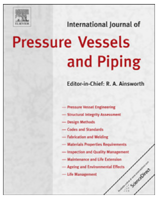
International Journal of Pressure Vessels and Piping 120-121 (2014) 152–161

Contents lists available at ScienceDirect

International Journal of Pressure Vessels and Piping

journal homepage: www.elsevier.com/locate/ijpvp



Integrated structural analysis tool using the Linear Matching Method part 2 – Application and verification

Haofeng Chen [a,*], James Ure [a], David Tipping [b]

[a] Dept of Mechanical and Aerospace Engineering, University of Strathclyde, 75 Montrose Street, Glasgow G1 1XJ, Scotland, United Kingdom
[b] Central Engineering Support, EDF Energy Nuclear Generation Ltd., Barnwood, Gloucester GL4 3RS, United Kingdom

ARTICLE INFO

Article history:
Received 29 January 2014
Received in revised form
2 May 2014
Accepted 7 May 2014
Available online 23 May 2014

Keywords:
Linear Matching Method
Load carrying capacity
Shakedown limit
Ratchet limit
Steady state cycle
Pipe header

ABSTRACT

In an accompanying paper, a new integrated structural analysis tool using the Linear Matching Method framework for the assessment of design limits in plasticity including load carrying capacity, shakedown limit, ratchet limit and steady state cyclic response of structures was developed using Abaqus CAE plug-ins with graphical user interfaces. In the present paper, a demonstration of the use of this new Linear Matching Method analysis tool is provided. A header branch pipe in a typical advanced gas-cooled reactor power plant is analysed as a worked example of the current demonstration and verification of the Linear Matching Method tool within the context of an R5 assessment. The detailed shakedown analysis, steady state cycle and ratchet analysis are carried out for the chosen header branch pipe. The comparisons of the Linear Matching Method solutions with results based on the R5 procedure and step-by-step elastic–plastic finite element analysis verify the accuracy, convenience and efficiency of this new integrated Linear Matching Method structural analysis tool.

© 2014 Elsevier Ltd. All rights reserved.

1. Introduction

Many engineering structures and components subjected cyclic thermal and mechanical loads experience alternating plasticity leading to low cycle fatigue (LCF) or ratchetting which results in an incremental plastic collapse. The evaluation of the LCF, shakedown and ratchet limits have been researched and modelled extensively by plasticity theorists, materials scientists, mathematicians and engineers. Cyclic plasticity is a complex problem and in recent years significant advances have been made in characterising different responses.

Incremental Finite Element Analysis provides a powerful tool to simulate the elastic–plastic behaviour of structures subjected to a specified load history. This allows investigation of any type of load cycle but also requires significant computer effort for complex 3D structures. In addition, this approach does not predict a shakedown or ratchet limit, it simply shows whether elastic shakedown, plastic shakedown or ratchetting occurs. To calculate the specific shakedown or ratchet limit, a significant number of simulations at different load levels are required to establish the boundary between shakedown and non-shakedown behaviours. The designer ideally requires a shakedown/ratchet analysis method that (i) can be applied efficiently to complex 3D geometry under complex thermo-mechanical loading, (ii) only requires readily available computing facilities and (iii) unambiguously specifies shakedown and ratchet limits.

Hence adopting both the upper and lower bounding theorems [1,2], direct methods [3–8] have been developed to directly address the limit load, shakedown and ratchet limits required in a design situation. However, shakedown and ratchet analyses are often difficult to incorporate in a design process. Typically these advanced direct methods require specialist programs that are not available or supported commercially and the computing required to analyse practical structures is extensive and often impractical. In the absence of a robust and practical plastic analysis method, design for shakedown in practice is still based on simple solid mechanics models incorporating design factors sufficient to ensure an adequate “margin of safety” against ratchetting is present [9]. This often leads to excessive conservatism in a design, with obvious technical and economic implications.

In recent years, on the basis of previously developed non-linear programming techniques [10,11], the Linear Matching Method (LMM) [12–18], has been developed to generate approximate inelastic solutions for the steady cyclic state, and to answer specific design related issues with great efficiency and flexibility using

---

DOI of original article: http://dx.doi.org/10.1016/j.ijpvp.2014.05.004.

* Corresponding author. Tel.: +44 1415482036.
E-mail address: haofeng.chen@strath.ac.uk (H. Chen).

http://dx.doi.org/10.1016/j.ijpvp.2014.05.005
0308-0161/© 2014 Elsevier Ltd. All rights reserved.

standard finite element codes. It has been demonstrated that LMM has both the advantage of programming methods and the capacity to be implemented easily within a commercial finite element code, Abaqus [19]. The LMM provides a general-purpose technique for the evaluation of shakedown and limit loads, ratchet limit, plastic strain range for the low cycle fatigue (LCF) assessment associated with a steady state cycle.

To enable widespread adoption of the LMMs in industry, an integrated software tool is further developed to not only removes the requirement for manual subroutine alterations, but also provide additional functionality for subsequent life assessment calculations. In an accompanying paper [20], this new integrated structural analysis tool using the LMM framework for the assessment of load carrying capacity, shakedown limit, ratchet limit and steady state cyclic response of structures was presented, and this new software tool will serve two functions.

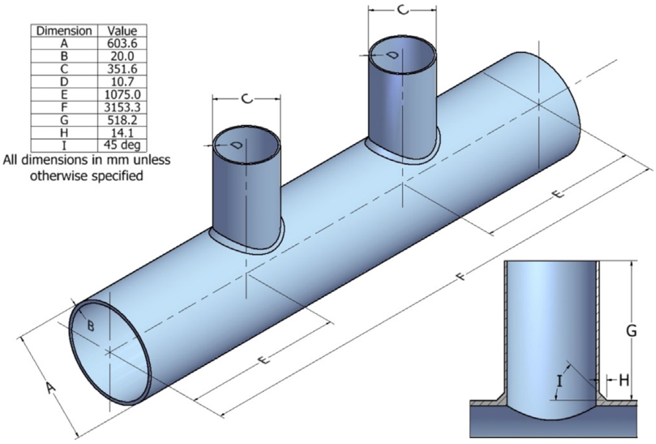
The first is to provide an appropriate Graphical User Interface (GUI) to the LMM, giving the industrial engineer an intuitive method for both inputting the data required for analysis and using results for subsequent analysis calculations. The pre-processor function of the GUI will be used for selection of analysis type, gathering load cycle data and conversion of the finite element model into a form required for the LMM analysis. Submission of the model for analysis from this pre-processor will automatically initiate the calculation procedure using the FORTRAN subroutines. Upon completion of the calculations, the GUI will then manage post-processing utilities for life assessment calculations of the structure beyond those available in commercial finite element software. The second function of the software tool is to use information given in the pre-processing function to automatically handle the required subroutine code changes according to the desired analysis type. Removing the need for the user to alter subroutines removes the possibility of human error in this task and helps its adoption by users who are accustomed to existing commercial finite element software.

The aim of the present paper is to demonstrate this new LMM software tool including practical application and verification through a header component typically used in an advanced gas-cooled reactor (AGR) power plant. In the cold reheat system of the AGR, it was required to demonstrate sufficient margin against ratchetting for the secondary header tees. Proof of shakedown in Ref. [21] proved problematic during the integrity assessment which makes this an ideal example for the demonstration of this LMM software tool. In the present paper, the important aspects of the background to the analysis conducted are summarised first, and then followed by a description of the finite element (FE) model. The analyses conducted in Ref. [21], which are based on the R5 procedure and elastic–plastic calculations, are described in Section 4. The setup and submission of the LMM analysis of the header is presented in Section 5, followed by a comparison of results with the R5 and incremental elastic–plastic finite element analysis (FEA) results.

2. Problem background and description

A schematic of such a header is shown in Fig. 1, where the main pipe has two parallel branch pipes. There are a number of these secondary headers in the system. They have all been designed with the same wall thicknesses, but two variations exist with regards to the distance between two branch pipes.

Non-destructive testing (NDT) was performed on a number of headers to determine current wall thicknesses. This inspection showed a significant variation in these wall thicknesses, where the minimum main and branch pipe thicknesses were found to be 20 mm and 10.7 mm respectively. It should be noted that these minimum thicknesses were not observed in the same header.



Dimension	Value
A	603.6
B	20.0
C	351.6
D	10.7
E	1075.0
F	3153.3
G	518.2
H	14.1
I	45 deg

All dimensions in mm unless otherwise specified

Fig. 1. Geometry and dimensions of header.

In order to prove shakedown in all of the headers whilst keeping the number of analyses to a minimum a worst case model was created. The minimum wall thicknesses observed from the NDT of all the headers were used in this model despite their occurrence in different headers. This gives an inherent conservatism in the model.

This worst case model also considered the possibility of an interaction between two branch pipes. There are two header geometries, the difference between them being the dimension *F* (3153.3 mm and 4169.3 mm) in Fig. 1. It was shown in Ref. [21] that the smaller of these two designs could show an interaction of stresses between two branches whereas the larger design would not. Therefore as a conservative approach the smaller branch geometry was used.

The design conditions of the header are an internal pressure of 4.55 MPa, which is limited by a safety relief valve upstream of the header, and a temperature of 382.2 °C. The analysis assumes that the pipework operates between two relatively steady state conditions of cold shutdown and hot pressurised, which was confirmed by plant temperature and pressure data. Therefore no cold-pressurised or thermal shock conditions are considered.

In addition to the pressure and temperature, headers experience bending moments due to interaction with the rest of the piping system. The applied bending moments at the cold shutdown and hot pressurised conditions were analysed using the pipe stress analysis software PSA5 [22] for the entire cold reheat piping system. There was a variation in bending moments seen across all the headers in the system, and so the worst case bending moments were chosen as a conservative option for this model.

3. Finite element model

3.1. Geometry

The dimensions of the header geometry used are shown in Fig. 1, and the model and mesh are created to match that of [21] as closely as possible. The weld is modelled as a 45° chamfer with a leg length of 14.1 mm. This gives a weld cap dimension of 20 mm, which was the minimum observed in the inspection data. Although symmetry exists in this geometry, the applied bending moments are not symmetrical. Therefore symmetry could not be used.

The FE model is meshed with the Abaqus quadratic brick element C3D20R, as shown in Fig. 2a. The mesh is biased to be denser in the region of the intersection and weld, resulting in a total

Download English Version:

https://daneshyari.com/en/article/787807

Download Persian Version:

https://daneshyari.com/article/787807

Daneshyari.com