FINITE ELEMENT ANALYSES OF AN INSTRUMENT GUIDE TUBE WITHIN A SUPERHEATER HEADER SUBJECT TO CYCLIC PRESSURE AND THERMAL TRANSIENT LOADINGS

Jinhua Shi, Pritesh Mistry

AMEC, Clean Energy - Europe

ABSTRACT

The boilers at Advanced Gas-Cooled Reactor (AGR) nuclear power stations in the UK are potentially life-limiting components. Enhancing the capability to monitor the boilers has been identified as having the potential to provide key evidence in justifying the extension of the generating lifetime of the stations.

It has been proposed to install new temperature measuring instrumentation to monitor reactor gas temperature and to provide additional data regarding boiler operating conditions. The modification will be to introduce thermocouples to the bore of an intact boiler tube to facilitate temperature measurement at or near to the locations of interest.

The modified boiler tube will be sealed at the feed header upstand. Between the upper surface of the superheater header tubeplate and the wall of the superheater header, the thermocouple bundle and sheath will be contained within a rigid stainless steel guide tube. The guide tube will be attached at both ends by welds, each forming a pressure boundary. At the tubeplate a weld will separate the bore of the sealed carrier tube from the steam space within the superheater header; a weld between the guide tube and the superheater header will separate the steam space within the superheater header from atmosphere outside the header.

In order to rank different design concepts, three 3-dimentional finite element models of the guide tube with the relevant pressure boundary components have been created using ABAQUS CAE. Quarter symmetry is used for all 3 designs to reduce computational size and time.

A series of pressure, boiler start-up and boiler trip thermal transient finite element stress analyses have been carried using ABAQUS to provide stress values for structural integrity assessments to be conducted using ASME III, Subsection NH and R5.

Boundary conditions have been applied appropriately to prevent artificial stiffening of the models. Constraints are applied at the symmetry planes and at the base of header in the translational direction normal to each plane to model symmetry.

The finite element mesh has been created from the geometry from approximately 18,000 quadratic 20-noded structured solid brick reduced integration elements (type C3D20R) for the elastic stress analyses. For the thermal transient analyses, the elements are set to heat transfer type (type

DC3D20). Refined mesh is applied at localised regions of interest for greater accuracy.

For the thermal transient analyses, the thermal transient temperatures are applied to the steam side and gas side inner and outer surfaces of the carrier tube with appropriate heat transfer coefficients (HTCs) applied to surfaces exposed to steam and gas. Thermal conductance defined between faces in contact to model heat transfer via conduction. Time periods of high temperature gradients are used for the thermal stress analyses as these can give rise to peak thermal stresses.

Furthermore, contact surfaces have been modelled where appropriate.

In this paper, the detailed finite element models will be presented. Detailed stresses obtained from all pressure and thermal transient loadings will be shown. After a detailed discussion on stresses calculated for each design, the final instrument guide tube design can be recommended from the stress analysis viewpoint.

SUGGESTED THEMES

Finite element, design, superheater header, pressure, thermal transient, heat transfer coefficient, boundary condition, contact.