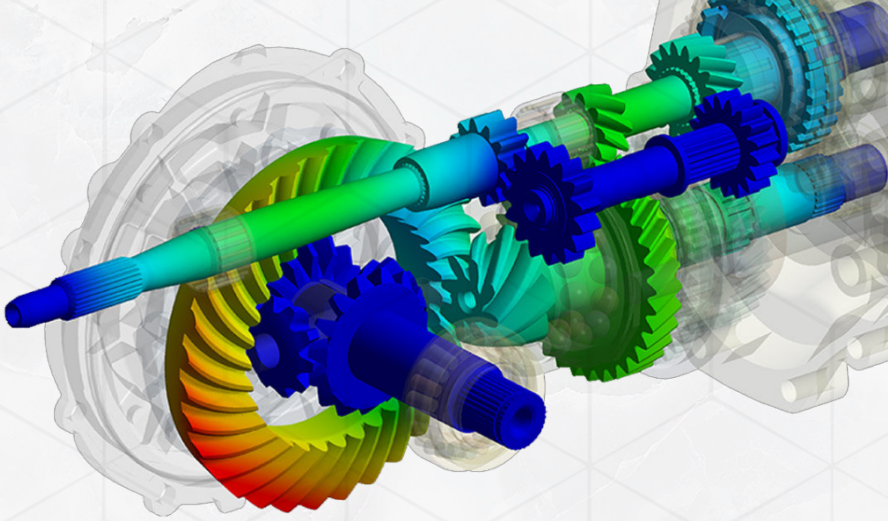

CASE STUDY

FLUIDCODES

SOFTWARE SUPPORT TRAINING CUSTOMIZATION

CONTACT OUR LOCAL OFFICES

UNITED ARAB EMIRATES	+971 4330 8666
SAUDI ARABIA	+966 13 8318182
EGYPT	+971 4330 8666
BULGARIA	+359 88 8813820
UNITED KINDOM	+44 20 3753 4607

sales@fluidcodes.com

consulting@fluidcodes.com

fluidcodes.com

FLUIDCODES

CASE STUDY

NPCC

FEA OF A HORIZONTAL PRESSURE VESSEL

CHALLENGES

In this study, the evaluation of nozzle- shell junction of a horizontal pressure vessel was performed using finite element analysis. High stress at the nozzle to head or shell junction area are common. Due to discontinuity of the geometry, the defect can occur and the junction region can become the weakest point, which will be the source of failure of the whole structure. Thus, a reliable and accurate analysis method for head or shell to nozzle junction is necessary.
The Sub-modeling technique in ANSYS was used for this study to accurately predict the stress concentration. The nozzle load data and internal working pressure loads were considered for the study.

ENGINEERING SOLUTION

The result of the analysis showed high-stress concentration developed at the Nozzle Shell junction due to an abrupt change in the geometry and the consequent change in stress flow. The maximum stress was less than the yield stress of the material.

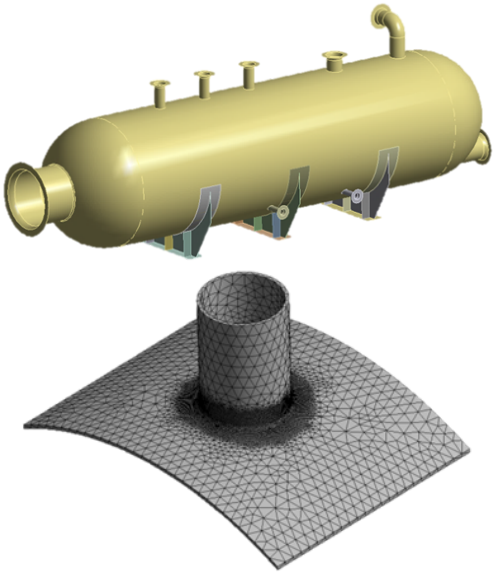

Figure 1. *Geometric Model of pressure vessel and submodel*

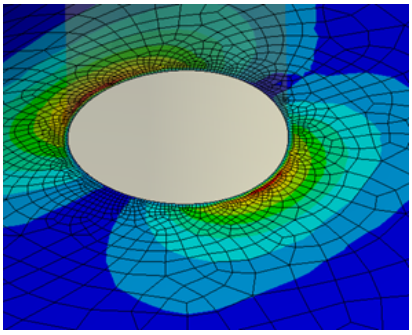

Figure 2. *Stress contour plot of global model*

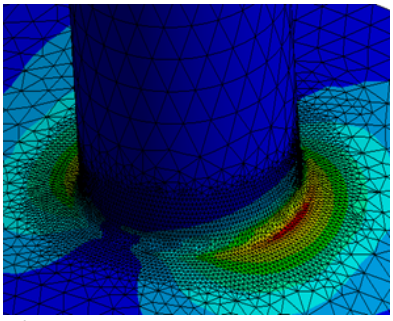

Figure 3. *Stress contour plot of submodel*