KNIGHTHAWK TECH NOTES

PHONE: (281) 282-9200 • FAX: (281) 282-9333
WEBSITE: www.knighthawk.com

March, 2017

Issue 17.01

"Pressure Containment Equipment / Vibration of a Centrifugal Blower"

You have just placed an order for a pressure vessel that operates in high temperature and pressure. The vessel is to be designed and constructed in accordance with ASME Section VIII, Division 1. There is one design problem that must be addressed in the vessel however. Hot and cold streams meet and there is a question about local differential thermal expansion. How is this problem best attacked to ensure there are no stress problems and the code is satisfied? One methodology is as follows:

- A finite element heat transfer model can be developed of the vessel to determine the temperature distribution in the metal. Heat transfer coefficients can be determined via hand calculations or computational fluid dynamics methodology.
- An elastic finite element stress model containing any mechanical loading, thermal loading, and pressure loading can be executed to determine the stresses.
- Classify the stresses in accordance with ASME Section VIII, Division 2, and Part 5. The stresses are classified as primary, secondary, and peak. The stresses must be linearized (For linearized stress, an algorithm is executed within the FEA tool. Although, there is technically a long definition of this, the layman's definition is that it is a methodology that bridges the gap between the advanced analysis methods which calculate stresses and the ASME code) in the areas of interest to separate the primary, secondary, and peak stresses. The classified stresses are then compared to the code allowable in Division 2 of the code.
- If the analysis passes this test the design can be certified to meet the code.

There are some cases where the design does not meet the code under the above elastic finite element analysis. For these cases were warranted a non-linear elastic-plastic analysis can be performed in accordance with the Division 2, Part 5 of the code. With the computing power we have today, it is not unusual to incorporate plasticity effects which would be an elastic/plastic model. In other words, the numerical model will be using plasticity of the metal, modeling close to what actually happens Care should be taken for this approach, in that the designer assumes that he or she has high knowledge of the actual load condition and the material properties. ***No analysis can be more accurate than the input data going into the analysis.***

Case Study: Vibration of a Centrifugal Blower

A multistage blower was just put into service at a new plant. It was tested in the shop at the factory and met performance criteria. The unit was installed in the third story of the petrochemical plant. The blower was tested before the startup of the plant under load at ambient air conditions and the unit performed well. The plant started up for the first time and the blower came on line as expected. To everyone's surprise the unit started vibrating at a damaging level and an inboard bearing failed. The bearing and rotor were replaced and the blower failed again shortly after startup. The process temperatures were over 500 °F.

Rotordynamics studies indicated the blower was operating close to a critical but in theory, but should operate O.K. It was known that the unit operated up in a structure and the unit operated up in a structure and therefore a sensitivity study was conducted with flexible base support boundary conditions. A structural finite element model was built eventually to find the actual support flexibility. After incorporating the flexibility of the structure into the rotordynamics model, results indicated the unit was running within 10% of a critical. But there was also a problem with the differential thermal growth of the unit due to the high temperature application. A finite element model was developed of the base to evaluate the heat transfer and deflection of the unit. It was found that there was a slight distortion in the unit, enough to amplify the vibration problem.

Modifications to the bearing spacing, structure, and unit base were made. The unit was started up and performed well. To solve this problem involved three numerical models; one rotordynamics model, a structural finite element model, and a model of the base. The cost of the numerical models was less than 1% of lost profit due to the plant shutting down. The plant walked away from this problem completely.

Cliff's Notes:

We are excited about 2017 at KnightHawk. Our value proposition for you is that we solve complex problems in industry for both static and rotating equipment. We have been doing this for over 25 years and we have seen a lot. What takes some groups months, can be a matter of days for KnightHawk. Call us and let us explain to you how we can solve your problem.

March madness is here. Hope your team makes it to the final four.

Take care and God Bless,

Cliff Knight

cknight@knighthawk.com

▼ ▼ ▼ ▼ ▼ ▼ ▼ ▼

KnightHawk Project Update

- Rotordynamics of a centrifugal compressor
- Non-linear finite element of a vessel
- Failed tank investigation
- Finite Element Analysis of reactor jacket
- Startup up support
- Brittle Fracture Analysis
- Heat Exchanger Rerate Analysis
- Failure Analysis of Bearing
- Pipe stress analysis of large bore piping system
- Finite Element Heat Transfer of Reactor Refractory
- Gear Box Failure Analysis
- Swing Gate Valve Failure Analysis
- Heat Exchanger Rating evaluation
- Boiler Header Failure Analysis
- Reverse Engineering of Medical Devise
- PMI and Materials Consulting
- Boiler Tube Failure Analysis
- Silo Rerate Analysis
- Failure Analysis of PSV Spring