JOURNAL OF ACOUSTICS AND VIBRATION RESEARCH (JAVR)
ISSN: XXXX-XXXX
VOL. 2, ISSUE 1, 50 – 61

Acoustic induced vibration assessment on process piping using coupled acoustical/mechanical finite element approach: improvement of existing method on branch

Rémi Salanon*[1], Loïc Ancian[1], Najib Fazail[2]

[1]Vibratec, 28 chemin du petit bois, Ecully Cedex BP36-69131, France
[2]Vibratec Asia Pacific, Jalan Bangsar Utama 1, Bangsar, 59000 Kuala Lumpur, Malaysia

*Corresponding email: remi.salanon@vibratec.fr

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 numerical approach presented for under-standing main phenomena that may lead to a fatigue failure due to AIV is based on a dynamic stress evaluation at pipe discontinuities (welded connections). This evaluation is performed through a one-way fluid-structure coupling Finite Element Analysis. Pressure fluctuations inside the pipe are predicted and coupled with a pipe structural analysis in order to determine the dynamic stress levels. Considering a random fatigue analysis, fatigue damage can be computed and several configurations were compared. This method was used to highlight the influence of branch thickness on branch connection fatigue integrity. In fact, this parameter is not taken into account in existing assessment methodology but may have an unneglectable influence on fatigue life of the connection. The results are in accordance with the fact that a failure is likely to occur within few minutes in a high sound power level area. This first result gives confidence in the fact that this finite element approach is consistent with energy institute guidelines and can be used to compare different geometries or mitigations.

Article History

Received: 09/06/2024

Revised: 28/10/2024

Accepted: 15/11/2024

Published: 12/12/2024

Keywords: Acoustics; vibration; piping; finite element.

INTRODUCTION

Acoustic-Induced Vibration (AIV) has been a subject of concern since the late 1970s and the characterization of the phenomenon remains complex. AIV refers to the high acoustic energy generated by pressure reducing devices that excites pipe shell vibration modes. Induced vibration may produce excessive dynamic stress levels when facing non-axisymmetric welded discontinuities such as branch connections, supports etc. AIV assessment methodologies were developed in order to cope with this risk from earlier design stage. Current methodologies are used systematically by engineers since they allow to perform the assessment on a large amount of line with a limited amount of time. However, these methods preset some limitations due to their simplicities and limited number of input parameters taken into account. A focus on main existing screening method is firstly done in order to highlight their limitations.

A first methodology was widely spread in 1982 by an ASME publication by Carucci and Mueller [1]. Thin- walled piping failures were investigated on 36 real cases, 9 of which were failures. The formula for calculating sound power level (PWL) at the pressure-reducing device is defined in this document and Table 1 shows the PWL calculation input data:

$$PWL = 10\, log_{10}\left[\left(\frac{P_1 - P_2}{P_1}\right)^{3,6} \boldsymbol{W}^2 \left(\frac{T_e}{M_w}\right)^{1,2}\right] + 126,1 + SFF \tag{1}$$

Table 1. PWL calculation input data.

Item	Symbol	Unit
Sound Power Level	PWL	dB
Upstream pressure	P1	Pa abs
Downstream pressure	P2	Pa abs
Gas flow rate	W	kg/s
Upstream gas temperature	Te	°K
Sonic fluid factor	SFF	dB

A "Recommended Safe Limit" curve as a function of calculated PWL vs. maximum pipe outside diameter downstream of the acoustic energy source was presented. The drawback of this method is that only the main pipe diameter is taken into account as a structural parameter. Other parameters that may have an influence on welded connection fatigue life are not considered (branch diameter, run pipe and branch thickness, connection type.

In 1996, Eisinger [2] suggested a straight-line fatigue limit that can be easily calculated. Eisinger plotted the original Carucci & Mueller data set as a function of the ratio of the internal diameter to wall thickness. A new design limit curve and a new failure limit curve was designed. This fatigue limit boundary can be expressed by the expression:

$$PWL_F = 176.6 - 0 - 125\frac{D_{int}}{t} \tag{2}$$

where D_{int}= pipe internal diameter and t = pipe wall thickness.

Using a safety factor of 2 (3 dB), the allowable design limit is given by:

$$PWL_{allow} = 173.6 - 0 - 125\frac{D_{int}}{t} \quad (3)$$

Due to its simplicity, this formula is now the most used form of the 'D/t' method. The Eisinger method presents some limitations as the Carucci and Mueller method:

- Limitation of D/t ratio between 46 and 115,
- No consideration of fitting type and size.
- No consideration of support type.

In 2008, the Energy Institute (EI) published the 2nd Edition of the Guidelines for the Avoidance of Vibration Induced Fatigue in Process Pipework [3]. The original publication was intended principally for use at the design stage; in the period since the first issue, more experience was gained in practical applications and a number of improvements were identified. It should be noted that the EI methodology is based largely on the original research conducted by Carucci and Mueller. The EI expands the assessment to consider fatigue life curves for a range of pipe fittings and piping materials. EI screening produces a likelihood-of-failure (LOF) number those defaults to 1 for any system screened at a LOF of 1 or greater. Systems with a LOF = 1 are deemed to be at risk and need to be redesigned. The methodology was developed for pipes up to 24" diameter. Experience shows that flare headers can often exceed this diameter and are therefore not directly covered by this guideline. The EI method is limited to certain forms of welded discontinuity. The only type of connection considered is the weldolet, leaving a lot of uncertainty for a similar but slightly different fittings such as sockolet, sweepolet etc. Welded supports are also not explicitly covered by the method. None of current approaches (D/t method, or EI guidelines) do consider the influence of branch thickness, t and would give the same score as function of the thickness. This paper will focus on demonstrating the need to take into account branch thickness when performing the design of a piping network subject to AIV or solving an AIV issue on an existing facility. A numerical approach will be used to achieve this purpose

METHODOLOGY

Analysis of the fatigue phenomenon in the singularity was performed to more accurately assess the failure risk at the singularities. An innovative computational approach was undertaken to identify the AIV damage based on a dynamic stress evaluation at pipe welded connections. An acoustic Finite Element Model (FEM) of the piping system is created. Structural FEM was also built with particular attention to the welded area. The stress evaluation is performed through a fluid-structure coupling Finite Element Analysis using MSC ACTRAN software. Pressure fluctuations inside the pipe are predicted and coupled with a pipe structural analysis to get the resulting dynamic stress levels. Validation of the method was given in [4].

STRUCTURAL MODELLING

In order to perform the fatigue evaluation through Finite Element Analysis, a structural Finite Element Model of the connection was built using NASTRAN shell elements (linear triangle and quadrangle). Shell elements are considered at run and branch mid-surfaces. An example of considered FEM is given in following Figure 1:

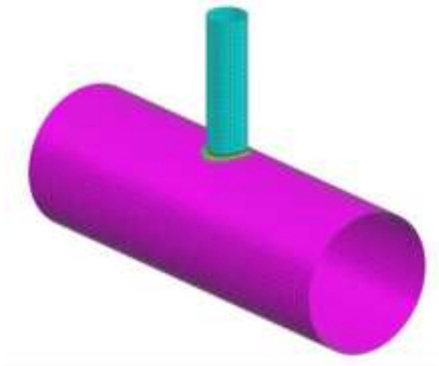

Figure 1: View of structural FEM.

A particular attention was made on the welded connection. The weld modeling is presented on a standard pipe to pipe connection and was defined considering post-processing method requirements. The hot spot stress was selected as post-processing method for the fatigue analysis. The "Hot Spot" or geometric stress takes into account the geometrical effects of the assembly. This stress should not be influenced by the detail of the local shape of the weld joint. It can be considered as the stress at the root of the weld without local effect. Using hot spot stress method is a robust method that make easier comparison between two studied cases. 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].

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

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.

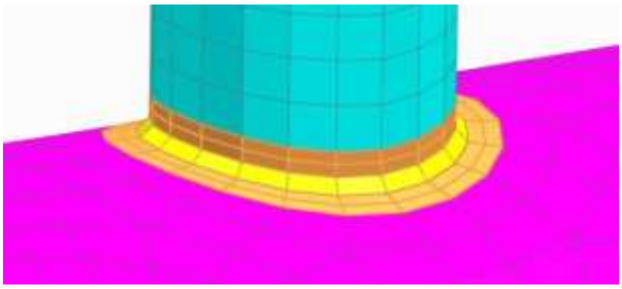

Figure 2: Weld modeling on a standard case.

Weld characteristics are taken into account in the Finite element model. The considered welds are full penetration weld. Weld joints are modeled by shell element at their mid-surface. Both pipes are linked at their extremities at coincident nodes. The weld is modeled by shell element considering the weld joint thickness.

ACOUSTICS MODELLING

An acoustic Finite Element Model (FEM) of the piping system is created. This model is made of solid elements that represent the fluid (here gas) with a given speed of sound. All walls are considered as rigid (zero displacement) except the upstream face, downstream and branch end with possible propagation of waves. A constant random PSD pressure source is applied on the upstream side of the pipe in the [100 – 2500] Hz frequency range (red surface in Figure 3). Waves will propagate in the system differently depending on the frequency and pipe diameter. Indeed, elementary waves can only propagate above their so called "cut-off" frequency, depending on the wave length and pipe diameter. As a result, at low frequencies, only plane waves will propagate below the cut-off frequency. Then, at given frequencies, additional waves will also transport the energy through the pipe with different shapes. These waves will propagate in the piping system with a helix shape as illustrated in Figure 4.

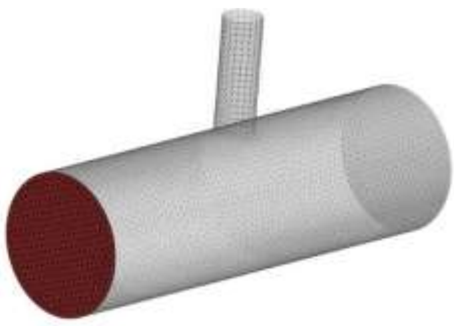

Figure 3: View of acoustic FEM.

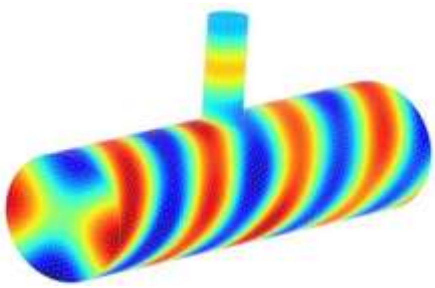

Figure 4: Example of acoustic propagating mode at a given frequency.

FLUID-STRUCTURE INTERACTION

The pressure loads from the acoustic computations are transferred to the structural finite element model. The transfer is a "one way" coupling: it is considered that pipe vibrations will not affect the acoustic behavior inside the pipe. It could be the case if the fluid is heavy; in the present study, the fluid is gas: this assumption was verified considering two cases. The first considered computation case is performing a direct acoustic response and structure modal response with 1-way coupling. The second studied case is a full direct response for both acoustic and structural analysis and in this case a 2-way coupling was considered. The first case is computationally less expensive by using a modal response. Pressure and vibration maps were compared at several frequencies. Results are similar: there is very little influence on the pressure loads and structural response for the 2 configurations. Therefore, the use of a 1 way-coupling is preferred and used in this paper.

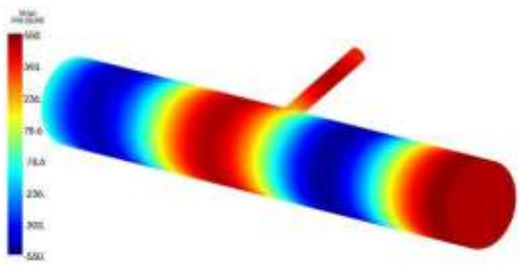

Figure 5: Pressure at 200 Hz – 1-way coupling.

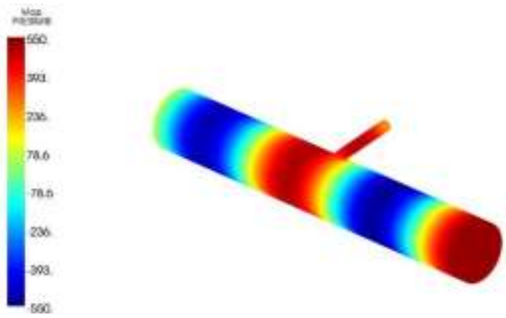

Figure 6: Pressure at 200 Hz – 2-way coupling

RESULTS AND DISCUSSION

Fatigue Analysis

Fatigue damage is traditionally determined from time signals of loading, usually in the form of stress or strain. This approach is satisfactory for periodic loading, but requires very large time records to accurately describe random loading processes. These may be prohibitive for many finite element analyses, especially when per- forming dynamic response. In most cases, the structure response under random excitation is performed in the frequency domain, the loading and the response are categorized using Power spectral density (PSD) and the dynamic structure is modelled as a linear transfer function. In 1985, T. Dirlik [5] proposed a method that is widely applicable and constantly outperforms all of the other available methods for random fatigue estimation. Until now, this method is still the most widely used and the most efficient. From a stress power spectral density (PSD) function, this method will estimate the number of cycles of each amplitude. This information is directly usable with a fatigue curve to compute the equivalent damage. The fatigue SN curve applied in this present study is the Hot spot stress SN curve taken from DNV-RPC-203 (2).

Case Study

The reference model was defined as a 24” x 6” pipe connection as presented in Figure 7. Four structural pa- rameters can potentially affect the stress results and therefore the expected failure time (main pipe thickness (T), branch thickness (t), main pipe diameter (D), branch diameter (d)). There are two potential crack initiation locations at the welded connection: on branch side or on run side (refer to Figure 8). Results will be post-processed for each location.

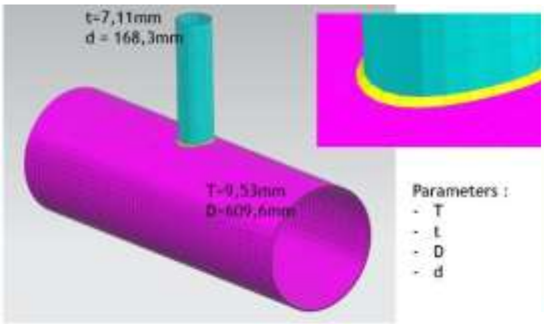

Figure 7: 24 inches reference case with corresponding dimensions.

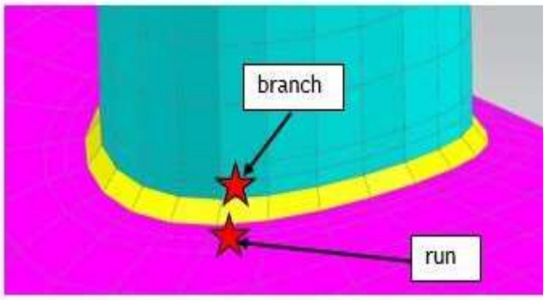

Figure 8: Crack initiation locations.

To make comparisons between branch and header side, results are presented considering a PWL of 154 dB. It has to be noted that this acoustic level is usually considered as low for most of the codes (D/t, EI guidelines) and should not lead to fatigue failure. A stress PSD was obtained for each element from the simulation, providing information about the frequency content. The following Figure 9 shows maximal hotspot stress PSD on each side of the welded connection.

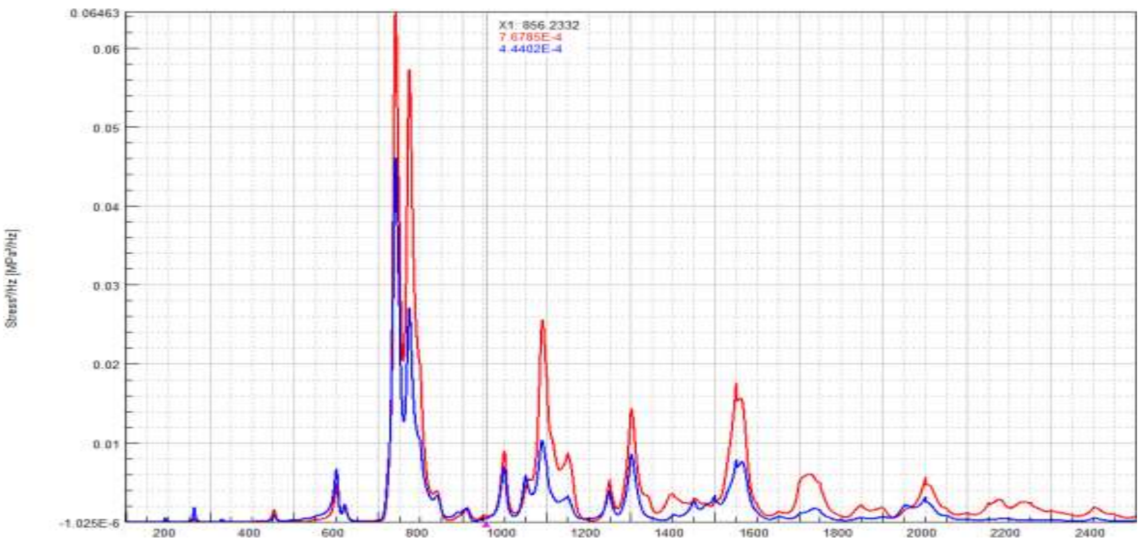

Figure 9: Stress PSD (Mpa²/Hz) for the 154 dB reference case (branch side in red – run side in blue).

For the damage to occur, the mechanical natural frequencies need to coincide with the acoustic modes propagating in the pipe. Based on maximal stress PSD comparison, it appears that failure should occur first on branch side (highest stress peak are observed on branch side). This conclusion is confirmed by the fatigue analysis. The two main modal contributors in term of stress on the weld are presented in Figure 10 and Figure 11. It corresponds to high order local modes. Duct modes responsible for high stress levels is also presented in Figure 12.

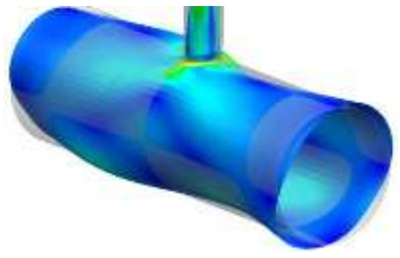

Figure 10: Mechanical natural frequencies at 741 Hz.

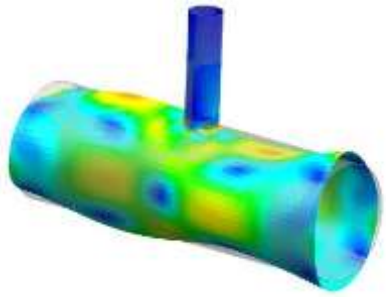

Figure 11: Mechanical natural frequencies at 774 Hz.

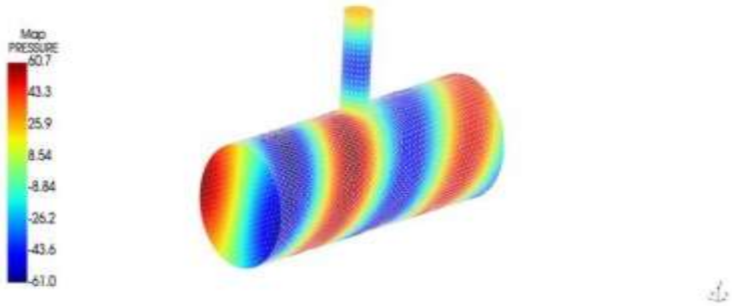

Figure 12: Duct mode 1,1 and corresponding pressure map at 750 Hz.

The total damage is computed as the sum of unit damage for each cycle. A histogram presenting the expected number of cycles and their amplitude is given in Figure 13.

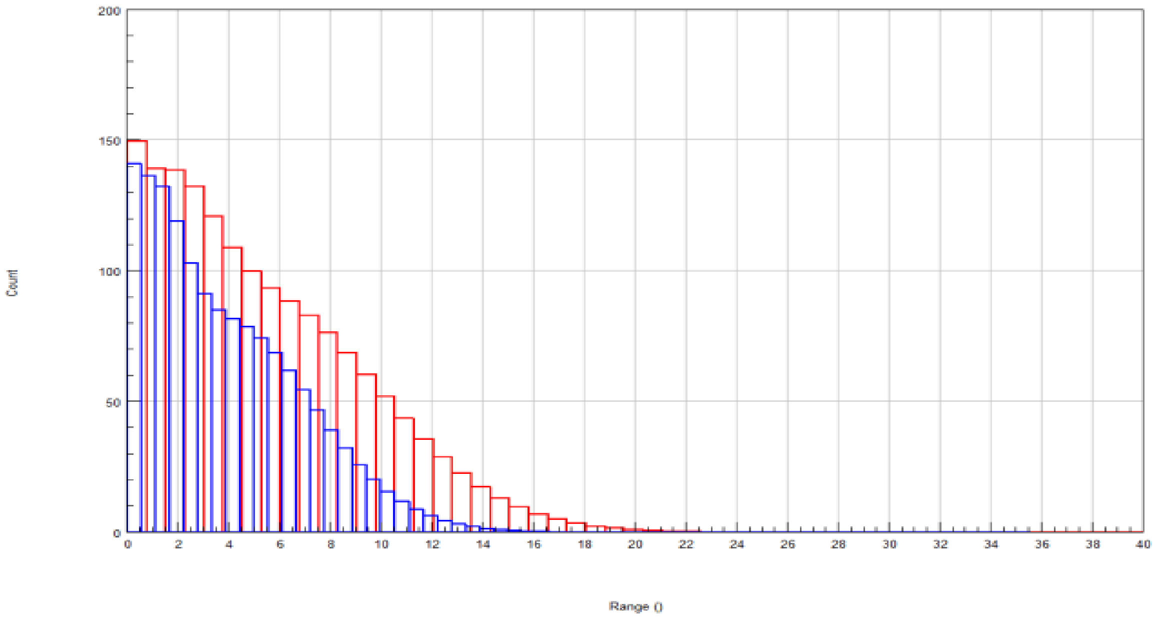

Figure 13: Life cycle histogram (branch side in red – run side in blue)

The damage is computed for 1 second of signal. The total life can be computed simply by inverting this damage value. Results are summarized in following Table 2:

Table 2: Damage and life duration results for the 154 dB.

Connection Side	Computed Damage for a PWL of 154 dB	Life duration (days)
Branch	$3.18\ 10^{-8}$	364
Run	$6.29\ 10^{-9}$	1840

Based on the numerical study results, it appears that fatigue life is lower on branch side, failure is therefore expected first on the branch side. The simulation shows that theoretically, the expected life of this pipe before crack initiation is minimum one year of continuous operation at 154 dB and three years on the run side of the weld. This duration may allow to conclude that the connection as an infinite fatigue life in comparison with operating duration under AIV conditions (only few hours). This conclusion is in-line with current methodology since a PWL of 154 dB is not considered as critical and should not lead to fatigue failure.

The EI guidelines AIV methodology was applied to this case in order to determine what the PWL limit value would induce a LOF of 0.5 and LOF of 1. An LOF score of 1 indicates that fatigue risk is important and corrective actions shall be undertaken such as a redesign of the concerned line. An LOF score below 0.5 indicates that the line can be considered as safe regarding AIV phenomenon. Table 3 below shows the detailed AIV screening calculation considering EI guidelines assessment.

Table 3 : PWL for LOF=0.5 and LOF=1

	LOF=0.5	LOF=1
OD Header (“)	24	24
OD branch	6	6
T header (mm)	9.53	9.53
α	1.005725	1.005725
s	27.93358	27.93358
B	157.1296	165.3162
Log(N)	8.60938	6.935913
N	4.07E+08	8,628,057
FLM1	1.13438	1.13438
FLM2	1	1
FLM3	1	1
N with FLM	4.6E+8	1E+7
PWL	159.86	167.8

In a second step, these PWL value will be applied to our studied case to determine the fatigue life for both PWL on both side of the welds as shown in Table 4.

Table 4: Fatigue life results

EI LOF	PWL	Run	Branch
0.5	159.9 dB	45 hours	9 hours
1	167.8 dB	7 minutes	1.2 minutes

These results are well in accordance with the fact that a failure is likely to occur within few minutes for a high PWL. This result gives confidence in the fact that this finite element approach is not too conservative and can be used to compare different geometries or mitigations.

Branch thickness sensitivity

The influence of branch thickness was evaluated considering a 24” run pipe and a 6” branch. Six configurations were tested and are listed in the Table 5 below and Main sensitivity results were plotted in the following Figure 14:

Table 5: Branch thickness evaluation – characteristics.

	RUN		BRANCH	
Case	D	T	d	t
1	609.6	9.53	168.28	3.404
2				5
3				6
4				7.11
5				10.973
6				14.25

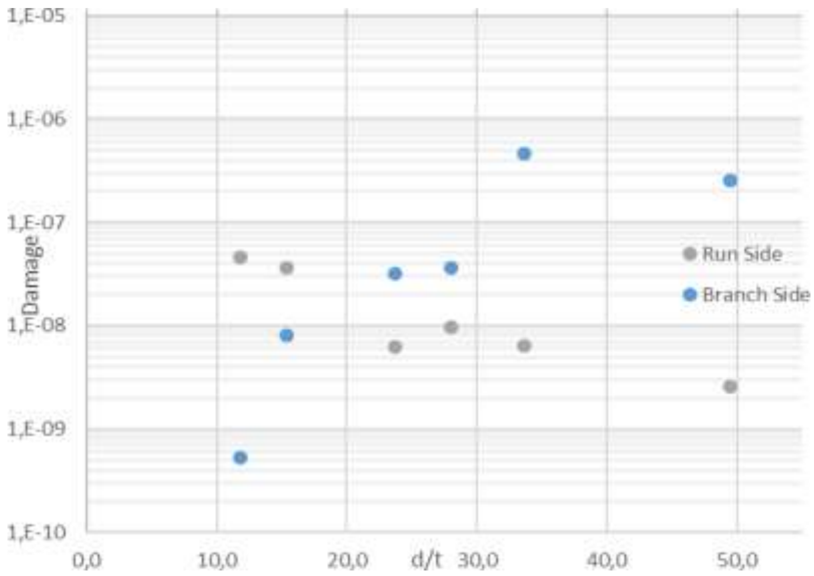

Figure 14: Damage vs d/t.

Branch thickness has an influence on damage on both the branch and run sides. As branch thickness in- creases, the branch side damage decreases while the run side damage increases. The failure will theoretically always occur on the thinnest pipe side of the branch.

RECOMMENDATION

As seen in the previously, failure may occur on the branch side in some cases and on run side for other cases. Behavior is quite different for both sides; therefore, the idea would be to estimate the damage on both sides of the branch connection. Maximal damage should be then used to determine fatigue life as shown in Figure 15.

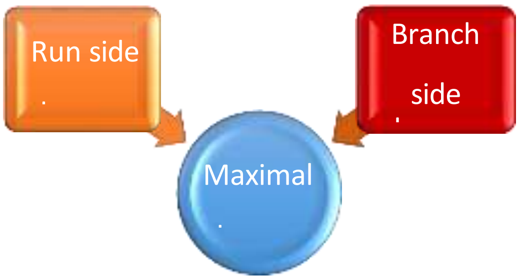

Figure 15: Methodology flowchart.

For a more practical point of view, a new FLM should be introduced to take into account branch thickness influence in connection fatigue life. At this time, no simple quantitative rule can be established to reach this objective. Further extensive computation studies would be needed. From a qualitative point of view, some recommendations can be made to take into account branch thickness specificity. As failure seems to occur on the thinnest thickness, it seems to be important when dealing with mitigation and when the increase of run pipe thickness is recommended to increase also the branch thickness. In a similar way, it would be recommended to consider a wrap around on both side of the weld. Good practice would be to have a wrap length of 2D on the run pipe (1D upstream the branch and 1D downstream). The wrap on the branch side should have a length of 2d.

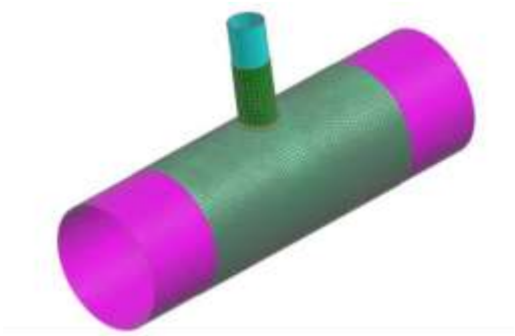

Figure 16: Example of a wrap on the branch.

CONCLUSIONS

A reference model consisting of a 24” x 6” branch connection was defined. Four mechanical parameters potentially affect stress results: (T = 9.53 mm, t = 7.11 mm, D = 609.6 mm, d = 168.3 mm), “t” not being considered by the Energy Institute Guidelines for detailed screening. Results for reference cases are in accordance with the fact that a failure is likely to occur within few minutes for a high PWL. This first result gives confidence in the fact that this finite element approach is consistent with EI guidelines and can be used to compare different geometries or mitigations. Sensitivity studies were performed to determine which parameter has an influence on branch fatigue life. The following conclusion can be made:

- The damage evolution is different on the branch side than on the run pipe,
- Highest damage is always computed on the pipe with the smallest thickness,

Some qualitative recommendations can be done to deal with branch thickness specificity regarding AIV. To establish quantitative assessment, further work would be needed.

.

REFERENCES

[1] Carruci & Mueller., Acoustically Induced Piping Vibration in High capacity Pressure Reducing Systems, ASME Paper 82-WA/PVP-8, 1982.
[2] Esinger F.L., Designing Piping System Against Acoustically Induced Structural Fatigue, Journal of pressure vessel technology 119, pp. 379-383, 1997.
[3] Energy Institute, Guidelines for the Avoidance of Vibration Induced Fatigue Failure in Process Pipework. London : Energy Institute, 2008.
[4] Coulon A., Salanon R., & Ancian L., Innovative numerical fatigue methodology for piping systems: qualifiying Acoustic Induced Vibration in the Oil & Gas industry. Procedia Engineering 2018.
[5] Dirlik T., Application of Computers in Fatigue Analysis. s.l. : The university of Warwick, 1985.
[6] DNV-RP-C203, Fatigue design of offshore steel structures. 2010.