CASE STUDY /

Ansys + FMC Technologies

"Ansys Parametric Design Language's superior customization capabilities in combination with Workbench Scripting Capabilities provide for visualization of damage. This provides critical engineering insights and enables product design of subsea oil and gas equipment to be qualified against ASME BPVC VIII, Division 3."

Abhineet Gupta
Engineer II | PSG-FEA / FMC Technologies India Pvt. Ltd.

CASE STUDY

Design of subsea oil and gas equipment has moved to high pressure and high temperature domains. With pressures beyond 15,000 psi and temperatures beyond 350 F, designs have to be qualified per ASME BPVC VIII, Division 3, which requires the cumulative damage on the equipment to be below 1. Local damage check evaluates the equipment's design against local strain rupture.

/ Company Description

FMC Technologies is a global market leader in subsea systems and a leading provider of technologies and services to the oil and gas industry. We help Exploration and Production customers overcome difficult challenges with onshore and offshore drilling, completion and production technologies and services to improve their returns.

/ Technology Used

- Ansys® Mechanical™
- Ansys® Workbench™

/ Business Challenges

Due to increasing depth of oil extraction, the structural loading requirements for subsea components are increasing, but there are also space and weight constraints on these components. To reduce over-engineering and promote value engineering, engineers must visualize damage distribution in each component. Because Ansys Structural does not have a method to plot damage, the challenge was to develop a customized method using Ansys's scripting capabilities.

/ Engineering Solution

- Ansys Mechanical or Workbench was used to determine loading conditions experienced by the product using FEA.
- Ansys Mechanical macros were developed to evaluate cumulative damage as per ASME BPVC VIII Division 3, Article KD 232.1.
- Ansys Parametric Design Language was used to customize the analysis to provide for damage visualization.
- Ansys Parametric Design Language Macro code was executed; the Macro code wrote an additional .RST file containing the elemental-nodal value of ASME's damage value for each node.
- The effects of the damage value on the components were evaluated.
- If any improvement in product performance was required, engineers made informed decisions based on the damage information in the .RST file.

Casing hanger cumulative damage distribution.

/ Benefits

- Ansys Macro capabilities enabled evaluation of derived results via scripting.
- Results such as principal stresses were extracted from the existing Ansys result files. These were later used as parameters to evaluate limiting tri-axial strain for a material, as per ASME Sec. VIII D3, under a particular loading condition.
- Ansys Parametric Design Language scripting allowed for flexibility to compute derived parameters and evaluate cumulative damage. These results were stored in a separate result file and saved for access.
- Ansys post-processing tools were used to plot damage and identify areas in the design that did not satisfy ASME Sec. VIII D3 criteria.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

Visit www.ansys.com for more information.