International Journal For Technological Research In Engineering
Volume 7, Issue 7, March-2020 ISSN (Online): 2347 - 4718

OPTIMIZATION OF SPIRAL CASING OF FRANCIS TYPE LIS PUMP THROUGH STATIC STRUCTURAL ANALYSIS

Umesh Kumar[1], Arun Patel[2]
[1]PG Student, [2]Asst. Professor
[1]Department of Mechanical Engineering
[1]NRI Institute if Science & Technology, Bhopal, Madhya Pradesh, India

Abstract: For The Francis type pumps impellers are equipment which transfers hydraulic energy to the fluid, which are generated by the coupled motor. These high capacity motors are powered by the connected grids. Spiral Casing is part of Francis type Pump installation which completely surrounds the pump and which has a continuously increasing cross sectional area in direction of flow to maintain a uniform flow velocity.
In this work an attempt is made to determine the optimized value of thickness of all sections of spiral casing. For the optimization of thickness of sections detailed calculation to be done for each section of spiral casing. The static structural FE analysis to be performed using FE analysis tools like UGNX-NASTRAN/ ANSYS software. The designed spiral is also to be simulated on the Test pressure for optimization of plate thickness of the sections of spiral casing using UGNX-NASTRAN/ ANSYS software. Based on this analysis and considering corrosion allowance the final thickness of spiral casing has been designed.
Index Terms - Pelton turbine, Distributor, Wye branch.

I. INTRODUCTION

At the present time concerns over disruptive fossil fuel markets and uncertain pricing, the current decline of nuclear energy as a viable energy source and the significant environmental consequences of thermal energy sources have placed greater emphasis on sustainable energy policies that include the significant development of nonpolluting, environment friendly, cheaper conventional energy and renewable energy supplies.
A pump casing for a centrifugal pump, which comprises an inlet opening, a discharge outlet, and a transition surface extending between an inner peripheral surface of the main pumping chamber and an inner peripheral surface of the discharge outlet, the transition surface arranged for separating an in use exit flow of material in the discharge outlet from an in use recirculation flow of material in the main pumping chamber. The transition surface has a cutwater having a profiled section which comprises a protrusion which extends irregularly from an otherwise generally rounded arched or U-shaped transition surface and is configured such that, in use, the velocity and/or turbulence resulting from the in use flow of the material being pumped in the main pumping chamber is reduced.

FUNCTIONS OF SPIRAL CASING AND ITS BASIS OF DESIGN

In Spiral casing is a spiral shaped water passage which completely surrounds the pump and which has a continuously increasing cross sectional area in direction of flow to maintain a uniform flow velocity. During pumping operation, the spiral casing collects the water being pumped out by impeller and discharges it into the discharge pipe. Access to the spiral casing is through a manhole on inlet pipe to discharge valve. Spiral casing has been designed in segments considering the transport limitations and ease of erection at site. The segments are made by rolled plate sections which are welded together to form a segment. In between the main segments, a makeup segment is provided with fitting margin on both sides for adjustment at site. The segments are welded to each other and to stay ring at site.

Spiral Casing

The spiral casing is fabricated assembly from steel plates in minimum number of section with a view to keep welding at site to the minimum feasible. In design of spiral casing limitation of transport dimensions i.e. designated path from manufacturing works to site location and handling of the sections according to site condition and station layout is also considered. It shall be designed to withstand maximum pressure during operation. This pressure is taken as static delivery head of the pump.
Hydraulic testing of spiral casing at site is to be performed at 1.5 times of the static delivery head. It is designed so as to withstand maximum allowable deflection and stresses in the plates. To avoid the failure of the spiral, deflection and stress intensity should be kept up to safe allowable limits.
The spiral casing is completely embedded in concrete when installed in pumphouse. The pressure of water is transferred to concrete through spiral casing. However, to transfer large axial forces to pump house an extension must be welded to outlet. This is done for reducing the specific pressure on concrete.

II. CASE HISTORY

The Spiral casing has an intricate geometry. This being the water delivery system of the pump, the efficiency of the pump depends on water path as well as structural strength of the spiral casing. The spiral casing of a pump consists of small segments arranged in spiral geometry which acts as a water delivery to the outlet. It is also subjected to very high stresses. Earlier, spiral casing were made by concrete. Later on, the design of spiral casing with fabricated interconnecting pipes was adopted. In the present work, a fabricated spiral casing, is being designed & developed.
The objective of this paper is to optimize the design of spiral

casing, to describe how different plate segments required to fabricate segments are developed and to find the stress pattern to ensure greater reliability and safe working condition .The water path has a gradual transition of sections; hence it is very important to obtain an accurate development of the plate segments, which when rolled and welded together will form the required structure. In this paper, spiral casing is modelled with various arrangements of stiffeners and sickle plate and analyzed to get optimized final model.
The stress analysis of the spiral casing is invariably complex and it is extremely difficult and tedious to obtain analytical solutions. In these situations, engineers usually resort to numerical methods to solve the problems. With the advent of computers, one of the most powerful techniques that has been developed in the realm of engineering analysis is the finite element method and the method being general can be used for the analysis of structures / solids of complex shapes and complicated boundary conditions.
The introduction of CAD software like UNI-GRAFIX, I-DEAS, NASTRAN, ANSYS, CATIA, etc. have cut down the time requirement for modeling and analysis, which are quite suitable for many industrial problems particularly engaged in the design field. These commercial software are fairly versatile. The results obtained through these software are moderately accurate.
The present competitive environment not only makes the use of finite element method (FEM) in designing of critical components like spiral casing a necessity but also as a tool to explore new shapes and geometries for least mass and highest stress bearing capacity. Use of FEM technique for designing spiral casing has helped in increasing the strength & rigidity of spiral casing by optimum utilization of material. This has helped in reducing the weight of spiral casing without sacrificing strength by eliminating the redundant material.

III. METHODOLOGY

3.1 PROJECT DATAS FOR DISTRIBUTOR

For analysis of stresses, a practical case of a spiral casing of a Francis type LIS pump has been taken. The required input data of the spiral casing for the analysis are given here:

- Type of Pump : Vertical Francis Impeller
- Direction of Rotation : Anticlockwise
- (viewed from Motor end)
- Static Head : 131.287 m
- Dynamic Head : 137 m
- Design Discharge of Pump : 75 m3/sec.
- Maximum Power Input : 123.5 MW
- Maximum Motor Rating : 145 MW
- Speed : 250 rpm
- Runaway Speed Ratio : 1.5 times
- Impeller Outlet Dia. : 4.46m
- Inner Top Cover Dia. : 5.0 m
- Discharge Valve Dia. : 3.2 m
- Elevation of Guide Vane Center line : 391 m from M.S.L.
- Spiral Outlet Dia. : 3125 mm
- Velocity at Spiral Casing exit = 9.78 m/s condition

3.2 MAKING A SOLID MODEL

For making a model of Spiral Casing a practical case of a Vertical Francis Pump of Dynamic Head 137m and Design Discharge 75 cusec. has been taken. The Outlet dia. of the Spiral casing is 3125 mm (i.e. at just inlet of Discharge valve). The diameter of the Spiral Casing varies at different sections from a maximum value of 3125 mm to a minimum value of 1082 mm. The preliminary thickness is assumed on the basis of previous projects which have the nearly same head & output. The various plate thicknesses used at different section are 16, 20, 25, 32, 36, & 40 mm.

The total weight of spiral casing model is 41 tons. Since spiral casing is a heavy equipment with large dimensions it creates some problems during meshing and FE analysis. So it is suggested to fragment the model in multiple segments keeping in to view of the parametric feature of individual model. But to keep the FE analysis results to be accurate, the Spiral casing is modeled and analyzed for whole structure. The modelling and simulation is done on both UGNX and Ansys platform to get best results.

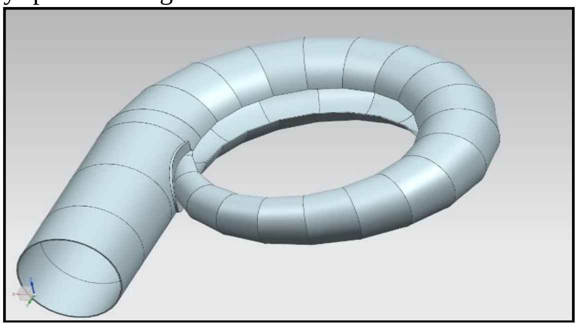

Figure 3.2.1 Solid Model of Spiral casing

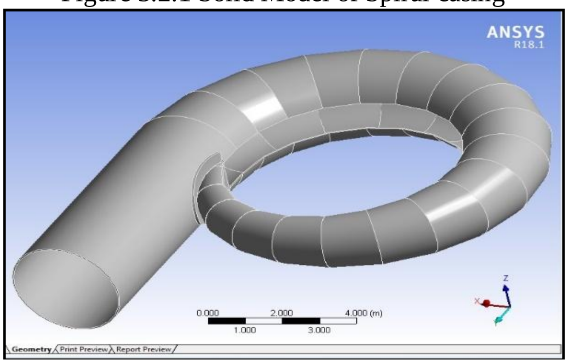

Figure3.2.2 Solid Model of Spiral casing (Ansys)

3.3 MESH GENERATION

Meshing is the stage of the finite element modeling process in which user divide a continuous structure (model) into a finite number of regions. These regions are known as elements and are connected together by nodes. Each element:

- Is a mathematical representation of a discrete portion of the model's physical structure.
- Has an assumed displacement interpolation function.
- Creating a good finite element mesh is one of the most critical steps in the analysis process, as the accuracy of finite element results depends partly on the quality of the mesh.
- The meshing capabilities available in Advanced Simulation which user automatically generates:
- 0D elements on selected points.
- 1D (beam) elements on edges.
- 2D (shell) elements on faces.
- 3D (solid) elements on volumes.

Advanced Simulation also contains a number of tools that helps for user to create specialized types of meshes. User can also use tools, such as Mesh Mating Condition to connect two separate meshes at a given interface.

The software creates all meshes directly on the model's polygon geometry. The software stores all meshes and mesh related data, such as the mesh's material and physical properties, in the FEM file

3.4 APPLYING BOUNDARY CONDITIONS

The fixed boundary condition is applied at the edges of the meshed model of spiral casing. Now for loading condition internal pressure of 16 kg/cm² is applied all over the model. Fig.6.3.1 shows the boundary condition applied model Spiral Casing, when the boundary & loading condition has been applied.

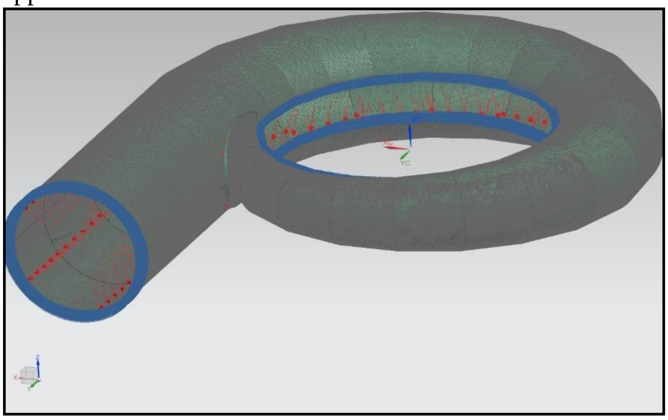

Figure3.4.1 Loading & Boundary condition on spiral casing

IV. SIMULATION

When user create a solution, user select the solver (such as NX Nastran), analysis type (such as Structural), and solution type (such as Linear Statics). A solution, which is stored in the Simulation file, contains a set of loads, constraints, and simulation objects. User can then solve using these conditions, or create new solutions defined by different conditions. User can use an unlimited number of solutions per Simulation.

For each solution, the selected solver determines which options are displayed, as well as the language used on the dialogs. Examples of solver-specific data include:

- Loads
- Constraints
- Simulation objects
- Solution options
- Solution parameters
- Solution Steps or Subcases

Each solution contains additional storage elements called steps or subcases, depending on the solver. Each step or subcase holds solution entities such as loads, constraints, and simulation objects.

- NX Nastran, MSC Nastran — For structural solves, constraints can be stored in the main solution or in the subcases; loads are stored in subcases. For thermal solves, loads and constraints are both stored in subcases.
- ANSYS — Constraints are stored in the main solution and loads are stored in sub steps, except for Nonlinear Statics and Thermal solves, where constraints are stored in sub steps.
- ABAQUS — Loading histories are divided into steps. For linear analyses, each step is essentially a load case. All loads and constraints are grouped in named steps. Steps may contain any number of loads and constraints of any type.

To simulate problems in which the results of one step are the initial conditions for the next step, user must ensure that the loads and boundary conditions of the previous step are also included in subsequent steps.

V. RESULTS AND DISCUSSION

Stress analysis for all the segments of distributor are done successfully using NASTRAN SOLVER. For this purpose, the models are prepared using parametric modeling method. Advantage of parametric modeling is that any change is any dimension, model updated automatically.

The result are shown in figures 7.1 to 7.5 in which

Red Color- shows region of maximum stress.

Blue Color- shows region of minimum stress.

Intermediate Color - shows regions having stress values between maximum and minimum.

If the maximum stress as indicated by red color exceeds the maximum allowable stress, then plate thickness is increased by next available value of plate thickness. Again the program is run with new sets of inputs and analysis is performed. The process is repeated again & again until the maximum stress in all the segments of the distributor is less than the maximum allowable stress. The last set of inputs values gives the minimum thickness of various segments of the distributor.

The various other important parameters which govern to selection of optimum thickness for particular segment of spiral casing are following: -

1. No. & position of ribs
2. Thickness of Ribs.

3. Thickness of sickle plate.

- The final value of plate thickness used in distributor model are 16, 20, 25, 32, 36, & 40 mm.
- Thicknesses of sickle plate are 100 mm.

The thickness is assumed on the basis of previous projects which have the nearly same head & output. The preliminary thickness is also calculated by hoop stress formula, which is given below: -
Hoop Stress: - pd/2t
Here, p – Design pressure inside spiral (i.e.16 kg/cm²)
d -dia. of particular section
t- thickness of particular section

As per the design norms, if the average stress is less than 1/5th of the ultimate tensile strength of material or 1/3rd of the yield strength of the material it is taken as acceptable design.
The acceptable value of stress is
Ultimate tensile strength = 4100 kg/cm²
1/5th of the ultimate tensile strength = 820 kg/cm²
Yield strength = 2400 kg/cm²
1/3rd of the yield strength = 800 kg/cm²
Hence Acceptable stress value= minimum of 1/3rd of the yield strength and 1/5th of the ultimate tensile strength
= 800 kg/cm²

Now thickness is refined by doing various no. of iterations.
For first iteration the model has been prepared with the thickness calculated using the above formula. The initial thickness was taken maximum 56mm and minimum 20mm. The FEM & stress analysis is carried out. Some local stresses are observed at the joints of segments. But almost whole structure is looking safer side. We need to do more iteration for better optimization.
For second iteration the thickness of each section were reduced by 3 mm and for bigger section the thicknesses are reduced by 5 mm. now again FEM & stress analysis is carried out. Again the structure is looking on safer side. We need to do more iteration for better optimization.
For third iteration thickness of each section were reduced again and taken maximum 44mm and minimum 16mm. Now again FEM & stress analysis is carried out. This time I found that every section was almost optimized some bigger sections thickness still needs to be reduced.
For fourth iteration thickness of each section were reduced again and taken 16, 20, 25, 32, 36, & 40 mm for different sections. Now again FEM & stress analysis is carried out. This time I found optimized results. The average stresses comes in the range of 700 kg/cm². Maximum stress on some local elements were higher but that can be neglected.
Here, as the value of average stress (i.e.700 kg/cm²) is below the 1/5th of the ultimate tensile strength of material (i.e.820kg/cm²) or 1/3rd of the yield strength of material (i.e.800kg/cm²). So these value of thicknesses has been accepted.

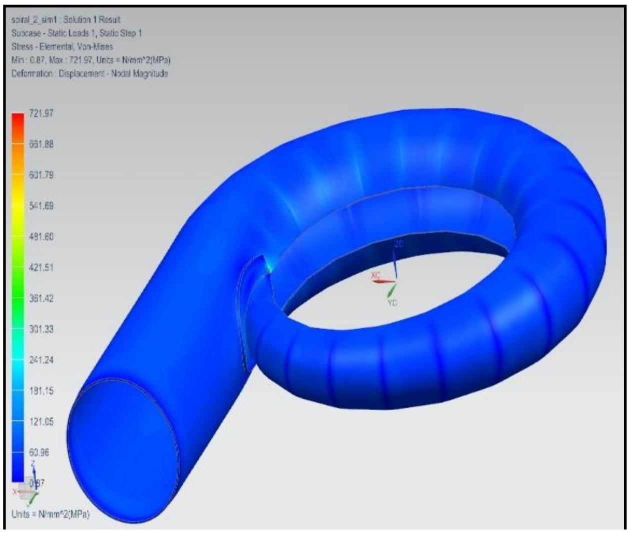

Figure5.1 Von Mises stress

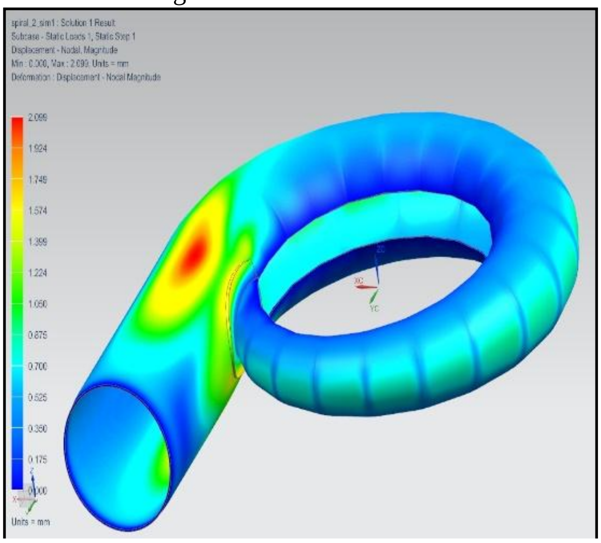

Figure5.2 Displacement results

VI. CONCLUSION

In the present work Spiral Casing of Vertical Francis type pump has been analyzed to check effectiveness f various design features and the optimization of structure under various conditions. From the analysis it is revealed that

- FEM has a very effective tool for designers in the field of structural design of hydraulic components.
- Area of stress concentration are recognized & stresses are reduced under 1/3rd of yield point of material.
- Better safety margins achieved.
- Stress pattern to be obtained for various plate thickness of distributor plate.
- Average stress has been found 700 kg/ cm² or below for different segments of the distributor against the ultimate tensile strength of material (i.e.4100kg/cm²) or yield strength of material (i.e.240kg/cm²).

Optimum thickness has been obtained for given boundary & loading conditions

REFERENCES

[1] AndriiKelin ; Oleksiy Larin ; RaisaNaryzhna ; OleksandrTrubayev ; Oleksii Vodka ; MariyaShapovalova, 2019 IEEE 14th International Conference on Computer Sciences and Information Technologies (CSIT), Lviv, Ukraine, Ukraine, 17-20 Sept. 2019, 19250460.
[2] Kevin Edward Burgess, Wen-Jie Liu, Luis MoscosoLavagna, Garry Bruce Glaves. “Pump Casing” Weir Minerals Australia Ltd. Jun 16, 2015, US9057385.
[3] Daniel GEBERT, Oliver Haaf, “Spiral Casing” EbmPapstMulfingen GmbH and Co KG. Jul 11, 2017, USD791840.
[4] Clough R. W., “The Finite Element Method in Plane Stress Analysis,” Proceedings of 2nd ASCE Conference on Electronic Computation, Pittsburgh, PA, September 8–9, 1960 pp. 345-378
[5] PushpaKhot, Dr. S. G. Taji, H. S. Bawiskar, “Quantifying the Role of Stiffener Ringsin Pressure Vessels using FEA” International Journal of Science, Engineering and Technology Research (IJSETR)Volume 5, Issue 1, January 2016, pp. 332-335
[6] 6. Cook R.D., Malkus D.S. &Plesha M.E. Concepts and applications of FEA. John Wiley& sons, New York, (2Ed) 1981.
[7] Dietrich D. FE Analysis of body. Mechanical Engineering, 109(3), 1987, 48-52.
[8] Finnigan P., Mathaway A., Lovenson W. CAT with FEM, Mechanical Engineering, 112(7), 1990, 32-38.
[9] Turner M. J., Clough R. W., Martin H. C., and Topp L. C., “Stiffness and Deflection Analysis of Complex Structures,” J. Aeronaut. Sci., Vol. 23, No. 9, 1956, pp. 805–823, 854.

