FINITE ELEMENT OPTIMIZATION OF STATOR BY CASTED AND WELDED STRUCTURES

Ganapati Kapashi[1], Prashant Mulimani[2], Veeranna D. K.[3], S. P. Dodamani[4]

[1]Assistant Professor, Dept. Of Mech. Engg., SCET, Belgaum-591156
[2]Assistant Professor, Dept. Of Mech. Engg., SCET, Belgaum-591156
[3]Associate Professor, Dept. Of Mech. Engg., SCET, Belgaum-591156
[4]Assistant Professor, Dept. Of Mech. Engg., SCET, Belgaum-591156

Abstract

Casting and welding are the major manufacturing process in the industry. Each method has its advantage and disadvantage. Due to the advances in computer based numerical techniques, simulation can be carried out to find the applicability of the method for the given problem can be carried out and the particular process can be considered for the manufacture which will reduce the cost of the product, inventory and manufacturing time. Finite element methods with its advanced softwares like ansys, Nastran and LsDyna can be used for manufacturing process simulation for better product design and optimization.

In the present work, a casted and weldment structure is considered for the same product(Stator) for analyzing for better strength with the low material requirement. Initially the geometries are built using catia software and dimensions are represented using Catia drafting software for further understanding. The structures are meshed using Hypermesh for good quality mesh satisfying the criteria required for good meshing which gives much better results. Shell mesh is adopted for optimized results. Shell mesh has the advantage of variation of thickness during optimization process. In the casted structure, total of 11 sections are considered for optimization process. Total of 30 design sets are obtained satisfying the state variable requirement. The best set shows 504kg weight requirement for the problem. The graphs are represented to shows the effect of design variables on the weight of the structure. Further analysis on welded structure shows, the initial design with higher weight of 780 kgs failing to satisfy the design requirement of displacement limitation of 0.26mm. So casted structure is better then welded structure in load carrying capacity for the same weight even factor of calculations shows lesser displacement and lesser stress for casted structure compared to the welded structure.

Keywords: *FEA, Optimization, casted stator, welded stator, ANSYS.*

---***---

1. INTRODUCTION

1.1 Optimization

Optimization is a popular subject in finite element analysis, and is becoming more important goal in the product development process analysis. This trend is facilitated by the ever-increasing computing power used to solve analysis problems. For the design engineer, it is often the real end goal.

1.2 Basic Concept of Optimization

Optimization is quite an interesting aspect of engineering practice that cuts across all branches of engineering. In the production sector, for example, the reduction of material (Figure 1.1) used in manufacturing is possible when optimization is incorporated beforehand.

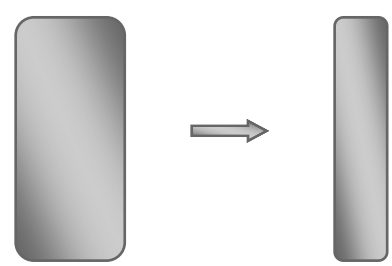


Fig 1.1: Material Reduction

1.2.1 Definition of Optimization.

Optimization can be defined as the process of finding the conditions that give maximum or minimum value of a ‘function’. Where effort required or benefit desired for a given practical situation is expressed as a ‘function’ of certain design variables [13]. This is illustrated in the Figure 1.2.

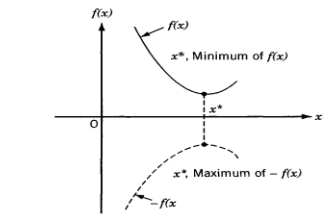


Fig 1.2: Minimum or Maximum Value of an Expression

2. PROBLEM DEFINITION AND FINITE ELEMENT MODEL DEVELOPMENT

2.1 Problem Definition

Design optimization of stator structures made of casting and welding and a comparative study to take the loads. The objectives include

- Modeling of the casted and welded structures
- Meshing and analysis(Shell mesh is the best suitable for optimization)
- Optimization of both the structures to take the same load

Requirement

The stator structure is required for the Generator section for power generation. Since the generator contains rotating rotor structure, the loads will be transferred to the stator structure. So the structure should withstand this loads without failure. So analysis should be done for better strength with lesser weight to the structure. Generally casted structure has higher investment, but with the better properties. Generally yield strength is much higher compared to the welded configuration. Also it is free from improper welding process. If welding is not done properly, premature failure may takes place with the weldment structures. The advantage of weldment is cost is less. Also members can be added wherever weakness is observed in the problem. Additional of additional parts is not possible with casted structure. Also casted structure requires higher inventory of dies and other manufacturing equipment. The present work is to compare both the designs as specified by autocad drawings.

2.2 2D Representation of the Stator Structure:

Fig 2.1: Cast Drawing

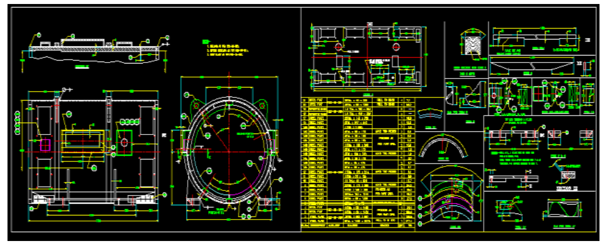


Fig 2.2: Stator Weld Drawing

2.3 3D Model Representation

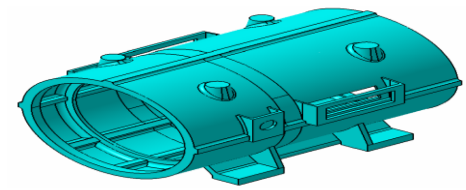


Fig 2.3: Casting structure

The figure shows modeled casted structure. Cast steel has good structural strength and due to integrity of the structure, it has lesser stress concentration effects. The ribbing is done to increase the strength of the casing. The model is done for the given dimensions specified in the autocad drawing. Catia software is used for model representation. Initially the structure is built using sketcher and later converted to three dimensional model using part modeling option. Finally the structure is integrated using assembler.

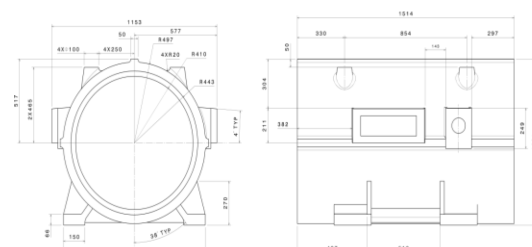


Fig 2.4 Dimensional Representation of Cast Structure

The figure shows major dimensions of the casted strucrure. Cating drafting module is used for representation of the model.

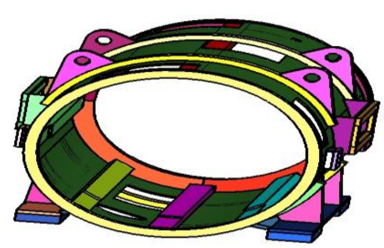


Fig 2.5: Weld Frame Structure

Similarly the weld frame structure is also built using catia software using catia as per the 2 dimensional drawing representation. Generally weld structure has the flexibility of joining the members by welding. This gives more flexibility to the designer to add the ribs whenever it is required.

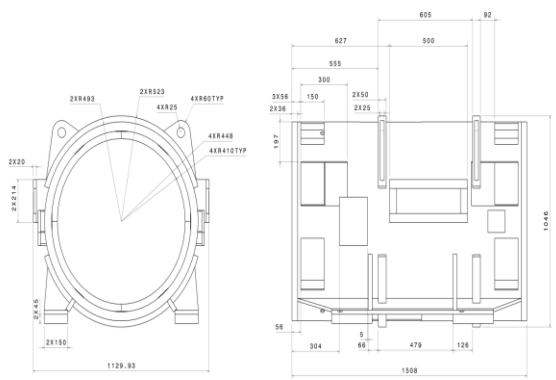


Fig: 2.6 Dimensional specification of fabricated weld structure

2.4 MESHED PLOTS:

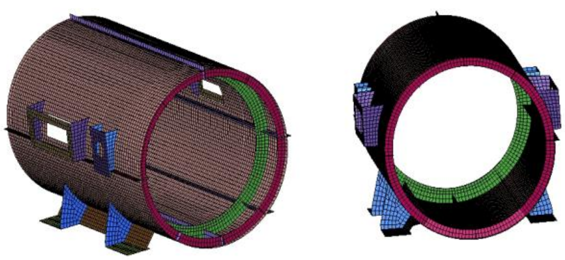


Fig 2.7: Cast Mesh

The figure shows meshed structure. Shell mesh has been carried out and the shell geometry is assigned with different properties. Hypermesh gives the advantage of grouping the components into separate members to assign required properties based on the geometry.

2.5 Materials

Cast steel fore Cast structure
Mild Steel for fabricated weld structure

Cast steel Properties
Yield Stress: 550N/mm2.
Allowable stress: 220Mpa with Factor of Safety 2.5

Weld Frame Structure:
Mild Steel
Yield stress: 250Mpa
Allowable stress: 100Mpa with Factor of Safety 2.5
Young’s Modulus: 206Gpa
Poison’s ratio=0.3

2.6: Design Requirements

- Structure stress should not exceed allowable stress of the materials
- Overall Deflection should not cross 260 microns or 0.26mm.
- Design should take a load of 15000N vertical and 410N-m torque load
- The stator inner faces are subjected to a pressure load of 5 bar pressure.

2.7 Boundary Conditions

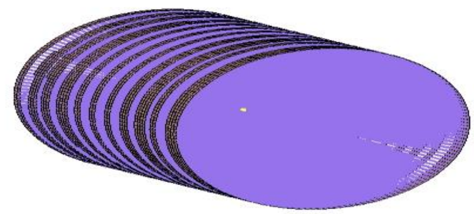


Fig 2.8: Boundary Conditions Plot

The figure2.8 shows applied boundary conditions on the problem. The load is applied through RBE3 element. The rotor loads are transferred to the stator structure through RBE3 element. The RBE3 element is connected to the inner walls of the stator frame. RBE3 element simulates rotor loads transferred to the stator frame. RBE3 element is the finite element methodology of load transfer with out actual model being made. The inner faces are selected and a internal pressure of 5 bar is applied. Through RBE3 element, both torque(410N-m) and translational loads(15000N) are applied as specified in the design input.

2.8 Weld Structure Mesh

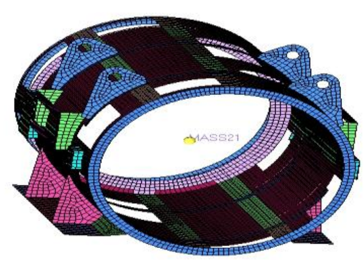


Fig: 2.9 Weld mesh Plot

Here also shell mesh is used for representation of the welded structure. A central mass element is created for application of loads through RBE3 element. Here also depends on geometry the members are grouped to separate collectors. The extensive built up of shell mesh helps in creation of scalar parameters required for design optimization process. 16368 elements and 16614 nodes are created for the mesh.

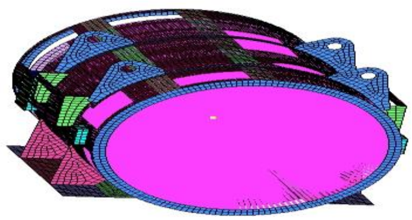


Fig 2.10: Weld Structure boundary conditions with RBE3 element

The figure shows welded structure with RBE3 element for load application. The inner wall nodes are connected to the rotor element (RBE3) for load transfer. This element is the most accurate element for load transfer. 16368 elements and 16614 nodes are created for the mesh. Initially mid surfaces are extracted and later surfaces are repaired to create an exact mid surface of the problem. Quality checks are carried out for warpage, skew angel, aspect ratio and jacobian etc. Jacobian is maintained above 0.7. Aspect ratio is limited upto 5. Warpage is allowed upto 50. Skew angle is allowed upto 300. The included angles are allowed upto 700 to 1200. Number of triangular elements is limited.

2.9 Element Type

2.9.1 Shell63

Generally shell elements are best suitable for thin shell structures where width/thickness ratio greater than 10. Here mid surfaces are extracted and problem is solved in two dimensional domain by which meshing and solving difficulties will be reduced. Also accuracy of the problem is more as shell elements consider the twisting effects due to its higher degree of freedom(6) compared to 3 degree of freedom of solid elements.

Shell63 Element Description:

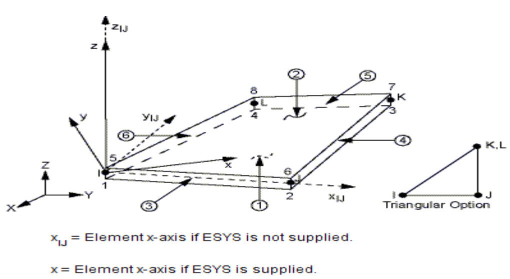


Fig 2.11: Shell63 element

SHELL63 has both bending and membrane capabilities. Both in-plane and normal loads are permitted. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. Stress stiffening and large deflection capabilities are included.

2.9.2 RBE3

RBE: RBE stands for Rigid Body Element. It is used to distribute force/moment from master node to slave nodes proportional to the weighting factors.

Master: Node at which the force/moment to be distributed will be applied. This node must be associated with an element for the master node to be included in the DOF solution.

DOF: Refers to the master node degrees of freedom to be used in constraint equations. Valid labels are: UX, UY, UZ, ROTX, ROTY, ROTZ, UXYZ, RXYZ, ALL.

Slaves: The name of an array parameter that contains a list of slave nodes. Must specify the starting index number. ALL can be used for currently selected set of nodes. The slave nodes may not be collinear, that is, not be all located on the same straight line.

Wtfact: The name of an array parameter that contains a list of weighting factors corresponding to each slave node above must have the starting index number. If not specified, the weighting factor for each slave node defaults to 1.

Menu Paths
Main Menu > Preprocessor > Coupling / Ceqn > Dist F/M at Mstr The force is distributed to the slave nodes proportional to the weighting factors. The moment is distributed as forces to the slaves these forces are proportional to the distance from the center of gravity of the slave nodes times the weighting factors. Only the translational degrees of freedom of the slave nodes are used for constructing the constraint equations. Constraint equations are converted to distributed forces/moments on the slave nodes during solution.

RBE3 creates constraint equations such that the motion of the master is the average of the slaves. For the rotations, a least-squares approach is used to define the "average rotation" at the master from the translations of the slaves. If the slave nodes are collinear, then one of the master rotations that is parallel to the collinear direction cannot be determined in terms of the translations of the slave nodes. Therefore, the associated moment component on the master node in that direction cannot be transmitted. When this case occurs, a warning message is issued and the constraint equations created by RBE3 are ignored. Applying this command to a large number of slave nodes may result in constraint equations with a large number of coefficients. This may significantly increase the peak memory required during the process of element assembly. If real memory or virtual memory is not available, consider reducing the number of slave nodes.

2.10 Assumptions

- The material is assumed to be isotropic and homogenous

- Analysis is done with in elastic limits
- Shell and RBE3 elements are used for finding the solution
- All approximations applied to finite element solutions apply to this problem.
- Sub problem approximation is considered for design optimization.
- Complete connection is considered for analysis

3. RESULTS & DISCUSSION

Both casted and fabricated structures are imported to Ansys for analysis. Initially the casted structure has been analyzed and results are obtained. The analysis is further continued to find optimum dimensions for the cast structure to find minimum weight. Later analysis for welded structure is carried out. The results are as follows.

3.1 Components of Design Optimisation

For design optimization, minimum requirement is to represent geometry to optimised in scalar parameters. Then design optimiser, by using sub-problem algorithm, checks for thickness variation in the geometry to the limiting state variable requirements. So initially geometry to be optimised should be represented by scalar parameters.

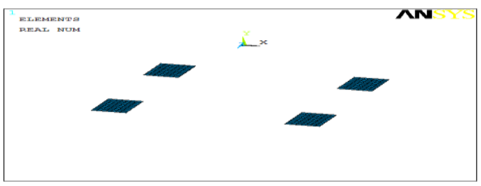


Fig 3.1: Optimisation Variable 1(Base support Plates)

Fig 3.2: Optimisation Variable 2

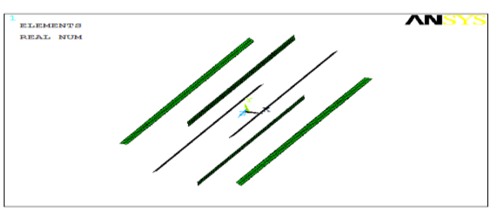


Fig 3.3: Optimisation Variable 3

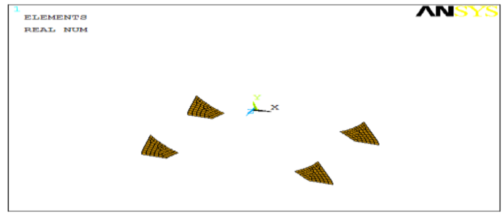


Fig 3.4: Optimisation Variable 4

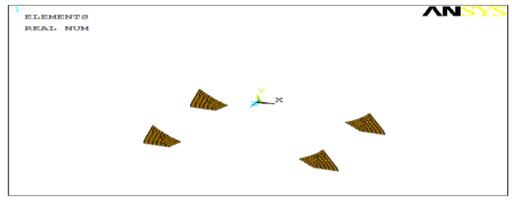


Fig 3.4: Optimisation Variable 4

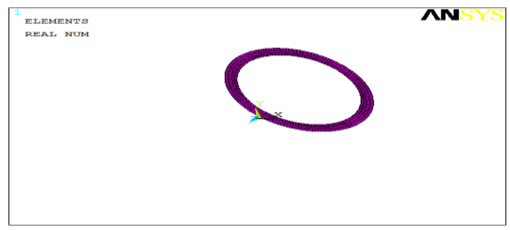


Fig 3.5: Optimisation Variable 5

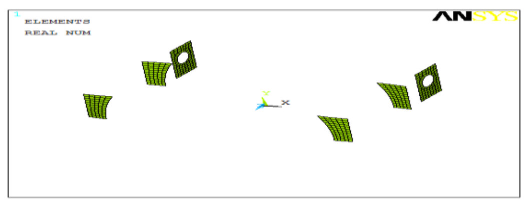


Fig 3.6: Optimisation Variable 6

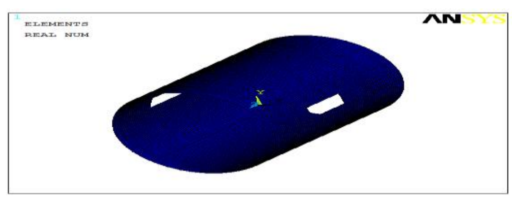


Fig 3.7: Optimisation Variable 7

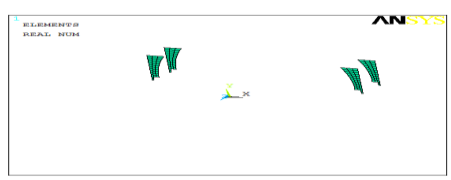


Fig 3.8: Optimisation Variable 8

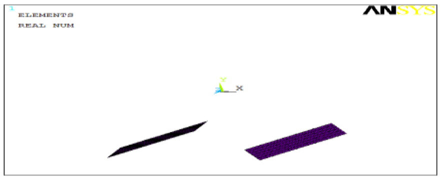


Fig 3.9: Optimisation Variable 9

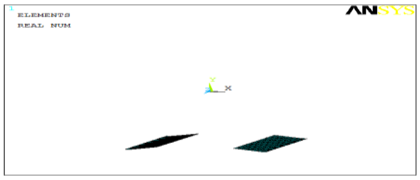


Fig 3.10: Optimization Variable 10

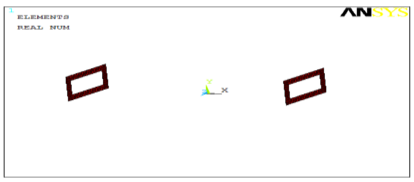


Fig 3.11: Optimisation Variable 11

3.2 Initial Structural Results

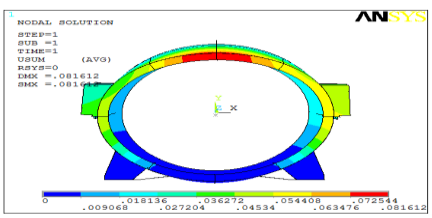


Fig 3.12: Overall Deflection (Maximum deflection 0.081mm)

The figure 3.12 shows developed displacement of 0.81mm in the strucuture due to loading. Maximmum deforamation is taking place at the top of the structure. This can be understood from the cantilever concept where the end away from the fixed is subjected to maximum deformation. The deformation represents the stiffness of the structure. Since the strucure deformation is only 81microns, the structure has the possibility of optimisation. Using the scalar parameter representation of design variables, the structure is optimised.

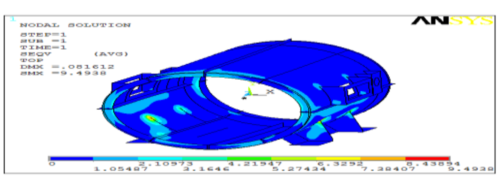


Fig 3.13: Vonmises Stress Plot(Maximum stress: 9.49Mpa)

The figure 3.13 shows overall vonmises stress in the structure. Maximum vonmises stress is around 9.49Mpa and shown by red colour marks. The status bar indicates the variation of stress in the members. Stresses are maximum at the base weldment joining locations. This can be attributed to stress concentration effects in the problem. The stress of 9.49Mpa shows very low stress development in the structure. Almost a factor of safety of 58 is maintained in the problem. But allowing more stresses and deformations also results to bent form of rotor which creates vibration problem. So generally a fixture like stators are built for deformation design.

4.2.1 Other Component Stresses

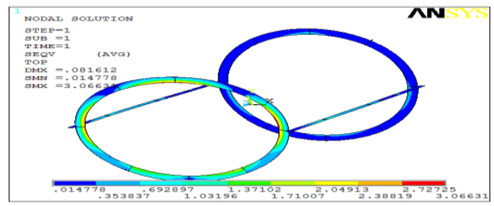


Fig: 3.14: Stresses in Component 2 (Maximum stress 3.06Mpa)

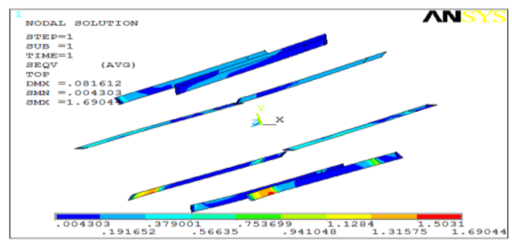


Fig: 3.15: Stresses in Component 3(Maximum stress 1.69Mpa)

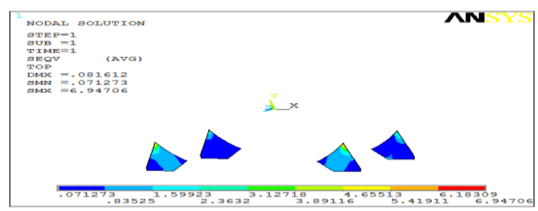


Fig: 3.16: Stresses in Component 4(Maximum stress 6.947Mpa)

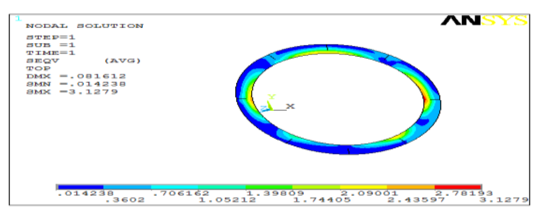


Fig: 3.17: Stresses in Component 5(Maximum stress 3.12Mpa)

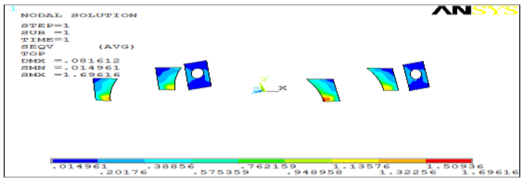


Fig: 3.18: Stresses in Component 6(Maximum stress 1.696Mpa)

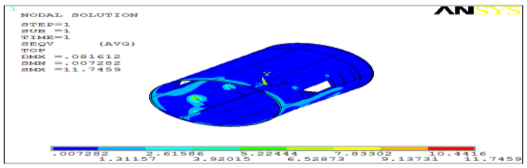


Fig: 3.19: Stresses in Component 7(Maximum stress 11.7459Mpa)

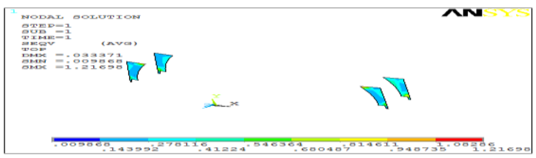


Fig: 3.20: Stresses in Component 8(Maximum stress 1.216Mpa)

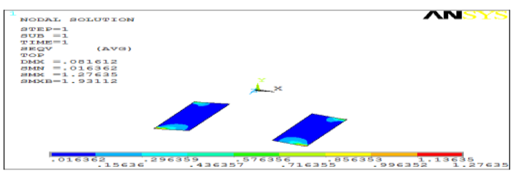


Fig: 3.21: Stresses in Component 9(Maximum stress 1.27Mpa)

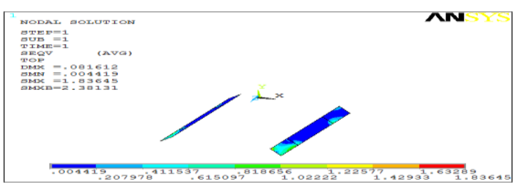


Fig: 3.22: Stresses in Component 10(Maximum stress 1.836Mpa)

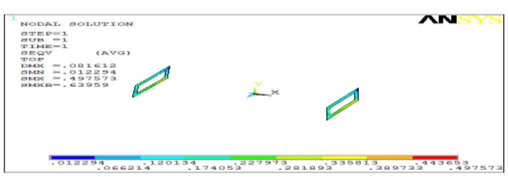
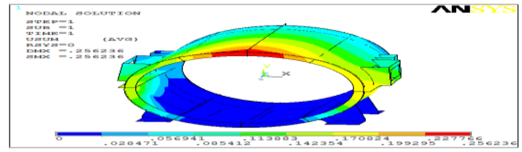


Fig: 3.23: Stresses in Component 11(Maximum stress 0.4975Mpa)

3.3 Final Set Results



Fig; 3.24 Final Structure Displacements (Maximum Displacement 0.256mm)

The figure 3.24 shows developed displacement 0f 0.256mm due to the applied loads after optimisation. Here also maximum displacement is observed at the top side. Minimum displacements are observed at the base. The displacements are more then the initial displacement but are less then the allowable deflection(0.26mm) for the problem. The status bar in 9 colors represents variation of displacement in the problem.

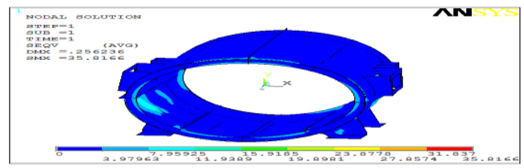


Fig 3.25: Vonmises Stress Plot (Maximum stress 35.816Mpa)

The figure 3.25 shows developed set for the final optimized set. The maximum stress is about 35.8Mpa at the regions of sharp geometrical variations or at the supports. But this stress is less then the allowable stress for the problem. So structure is safe for the given loading conditions. The problem is optimized based on deflection limitation in the problem. From the results it is observed that greater factor of safety is observed in the stress plots. But the deflection is almost reaching to the limiting deflection of the problem. The deflection limitation is specified based on the rotor vibration requirements.

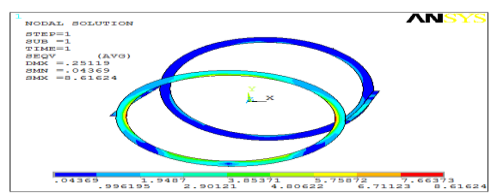


Fig 3.26: Stresses in Component 2(Maximum stress 8.6162Mpa)

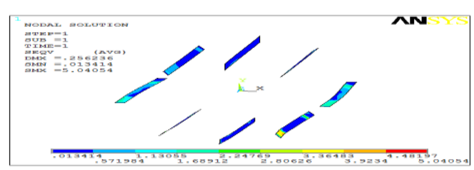


Fig 3.26: Stresses in Component 3(Maximum stress 5.04Mpa)

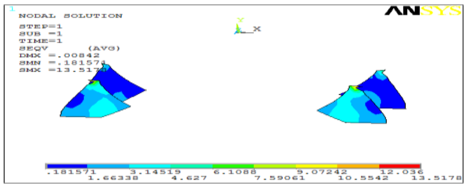


Fig 3.26: Stresses in Component 4(13.5178Mpa)

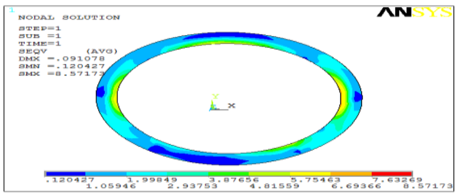


Fig 3.26: Stresses in Component 5(Maximum stress 8.57173Mpa)

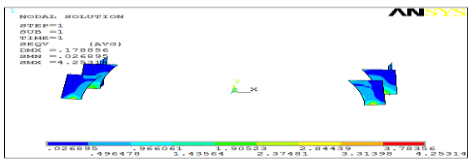


Fig 3.26: Stresses in Component 6(Maximum stress 4.25314Mpa)

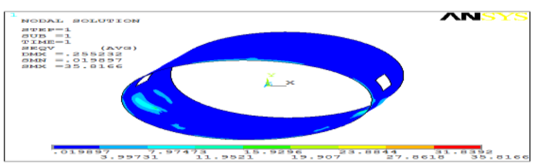


Fig 3.26: Stresses in Component 7(Maximum stress 35.8166Mpa)

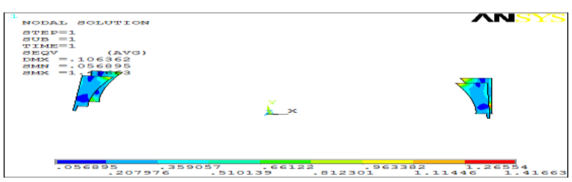


Fig 3.26: Stresses in Component 8(Maximum stress 1.4166Mpa)

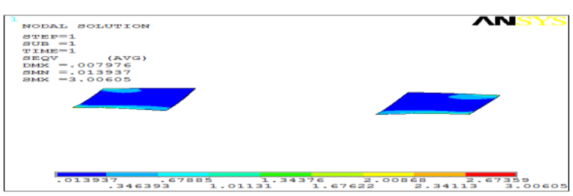


Fig 3.26: Stresses in Component 9(Maximum stress 3Mpa)

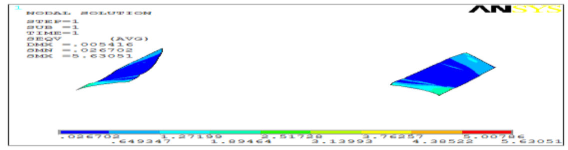


Fig 3.26: Stresses in Component 10(Maximum stress 5.63Mpa)

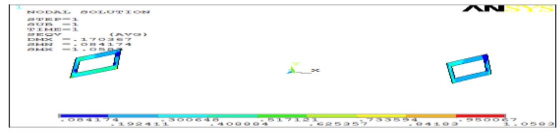


Fig 3.26: Stresses in Component 11(Maximum stress 1.05Mpa)

The figures shows complete safety of the structure for the given loads. All the member stresses are well below the allowable stresses. So the optimum design set satisfies the requirements of design.

3.4 Optimisation Graphs

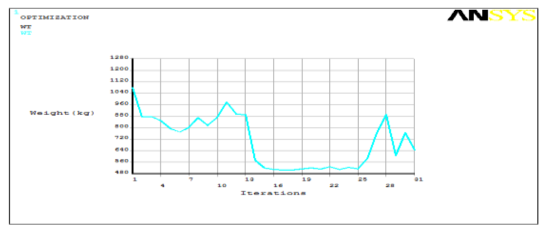


Fig 3.27: Iterations Vs Weight

The figure shows variation of weight with the iterations. Weight value is reducing near the optimum sets and is increasing for other sets. The weight depends on the scalar parameters represented for design variables.

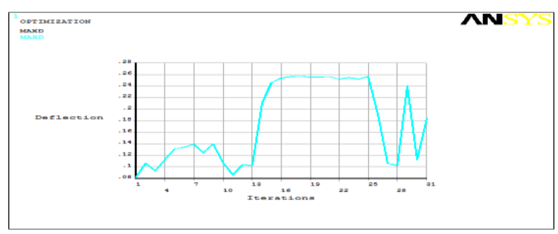


Fig 3.28: Iterations Vs Deflection

The figure shows iterations to deflection. The deflection value is increasing and reaching to the limiting deflection of 260 microns. Exactly where the weight is minimum deflection is more in the problem. Similarly when weight is more, the deflection value is less.

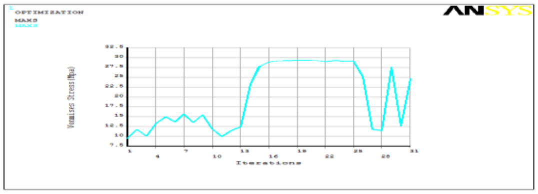


Fig 3.29: Iterations Vs Vonmises Stresses

The graph shows stress variation during the optimization process. The stress is maximum where the weight is optimum. Since the structure has reached displacement optimum values, the stress is not optimized. The design iteration will close, if any one state variable is reaching to the critical value.

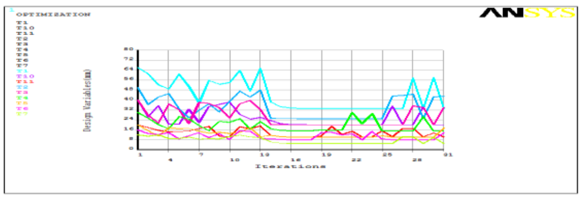


Fig 3.30: Iterations Vs Design Variables

The figure shows influence of the design variables on the iterations. The change of design variables influence the weight , deflection and stress generation in the problem.

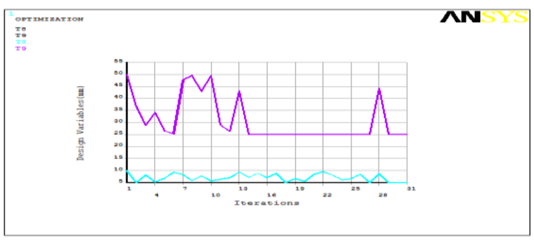


Fig 3.31: Iterations Vs Design Variables(T8,T9)

The figure shows variation of T8 and T9 during the iteration process. More fluctuation can be observed for variable T9.

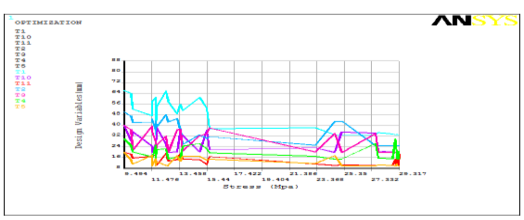


Fig 3.32: Stresses Vs Design Variables

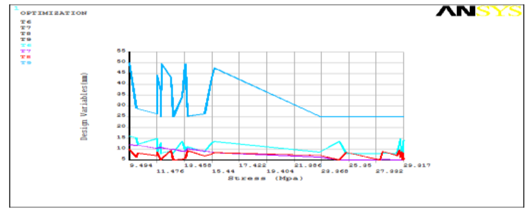


Fig 3.33: Stresses Vs Design Variables(T6,T7,T8,T9)

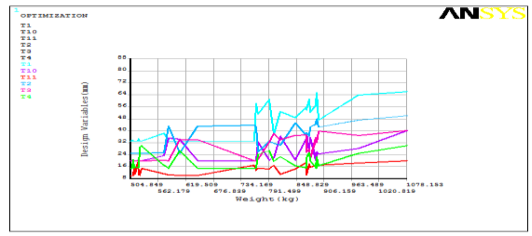


Fig 3.34: Weight Vs Design Variables

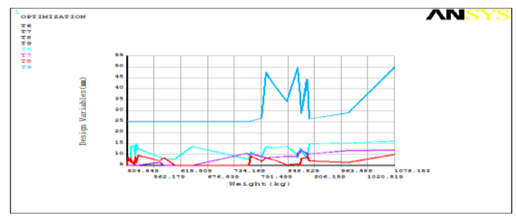


Fig 3.35: Weight Vs Design Variables(T6,T7,T8,T9)

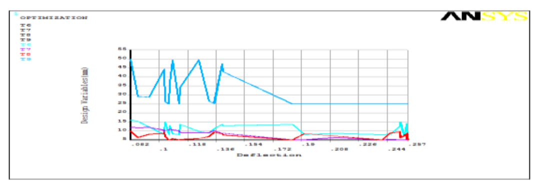


Fig 3.36: Deflection Vs Design Variables

The figures show variation and effect of design variables on the deflection, stress generation and weight of the structure. From the graphs the iteration can be observed and maximum effect values can be analyzed and the design can be further optimized for better product.

3.5 Welded Structure Results

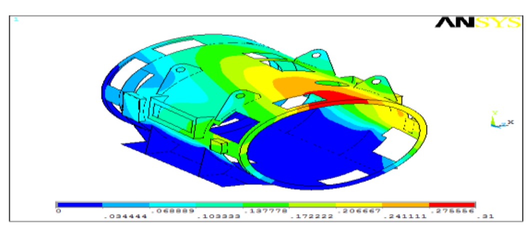


Fig 3.37: Displacement Plot

The figure shows displacement of initial design as 0.31mm or 310 microns. Here also maximum displacement is observed at the top. The displacement is minimum at the base. The status bar shows variation of displacement along the arrangement of the members.

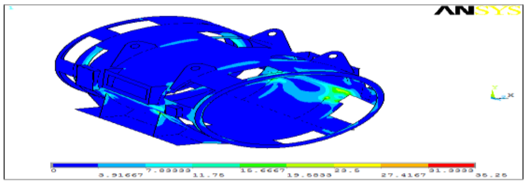


Fig 3.38: Vonmises Stress Plot

The figure 3.38 shows variation of vonmises stress in the problem. Maximum stress is around 35.25Mpa at the support weld regions and glass openings. Excepting the stress concentration region, remaining regions are free from stress condition.

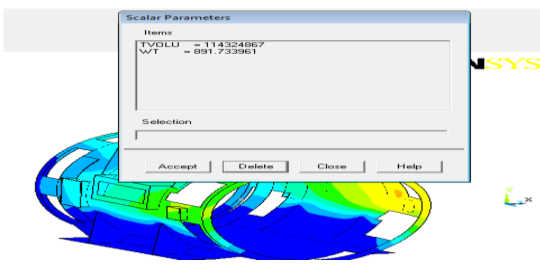


Fig 3.39: Weight of the Structure

The figure 3.39 shows weight of the structure as 891.7Kg. So this weight is more than the best design set obtained for case set. So cast design is far better than weld design specified by the two dimensional drawing. So cast structure can be selected for the given loads. Also cast structure has the advantage higher yield and ultimate strength compared to the welds of mild steel whose yield stress is much smaller(250Mpa). Also deflection development is smaller compared to the welded structure.

Table 1: Comparative Results for Casted and Welded Configurations

Structure	Weight	Displacement(mm)	Vonmises Stress(Mpa)
Casted Structure	504.85	0.256	35.81
Welded Structure	891.7	0.31	35.25

The table shows comparative results for both the configuration. The results shows casted structure has lesser weight compared to the welded structure. Also casted structure is satisfying the functional variables like displacement constraint of 0.26mm. Even though stresses are same, the welded structure is taking more weight and is not satisfying the design requirement for displacement. So casted structure can be considered for manufacturing. Also factor of safety in the weld structure (550/35.81=15.35) is much higher compared to the welded structure (250/35.21=7.1). So casted structure is better in weight, deflection and stress limits with factor of safety compared to the fabricated weld structure for the present stator built up for given loads and design conditions

3.6 Discussion

General impression in the manufacturing industry is to use either casted or welded structures in the applications. Present analysis is an application of finite element application in the manufacturing industry. A casted structure is free from weld disconnections and has better strength compared to the weld structures. But this requires lot of inventory and machinery for casting. The weldments are the most used in the manufacturing industry due to the advantages of weld additions at the required critical regions. It requires minimum inventory. But it is suffering with the problem of skilled weld person, weld preparation, quality of weld etc. Also weld material is generally mild steel and subjected to minimum yield stress. The cast steel yield strength is almost double to the weld metal yield stress. So aquatically minimum cross sections are required for the same loading capacity. But cast has certain brittleness compared to the near perfect ductile nature of mild steel. So crack generation possibility is more with casted structure compared to the weld structure. In the present work, a comparative analysis has been carried out for welded and casted structure for the given autocad dimensional representation.

Initially casted structure is modeled and meshed with shell elements. 11 design thicknesses are considered to reduce the weight. Initial iteration shows lesser stress 9 development with a weight of 1078kg. The deflection development is 0.081mm. the results shows need of optimization of the structure to satisfy the state and design variables. Selecting 11 design members which influence the structural weight is represented in scalar variable form. Sub problem approximation is considered for optimization. The results shows reduction of weight to 504 kg(Almost 53% reduction in weight). The deflection is increased from 0.081mm to 0.256mm (almost 216% increase in deflection). The stress also increase from 9Mpa to 35.8Mpa (almost 298% increase in stress). But the final design set satisfying the design requirements. Further analysis on welded structure shows, initial design failure to meet the design requirement of displacement. The results show a displacement development of 0.31mm compared to the allowable displacement of 0.26mm. So the design is not possible, as the casted structure is giving maximum displacement of 0.256mm at 504 kg weight where the fabricated structure with 891.7 kg is not satisfying the requirements. So casted structure is better then the welded structure. Also factor of safety calculations shows higher strength of casted structure compared to the welded structure.

4. CONCLUSIONS AND FURTHER SCOPE

4.1 Conclusions

Structural analysis has been carried out to find optimum structure for the Generator stator structure. A comparative designs has been developed to find the best set with minimum weight for the stator structure. The overall summary is as follows.

- Initially the structures has been built as per the specification given by the structural drawings for the stator structure.
- Both the models are built using Catia software and meshed after extraction of mid surfaces in Hypermesh software.
- Necessary quality criteria have been satisfied for better results.
- Initially the casted structure has been imported and analysed for the structural strength. Initial structure weights around 1078 kg. The stress values are around 9Mpa and deflection is around 81 microns. So this structure can be optimised for better results.
- Total of 11 design variables are specified for design optimisation and subspace optimiser tool is used for faster convergence.
- A total of 30 sets are obtained and best set is available in the 18 set.
- The initial and final configuration results are presented for stress condition to check the nearness to design requirements.
- The graphical plots are represented to show variation of weight, stress and deflection with the number of iterations. Also different graphs are presented to analyse the effect of the particular variable on the required parameter.
- Further analysis has been carried out on welded structure. But this structure by initial design, unable to satisfy the functional requirement to maintain 260 microns displacement limit. Also welded structure weight is more compared to the optimised set of casted structure. So Casted structure gives better results compared to the welded structural design with minimum weight. So casted structure can be considered for the stator.

4.2 Further Scope

- Design can-be further extended with dynamic analysis
- Design can be done with thermal loads
- Rotor dynamic study can-be carried out on the structure
- Topology optimisation can-be carried out
- A mix of cast and weld can be considered for better design

REFERENCES

[1] R.J.Duffin, E.L.Peterson and C.Zener, Geometric programming: theory and applications, Wiley, New york, 1967.

[2] G.B.Dantzig Linear Programming and Extensions, Princeton University PressR. A. Gettatly and R. H. Gallagher, “A procedure for automated minimum weight structural design, Part

[4] I - Theoretical basis, Part II - Applications,” Aero. Quart. Part I, Vol. 17, pp. 216-230 and pp.332-342, 1966.

[5] M.M. Denn, Optimisation by variational methods, McGraw-Hill , New York 1969.

[6] G.S.G. Beveridge and R.S. Schechter, Optimisation : theory and practice, McGraw-Hill, New York , 1970J.L. Kuester and J.H. Mize, Optimisation techniques with FORTRAN, McGraw-Hill, New York, 1973.

[8] M.J. Panik, Classical Optimisation: foundations and extensions, North-Holland Publishing Co., Amsterdam, 1976.D. Koo, “ Elements of Optimisation” Springer-Verlag, New York, 1977.

[10] D.G. Carmichael, Structural modeling and Optimisation, Ellis Horwood Chichester, 1981.

[11] Morris,A.J**. Foundations of structural optimization: a unified approach**. John Wiley & Sons, 1st ed., UK, **1982**

[12] Y. M. Xie and G. P. Steven, “A simple evolutionary procedure for structural Optimisation,” Comp.Struct. Vol. 49, pp. 885-896, 1993.

[13] S.S. Rao – Optimization, Theory and Applications, Wiley inter-science, 1996

[14] Ernst Hustedt, AMES Ltd. 1999, (Air New Zealand Engineering).

[15] Jim Patterson (Hendrickson Trailer Suspension Systems). 2000.

[16] Finite Element Procedures – Klaus-Jurgen Bathe, Prentice Hall of India Pvt. Ltd.- Sixth Edition 2002.

[17] Introduction to the Finite Element Method, Desai/Abel – CBS publishers 2002.

[18] Finite Element Analysis – C.S. Krishnamoorthy, Second Edn, TMH – 2002.

[19] Hursha Narayan (Robert Bosch Corporation). 2002

[20] Joe Metrisin (Florida Turbine Technologies, Inc.). 2002