements), not included in Energy Institute guidelines scopes, are assessed. Comparison between computation results with and without mitigation measures makes it possible to quantify the impact of such modifications accurately and to establish a Likelihood of Failure adjustment coefficient when using these modifications.

2. Finite Element Analysis Methodology

Some methodologies using Finite Element Analysis to predict dynamic stress in pipes under AIV excitation already exist in the literature. However, one of main limitations to these methodologies is that only the structure (the pipe) is considered in the model, but not the fluid. Therefore, to overcome this, assumptions have to be made on the pressure distribution along the pipe wall. The usual assumption is to consider the acoustic pressure wave as a plane wave (pressure equally distributed along the wave) or to consider that the pressure distribution is identical to the pipe shell mode shape (very conservative as all the shell modes would be driven by the acoustic excitation).

This new approach still use an FE analysis, but with a fluid / structure coupling using ACTRAN FFT software. As the fluid considered during AIV is a gas (light fluid), a one-way coupling approach can be used (Pressure fluctuations inside the fluid generate pipe wall vibrations but not the other way around). Therefore, two finite element models are made: one for the fluid, one for the pipe.

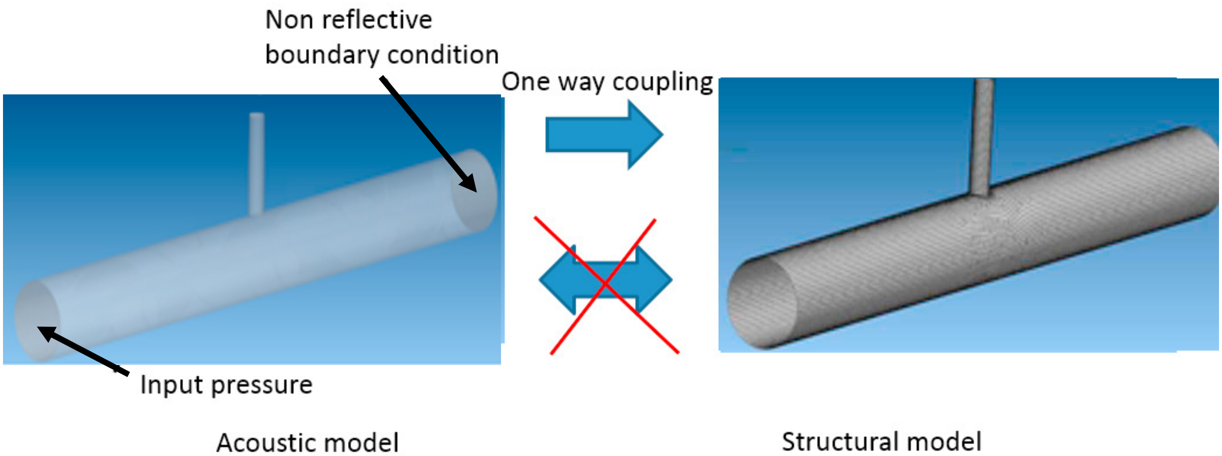


Fig. 2. Finite Element Methodology

Estimated acoustic energy generated by the pressure-reducing device is input at the entrance of the pipe and acoustic wave propagation is then computed. Calculation output is the pressure field inside the pipe at different frequencies. This estimated pressure field is then applied to the structural model (pipe wall), to predict dynamic stress at discontinuities.

The use of this methodology is illustrated through a case study based on field measurements.

3. Field Case

3.1. Case Background

The selected case is a vibration diagnosis performed by Vibratec on an FPSO for flare network pipe failure. During production flaring, after a short period of time, cracks appeared on two supports attached to a 16” pipe downstream pressure control valve. Fig. 3. Pipe arrangement and crack location
shows the pipe arrangement and the crack location for one of the supports.

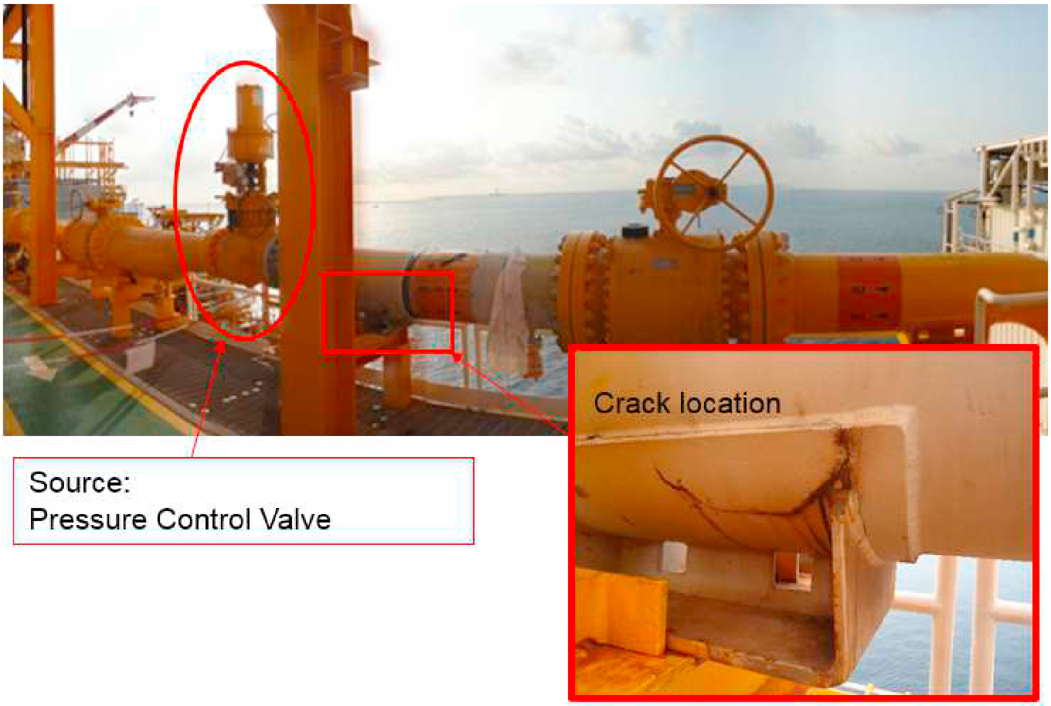


Fig. 3. Pipe arrangement and crack location

Measurement and investigations quickly showed that this case was AIV related. The Pressure Control Valve upstream the pipe had a high pressure drop with an important mass flow rate flowing through the pipe (LOF > 1). Vibration measurements all around the pipe section at the crack location showed that while the valve was operating, most of the vibration level (leading to fatigue failure) were caused by pipe shell vibration mode being excited. AIV is therefore the reason for this pipe failure.

Since this event occurred on a gas flaring line, it represents a major safety incident, although no gas was released in the atmosphere. Had the failure occurred on another pipe discontinuity (small bore connection for instance) rather than a support, consequences could have been catastrophic.

Since the vibration measurements were performed in known process conditions, this case was a good candidate to validate the methodology. The following chapters describe how the Finite Element analysis presented earlier was used to attempt to reproduce the measurement results and then validate the method.

3.2. Acoustic energy input to the model

Pipe vibration is calculated through the structural FE model with the pressure field from the acoustic FE model. The acoustic FE model requires a Sound Power Level (SWL) input at the pipe entrance. This SWL comes from the valve upstream. Based on Carrucci and Muller work [Carruci, 1982], acoustic power dissipated by the Control Pressure Valve can be estimated using this formula:

$$PWL = 10\log\left[W^2 \cdot \left(\frac{\Delta p}{P_1}\right)^{3.6} \cdot \left(\frac{T}{M \cdot W}\right)^{1.2}\right] + 126 \tag{2}$$

Where

W = Mass flow rate (kg/s)
Δp = Valve pressure drop (Pa abs)
P_1 = Valve upstream pressure (Pa abs)
T = Valve upstream Temperature (K)
MW = Molecular Weight (g/mol)

Based on process parameters for this line, the PWL was estimated at 168 dB during measurement condition.

Moreover, one of the main characteristics of this acoustic energy is that it is a broadband excitation, therefore the Sound Power was input as a Power Spectral Density (PSD). Excitation shape was tuned on measurements.

3.3. Acoustic FE model

A finite element model of the fluid (gas) inside the pipe was developed (Fig. 4). The mesh size was selected as 15 mm. This size allows accurate wave propagation calculations up to 3000 Hz (more than 8 wave lengths per element at this frequency). AIV being a high frequency phenomena, the pressure field was computed between 150 Hz and 3000 Hz.

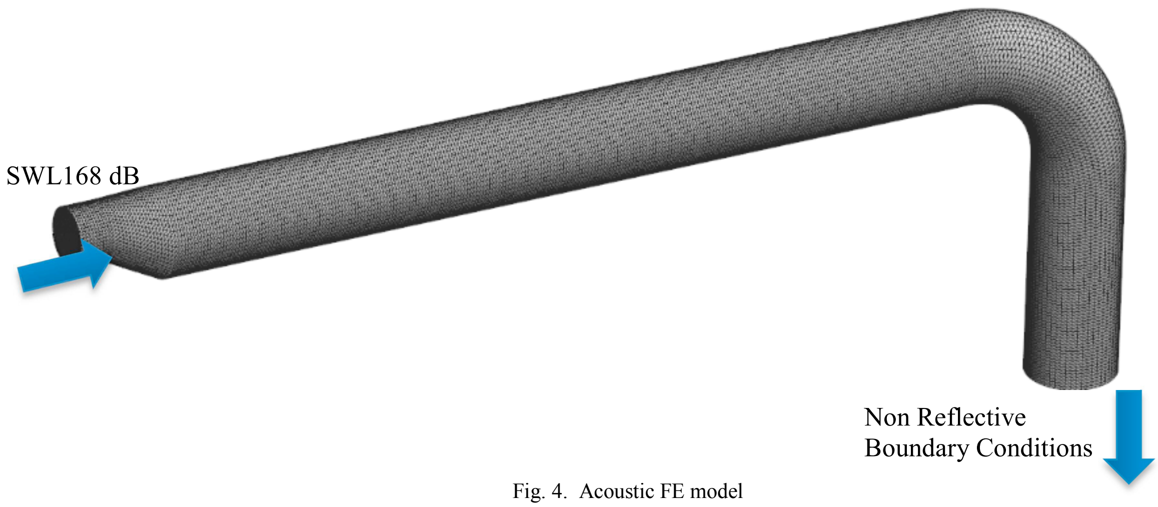


Fig. 4. Acoustic FE model

The acoustic pressure field obtained after computation is coherent with the theory: before the threshold frequency, only plane wave acoustic modes are present; after the threshold, transverse acoustic modes dominate (Fig. 5 and Fig. 6). The threshold frequency is given as follows:

$$f_{threshold} = \frac{0.5861 \cdot c}{D} \tag{3}$$

Where c is the velocity of sound (m/s) and D the pipe internal diameter (m). For this case, the threshold frequency was evaluated at 660 Hz.

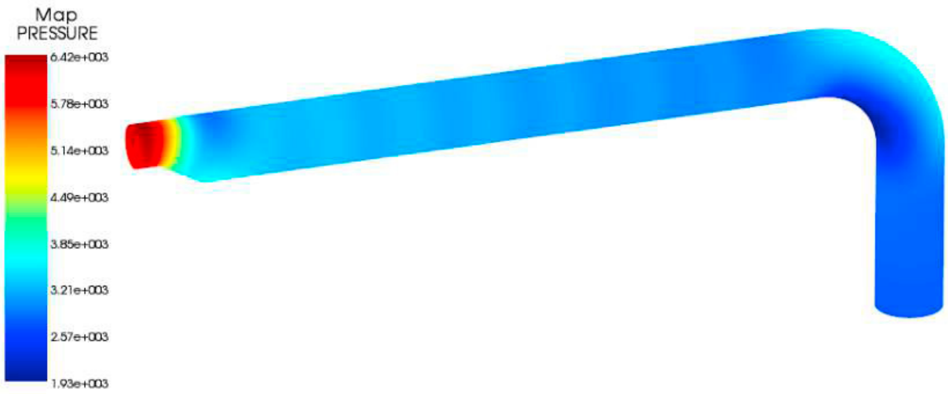


Fig. 5. Acoustic pressure field (Pa) in the pipe at 470 Hz – below the threshold frequency (plane wave acoustic mode)

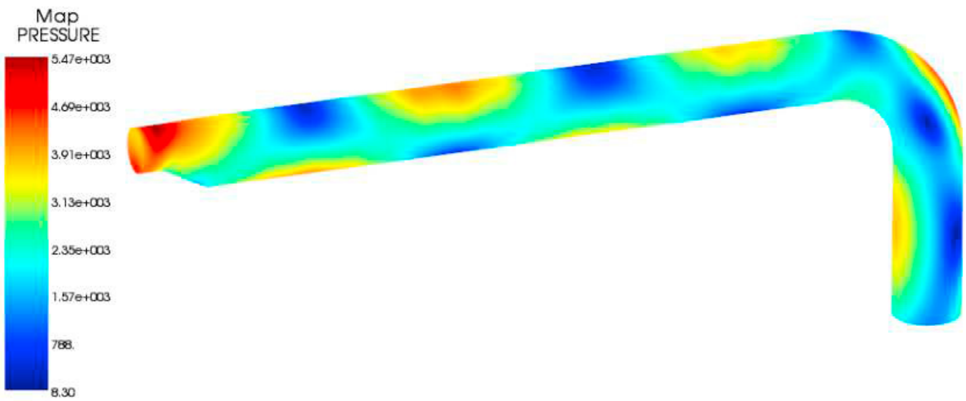


Fig. 6. Acoustic pressure field (Pa) in the pipe at 980 Hz – above the threshold frequency (transverse acoustic mode)

3.4. Structural FE model

Using the previously computed acoustic field with the acoustic FE model, the aim was to predict the vibration levels along the pipe wall and to compare them with measured ones. For this purpose, a structural FE model was developed and is shown in Fig. 7. Pipe, supports and flanges were modeled as shell elements. The large valve was modelled as rigid with mass, assuming that its thickness compared to the pipe thickness would not influence results.

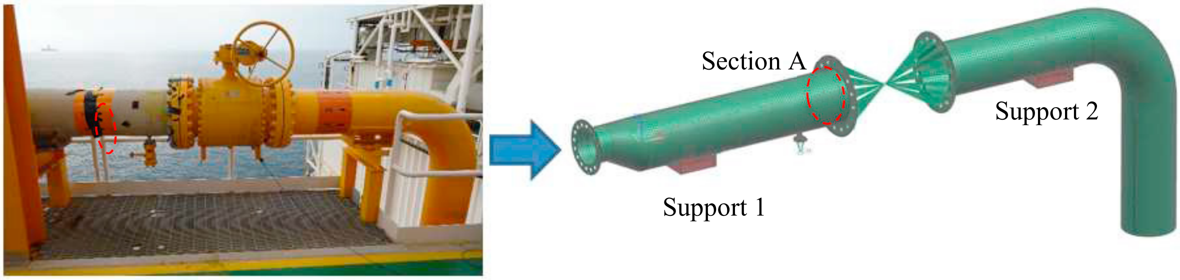


Fig. 7. Structural Pipe FE model

The damping ratio used for the model was chosen based on field measurements that show that for the pipe vibration shell mode, the damping ratio ξ is between 0.5 % and 1.5 %. A value of ξ = 1 was chosen for the model.

The model was first used to extract all the structure's natural frequencies below 3000 Hz. In the AIV frequency range of interest (150 Hz to 3000 Hz here), the modal response is mainly driven by pipe shell modes as illustrated in Fig. 8.

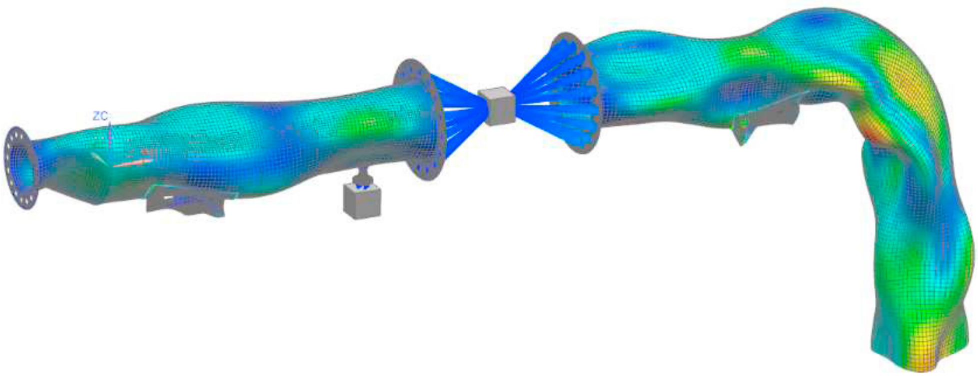


Fig. 8. Pipe deformed shape mode at 770 Hz

Once the modal basis was extracted, pipe shell modes were associated with the acoustic pressure field calculated previously in a response calculation to predict vibration and stress levels on the pipe wall and at discontinuities.

Measurements available for validation are vibration measurements. Vibration probes were placed on several sections along the pipe (Fig. 9).

Fig. 9. Vibration probe mounting

Vibration levels were measured at the section where the crack initiated, however the crack along the support saddle was not modelled in the FE model, therefore it would not be representative to compare vibration levels at this location. Therefore a section slightly further was chosen for comparison (section A on Fig. 7). Vibration levels measured in this area present a good correlation with the simulation. They are not presented in this paper, only the fatigue analysis is compared between measurement and simulation.

Stress levels were computed and compared with the fatigue endurance limit from BS7608 for two location:

- At the first support saddle welding line (crack initiation location) : Support 1 (ref. Fig. 7)
- At the next support saddle (no crack initiation) : Support 2 (ref. Fig. 7)

Maximum principal stress was extracted from the FE element model and plotted over frequency in Fig. 10.

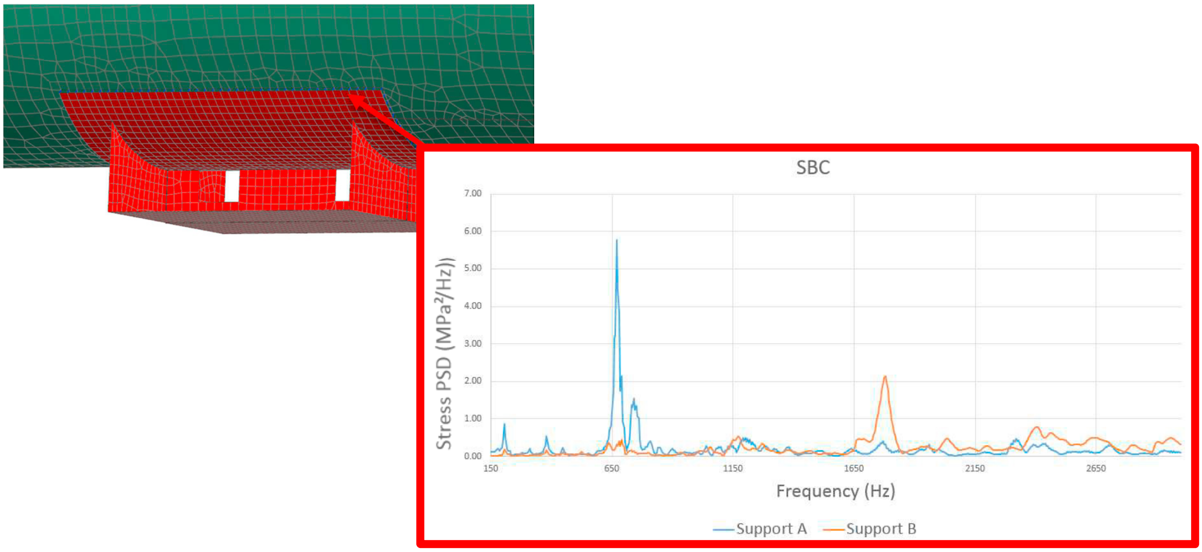


Fig. 10. Normal stress PSD at crack initiation location for support 1(crack initiation, blue) and support 2 (orange)

Crack occurred on support A and no crack initiation was reported on support B. Numerical stress evaluation in Fig. 11 is in accordance with what occurred on the field, maximum stress level at support B is much lesser than maximum stress level at support A.

The global RMS value is calculated as 11.5 MPa, corresponding to a maximal peak-peak value of 68.9 MPa, more than the fatigue endurance limit of 35 MPa at 10^7 cycles from curve F2 of BS7608. Numerical estimation from the methodology are in line with fatigue failure prediction. This, combined with a good comparison between vibration measurements and calculations, gives confidence in this method.

4. Improvements to the existing screening methodology

Although Finite Element analysis provides detailed and precise results for assessing AIV risk at pipe discontinuities, this method cannot be widely used during offshore plant design. During this phase, a huge number of valves (and associated downstream pipes) needs to be assessed for AIV risk and a Finite Element approach is not feasible for economic reasons. To overcome this, an analytical methodology has been developed to estimate the AIV risk using few parameters. As mentioned previously, the Energy Institute guidelines, widely used currently, are very suitable for the design phase to quickly assess a large number of valves / lines by calculating a Likelihood of Failure (LOF) number between 0 and 1 representing the risk of AIV failure for each discontinuity. However, the guidelines are not able to quantitatively assess mitigation actions since the LOF calculations do not use the same parameters as those impacted by the mitigation.

In order to overcome this limitation, the Finite Element analysis described previously was used to quantitatively assess the impact of some of the "mainstream" AIV mitigation actions on LOF calculation.

Not all main pipe / branch configurations could be represented in a short paper; two representative cases that can be encountered during AIV analysis are presented.

4.1. Small Bore connection (SBC) on Valve Tailpipe

Many SBC serving different purposes (e.g. pipe purging) can be found along valve tailpipes. The case evaluated below is a 16" diameter SCH 40s tailpipe with a 1" SCH 40s diameter SBC with a purging valve and blind flange attached. The following configurations were studied:

- Use of Flangeolet (basic configuration),
- Use of Sweepolet (1st mitigation action),
- Use of Flangeolet with a Full Wrap-Around Pad (2nd mitigation action).

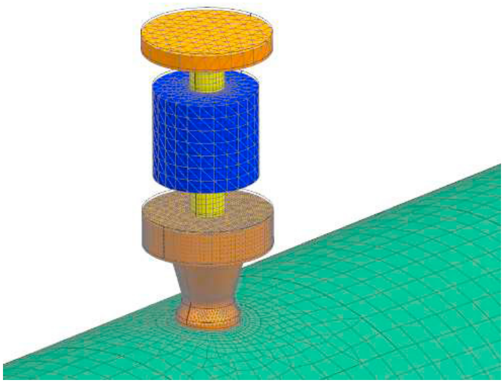
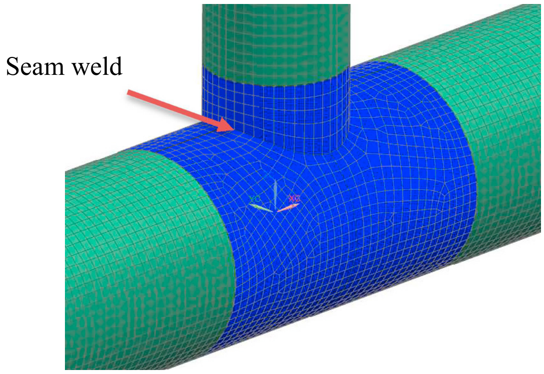
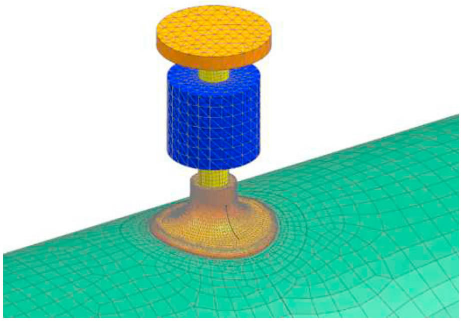
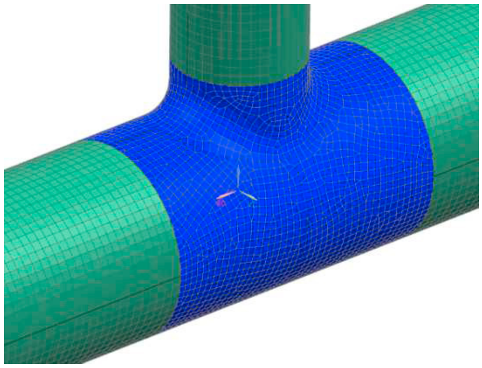
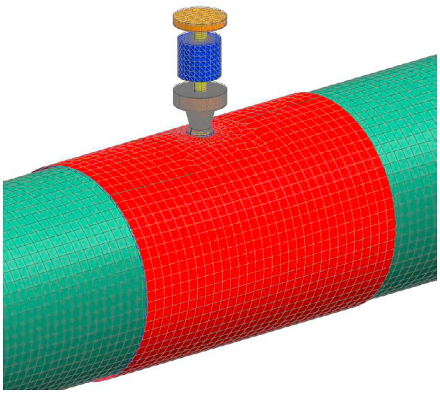
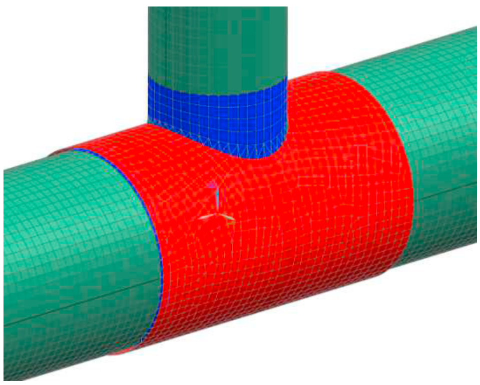
4.2. Valve Tailpipe 90° Tee connection on Flare Header

Another source of discontinuities when studying AIV is when the valve tailpipe is connected to another pipe or the flare header. The case evaluated below is a 12" diameter SCH 40s flare header with an 8" SCH 40s diameter tailpipe, 90° Tee connection. The following configurations were studied:

- Use of unreinforced fabricated tee (seam weld, basic configuration),
- Use of wrought butt-welding tee (1st mitigation action),
- Use of fabricated tee with Full Wrap-Around Pad (2nd mitigation action).

FE models are shown in table 1.

Table 1. FE models

Small bore connection	90° Tee
Basic configuration	
	
Use of flangeolet	Use of unreinforced fabricated tee
1st mitigation action	
	
Use of Flangeolet with Full Wrap-Around Pad	Use of fabricated tee with Full Wrap-Around Pad
2nd mitigation action	
	
Use of Flangeolet with Full Wrap-Around Pad	Use of fabricated tee with Full Wrap-Around Pad
N.B.: The Full wrap around pad is added on top of the main pipe as an extra shell and welded at each side of the main pipe and the connection bore.	

4.3. Weld modeling

The weld modeling is presented on a standard case, this modeling is then adapted on each cases studied here. The weld studied are full penetration weld.
Pipe and weld joint are modeled by shell element at their mid-surface. Both pipe are linked at their extremities at coincident nodes. The weld is modeled by shell element considering the weld joint thickness.

The hot spot stress is used for the fatigue analysis. The hot spot stress is determined by linear extrapolation at specified reference point at 0,5t and 1,5t (t = plate thickness) according to equation (4) from [5].

$$\sigma_{hs} = 1{,}5 \cdot \sigma_{0.5t} - 0{,}5 \cdot \sigma_{1.5t} \tag{4}$$

The element lengths adjacent to the weld joint are determined by the reference points. The next two element rings from the weld toe are modeled respectively with a length of 1t and the normal stress is evaluated at their center, see Fig.11.

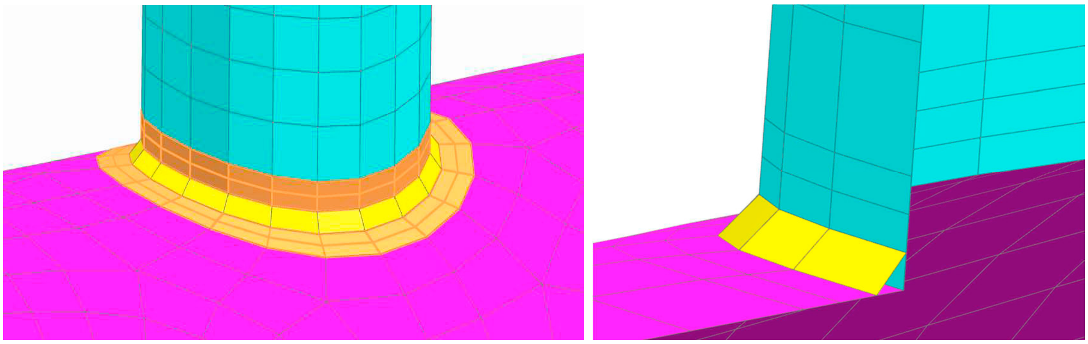


Fig. 11. Weld modeling on a standard case

4.4. Results

In order to quantify the risk reduction from each mitigation action, the initial design case was compared to use of each mitigation method. For each FE model, the same nominal sound pressure level was input using the studied methodology and the maximum hot spot stress was computed for each configuration. The resulting stress reduction ratios are given in Table 2.

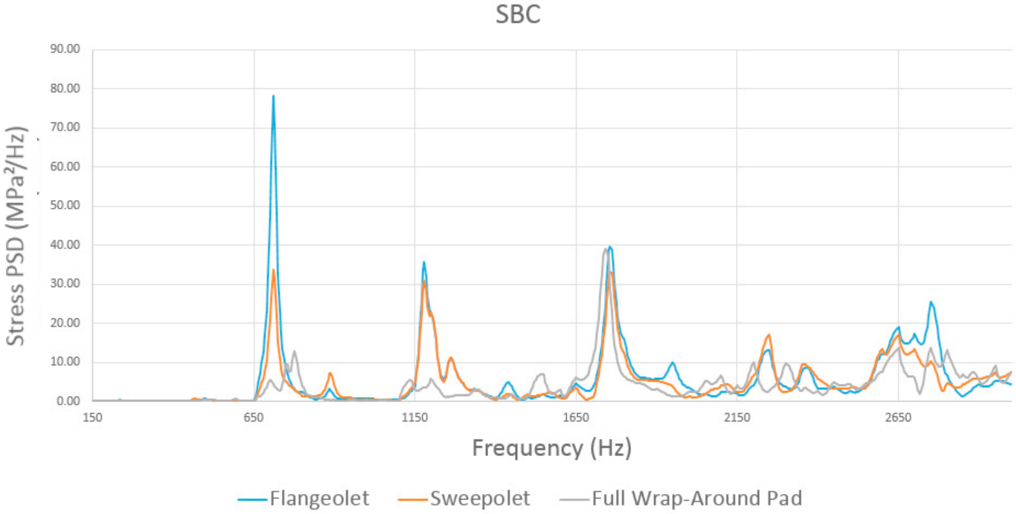


Fig. 12. Stress PSD for small bore connection mitigations

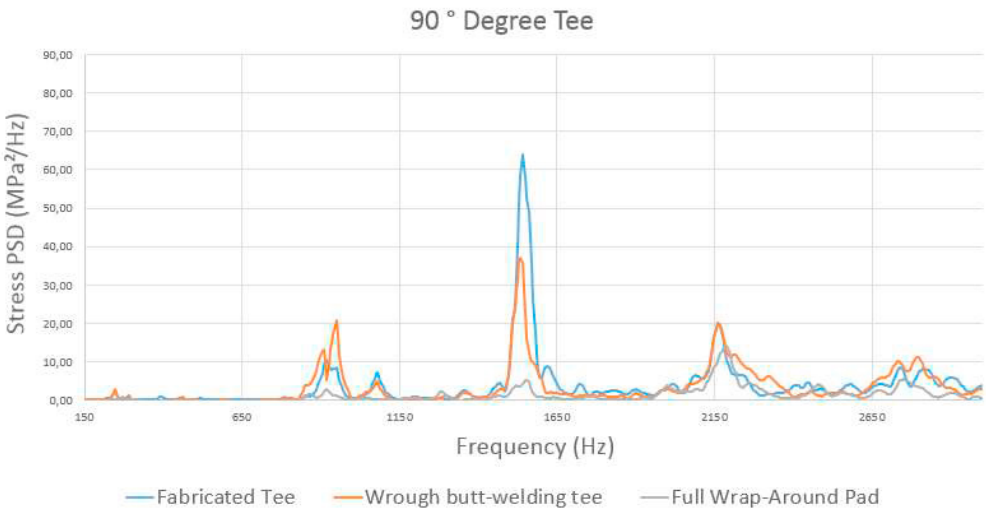


Fig. 13. Stress PSD for tee mitigations

Table 2. Stress reduction ratio

	SBC	90° Tee
Initial case	100%	100%
1st mitigation action case	73%	74%
2nd mitigation action case	60%	30%

As first approach, the stress reduction ratio is used to evaluate the LOF gain corresponding to each mitigation action.
The first step of the Energy Institute guidelines is to calculate a number N representing the number of cycles before failure. Using this number, the LOF is calculated with formula (1).

A LOF equal to 1 corresponds to a nominal fatigue life of 10^7 cycles (fatigue limit) [Swindell, 2012]. If the calculated number of cycle before failure (N) is above 10^7, the LOF is below 1. The Energy Institute guidelines were developed based on BS7608 S-N curves [Swindell, 2012]. These curves and the stress reduction ratio make it possible to estimate an updated number of cycles before fatigue failure and then an updated LOF. This method was used for our 2 cases and an example LOF of 1 for the initial case. Results are presented in table 3.

Table 3. Updated LOF

	Small Bore Connection		90° Tee	
	1st mitigation Sweepolet	2nd mitigation Full Wrap-Around Pad	1st mitigation Wrought butt-welding tee	2nd mitigation Full Wrap-Around Pad
Initial LOF	1	1	1	1
Number of cycles before failure	10^7	10^7	10^7	10^7
Stress level on S-N curve	35	35	35	35
Stress level (with mitigation)	25.55	21	25.9	10.5
Updated number of cycles	2.58×10^7	4.65×10^7	2.48×10^7	3.72×10^8
Updated LOF	0.88	0.80	0.88	0.53

While strictly following the Energy Institute guidelines would have predicted the same LOF for each configuration, this new methodology makes it possible to quantify the use of each design improvement in terms of LOF. The impact of each mitigation action can now be justified and mitigation can be used with more confidence than before, when improvements were applied with no justification.

5. Conclusion

The Finite Element methodology introduced in this article has shown good agreement when compared to field measurements, giving confidence in the pertinence of this approach, even if the results are strongly dependent on the quality of the acoustic energy data input initially in the model for quantitative analysis. This method is a robust approach for performing comparative analyses.

Using this methodology, it is now possible to assess the direct impact of an AIV mitigation action. Using some of the most common solutions for AIV risk, quantitative analyses have been made, demonstrating the LOF reduction from each solution. However, the configurations studied in this article remain very specific to the model properties (valid only for one pipe size and schedule, a specific singularity…). A full set of models has to be made and analyzed in order to derive a global LOF modifier for each mitigation action.

A project coordinated by VIBRATEC, named “Acoustic-Induced Vibration: Improvement of the LOF calculation” started in February 2017 and is sponsored by SAIPEM, TECHNIP and TOTAL. The aim of the project is to analyze the fatigue phenomena in the singularity in order to improve mitigation action efficiency and to more accurately assess the risk at the singularities. During the project, several types of singularity will be studied: supports, branches and small bore connections. To improve the Energy Institute guidelines two issues that are independent of the singularities but problematic will be studied too: Extension to large pipes and sound propagation through branches. The project is still in progress and will end in February 2018.

References

[1] BS7608, 1993. Fatigue design and assessment of steel structures.
[2] Carucci, V.A., Mueller R.T., 1982. Acoustically Induced Piping Vibration in High Capacity Pressure Reducing Systems. ASME 82-WA/PVP-8.
[3] Energy Institute, 2008. Guidelines for the avoidance of vibration induced fatigue failure in process pipework, second edition.
[4] Swindell, R., 2012. Acoustically induced vibration – development and use of the 'Energy Institute' screening method. Internoise 2012.
[5] A. Hobbacher, Recommendations for fatigue Design of Welded Joints and Components, document IIW-1823-07, IIW International Institute of Welding.