Design of a Knife Valve and Stress Analysis using Finite Element Analysis

Mr. Ashish Mule
Research Scholar
Department of Mechanical Engineering
Dr. J. J. Magdum College of Engineering, Jaysingpur

Mr. Y. R Patil
Associate Professor
Department of Mechanical Engineering
Dr. J. J. Magdum College of Engineering, Jaysingpur

Pranav Abhyankar
CAE Analyst
Abhyankar Consultancies, Pune

Amit Hingane
Engineer
Brose India Automotive Systems Pune

Abstract

A valve is a device that regulates, directs or controls the flow of fluid by opening, closing or partially obstructing various passage ways. Knife valve is a type of gate valve. These types of valves are commonly used in pulp industries, pharmaceutical industries, where these valves handle slurry type fluids. The advantage of this type valve is easy operation and maintenance. In the world of advanced technology and globalization the organizations are moving towards the optimization using advanced technique. The objective of the paper is to design the knife valve for design condition. The finite element analysis is used to check the design adequacy of the model. The stress analysis of valve is done using finite element software ANSYS 17.

Keywords: Gate Valve, Knife Valve, Finite Element Analysis, Stress Analysis

I. INTRODUCTION

In the world of advanced technology and globalization the organizations are moving towards the optimization using advanced technique. A valve is a device that regulates, directs or controls the flow of fluid by opening, closing or partially obstructing various passage ways they are used in various applications like industrial (oil, gas, power generation, mining chemical manufacturing), military, commercial, residential. Valves are various types having wide range of size and applications.

The design of the valve were previously done using the past design or on the conventional procedures. The development of advanced technique like finite element analysis the use of such techniques was done for optimizing the designs of the valve bodies. Some the research work discussed below. X G Song [1] in his paper Analysis and optimization of a butterfly valve disc discussed about the conventional design for butterfly valve. This valve is used for chemical industries. The CAD model was created for a butterfly valve. Use of CAE commercial software was done for finding the stresses induced on the valve. By using design of experiments the optimum valve of the variables in design were finalized. Dr. K.H. Jatkar and Sunil S. Dhanwe [2] have worked on classical theory and finite element analysis of the gate valve. Finite element analysis carried out by using Ansys software. Stress value of classical and finite element analysis compared and it matches approximately with each other and that can be used for further development of the gate valve. The analysis of knife gate valve was not observed in the literature reviewed till now.

The literature review confirms that use of finite element analysis is useful for the optimization of the design. The analysis was carried out using ANSYS work bench. The CAD model was created using CAD software Pro-E. The element type used for analysis is solid 185. The contacts were established in between different parts.

II. THEORETICAL BACKGROUND OF KNIFE EDGE VALVE

Knife gate valves were originally designed for use of pulp and paper industry. By using a sharp, beveled edge, a knife gate was ideally designed to cut through the chewy pulp encountered in the pulp and paper industries. The benefits of knife gates also include that they are easy to actuate/operate and cheap to produce. As a result, knife gate valves was widely used into numerous other markets, including wastewater treatment, oil and gas, mining, and power, in a short period of time. The main parts of the most usual type of valve are the body and the bonnet. These two parts together form the casing; that holds the fluid going through the valve.

III. METHODOLOGY

The methodology for designing the knife valve is discussed below.
A. Operation of the Valve

These types of valves are commonly used in pulp industries, pharmaceutical industries, where these valves handle slurry type of fluids. The advantage of this type valve is; easy operation and maintenance. The gate plate used in Knife valve, has a shape similar to knife. The stem is attached to the knife gate and yoke sleeve. The stem has a screw threading and yoke sleeve as a tapping done on it. Hand wheel is mounted on the stem. As the hand wheel is rotated the stem moves up and down. As stem is connected to the gate rotary motion is transferred to linear motion of the plate. The valve gets shut off when plate comes down. Seal ring is generally provided to avoid leakage after shut off.

B. Design Procedure for Knife Valve

1) Design of the body
The body is generally made from cast steel or hastelloy. The valve has a flange with internal treading on it. The bore diameter of the body is the primary specification for a valve. The wall thickness of the body decides the strength and weight of the body. As per classification of pressure vessels one can consider body as a thick shell subjected to internal pressure.
Considering Lame’s Equation,
Wall thickness \( t_1 \) can be determined as follows,
\[
t_1 = \left( \frac{d_i}{2} \right) \times \left\{ \sqrt{\frac{(f t + p i)}{(f t - p i)}} - 1 \right\}
\]
Where \( d_i = \) Bore diameter (mm)
\( f = \) Permissible tensile strength
\( p_i = \) Internal pressure (MPa)

2) Design of Valve Stem
The valve stem is simple threaded rod fitted in the yoke sleeve. The gate is attached to this stem, hence lowering or rising of the stem causes motion of the gate valve. The stem should be safe for shear strength. Using general theory of strength of material one can find the safe diameter for applied torque.
As direct stress applied on hand wheel is zero (i.e. \( \sigma = 0 \))
So, \( \tau_{\text{max}} = \tau \)
For safety of hand wheel against shear failure,
\( (S_{sy}) = 0.5 \times S_{yt} = 122.5 \text{ MPa} \)
\( S_{sy} = \) yield strength in shear
\( S_{yt} = \) yield strength in tension
Considering, factor of safety = 3
Allowable torsional stress \( (\tau) = \frac{S_{sy}}{FOS} = 40.88 \text{ MPa} = \tau_{\text{max}} \)
For tensional shear strength from solid mechanics one can say,
\[
\tau_{\text{max}} = \frac{16T}{\pi d^3}
\]
From above relation one finds the value of diameter \( d \) for stem = 14.645 mm rounding up to 16 mm.
Other parts like flange and bolts is done considering the codes and standards. The code ASME B 16.5 (page no. 74) was used for finalizing the required dimensions. The CAD model was created from the dimensions considering the different manufacturing feasibilities and tolerances.

IV. Finite Element Analysis of Knife Valve

The Finite Element Method only makes calculations at a limited (Finite) number of points and then interpolates the results for the entire domain (surface or volume) by using shape function. CAD model was created using CAD software. The 3D CAD model is used for analysis.

Fig. 1: CAD Model
The finite element analysis has following steps;

**A. Pre-Processing**

In this stage the material properties and geometric model is developed. The model is then discretized or meshed using proper type of element. The boundary conditions are applied like constrains, loadings etc.

![Fig. 2: Mesh](image)

![Fig. 3: Boundary conditions](image)

**B. Solution**

In this stage the matrices are generated and solved for the unknown variables.

**C. Post Processing**

In this stage the results are reviewed. The results like stresses and deformation are viewed. The analysis was carried out using ANSYS work bench.

**V. RESULTS AND DISCUSSIONS**

Post-processing is the most important step in analysis. One may require making design decision based on the result. Post processor is used to review the result carefully and check the validity of the solution.

![Fig. 4: Deformation in valve](image)

![Fig. 5: Stress in valve body](image)
From the results obtained from finite element analysis it was observed that deformation in gate is maximum. The values observed in finite element analysis are near to the values obtained from analytical procedure. The deformation is within allowable value. Also stress in the valve body is below yield. Hence one can say that the design is safe. Further there is chance of optimization in valve body by doing different iterations.

VI. CONCLUSION

From this paper the design approach for knife valve is discussed. The design was verified by using the numerical approach of finite element analysis. The limitations of analytical approach for designing the complex shapes and parts can be overcome by use of advance techniques. It has observed that there is a scope for optimization in the design. By use of finite element method one can easily do iterations to get an optimized design.

REFERENCES