A simplified method to predict thermal fatigue in mixing tees of nuclear reactors

Frederic Blom, Martin Church & Sander Willemsen

NRG Petten, The Netherlands

Abstract The phenomenon thermal fatigue is investigated in a mixing tee. The mixing tee configuration is used in nuclear reactors where thermal fatigue is a safety related phenomenon. Turbulent mixing of hot and cold fluid flows causes temperature fluctuations in the pipe wall, which cause stress fluctuations. When the amplitude of these thermal stresses is large enough the wall can suffer from fatigue resulting in cracks in the wall. The influence of local temperature gradients on the pipe wall of the Fatherino model is determined by Computational Fluid Dynamics (CFD). More specifically, Large Eddy Simulation (LES) is used to predict the mixing flow. The results from the CFD calculation are transferred to a finite element code, where the stresses due to the temperature gradients are calculated. The results from the CFD model show fluctuating wall temperatures. The results are presented in a specific style for thermal fatigue namely the temperature differences, and not the absolute temperatures. These temperature differences are the source of the fluctuating stress and fatigue in the pipe wall. This alternative presentation shows that the fatigue stress is caused by moving temperature spots, which are convected by the main flow velocity. In order to simplify the calculation method, the three dimensional model is transferred into a one-dimensional model through several intermediate steps. This approach gives insight into the model simplifications and its influence on the results. It is shown that a one-dimensional simplified model can conservatively predict the fatigue behaviour of a three-dimensional model loaded by a moving temperature spot. For the coupling to the unsteady CFD model, it is shown that the treatment of the interface temperatures is extremely important. This method uses the information of the three-dimensional CFD results in a one-dimensional model and opens a new opportunity to predict thermal fatigue in mixing tees by a simplified method.

1 INTRODUCTION

Thermal fatigue is a special form of fatigue where the loading is caused by thermal fluctuations in a solid material. The resulting cyclic stresses are caused by temperature gradients in the material. The phenomenon thermal fatigue is investigated here in a mixing tee, where the fatigue phenomenon is caused by turbulent mixing of hot and cold fluid flows. The configurations are used in nuclear reactors where thermal fatigue is a safety related phenomenon. The pipe wall downstream of the mixing tee faces temperature fluctuations and gradients, which cause fluctuating thermal stresses. When the amplitude of these thermal stresses is large enough the wall can suffer from fatigue resulting in cracks in the wall.

In the current research Computational Fluid Dynamics (CFD) is used to investigate the influence of local temperature gradients on the pipe wall. The results from the CFD calculation are transferred to a finite element code, where the stresses due to the temperature gradients are calculated. Since this approach is time consuming in terms of preparation and computer time, the differences and correspondences with the one-dimensional model are investigated. The analysis systematically considers 1D, 2D and 3D simulation techniques as a means of assessing transferability. This approach gives insight into the model simplifications and its influence on the results.

2 GEOMETRY AND EXPERIMENTAL CONDITIONS

2.1 Fatherino 2 Facility

The Fatherino 2 facility of CEA is used to determine the thermal mixing in a T-junction during the THERFAT project [2]. The aim of this study is to measure the temperature of the fluid close to the wall, at locations where the highest fluctuations of the temperature appear. It is assumed that these fluctuations induce significant stresses on the structure, which can lead to irreversible damage. In Table 1, the average inlet temperatures and velocities are given.

Table 1 Average experimental conditions

Temperature hot inlet	T_h	80.7	°C
Temperature cold inlet	T_c	8.7	°C
Temperature difference	ΔT	72	°C
Hot fluid flow	V_h	2.56	m/s
Cold fluid flow	V_c	0.84	m/s
Total fluid flow	V_{tot}	3.4	m/s

3 CFD MODEL

3.1 Geometry and Mesh

Calculations have been carried out using a Large Eddy Simulation (LES). To set up such a calculation a specific procedure is used to reduce the calculation time. First, the pipe geometry is created with long inlet tubes, typically 25 pipe diameters (25D). With this elongated model a proper boundary condition for the shorter more detailed model (inlet tubes are 5D) is generated.

The mesh consists of 390995 fluid cells and 178032 solid cells. A fine boundary layer mesh is used, ensuring proper use of wall-functions.

3.2 Main Fluid Dynamics Model

In this project, the commercial CFD code FLUENT (version 6.2.16) has been used. The main fluid dynamics model has the following characteristics: Incompressible, Non-Isothermal, Conjugate heat transfer approach, Transient, Three-dimensional, Turbulent and Large Eddy Simulation. Several implementations of this model exist in the description of the subgrid scale eddy viscosity model. In this project the Dynamic Smagorinski-Lilly model has been used [3].

3.3 Initial Conditions

The initial conditions for the LES simulation on the short model, are generated in a steady-state Reynolds Averaged Navier-Stokes (RANS) simulation. Since temperature fluctuations will start developing in the transient LES simulation, a simulation is performed to allow the fluid and solid to equilibrate to this transient state.

4 RESULTS CFD MODEL

In Figure 1 a typical temperature profile is shown. It is clear that vigorous turbulent mixing occurs at the junction, where the cold and hot fluid meet. In the scale resolving LES simulation, the large eddies are clearly visible in the wavy interface of the hot and cold fluids. The waves are formed as the junction travels downstream with the flow. At the top of the pictures the lateral motion of the fluid is depicted. Here also large-scale flow structures can be observed. As explained in previous chapters, the most important factor for predicting thermal fatigue is the evolution of temperature at the T-junction walls. The temperature changes are caused by a fluctuating heat flux at this boundary. This heat flux is also depicted in Figure 1. Here, the formation of a heat flux spot can be observed. This insight will be used in subsequent stress analysis of the next chapter.

During validation it turns out that the average temperature is overestimated by the calculations. Besides the average temperature, also the amplitude of the temperature fluctuations is compared. Here the overestimation is even more pronounced. Before drawing conclusion on this comparison, it is important to consider the following:

1. The thermo couples are positioned very close to the wall, hence, this comparison only provides information on this specific location;
2. The thermo couples react relatively slowly (response time is 27 ms) to changes in the temperature, hence, fast fluctuations are filtered out.

Unfortunately, it is to be expected that the LES simulation is most likely to have problems in predicting the temperatures close to the wall since standard wall-functions had to be used to limit the computational time required by the simulation. Work is ongoing to establish the influence of thermo couple response time on the results. The CFD results are used to demonstrate the coupling to the finite element method and qualitative phenomena are used to develop the simplified method in this paper.

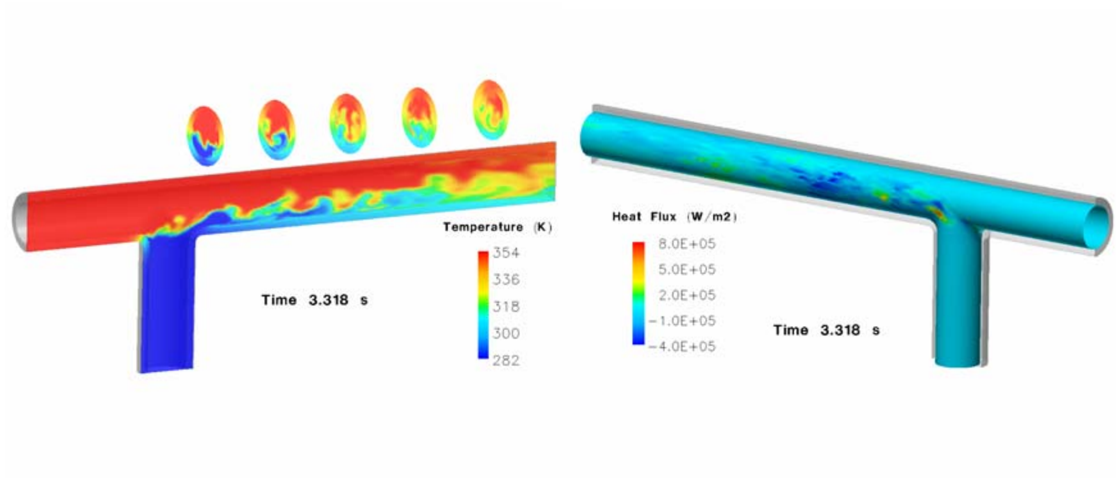

Figure 1. Fluid and solid temperature and wall heat flux at the fluid-solid interface

5 MECHANICAL MODELS

The Fatherino experiment is described by different models ranging from very simple 1D models to a full 3D model. The starting point is a 1D plate model approach adopted here for the assessment of high cycle thermal fatigue using the so-called "Sinusoidal Method" in line with the simplified assessment approach proposed by EdF and being considered further within the NESC Thermal Fatigue working Group [1,4,5]. The Sinusoidal Method assumes that local variations in the bulk fluid temperature close to the pipe wall during mixing conditions may be approximated by a sinusoidal variation with a given mean temperature and cyclic amplitude (both yet to be defined). Whilst this approach represents a very generalised characterisation of a very complex situation it is hoped, given other conservatisms in the overall approach, that the assessment route is relatively insensitive to this basic premise. In particular, response is considered over a range of postulated loading frequencies, with the thermal and fatigue response of the surface determined for each frequency. Lifetime is inferred on the basis of the most conservative prediction.

5.1 1D Plate Model

To allow flexibility in terms of applied local constraint, a simplified plate model has been developed as part of the present work to allow consideration of this issue. The thermal response of a component wall as occurs during turbulent mixing is likely to be very localised. In practice the heat transferred to the wall will be conducted away from the local hotspot not only in the through wall direction but generally though the wall material. There will be a temperature difference between the local hotspot and the surrounding and underlying wall material that will induce constrained thermal expansion. The plate model differs from the alternative cylinder model where response in the circumferential/radial plane is determined inherently on the basis of geometric force balance and compatibility. The routine generates surface stress levels for several symmetric constraint conditions: no constraint, full constraint, bending constraint and membrane constraint.

5.2 1D Tube Model

For the 1D tube model the thermal results are represented by the same model as for the 1D plate model. The stresses however are analytically calculated based on a cylinder model.

The model is implemented in ANSYS as well in order to perform validation simulations. Moreover, the model can be extended to more complex geometrical configurations, which is shown in the subsequent sections.

5.3 2D Model, Partly Heated

The next level in refinement to the Fatherino test model is a two dimensional model, which is partly heated along the circumference. The model is shown in
with the thermal boundary conditions on a part of the interior surface, indicated by the arrow. The surface angle for heating is 30°.

5.4 3D Tube Model, Spot Heated

The three dimensional model and mesh of the pipe is shown in

. The thermal boundary conditions are shown in the spot, where the tube is heated. The angle where the surface is heated is 30° and the length of the spot is 55 mm. The mesh is refined in the radial direction to obtain more accuracy at the inner surface of the pipe, where the thermal and stress cycle, are higher.

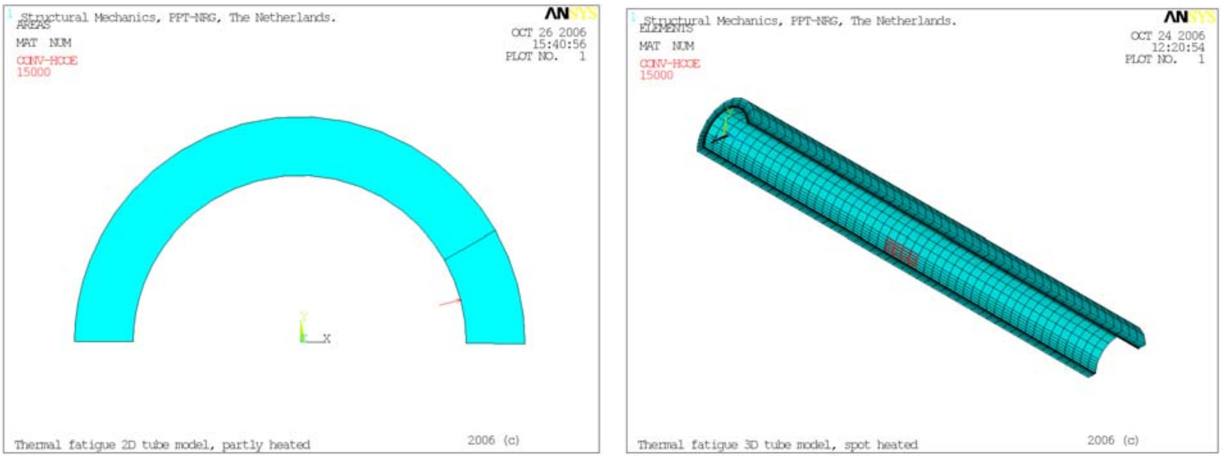

Figure 2. 2D model, partly heated and 3D model spot heated

5.5 3D Tube Model, Travelling Spot

The 3D tube model is next used to calculate the thermal stresses due to a moving spot over the length of the tube. The same boundary conditions are used as for the previous model from section 5.4 but in this case a heating spot is applied to the model with a constant velocity consistent with the mean flow speed (V=3.4 m/s) in the Fatherino experiment.

5.6 3D Fatherino Model, Travelling Spot

The Fatherino model and mesh with thermal boundary conditions is then used to perform the simulation. The angle where the surface is heated is 30° and the length of the spot is 55 mm. The spot travels over the T-junction, in order to investigate the influence on the temperatures and stresses. The mesh is very fine since the model is taken from the CFD calculations, which require a fine mesh in order to resolve the flow and its turbulent structures.

5.7 3D Fatherino Model, CFD Results Coupling

In order to calculate the stresses from temperatures calculated from the CFD model the temperatures resulting from this model are used in the finite element model in ANSYS. Therefore, an interface is created through file exchange. The temperatures from the CFD model are written to a Nastran file by means of a user written C-code program.

The nodes and elements are read into the ANSYS input file. In order to obtain fluctuating stresses results at different times have to be subtracted to obtain the stress amplitude. Since the calculations are linear, the stress amplitudes can be obtained as well by using the temperature difference from the CFD input. It is chosen to use this type of loading since then, the temperature difference, which causes the fatigue can be monitored directly. In order to obtain temperature differences, the first thermal solution from the series is used as a reference point. The other temperature solutions are subtracted from this first solution.

6 MECHANICAL RESULTS

The mechanical results are calculated using the thermal/mechanical conditions as shown in Table 2.

Table 2. Conditions for thermal/mechanical models

Heat capacity	C_p	500	J/kgK
Heat conduction	λ	14.57	W/mK
Density	ρ	8000	Kg/m^3
Thermal expansion	α	18.02×10^{-6}	1/K
Young’s modulus	E	184	GPa
Poisson ratio	ν	0.3	-
Heat transfer coefficient	a	15000	W/m^2K
Temperature amplitude	$0.5\Delta T$	36	°C
Total fluid flow	V_{tot}	3.4	m/s

6.1 1D Plate Model

The results for the 1D plate model are shown in Figure 3. At high frequencies the stress amplitude tends towards zero independent of the constraint option chosen. This is consistent with the expectation that temperature fluctuations are sufficiently rapid that the wall has no time to respond thermally. At intermediate and lower frequencies surface response is seen to be sensitive to the chosen constraint model. At low frequencies quasi-static conditions are achieved with essentially a stationary, uniform temperature distribution through wall. Under full constraint, membrane stress conditions are achieved, with through wall expansion fully suppressed. With no constraint, stress free conditions exist. At intermediate frequencies, variations in stress amplitude reflect the constraint options and though-wall force and compatibility requirements. The full constraint option is consistently the most conservative. Thereafter, in the range 0.1-10 Hz bending restraint provides peak surface stress amplitudes essentially reflecting temperature changes local to the heated surface. Membrane stresses only dominate at lower frequencies when through wall temperature gradients close to the heated surface become less severe.

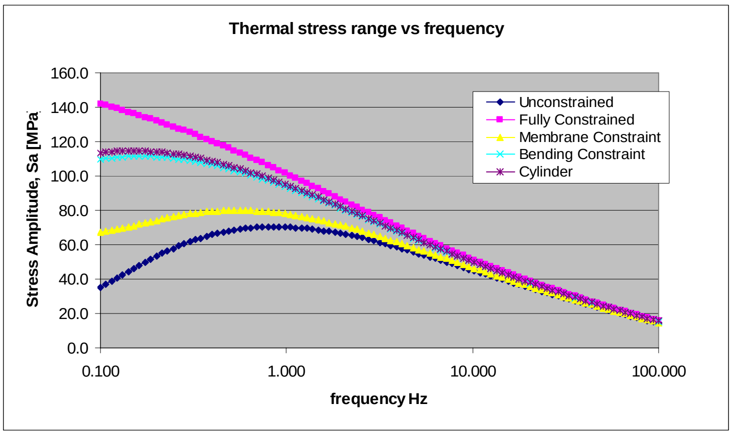

Figure 3. Stress amplitude as a function of frequency, 1D plate model

6.2 1D Tube Model

The stress amplitude as a function of the frequency for the analytical cylinder model is shown in Figure 3 as well. The stress values are comparable to the 1D plate model with bending constraint.

6.3 Comparison of Temperatures and Stresses for 1D, 2D and 3D Models

The model results for the different 1D, 2D and 3D models are compared in Table 3 for a frequency of 0.1 Hz, which corresponds to the maximum stress amplitude response from the 1D analysis. The maximum temperatures and stresses at step 4 (t=2s) are compared for the ANSYS models. This table shows that the 1D simulation is conservatively representative for the 2D and 3D models. This analysis shows that a 3D model subjected to cyclic heating in a spot can be conservatively represented by the 1D model.

Table 3. Comparison of temperatures and stresses for the different models, 0.1 Hz

		Step 4 (ANSYS only)		Total	
Dimension	Model	T_{max} [°C]	σ_{max} [MPa]	ΔT [°C]	Δσ [MPa]
1D plate	Analytical bending	-	-	55.7	119.6
1D tube	Analytical	-	-	55.7	106.1
1D tube	ANSYS	24.6	101.0	53.6	116.0
2D tube	ANSYS	24.6	91.0	53.6	99.5
3D tube	ANSYS	24.7	92.7	53.6	104.0

6.4 3D Tube Model, Travelling Spot

The thermal results of a travelling spot at a fixed position on the symmetry plane of the pipe are shown in Figure 4. This figure shows that the temperatures are quickly increasing to a maximum of 6.3°C. When the spot is passed, the temperature slowly decreases. The calculated stress is maximal at the point where the temperature is maximal. The maximum stress is 27.7 MPa.

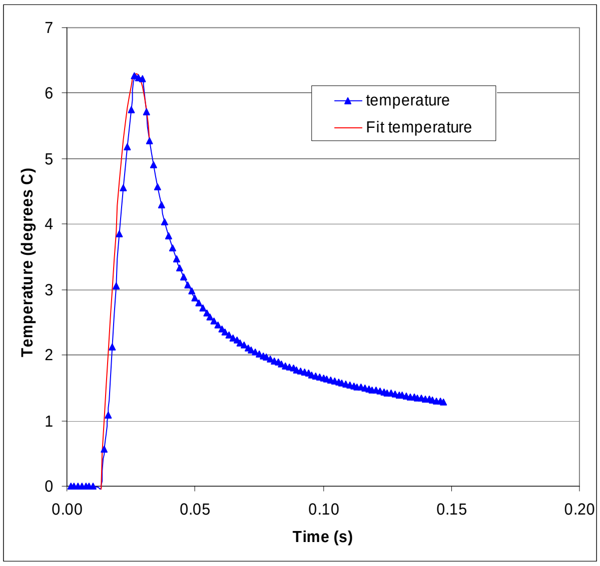

Figure 4. 3D tube model, travelling spot, thermal results, time resolved + temperature fit

6.5 3D Fatherino Model, Travelling Spot

In order to check the possible stress concentration influence of a T-branch on temperature and thermal stresses, the same travelling spot simulations are performed on the Fatherino test model. The thermal results show the same thermal and stress response as for the straight pipe; therefore, the T-branch can be represented by a straight pipe for the moving spot problem.

6.6 Comparison of Temperatures and Stresses for the Different Models

The travelling spot problem is investigated in more detail. In order to couple the sinusoidal method to the travelling spot, a sine function is fit into the temperature solution. This function is shown in Figure 4. The frequency of the sine fit is 18 Hz. By means of this frequency the relation to the sinusoidal function fit can be traced. The maximum temperature from the travelling spot solution is 6.3 °C and the corresponding stress (Von Mises) is 27.7 MPa.

The comparison of the different models to this outcome using the fitted frequency of 18 Hz is shown in Table 4. Besides the temperature range ΔT, also the maximum temperature T_{max} is shown in this table. The initial temperature rise is comparable to the rise of temperature when the spot travels past a point on the tube. The time needed for the temperature rise causes a thermal gradient in the wall, which results in the thermal stress, therefore only the first part of the sine is required to represent the time response of the travelling spot in the sinusoidal method.

Table 4 shows that the temperatures and stresses are slightly higher for the sinusoidal method than for the corresponding travelling spot. This comparison shows that the travelling spot problem can be conservatively represented by means of the 1D, 2D and 3D sinusoidal models. The analysis shows as well that only the first part of the sine is needed to calculate the influence of the rising temperature on the thermal stress.

Table 4. Comparison of temperatures and stresses for the different models, 18 Hz

Dimension	Model		T_{max} [°C]	ΔT [°C]	Maximum σ_{eqv}[MPa]
1D plate	Analytical bending	Sinusoidal	7.0	12.1	27.8
1D tube	Analytical		7.0	12.1	28.3
1D	ANSYS		6.68	11.5	31.0
2D	ANSYS		6.68	11.5	30.4
3D	ANSYS		6.64	11.5	31.2
3D travelling spot	ANSYS		6.3		27.7

6.7 3D Fatherino Model, CFD Results Coupling

In order to calculate the stresses originating from the temperature fields calculated by the CFD model, first the wall temperatures are transferred to the ANSYS model. These wall temperatures are shown in Figure 5. This figure shows the creation of a temperature spot, which is transported with the mean flow velocity and finally fades away (not in figures). The temperature differences vary between -10 °C and 18 °C. The maximum stress is located above the corresponding temperature spot, as in the travelling spot problem. The maximum stress is 77 MPa.

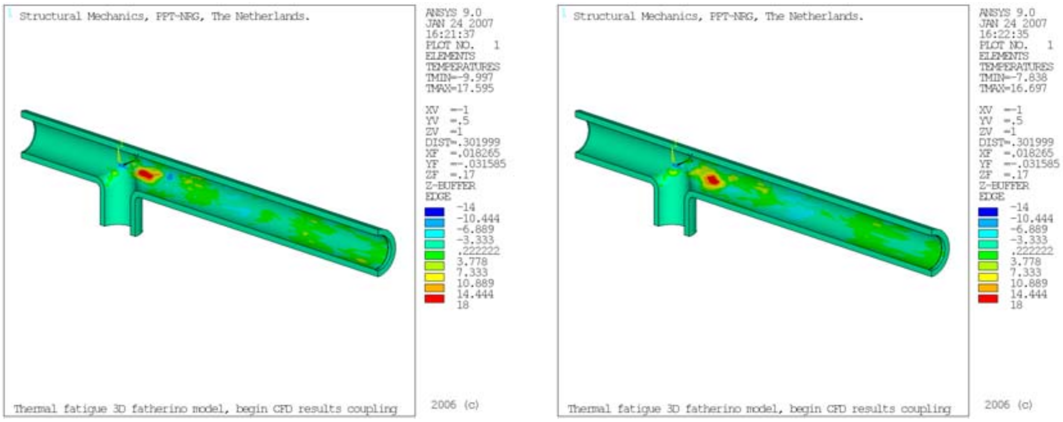

Figure 5. Temperature differences transferred from CFD model to ANSYS

The comparison to the sinusoidal method is made by fitting a temperature curve to a time resolved temperature at the spot centre, similar to the travelling spot problem. The result of the fit of the spot centre of Figure 5 is shown in Figure 6. The fitted frequency is 25 Hz. Next, the sinusoidal method is used to calculate the temperature rise and resulting thermal stress corresponding to a frequency of 25 Hz.

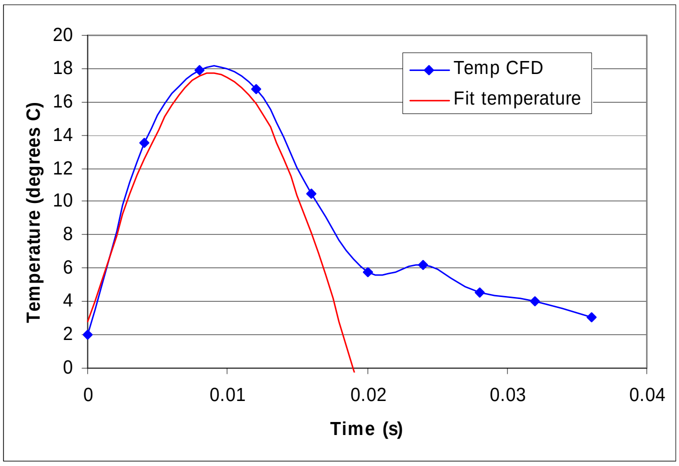

Figure 6. CFD model thermal results (spot centre Figure 5/3), time resolved + temperature fit

The sinusoidal method uses a heat transfer coefficient, which is set to 15000 W/m^2K for the previous problems. In order to make the heat transfer coefficient correspond to the CFD model, the heat transfer itself is used to fit this parameter. The heat transfer distribution maximum is equal to 7.72×10^5 W/m^2 (see Figure 1). The bulk temperature amplitude used in the sinusoidal method is ΔT=36 °C. Using this information, the heat transfer coefficient can be calculated by $a=HF/\Delta T=7.72\times10^5/36=21444$ W/m^2K.

Table 5. Comparison of temp. and stresses for the different models, 25 Hz/α = 21444 W/m^2K

Dimension	Model		T_{max} [°C]	ΔT [°C]	Maximum σ_{eqv}[MPa]
1D plate	Analytical bending	Sinusoidal	7.9	13.7	32.0
1D tube	Analytical		7.9	13.7	31.5
1D	ANSYS		7.6	13.2	26.5
2D	ANSYS		7.6	13.2	34.7
3D	ANSYS		7.6	13.2	35.5
3D CFD coupling	ANSYS / Fluent		18	-	77

The heat transfer coefficient α=21444 W/m^2K is used to calculate the temperatures and stresses by means of the sinusoidal method. The results of the comparison are shown in Table 5. This table shows that the temperature and stress in the CFD model is much higher than that of the sinusoidal method.

The higher temperature requires further attention, since the heat flux is correlated to the CFD model, where the temperature distribution is originating. In order to obtain a temperature rise, a certain heat flux is required during a given period of time. This process is about the same for the CFD model and the sinusoidal method. Therefore, a closer analysis is performed on the CFD temperature results. The CFD temperatures with details through the wall are in fact a fluctuation, since the temperature distribution is calculated by the temperature differences in time. It appears that the temperature rise is located in a single cell at the inner wall of the CFD model. Therefore, the numerical determination of the temperature value is important. It has been discovered that the cell centred and nodal values of the temperature are very different. When the nodal value at the fluid/structure interface is taken as the mean between the fluid and structure cell, the temperature is not correct since fluctuations can be much faster in the fluid domain. This phenomenon is strongly related to the transient character of the problem since the timescales in the fluid are different from the time scales in the solid. Further is ongoing to solve this problem.

7 CONCLUSIONS

The phenomenon thermal fatigue is investigated here in a mixing tee. Turbulent mixing of hot and cold fluid flows is investigated as the cause of the thermal fatigue phenomenon in this mixing tee. The configurations are used in nuclear reactors where thermal fatigue is a safety related phenomenon. The pipe wall downstream of the mixing tee faced temperature fluctuations and gradients, which cause thermal stresses, which fluctuate. When the amplitude of these thermal stresses is large enough the wall can suffer from fatigue resulting in crack initiation and growth in the wall.

The Fatherino model is used as a case study. Computational Fluid Dynamics (CFD) is used to determine the influence of local temperature gradients on the pipe wall. The results from the CFD calculation are transferred to a finite element code, where the stresses due to the temperature gradients are calculated. Since this approach is time consuming in terms of preparation and computer time, the differences and correspondences with the one-dimensional model are investigated. The analysis systematically considers 1D, 2D and 3D simulation techniques as a means of assessing transferability. This approach gives insight into the model simplifications and its influence on the results.

Whilst details of the CFD analysis have yet to be fully resolved, it is shown that the one-dimensional simplified models can conservatively predict the fatigue behaviour of a three dimensional model.

8 REFERENCES:

1. Church J.M., "Thermal Fatigue During Turbulent Mixing, Implementation and Case Study Investigation of the Sinusoidal Method", NRG report 21858/06.76729/P, 2006.
2. Schippers, M., "D10: Numerical Analysis of thermo-hydraulic tests", THERFAT R(13)/D10/2003, 2005.
3. Germano, M., Piomelli, U., Cabot, W.H. "Dynamic Subgrid Scale Eddy Viscosity Model", Physics of Fluids A, Vol. 4, No. 3, 1991.
4. Chapuliot, S., Gourdin, C., Payen, T., Magnaud, J.P., Monavon, A., "Hydro-thermal-mechanical analysis of thermal fatigue in a mixing tee", Nuclear Engineering and Design 235, 2005.
5. Dahlberg M. et al., "Development of a European procedure for assessment of high cycle thermal fatigue in light water reactors", Final report of the NESC-Thermal Fatigue project, NESCDOC (06)004, 2007.