iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 10 **Issue:** V **Month of publication:** May 2022

DOI: https://doi.org/10.22214/ijraset.2022.43079

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue V May 2022- Available at www.ijraset.com

Finite Element Analysis of Butterfly Valve

Mr. Sudatta Walmik Deshkar[1], Mr. H. N. Sayankar[2], Dr. Vinod S. Gorantiwar[3]
[1]*Student of M. Tech CAD/CAM*
[2]*Assistant Professor Mechanical Engineering*
[3]*Principal, Shree Sai College of Engineering & Technology, Bhadrawati, Chandrapur, Maharashtra, India-442902*

***Abstract:** The main objective of this project is shape optimization and structural stability of the butterfly valve for metallic and non-metallic materials butterfly valve is mostly used in the engine carburettors need to make structural stability and shape optimization plays the main role for this component, design modifications and material comparative analysis done in ANSYS Structural modules and find the optimized shape through stress, strain and deformation results. Valves for hydro power projects are installed for safety, maintenance, and shut-off, as well as for flow and pressure regulation. A Butterfly valve is a type of flow control device, which is widely used to regulate a fluid flowing through a section of pipe. This type of valve is mainly used as safety valve, turbine inlet valve, and pump valve for low to medium design pressures. They are operated by oil hydraulic systems for opening and closing or by closing weight and hydraulic pressure for opening. For turbine inlet valves, oil pressure can also be taken from the governor hydraulic oil system. The sealing system is of flexible, adjustable rubber/metal type to reduce leakage to a minimum. Water flow through the valve is possible in both directions. The main objective of this thesis work is to analyses the option of fabricated variant for door & body in place of casted, reduction in the material of valve body & door by structural design & FEM analysis & optimization in the material of valve component. The 3D modelling to be performs for butterfly valve by using CAD software. Further the stress & displacement FEM analysis of the butterfly valve to be performed by using ANSYS tool to evaluate the optimized result.*
***Keywords:** Butterfly valve, design optimization, engine components, FEA*

I. INTRODUCTION-BUTTERFLY VALV

A butterfly valve is a valve which can be used for isolating or regulating flow. The closing mechanism takes the form of a disk. Operation is similar to that of a ball valve, which allows for quick shut off. Butterfly valves are generally favoured because they are lower in cost to other valve designs as well as being lighter in weight, meaning less support is required. The disc is positioned in the centre of the pipe, passing through the disc is a rod connected to an actuator on the outside of the valve. Rotating the actuator turns the disc either parallel or perpendicular to the flow. There are different kinds of butterfly valves, each adapted for different pressures and different usage. The zero offset butterfly valve, which uses the flexibility of rubber, has the lowest pressure rating. The high performance double offset butterfly valve, used in slightly higher-pressure systems, is offset from the centre line of the disc seat and body seal (offset one), and the centre line of the bore (offset two). This creates a cam action during operation to lift the seat out of the seal resulting in less friction than is created in the zero offset design and decreases its tendency to wear. The valve best suited for high-pressure systems is the triple offset butterfly valve. In this valve the disc seat contact axis is offset, which acts to virtually eliminate sliding contact between disc and seat. In the case of triple offset valves the seat is made of metal so that it can be machined such as to achieve a bubble tight shut-off when in contact with the disc. Butterfly valves may be used for a wide variety of applications in the areas of water supply, waste management, fire protection or gas supply, gas and oil industries, in fuel handling systems, power generation etc. Some of the benefits for this type of valve are the basic structure that does not take up too much room, and the light weight and lower cost relative to other valve designs.

II. MATERIAL USED FOR BUTTERFLY VALVES

A. *Existing Material*

1) *Carbon Steel:* Carbon steel is an alloy of iron where the main alloying element is carbon. Generally, no other alloying elements are added to control the properties of the material. For butterfly valve construction, carbon steel is most often used to form the body and disc of the valve using the sand casting process. Carbon steels are available in several different grades. The most common grades used for valve bodies and discs are cast grades ASTM A216 WCB (Weldable Cast B-grade) and LCC (Low Carbon Content) steels. WCB material is most suitable for high temperature use, whereas LCC can be used at low (sub-zero) temperatures.

a) Main Advantage of Carbon Steel: The cost – carbon steels are relatively cheap, and valves produced from carbon steels provide a cost-effective solution in environments where other factors are considered less important than cost.
b) Main Disadvantage of Carbon Steel: Poor corrosion resistance. This can be overcome by surface protection such as paint, provided that the line media does not corrode the valve from the inside.

2) Stainless Steel: The definition of a stainless steel is an alloy of iron with a minimum chromium content of 10.5%. The effect of the chromium is to form a self-healing layer of chromium oxide on the surface of the material. When the surface is broken by mechanical damage such as scratching, the chromium quickly reacts with oxygen in the air, so preventing the oxygen from reacting with the iron and forming iron oxide (rust). An ever-increasing number of stainless steels are available, of which the simple iron-chromium-nickel grades are most often termed 'stainless steel'. Stainless steels can be further classified as ferritic, austenitic, martensitic, duplex and precipitation hard enable. This classification is based on the microstructure that is developed in the material by varying the alloying elements present. For valve construction, the most common grades used are austenitic and duplex. These are described briefly below.

3) Austenitic Stainless Steels: Austenitic stainless steels, in addition to chromium, contain elements such as nickel, which have the effect of retaining the high temperature face-centred-cubic austenitic structure at temperatures where it would normally have transformed to the ferritic body-centred-cubic structure. This face-centred cubic structure gives the material improved toughness and ductility compared to the ferritic grades. Depending on the nickel content, the tough austenitic structure can be retained even at extremely low temperatures, allowing the material to be used in cryogenic applications. Improved resistance to pitting corrosion can be achieved by adding molybdenum to the alloy.

4) Duplex Stainless Steels: Duplex stainless steels contain a balanced structure of both the austenitic face-centred cubic and ferritic body-centred structure of iron. This structure is developed by carefully controlling both the alloying elements and the heat treatment performed on the alloy to obtain a structure consisting of 50% austenite and 50% ferrite. The result is an alloy that combines the higher strength of ferrite with the improved toughness of austenite. The super duplex grades contain higher levels of chromium and molybdenum to enhance their resistance to pitting and crevice corrosion.

III. INTRODUCTION TO CAD

Throughout the history of our industrial society, many inventions have been patented and whole new technologies have evolved. Perhaps the single development that has impacted manufacturing more quickly and significantly than any previous technology is the digital computer. Computers are being used increasingly for both design and detailing of engineering components in the drawing office. Computer-aided design (CAD) is defined as the application of computers and graphics software to aid or enhance the product design from conceptualization to documentation. CAD is most commonly associated with the use of an interactive computer graphics system, referred to as a CAD system. Computer-aided design systems are powerful tools and in the mechanical design and geometric modeling of products and components.

There are several good reasons for using a CAD system to support the engineering design function:
1) To increase the productivity
2) To improve the quality of the design
3) To uniform design standards
4) To create a manufacturing data base
5) To eliminate inaccuracies caused by hand-copying of drawings and inconsistency between
6) Drawings

A. CAD/CAM History

The concept of CAD and CAM is relatively new. The usage is linked with the development of computers. The actual application of CAD/CAM in industry, academia and government is only approximately 30 years old. Formal courses in CAD and Finite Element Analysis (FEA) were introduced in 1970's. The major application thrust of CAD came in 1980's, with the availability of PCs and workstations. In its early stage of usage, very few engineering companies could afford the expense of mainframe computers; however, PCs and workstations have evolved into affordable and adequate platform to support comprehensive CAD packages that initially were designed to run on the mainframe platform. A brief history of the evolution of CAD/CAM, according to the decade and the major CAD/CAM developments.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue V May 2022- Available at www.ijraset.com

IV. INTRODUCTION TO CATIA

CATIA also known as Computer Aided Three-dimensional Interactive Application and it is software suit that developed by the French company call Dassult Systems. CATIA is a process-centric computer-aided design/computer-assisted manufacturing/computer-aided engineering (CAD/CAM/CAE) system that fully uses next generation object technologies and leading edge industry standards. CATIA is integrated with Dassult Systems Product Lifecycle Management (PLM) solutions. It allows the users to simulate their industrial design processes from initial concept to product design, analysis, assembly and also maintenance. In this software, it includes mechanical, and shape design, styling, product synthesis, equipment and systems engineering, NC manufacturing, analysis and simulation, and industrial plant design. It is very user friendly software because CATIA Knowledge ware allows broad communities of user to easily capture and share know-how, rules, and other intellectual property assets.

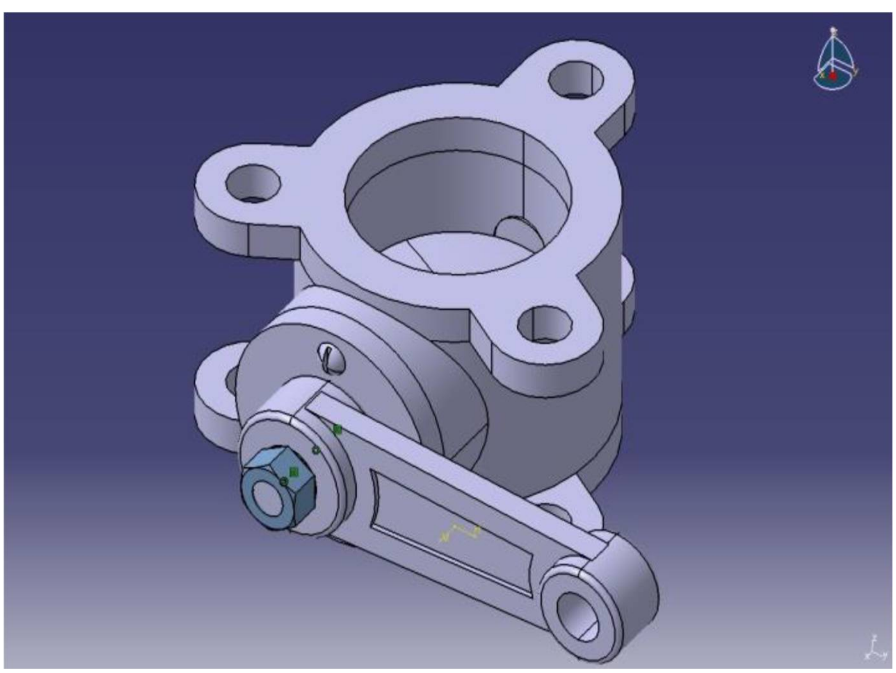
A. Engineering Design

Catia V5 offers a range of tools to enable the generation of a complete digital representation of the product being designed. In addition to the general geometry tools there is also the ability to generate geometry of other integrated design disciplines such as industrial and standard pipe work and complete wiring definitions. Tools are also available to support collaborative development. A number of concept design tools that provide up-front Industrial Design concepts can then be used in the downstream process of engineering the product. These range from conceptual Industrial design sketches, reverse engineering with point cloud data and comprehensive freeform surface tools.

B. Different Modules in Catia 5

- Sketcher
- Part Modeling
- Surfacing
- Sheet Metal
- Drafting
- Manufacturing
- Shape designs

C. Design Models



V. ANSYS INTRODUCTION

The ANSYS program is self-contained general purpose finite element program developed and maintained by Swason analysis systems Inc .the program contain many routines, all inter related and all for main purpose of achieving a solution to an engineering problem by finite element method.

ANSYS finite element analysis software enables engineers to perform the fallowing tasks:

1) Build computer model or transfer models of structures, products, components, or system.
2) Apply operating loads or other design performance conditions.
3) Study physical responses, such as stress levels, temperature distributions, or electromagnetic fields.
4) Optimize a design early in the development process to reduce production costs.
5) Do prototype testing in environments where it otherwise would be undesirable or impossible.

The ANSYS program has a compressive graphical user interface (GUI) that gives users easy, interactive access to program functions, commands, documentation and reference material. An intuitive menu system helps users navigate through the ANSYS program. Users can input data using a mouse, a keyboard, or a combination of both. A graphical user interface throughout the program, to guide new users through the learning process and provide more experienced users with multiple windows, pull-down menus, dialog boxes, tool bar and online documentation.

VI. ORGANIZATION OF THE ANSYS PROGRAM

The ANSYS program is organized into two basic levels:

1) Begin level
2) Processor (or routine) level

Begin level acts as a gateway into and out of the ANSYS program. It is also used for certain Global program controls such as changing the job name, clearing (zeroing out) the database, and copying binary files. When we first enter the program, we at the begin level.At the processor level, several processors are available; each processor is a set of functions that perform a specific analysis task. For example, the general preprocessor (PREP7) is where we build the model, the solution processor (SOLUTION) is where we apply loads and obtain the solution, and the general postprocessor(POST1) is where we evaluate the results and obtain the solution. An additional postprocessor (POST26), enables we to evaluate solutions results at specific points in the model as a function of time.

VII. PERFORMING A TYPICAL ANSYS ANALYSIS

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analysis for different engineering disciplines.

A typical ANSYS analysis has three distinct steps:

1) Build the model
2) Apply loads and obtain the solution
3) Review the results

A. Pre-Processor

The input data for an ANSYS analysis are prepared using a preprocessor. The general preprocessor (PREP 7) contains powerful solid modelling and mesh generation capabilities, and is also used to define all other analysis data with the benefit of data base definition and manipulation of analysis data.

Parametric input, user files, macros and extensive online documentation are also available, providing more tolls and flexibility for the analyst to define the problem. Extensive graphics capability is available throughout the ANSYS program, including isometric, perceptive, section, edge and hidden-line displays of three-dimensional structures-y graphs of input quantities and results, and contour displays of solution results.

The following table shows the brief description of steps followed in each phase.

Pre-processor	Solution processor	Post processor
Assigning element type	Analysis definition	Read results
Geometry definition	Constant definition	Plot results on graphs
Assigning real constants	load definition	view animated results
Material definition	solve	
Mesh generation		
Model display		

The Pre-processor stage involves the following:

1) Specify the title, which is the name of the problem. This is optional but very useful, especially if a number of design iterations are to be completed on the same base mode.
2) Setting the type of analysis to be used e.g., structural, thermal, fluid, or electromagnetic, etc.
3) Creating the model: The model may be created in pre-processor, or it can be imported from another CAD drafting package via a neutral file format.
4) Defining element type: these chosen from element library.
5) Assigning real constants and material properties like young`s modules, Poisson’s ratio, density, thermal conductivity, damping effect, specific heat, etc.
6) Apply mesh: Mesh generation is the process of dividing the analysis continuum into number of discrete parts of finite elements.

B. Solution Processor

Here we create the environment to the model, i.e. applying constraints & loads. This is the main phase of the analysis, where the problem can be solved by using different solution techniques. Here three major steps involved:

1) Solution type required, i.e. static, modal, or transient etc. is selected.
2) Defining loads: The loads may be point loads, surface loads, thermal loads like temperature, or fluid pressure, velocity are applied.
3) Solve FE solver can be logically divided into three main steps, the pre-solver, the mathematical-engine and post-solver. The pre-solver reads the model created by pre- processor and formulates the mathematical representation of the model and calls the mathematical-engine, which calculates the result. The result return to the solver and the post solver is used to calculate strains, stresses, etc. for each node within the component or continuum.

C. Post Processor

Post processing means the results of an analysis. It is probably the most important step in the analysis, because we are trying to understand how the applied loads affects the design, how food your finite element mesh is, and so on. The analysis results are reviewed using post processors, which have the ability to display distorted geometries, stress and strain contours, flow fields, safety factor contours, contours of potential field results, vector field displays mode shapes and time history graphs. The post processor can also be used for algebraic operations, database manipulators, differentiation and integration of calculated results. Response spectra may be generated from dynamic analysis. Results from various loading may be harmonically loaded axis metric structures.

VIII. REVIEW THE RESULTS

Once the solution has been calculated, we can the ANSYS post processor to review the results. Two post processors are available: POST 1 and POST 26. We use POST 1, the general post processor to review the results at one sub step over the entire model or selected portion of the model. We can obtain contour displays, deform shapes and tabular listings to review and interpret the results of the analysis. POST 1 offers many other capabilities, including error estimation, load case combination, calculation among results data and path operations. We use POST 26, the time history post processor, to review results at specific points in the model over all tome steps. We can obtain graph plots of results, data vs. time and tabular listings.

Other POST 26 capabilities include arithmetic calculations and complex algebra. In the solution of the analysis the computer takes over and solves the simultaneous set of equations that the finite element method generates, the results of the solution are:

1) Nodal degree of freedom values, which form the primary solution.
2) values which form the element solution.

A. Meshing

Before meshing the model and even before building the model, it is important to think about whether a free mesh or a mapped mesh is appropriate for the analysis. A free mesh has no restrictions in terms of element shapes and has no specified pattern applied to it.Compare to a free mesh, a mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped area mesh contains either quadrilateral or only triangular elements, while a mapped volume mesh contains only hexahedron elements. If we want this type of mesh, we must build the geometry as series of fairly regular volumes and/or areas that can accept a mapped mesh.

B. Structural Static Analysis

A static analysis calculates the effects of study loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time varying loads. A static analysis can however include steady inertia loads and time varying loads that can be approximated as static equivalent loads. Static analysis is used to determine the displacements, stresses, strains and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed, i.e. the loads and the structure's responses are assumed to vary slowly with respect to time.

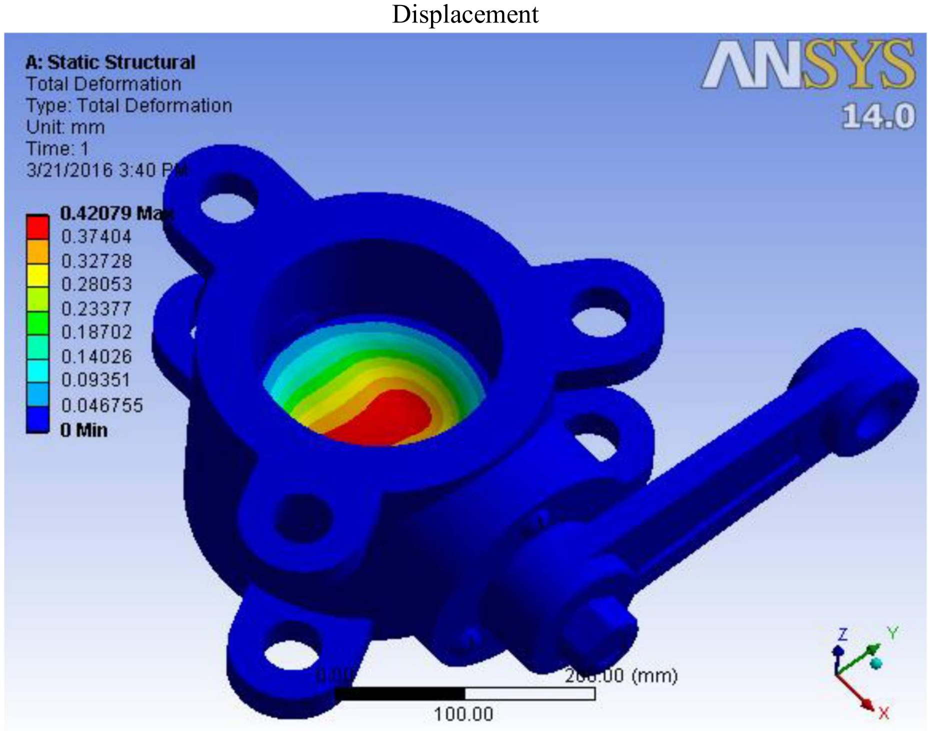
The kinds of loading that can be applied in static analysis include:

1) Externally applied forces and pressures.
2) Steady state inertial forces.
3) Imposed displacement.
4) Temperatures.
5) Fluencies (for nuclear swelling).

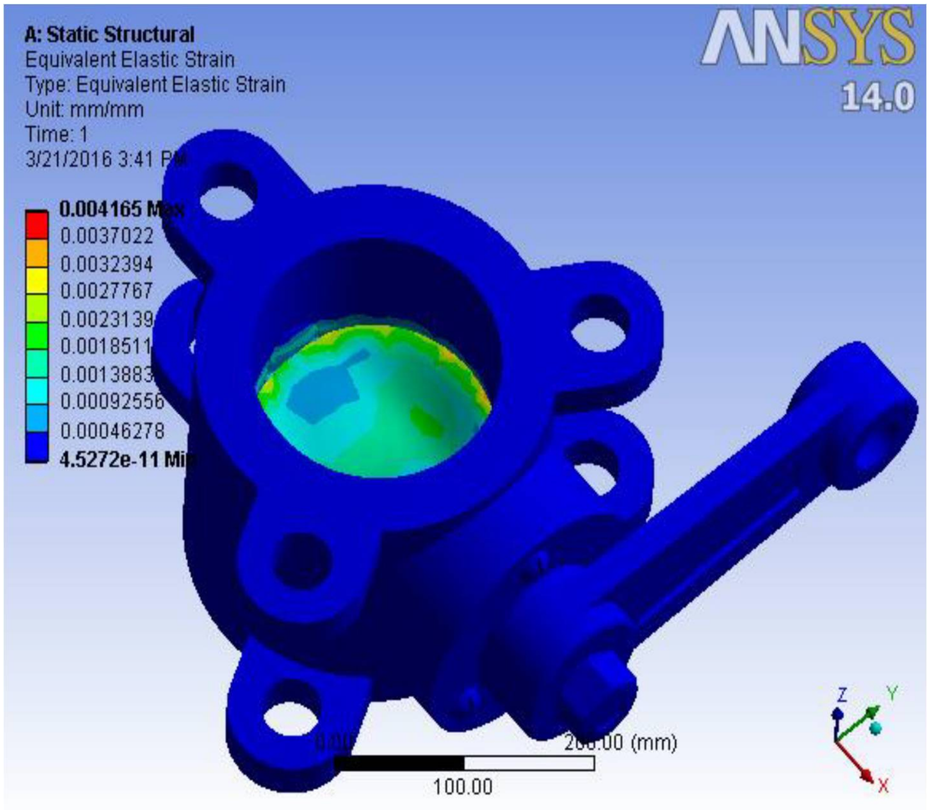
IX. ANSYS RESULTS

A. Structural Analysis Of Carbon Steel

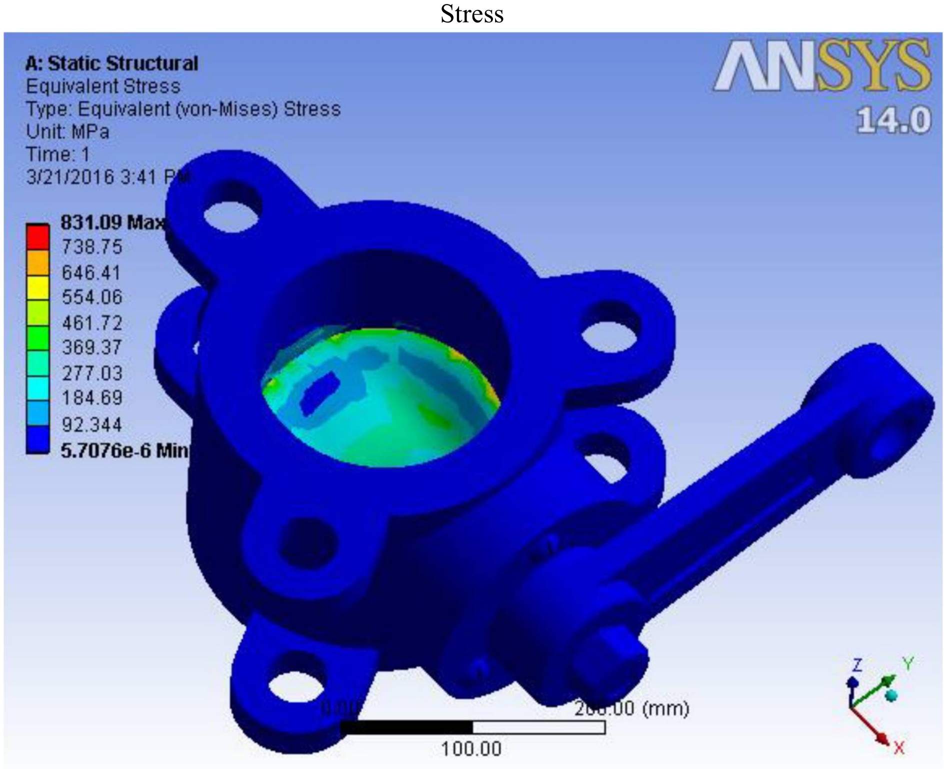
Displacement



Strain

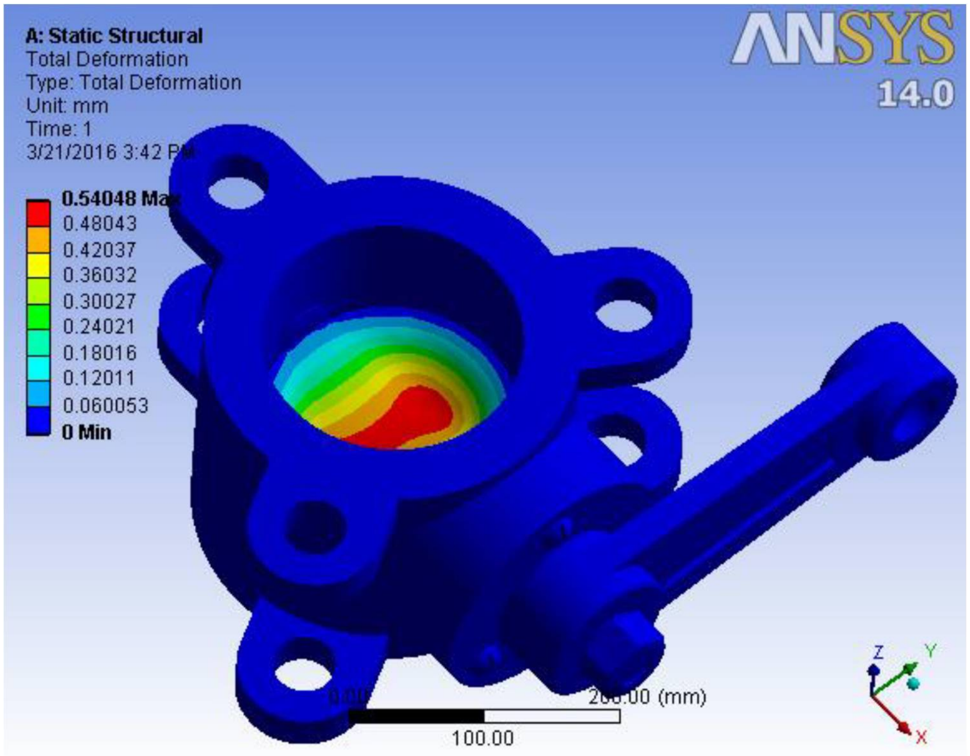


Stress

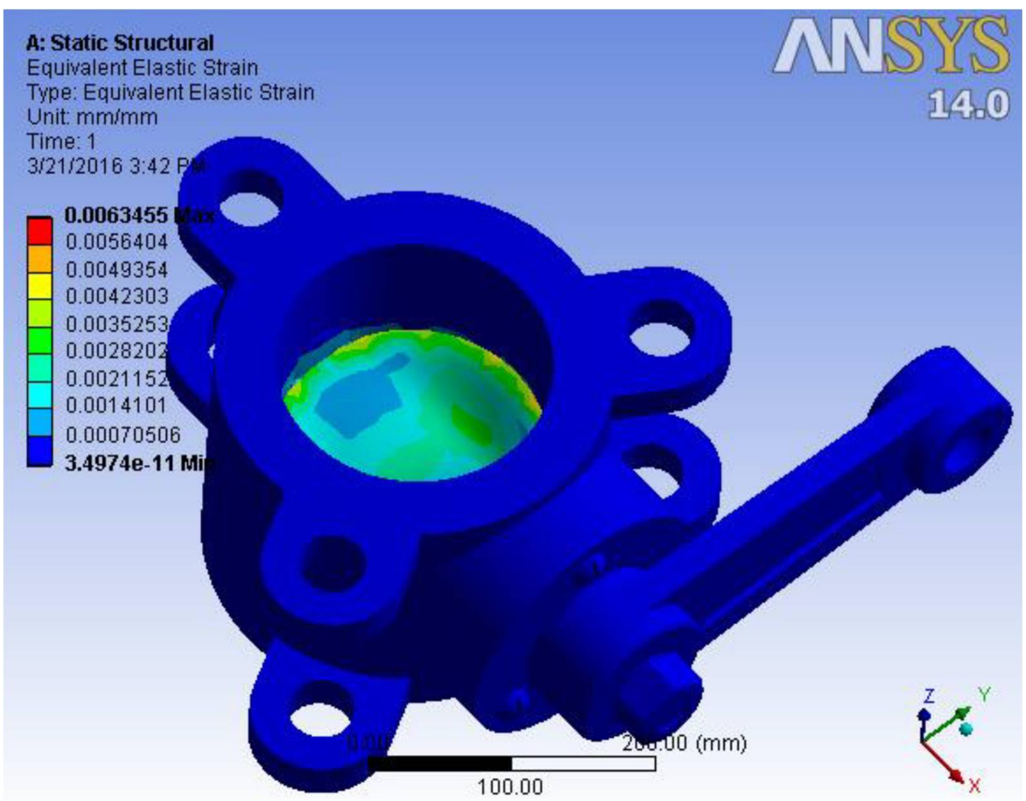


B. Structural Analysis of Nickel Alloy Bronze

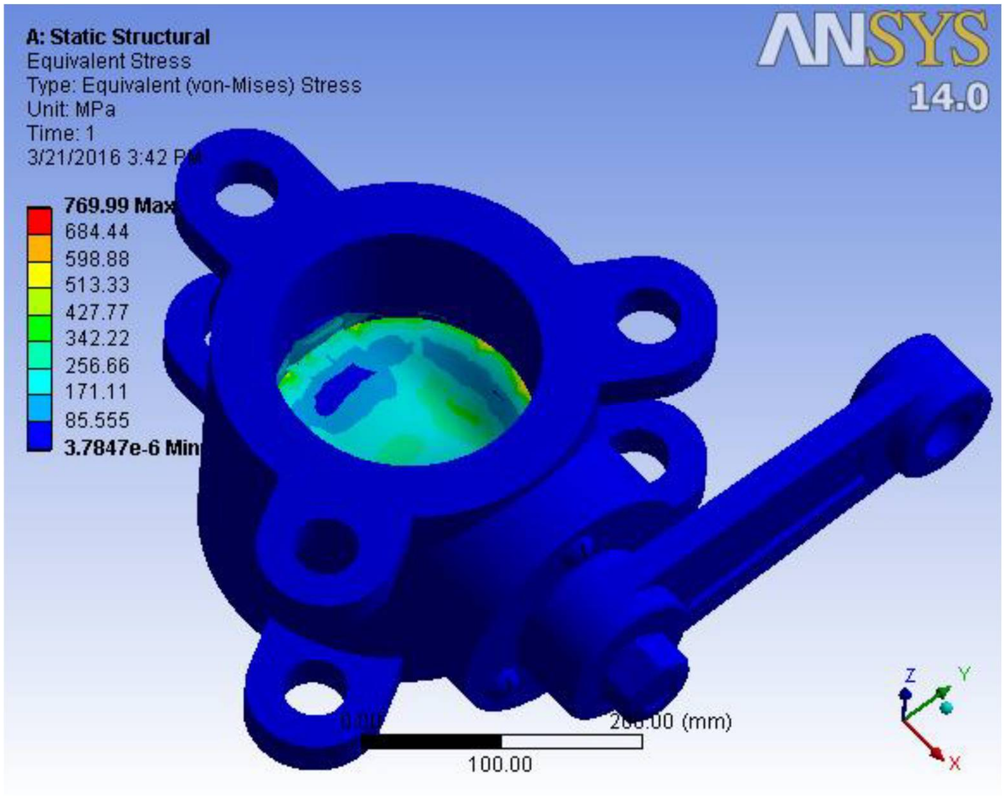
Displacement



Strain

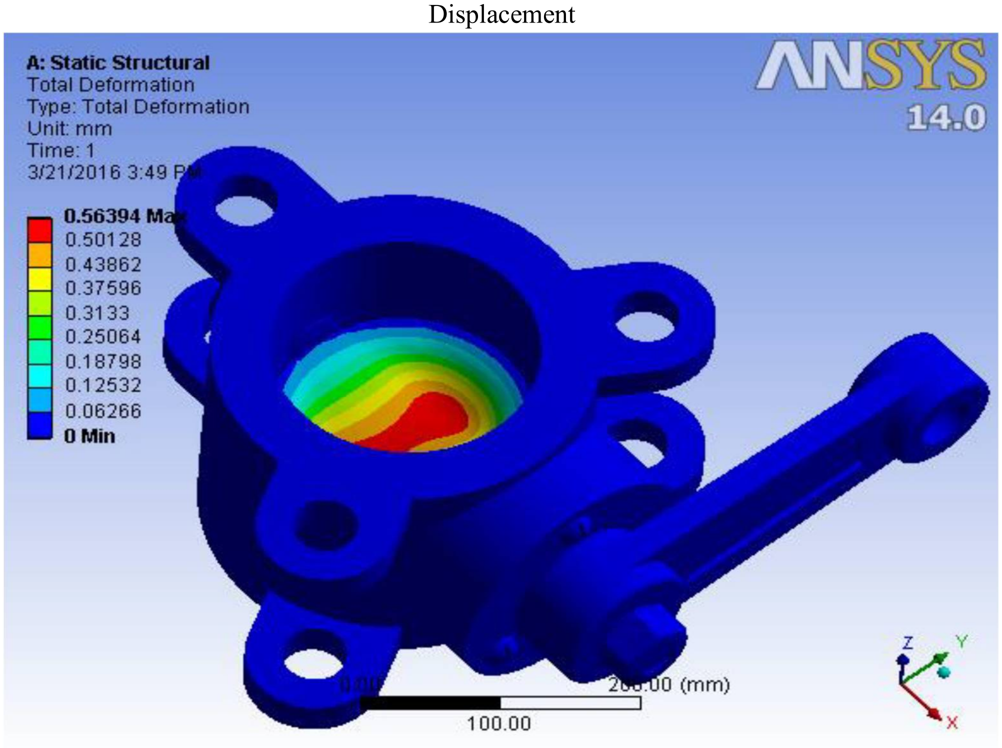


Stress

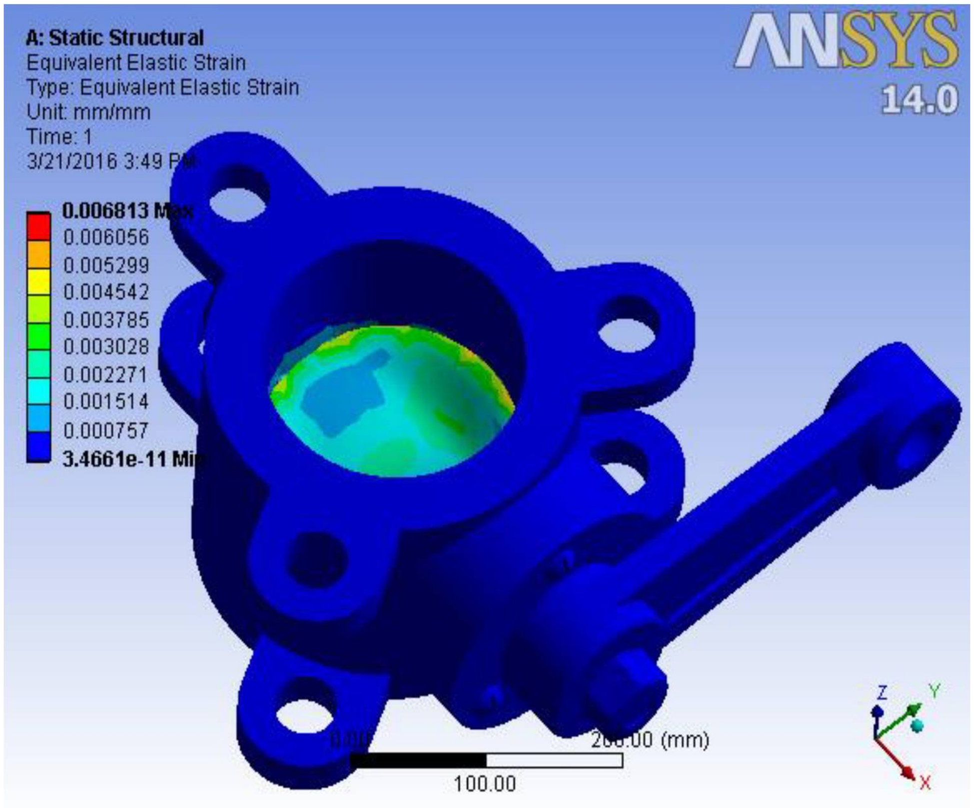


C. Structural Analysis Of Titanium Bronze

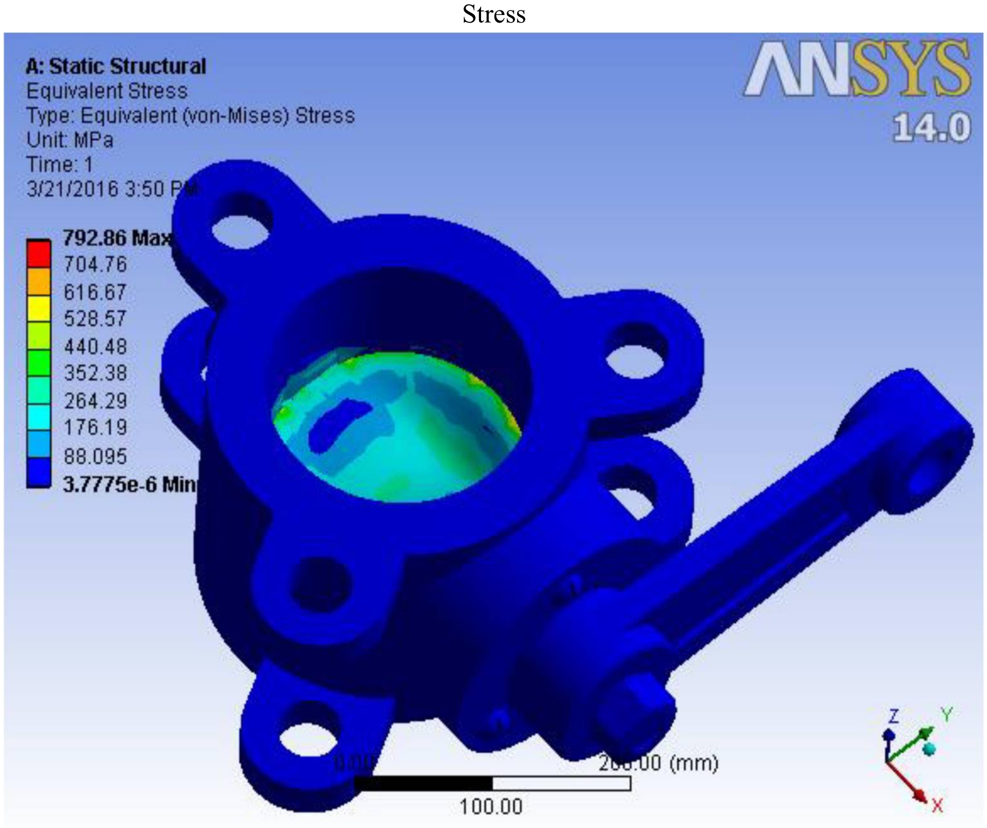
Displacement



Strain



Stress



ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue V May 2022- Available at www.ijraset.com

X. RESULTS AND CONCLUSION

A. Results

1) Weight of butterfly valve with existing design
a) Titanium bronze : 92.1924 kg
b) Nickel aluminum bronze : 93.574 kg
c) Carbon steel : 93.604 kg
2) Weight of butterfly valve with new design
a) Titanium bronze:93.82 Kg
b) Nickel aluminium bronze : 94.67 kg
c) Carbon steel: 94.73 kg

Results Table

Material	Displacement (mm)		Stress (MPa)		Strain (mm/mm)	
	Min	Max	Min	Max	Min	Max
Carbon steel	0	0.4207	5.707e-6	831.09	4.527e-11	0.004207
Titanium bronze	0	0.5639	3.77e-6	792.86	3.466e-11	0.00681
Nickel aluminium bronze	0	0.54048	3.789e-6	769.99	3.497e-11	0.634

B. Conclusion

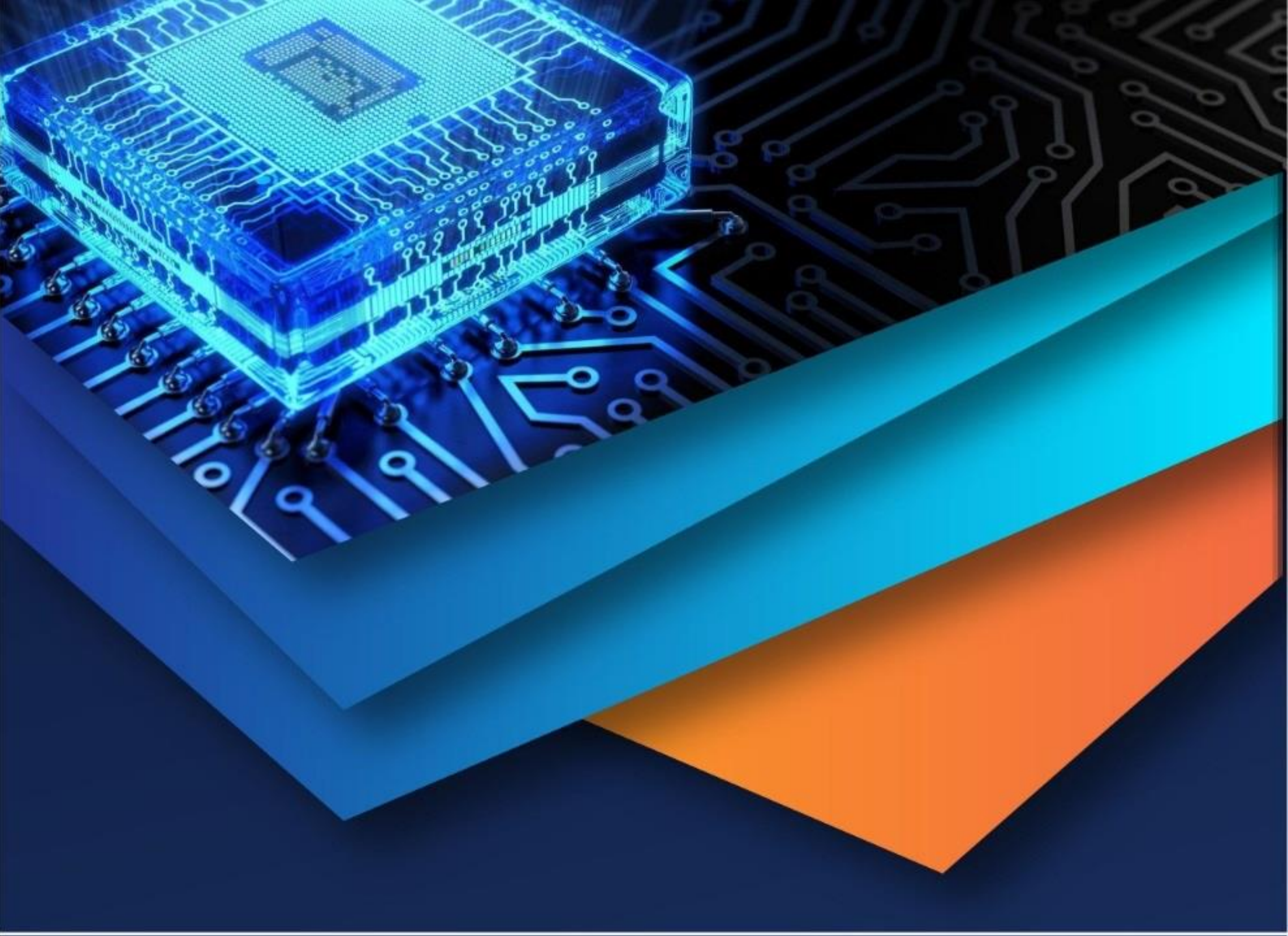
From the results it is concluded that the results obtained with new design is best than previous or existing design. As shown in results the stress value find out with help of carbon steel is 831.09 MPa where as stress value obtained with the nickel al bronze is 769.99 MPa . And stress value obtained with titanium bronze is 792.86 MPa . And also displacement value obtained with carbon steel is more than nickel al bronze and titanium bronze. So from results it is concluded that the best material used for butterfly valve amongst three is nickel aluminium bronze.

XI. FUTURE SCOPE

Now a day's Plastic valve are also manufacturing by most of the companies available in market. In future we can take plastic valve and its Part material for study. Its design as well as analysis using above software and also with other latest software and also try to compare it with above material.

REFERENCES

[1] Ogawa, K. and Kimura,T. Hydrodynamic characteristics of a butterfly valve – prediction of torque characteristic. ISA Trans., 1995, 34, 327–333. Proc. IMechE Vol. 223 Part E: J. Process Mechanical Engineering JPME236 © IMechE 2009 Downloaded from pie.sagepub.com at DONG A UNIV LIBRARY on July 11, 2011 Analysis and optimization of butterfly valve disc 89.
[2] Huang, C. D. and Kim, R. H. Three-dimensional analysis of partially open butterfly valve flows. Trans. ASME, J. Fluids Eng., 1996, 118, 562–568.
[3] Park, J. Y. and Chung, M. K. Study on hydrodynamic torque of a butterfly valve. J. Fluids Eng., 2006, 128(1), 190–195.
[4] Danbon, F. and Solliec, C. Aerodynamic torque of a butterfly valve – influence of an elbow on the time-mean and instantaneous aerodynamic torque. J. Fluids Eng., 2000, 122(2), 337–344.
[5] Caillé, V. and Laumonier, J. Effect of periodic aerodynamic pulsation on flow over a confined butterfly valve. Expl Fluids, 1998, 25(4), 362–368.
[6] Wojtkowiak, J. and Ole´skowicz-Popiel, C. Investigations of butterfly control valve flow characteristics. Found. Civil Environ. Eng., 2006, 7, 328–395.
[7] Yi, S. I., Shin, M. K., Shin, M. S., Yoon, J. Y., and Park, G. J. Optimization of the eccentric check butterfly valve considering the flow characteristics and structural safety. Proc. IMechE, Part E: J. Process Mechanical Engineering, 2008, 222(E1), 63–73.
[8] Amago, T. Sizing optimization using response surface method in FOA. R&D Rev. Toyota CRDL, 2002, 37(1), 31–36.
[9] Craven, P. andWahba, G. Soothing noisy data with spline functions: estimating the correct degree of smoothing by the method of generalized cross-validation. Number. Math., 1978, 31, 377–403.
[10] Jeong, S., Murayama, M., and Yamamoto, K. Efficient optimization design method using Kriging method. In the 42nd AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada, 2004, AIAA paper 2004-118.



10.22214/IJRASET

INDEX COPERNICUS

45.98

IMPACT FACTOR: 7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY