Procedure for Performing Stress Analysis by Means of Finite Element Method (FEM)

Colaboração dos engºs Patrício e Ediberto da Petrobras

1. Objective

This Technical Specification sets forth the minimum requirements for the supply of a mechanical and/or thermal stress analysis report by means of the Finite Element Method (FEM).

2. Scope

2.1. Whenever a Finite Element Analysis (FEA) is required, the stress analysis report shall be submitted to PETROBRAS for approval. The supplier responsible for the stress analysis report is hereinafter referred to as DESIGNER.

2.2. The Finite Element Analysis (FEA) shall be performed according to Part 5 of ASME Section VIII Division 2, hereinafter referred to as ASME VIII-2. For parts of the equipment designed according to ASME Section VIII Division 1, hereafter referred to as ASME VIII-1, the allowable stresses to be used in the stress analyses shall be obtained from Tables 1A and 1B of ASME Section II Part D, hereafter referred to as ASME II-D.

3. Stress analysis report content

The stress analysis report shall include, but is not limited to, the following items namely:
3.1- Analysis Description
3.2- Reference Documents
3.3- Analysis Data
3.4- Type of Analysis performed
3.5- Finite Element Model
3.6- Results
3.7- Results Review
3.8- Conclusions
3.9- Recommendations
3.10- References
3.11- Annexes

3.1. Analysis description

The DESIGNER shall describe:
- The equipment and/or component part to be analyzed, which may be showed by means of drawing or sketch;
- The reason for using FEA, justifying why an analytical analysis is not satisfactory;
- The structural-mechanical behavior of the component part analyzed;
- The types of analysis performed;
- All loads and load case combinations considered in the analysis;
- The name of the commercial finite element software used and its version.

3.2. Reference documents

All reference documents used in the analysis shall be listed, such as:
- The equipment Construction Code and Addenda;
- The drawings or data sheets (indicating the revision) used to build the geometry of the finite element model and to acquire the materials properties at operation and design conditions;
- Other sources of information used to elaborate the finite element model and/or to set the check conditions, e.g., data sheets with external loads on nozzles, standards, catalogs, fax messages, e-mails etc., shall be included in the Annexes.

3.3. Analysis data

3.3.1. The following input data (when applicable), shall be summarized in a table, with their values and source of information:
- Operation pressure, design pressure and MAWP (Maximum Allowable Working Pressure), or the pressurization curve;
- Temperatures: initial (Summer or Winter), operation and design, or the transient temperature curve;
- Upset conditions of temperature and/or pressure, informing the holding time;
- Static head from liquid or bulk materials (e.g. catalyst);
- Liquid specific gravity;
- Loads in all different analysis conditions including those that are a function of time;
- Wind loads;
- Loads on nozzles;
- Corrosion allowance;
- Material specification, including refractory linings and thermal insulation;
- Allowable stresses from ASME II-D (room temperature and evaluation temperature);
- Elasticity modulus (room temperature and evaluation temperature);
- Yield strength (room temperature and evaluation temperature);
- Ultimate tensile strength (room temperature and evaluation temperature);
- Joint efficiency;
- Material fatigue curve;
- Thermal expansion coefficients (varying with temperature when applicable);
- Film coefficient (varying with temperature when applicable);
- Internal and external heat transfer coefficients, with the respective bulk temperatures;
- Poison's ratio.

3.3.2. A table with the external loads for each load case shall be presented.

3.3.3. For nonlinear analysis, the material property model selected shall be justified, and the commercial package software commands and parameters values used shall be described.

3.3.4. For linear elastic analysis a table summarizing all allowable limits for primary and secondary stresses shall be elaborated to check every load case conditions evaluated.

3.3.5. If the refractory or insulation weight is considered by the use of equivalent density for the metal, a table with these values shall be elaborated.

3.4. Types of analyses performed
3.4.1. The DESIGNER shall describe all types of analyses performed, justifying why they were chosen and shall explain all conceptual simplifications and hypothesis considered.

3.4.2. For load cases with temperature gradients, a thermal analysis is required to obtain the temperature profile resulting from either axial or through-wall gradients on the component part evaluated in order to perform a coupled structural-thermal analysis.

3.5. Finite element model
3.5.1. The DESIGNER shall describe the finite element model elaborated and its dimensions, justifying and explaining all the simplifications hypothesis, disregarding and assumptions adopted in the geometry creation, such as: symmetry, axisymmetry, plane stress, plan strain, etc. Furthermore, it is necessary to explain the behavior hypothesis considered, like contact conditions/type, constraint equations, equivalent heat transfer coefficients etc.

3.5.2. The DESIGNER shall list all types of elements used and their names on the software elements library. Each type of element chosen shall be justified.

3.5.3. There shall be assembled a table containing the description of the model, with the following columns:

- Name given to each component of the model;
- Type of element adopted for each component and its reference number in the model;
- Reference name in the software elements library;
- Real Constant defined to each component (when applicable) and its reference number in the model;
- Component material and its reference number in the model;
- Number of elements of each component.

3.5.4. There shall be assembled a table containing the Real Constants (when applicable), with the following columns:
- Reference number of each real constant associated with the component;
- Thickness, in case of shell elements;
- Cross-section properties, in case of beam elements;
- Location of each real constant associated to the component.

3.5.5. There shall be assembled a material table, with the following items:
- Reference number of the component material;
- Material specification;
- Modulus of Elasticity as temperature function (when applicable);
- Other properties (thermal expansion coefficient, thermal conductivity, etc.), as temperature functions (when applicable).

3.5.6. There shall be presented at least the following graphical displays (color contour plotting) of the geometry model:
- The elements with the associated materials;
- The elements with their associated types;
- The elements with the associated real constants;
- The areas showing their normal, in case of shell elements model;
- The elements mesh, showing the refined regions.

3.5.7. The DESIGNER shall inform the elements mean size at regions of interest and the complete description of the mesh refinement criteria used, justifying the quality of the mesh discretization, showing the energy error within each finite element and expressing this error in terms of a global error energy norm.

3.5.8. Applying the Loads
3.5.8.1. All applied loads and load combination in each load-step evaluated shall be listed, including those that are a function of time.

3.5.8.2. It is necessary to describe and justify how the following loads are applied to the finite element model:
- Pressure;
- Temperature and/or temperature gradient;
- Wind loads;
- External loads on nozzles;
- Weight of internal or external parts;
- Pressure end load applied on the edge of the model to simulate the longitudinal stress, when applicable;
- Concentrated loads;
- Other surface loads (ex.: convection, heat flux);
- Inertia loads such as gravitational acceleration, angular velocity and angular acceleration.
- Imposed displacements.

3.5.8.3. The DESIGNER shall depict graphical displays (color contour plotting) to check how the above-mentioned loads are applied to the model.

3.5.9. Applying Boundary Conditions

3.5.9.1. The DESIGNER shall justify all boundary conditions (degree-of-freedom constraints) adopted in each load case, explaining how they were applied to the model.

3.5.9.2. The symmetry and antisymetry (when applicable) degree-of-freedom constraints on the model shall be justified, and the graphical displays (color contour plotting), showing the regions, planes and faces of symmetry/antisymetry shall be presented.

3.5.9.3. If the analysis considers the contact between parts, describe the type of contact used and describe the behavior expected for the model.

3.5.9.4. If constraints equations for coupling degrees of freedom are used, describe how it was used (methods of generating constraint equations) and justify the behavior expected for the model after loading.

3.5.9.5. Graphical display (color contour plotting) of the applied boundary conditions and degree-of-freedom constraints shall be provide to check if they were applied properly.

3.5.10. In case of nonlinear or transient analysis, the DESIGNER shall describe all control settings defined, such as: type of equation solver, the use of small or large displacements, prestress effects, iteration method, number of load-steps, number of iterations, loading application conditions (stepped or ramped), convergence criteria, termination criteria, etc.

3.6. Results

3.6.1. The DESIGNER shall present graphical displays (color contour plotting) of the deformed shape of the model for each load-step, comparing and commenting each deformed shape plots with its expected appearance with regard to the direction of application and type of loading for each load-step. The plots legend must show the percent error in structural energy norm (SEPC), which indicates the quality of the mesh refinement, and its value shall be less than 10% in the regions of interest.

3.6.2. The DESIGNER shall check if the reaction loads at constrained nodes (support reactions) for each load-step are consistent with the applied loads. The sum of the reaction loads in each direction shall be equal to the sum of the applied forces in that direction. There shall be searched for the existence of unexpected reaction loads not complying with the directions of the applied loads.

3.6.3. The Designer shall present for each load-step graphical displays (color contour plotting) of displacement UX, UY e UZ, when applicable, in an adequate plotting coordinate system. The points to be considered relevant in the plots of displacement results demonstrating the compliance with the loadings applied shall be identified and commented. The maximum displacements obtained shall be checked in order to satisfy their defined allowable values (when applicable).

3.6.4. The DESIGNER shall present for each load-step graphical displays (color contour plotting) of membrane and bending stress distributions for SX, SY, SZ, S1, S2 and S3 in an adequate plotting coordinate system. The points to be considered relevant in the plots of displacement results demonstrating the compliance with the loadings applied shall be identified and commented.

3.6.5. The DESIGNER shall demonstrate that the results of FEA performed comply with simplified analytical calculation, data based on experimental studies or data in the literature.

3.6.6. The DESIGNER shall present for each load-step graphical displays (color contour plotting) of von Mises equivalent stress (membrane and bend) distribution, categorizing the stresses according to ASME VIII-2, in case elastic stress analysis method. The maximums shall be indicated and compared to associated limiting values.

3.6.7. In case of transient analysis, used to determine the dynamic response of a structure under the action of any general time-dependent loads, the DESIGNER shall present time-varying graphs of stress, displacements, forces and temperature (when applicable) of specific locations (nodes) in the model. The chosen nodes shall be justified and the time-varying graphs commented. The maximums shall be indicated and compared to the associated limiting values.

3.6.8. When fatigue analysis is required, it shall be made on the basis of the number of applied cycles of a stress or strain range at specific locations (nodes) in the component. The controlling stress for the fatigue evaluation is the effective total equivalent stress amplitude calculated for each cycle in the loading histogram. The chosen nodes shall be justified, the cycle counting methods explained, and the controlling stress range commented.

3.6.8.1. The DESIGNER shall check whether the directions of the principal stresses at the point being considered change during the stress cycle.

3.6.8.2. The designer shall present time-varying graphs of the component stresses at specific locations (nodes) evaluated.

3.6.9. There shall be assembled tables summarizing the results for each load step, including the following columns:
- Name given to the component part;
- Material;
- Temperature;
- Stress categorization;
- Allowable limits;
- von Mises equivalent stress.

3.7. Results review
Each load case results shall be commented and analyzed comparing outputs with the expected results.

3.8. Conclusion
The conclusions of the stress analysis performed for all load cases shall be reported.

3.9. Recommendations
The recommendations relevant to the analysis shall be informed.
If the analysis performed shows that it is necessary to reinforce some component part, increase thicknesses or reduce/limit some type of loadings, this modifications shall be clearly informed and point out.

3.10. Bibliographic references
All reference sources used in the stress analysis report, such as, articles, books etc. shall be informed.

3.11. Annexes
All documents containing data used in the FEA report, such as: data sheets with external loads on nozzles, excerpts of standards, excerpts of catalogs, fax messages, e-mails, shall be attached.