ISSN: 2278 – 7798
International Journal of Science, Engineering and Technology Research (IJSETR)
Volume 5, Issue 11, November 2016

DESIGN AND FATIGUE ANALYSIS OF CHOKE VALVE BODY

N. SURESH [1], B. SRINIVASA REDDY[2]

Assistant professor, Geethanjali College of Engineering and technology, Cheeryal, Hyderabad, TS, India.

ABSTRACT

Design and Fatigue Analysis of Choke Valve body component mainly involves in Modeling of Component and Fatigue life Analysis. Fatigue occurs when a material is subjected to repeat loading and unloading. If the loads are above a certain threshold, microscopic cracks will begin to form at the stress concentrators such as the surface, persistent slip bands (PSBs), and grain interfaces. The Choke valve is used in oil and gas production wells to control the flow of well fluids being produced. Another Purpose that the choke valves allow fluid flow through a very small opening, designed to kill the reservoir pressure while regulating the well production. The Choke valve Body Subjected to utmost Working Pressure Conditions , to withstand the body under heavy temperature and pressure conditions , I have opted F22 Material for Fabrication of this Choke valve Body. This report is to evaluate the fracture mechanics and fatigue life of a choke valve body subjected to 5000 N/MM working pressure at 50 °F. The purpose of this study was to determine the number of cycles the body can withstand at working pressure and hydro static test pressure. To analyze the body, the body was first Modeled in CAD with certain dimensions and then designed body was analyzed using Finite Element Analysis software ANSYS.

Keywords: choke valve, F22, Slip band, Elasticity.

I. INTRODUCTION

The component used in this Project is a Choke valve Body. Choke valve is used in oil and gas production wells to control the flow of well fluids being produced. Another purpose that the choke valves allow fluid flow through a very small opening, designed to kill the reservoir pressure while regulating the well production.

Choke valve is a type of valve designed to create a choked flow in a fluid line in an automobile. Choked flow is a compressible flow effect. It means “Choked” or “Limited” is the fluid velocity. When a flowing fluid at a given pressure and temperature passes through a restriction such as convergent-divergent nozzle or Valve into a lower pressure environment the fluid velocity increases.

Figure: 1. Choke valve close position

ISSN: 2278 – 7798
International Journal of Science, Engineering and Technology Research (IJSETR)
Volume 5, Issue 11, November 2016

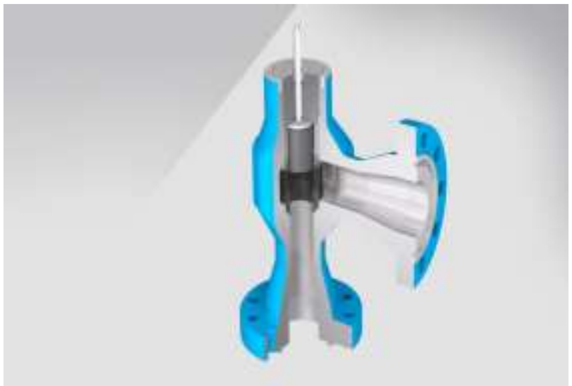


Figure :2. Choke valve open position

II.ABOUT SOFTWARE

The Software used in this Project is (CAD) Computer-Aided-Design for Body design and for Analysis commercially FEA software ANSYS was used for this body simulation. ANSYS is a Finite Element Analysis (FEA) code widely used in the Computer Aided Engineering (CAE) field. ANSYS software allows constructing computer models of structures, machine components or systems, apply operating loads and other design criteria, and study physical responses, such as stress levels, temperature distributions, pressure, etc. The ANSYS program has a variety of design analysis applications, ranging from automobiles to such highly sophisticated systems as aircraft, nuclear reactor containment buildings and bridges. There are 250+ elements derived for various applications in Ansys.

The ANSYS Element used in this Project is second order Tetrahedral Structural Solid (Solid-92)-3-D-10-Node. This Solid 92 has Quadratic displacement be haviour and is well suited to model irregular meshes.

A. METHODOLOGY USED:

Develop a 3D model from the available 2D drawings of Choke valve Body.

The 3D model is created using Computer Aided Design software.

The 3D model is converted into parasolid and important into ANSYS to do static analysis by applying the pressure.

Calculate stresses and deflections of the original model and check if the component is withstanding for the operating pressure.

Under working and hydrostatic pressures application, perform analysis to observe stresses and crack growth on the modified model.

B. INPUT FOR THE PROJECT:

To perform fracture mechanics analyses on a choke valve body. The body would be subjected to 20,000 psi internal working pressure at 450 °F during its operation. During hydrostatic test, the body would be subjected to 1.5 times the working pressure which is equal to 30,000 psi at room temperature (70 °F).

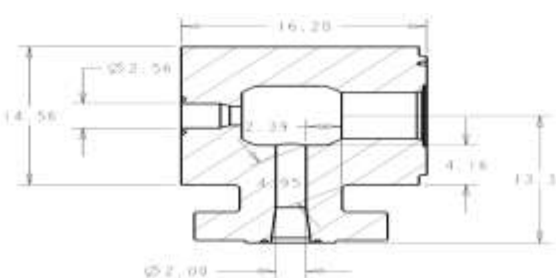


Figure: 3. Optimized studded outlet body design showing the studded outlet and flanged inlet

During the expected life of the valve, closing and reopening operations will be executed and therefore, the valve should withstand high pressure impact cyclic loadings for a certain minimum number of cycles. A highly possible failure mode caused by the cyclic loading, or the low cycle fatigue (LCF) of the material should be taken care for its design.

III FINITE ELEMENT ANALYSIS

A. Working pressure scenario:

Linear-elastic FE analyses were performed to compare the stress distribution in the studded outlet body with the flanged outlet body. Figure 5 shows the comparison of stress intensity distribution for the working pressure scenario. The maximum value in the legend was set to 2/3rd of Sy and with a duration factor of 0.85 to account for 450 °F temperature rating (i.e., 45,333 = 0.444*0.85* 8000).

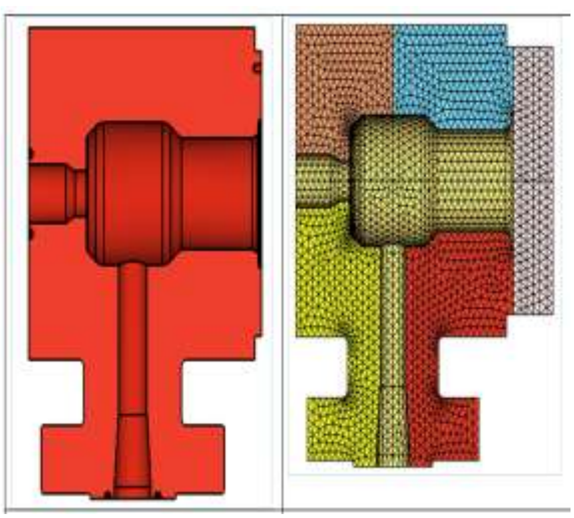


Figure: 4.The simpified geometry(left) and FE mesh used in modeling the working pressure scenario.

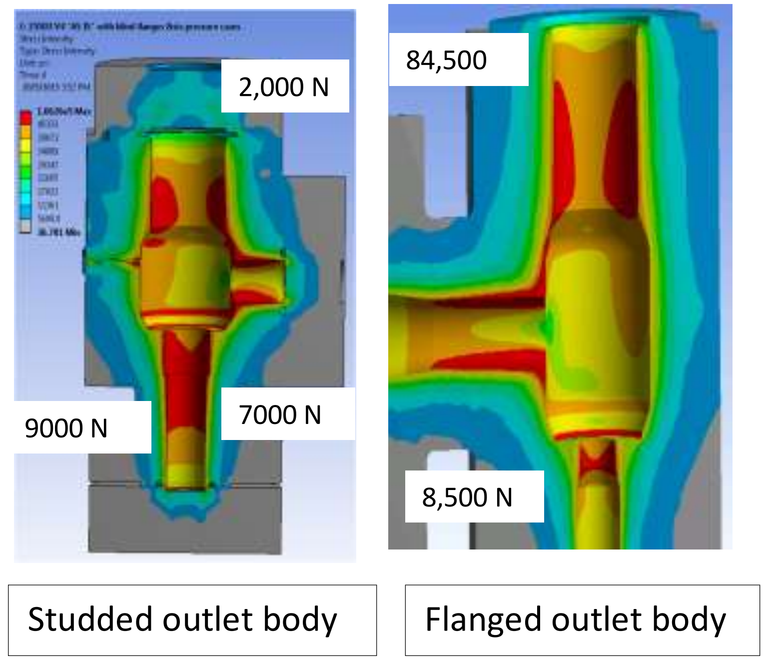


Figure:5. Comparison of stress intensity in the studded outlet body with flanged outlet body showing the similar stress distribution at the inlet, outlet and near intersecting cross bores for the working pressure scenario.

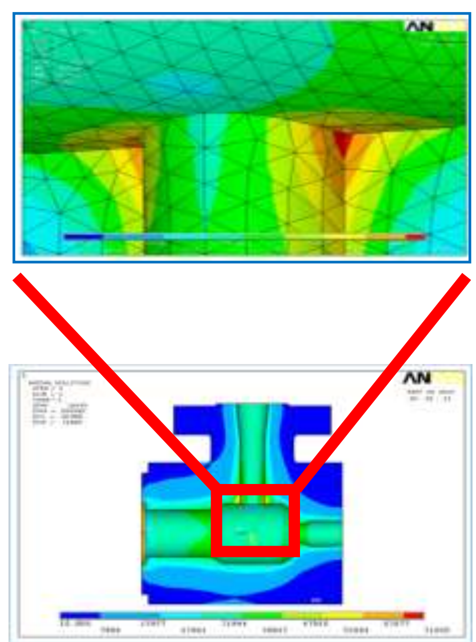
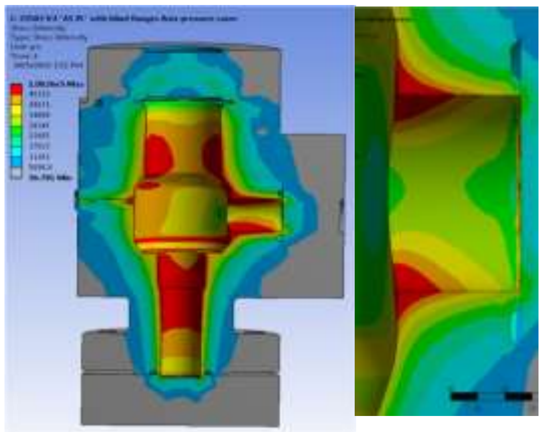


Figure: 6.stress contours obtained from non linear FE analysis in the studded outlet body

ISSN: 2278 – 7798
International Journal of Science, Engineering and Technology Research (IJSETR)
Volume 5, Issue 11, November 2016

showing the high stress location at the intersecting bore for the working pressure scenario.

B. Hydrostatic Test Scenario:



Studded outlet body Flanged outlet body

Figure:7.comparison of equivalent stresses in the studded outlet body with flanged outlet body showing the similar stress distribution at the inlet, outlet and near interesting cross bores for the hydrostatic test scenario.

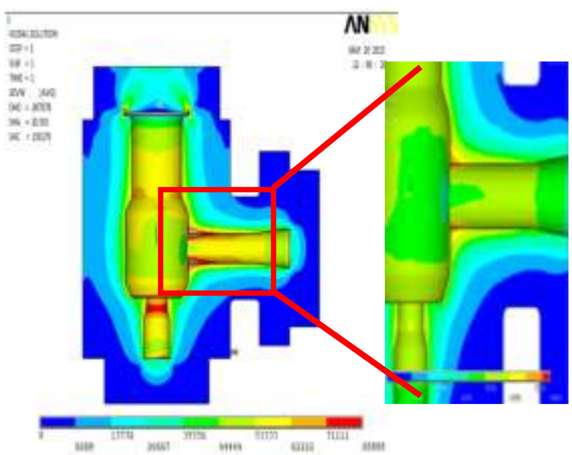


Figure: 8.stress contours obtained from non linear FE analysis in the studded outlet body showing the high stress location at the intersecting bore for the hydrostatic test scenario.

IV. FRACTURE MECHANICS EVALUATION

	Working pressure scenario	Hydrostatic Test Scenario
Fracture toughness kc	95.5	101.98
Critical crack size ac	1.475	1.098
Maximium allowable crack size 25% of ac	0.369	0.275

Table 1 Fracture Mechanics Parameters

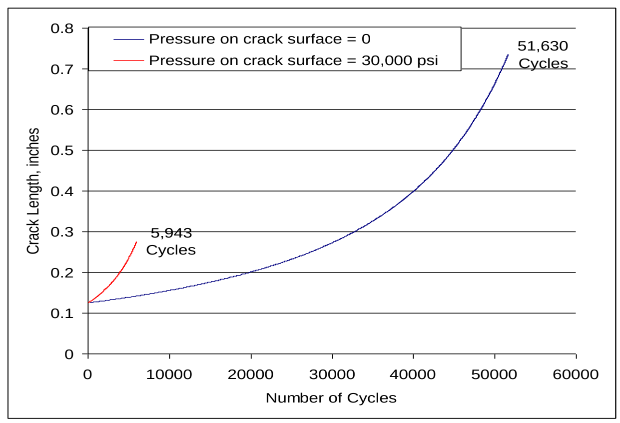


Figure: 9 Number of cycles versus crack growth for hydro test scenario obtained from ASME VII

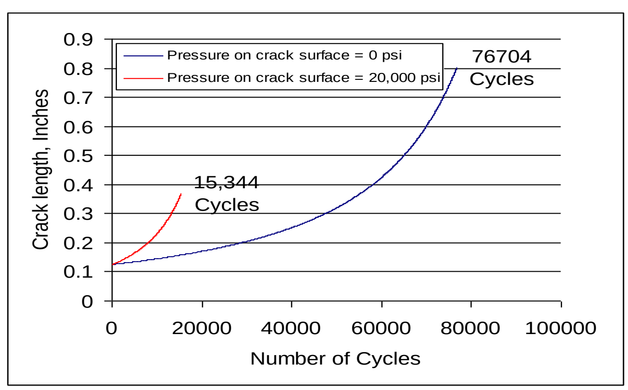


Figure: 10 Number of cycles versus crack growth for working pressure scenario obtained from ASME VII

V. CONCLUSION

The material properties of F22 needed for the Macro were not available from MFV. Therefore, a material that has properties such as yield strength, modulus of elasticity very similar to F22 was used in the Macro. Guidelines from ASME VIII-3 were used to determine the crack shape, initial crack size. These guidelines were also used to calculate critical crack size. However, the critical crack size was also calculated using the Macro as well as ASME VIII-3 method. The results from ASME VIII-3 method were more conservative than those obtained using Macro.

REFERENCES

[1] T. L. Anderson Fracture Mechanics: Fundamentals and Applications, 3rd edition, 2005.

[2] Van Stone, Gooden and Krueger. "Advanced Cumulative Damage Modeling," AFWAL-TR-88-4146 1988.

[3] ASME Boiler & Pressure Vessel Code, Section VIII, Division 3, Rules for Construction of Pressure Vessels, Alternative Rules for the Construction of High Pressure Vessels, 2010 edition, ASME, New York, New York.

[4] Cross-Reference of ASTM Material Specifications Covering Cast and Forged Valves, Fittings, Flanges and Unions

Author 1:

SURESH NUTHALAPTI,

M.Tech (CAD/CAM),

Working as assistant professor in Geethanjali College of Engineering and Technology, Hyderabad. And membership in International Association of Engineers (IAENG) NO: 174373.

Author 2: *B.SRINIVASAREDDY,* M.Tech, Working as a assistant professor in Geethanjali College of Engineering and Technology, Hyderabad.