

Stress Sensitivity

Stress analyses evaluation is undertaken on many pipelines. This is performed with a combination of hand calculations and computer software. The first step on any stress analysis is to assess the complexity of the analysis by collecting all the appropriate inputs. Most stress analyses start with basic stress equations which give an order of magnitude to the forces and displacements. For the Corrib pipeline two different types of software were used due to the levels of complexity of the operational and the hydrostatic pressure test cases.

Inputs

There are three basic types of input data in a stress model which are boundary conditions, soil data and design data. Boundary conditions are the inputs used to create the model. These can include items such as anchor blocks and, for the Corrib onshore pipeline a tunnel. Soil data is taken from geotechnical reports. The soil conditions are interpreted and the worst case scenario is chosen and if required multiple soil information cases are used. This soil data is then put into a set of equations to create the soil spring stiffness which is derived from the American Life Alliance (ALA) Guidelines for Design of Buried Steel Pipe.

The design data is the basis for the model analysis. This includes the internal pressure, installation temperature, minimum temperature, maximum temperature etc. This information forms the basis of the evaluation "cases" which will be analysed. The stress engineer will use hand calculations and experience to develop the most onerous cases. This is where most of the sensitivity of the model and the effect of stress in the pipeline is established.

Caesar II Software

Caesar II is a software created for calculating stress in piping systems. It was first released in 1984. The latest version is 5.2. This is an industry recognised software tool which has been used to for both above and below ground pipe work together with pipelines. This program uses beam equations and is suitable for low deflections and linear (elastic) stress models.

Finite Element Analysis Software (Abaqus)

Abaqus is an adaptable finite element analysis software which is used to simulate pipe segments opposed to beam members. It has the added ability of working within the linear (elastic) and non-linear (plastic) stress ranges. Within the linear range of the program it will produce the same results as Caesar II. This software was used for modelling the hydrostatic pressure test case in the tunnel to establish if the pipeline entered the non-linear range.

Software Validation

Both software packages have been used for many years in the oil and gas industry. The predicted stresses, strains and deflections have been validated against real world behaviour of pipeline systems and confirmed to give accurate results. These programs were originally built on accepted engineering equations but have since undergone industry tests to improve the modelling capability. Both software packages provide support contacts to resolve any modelling issues. Thus any errors introduced by the software package is negligible compared to the accuracy of any input data. Design factors used within the pipeline codes include allowances for these tolerances and uncertainties.

Stress Sensitivity

Modelling Process

For each stress analysis performed by the stress analyst, a complete validation is completed by a second engineer. This process involves checking the entire analysis from start to finish which include:

- assessing initial hand calculations
- verifying input data selected
- review of the model construction
- checking of model inputs
- review of sensibility check selected and their results
- review and checking of reports ensuring that a clear presentation is made for the complex analysis

JP Kenny considers that the stress analysis methodology adopted for the Corrib pipeline achieves the required accuracy to meet the code requirements.