ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 2, Issue 7, January 2013

Analysis and Optimization Solar Panel Supporting Structures Using F.E.M

Ravindra Naik, Vinayakumar B. Melmari, Adarsh Adeppa

Abstract--* The use of renewable energy resources is increasing rapidly. Following this trend, the implementation of large area solar arrays is considered to be a necessity. Several design approaches of the supporting structures have been presented in order to achieve the maximum overall efficiency. They are loaded mainly by aerodynamic forces. International regulations as well as the competition between industries define that they must withstand the enormous loads that result from large air velocities. Furthermore, they must have a life expectancy of more than 20 years. Optimization plays very important role in product design and prevent un-necessasary inventory satisfying the functional requirements. But optimization with proper design helps to built efficient products in the everyday competing market. Stress analysis plays important role in optimizing the structure. Due to the advances in computer based finite element software's design process is made simple by easier simulation methods fast replacing prototype built up and testing. In the present work, a solar panel supporting structure is designed to take rotational loads for 90 ^{0} for safe operation. So the design should consider the loads coming on the structure for 90 ^{0} rotation along with inertia effect of the rotating members. The mechanism should withstand the aerodynamic loads, inertia loads and rotation loads along with friction loads. The design should consider aerodynamic factors for load calculations and design should satisfy all the functional requirements.

Keywords: **FEA, Optimization, DV, SV, OCM CID, ANSYS.**

I. INTRODUCTION

A. 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.

B. Basic Concept of Optimization

Optimization is quite an interesting aspect of engineering practice that cuts across all branches of engineering.

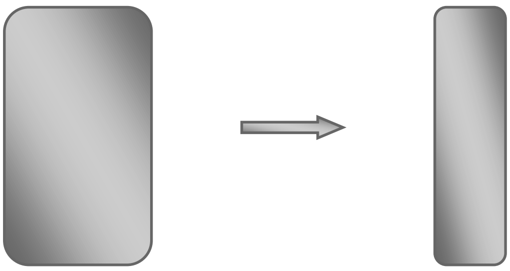

Fig.1: Material Reduction

In the production sector, for example, the reduction of material (Figure1) used in manufacturing is possible when optimization is incorporated beforehand.

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 2.

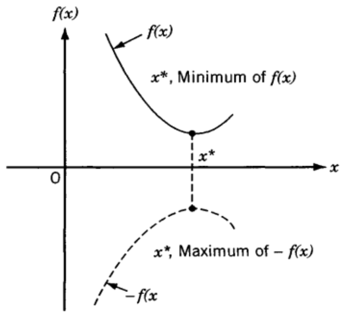

Fig.2: Minimum or Maximum Value of an Expression

C. Review of Optimization

Analysis software increasingly includes design optimization tools that would not have been possible just a few years ago. Increased raw computing power has reduced "single" analysis run times so that multiple analysis runs can be performed in reasonable time. Improvements in CAD software that allow for geometry models that can be easily and automatically changed have also helped, allowing for schemes of multiple analysis to be run without human intervention. More over advances in convergence techniques have allowed for some reasonable degree of accuracy in these automatic schemes. But the optimization is not an easy process. It is like the design process itself - potentially infinitely complex and perfectly unique for each specific situation. It's part art and part science. Successful design optimization requires complete understanding of the software limitations, a willingness to simplify the problem as much as possible, and skill in deciding which factors are important in improving the design. It's important to recognize that optimization problems increase in complexity exponentially with the number of variables used. This stems naturally from the mathematics of combinations in statistics. While a 2 or 3 variable Optimization scheme might seem like a simple problem, this is really close to the practical limit for any real FEA "pure" Optimization today. Often only 1 variable is possible, a case where Optimization is not even needed, since the same result can be obtained by performing a sensitivity

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 2, Issue 7, January 2013

study (as is often provided in FEA software) with the one variable or even by simply making a few different analysis runs individually. But the real work lies in working with multiple variables where problems are compounded. While working towards an optimum design, the ANSYS Optimization routines employ three types of variables that characterize the design process: design variables, state variables, and the objective function. These variables are represented by scalar parameters in ANSYS Parametric Design Language (APDL). The use of ADPL is an essential step in the Optimization process. Optimization modules and programs use mathematical techniques that integrate the manual design cycle into a computerized iterative process. This is done automatically until an optimum design is produced. To start the Optimization process, parameters are first defined. They are referred as the design set. These parameters include design variables, state variables and the model objective. Their values are modified throughout the Optimization process. Design variables are independent quantities that are constrained within a specified range and are changed during the Optimization analysis process. Optimization is a tool that can be used to better understand the model reactions in general conditions. The model can then be automatically modified to fit within the user-specified limitations. There are several advantages to the use of Optimization modules. The use of modules decreases the needed evaluation and modification time for each design option. It also allows for quicker and easier comparison of similar models that fit within the specified requirements. Once the parameters are defined, their values are easily changed both by the computer module and the user. Multiple models that fit within the stated specifications can easily be created and evaluated to determine which could work best under the necessary circumstances. Graphical displays of the results such as stress, temperature gradients and deflection help the user understand the reactions of the model to different conditions so that adaptations can be made to better-fit desired use of the design. With complex models problems start to arise, due to the numerous possible parameters, it is difficult to determine every possible parameter that might be considered when the model is initially created. The analysis results should not be trusted completely, as the analysis is only as accurate as the model used. The mathematical models that are utilized in each program vary in the degree of their accuracy and thus the computerized results should be checked.

D. Literature Survey

In the face of the ever limited resources available for structural design, construction and maintenance, the concept of optimization has always being a valuable principle guiding engineering practice. According to Schmit (1984), designers are always striving to evolve the 'best' or 'optimum' structural design in terms of cost, weight, aesthetics, or a combination of these. In the real sense of it, the above process can logically be termed either as maximizing desired benefits or minimizing efforts required .Rao [13] went further to say that "For a given practical situation, since the effort required or the benefit desired can be expressed as a function of certain design variables, then optimization invariably can be defined as the process of finding the conditions that give the maximum or minimum value of a function".

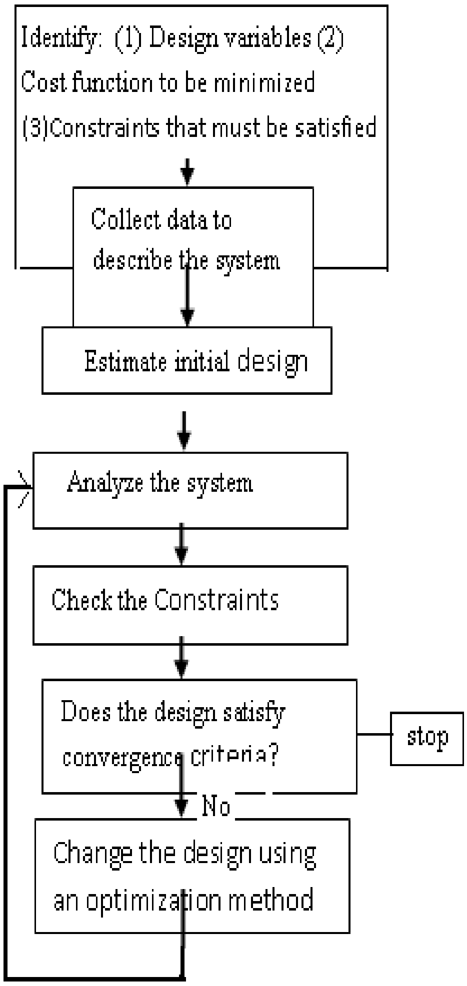

Fig 3: The Optimization Process

On the necessity of optimization process, Morris [11] further explained that most modern designs now require a long gestation period with much more emphasis being placed on innovation to compensate for long design cycle times. With the innovations having to meet the required degree of confidence and safety in order to ensure efficient designs, optimization processes play an important contributory role towards ensuring this goal.

E. Typical Optimization Methods

There are so many methods available to solve different structural optimization problems; here some typical methods shall be explained. Methods involving the Optimality Criteria Methods (OCM) and mathematical programming methods are going to be considered. It must also be stated that there are so many software tools available to solve many of the structural optimization problems.

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 2, Issue 7, January 2013

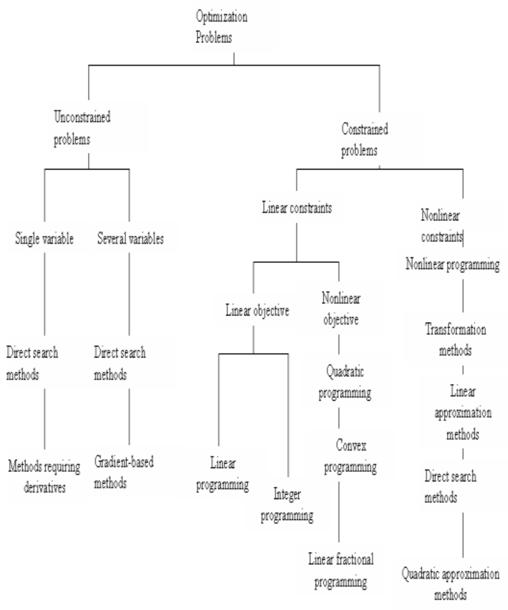

Fig 4. General Optimization Processes [24]

F. Non-Conventional Energy Resources

1. Introduction

Energy has been an important component to meet the day to day need of human beings. The degree of civilization is measured by the energy utilization for human advancement or needs. Energy has been defined as the capacity to do work or capability to produce an effort. It is expressed in N-m or Joules. The energy density is expressed as J\kg. Before the industrial revolution of the 18th century, most energy used relied on two important sources i.e., human and animal muscles, and the energy of wind and water available in nature. The chemical energy in fire wood was the main source of heat and light. Different forms of energy: Broadly speaking two main types of energy are heat and work. However, other forms of energy are 1) electrical energy, 2) mechanical energy, 3) chemical energy, 4) heat energy and 5) nuclear energy. Electrical energy is the most convenient form of energy because it can be easily transported, easily controlled and easily converted into other forms of energy at about 100% efficiency. The only short coming of electrical energy is that it can't be stored in large quantities.

2. Classification of energy resources

The various sources of energy can be conveniently grouped as, Commercial primary energy resources: Non-renewable sources of energy or conventional sources of energy are being accumulated in nature for a very long time and can't be replaced if exhausted. Nature gifted resources which are consumed can't be replaced. Eg: coal, petroleum, natural gas, thermal power, hydro power and nuclear power are the main conventional sources of energy.

3. Renewable sources of energy

Energy sources which are continuously and freely produced in the nature and are not exhaustible are known as the renewable sources of energy. Eg: solar energy, biomass and wood energy, geo thermal energy, wind energy, tidal energy and ocean energy. But main attention has to be directed to the following sources of renewable namely, a) solar photovoltaic, b) wind, and c) hydrogen fuel cell.

G. Advantages of Renewable Energy

a) These sources of energy are renewable and there is no danger of depletion. These recur in nature and are in-exhaustible.

b) The power plants based on renewable sources of energy don't have any fuel cost and hence negligible running cost.

c) Renewable is more sites specific and are used for local processing and application. There is no need for transmission and distribution of power.

d) Renewable have low energy density and more or less there is no pollution or ecological balance problem.

e) Most of the devices and plants used with the renewable are simple in design and construction which are made from local materials, local skills and by local people. The use of renewable energy can help to save foreign exchange and generate local employment.

f) The rural areas and remote villages can be better served with locally available renewable sources of energy. There will be huge savings from transporting fuels or transmitting electricity from long distances.

H. Disadvantages of Renewable Energy

a) Low energy density of renewable sources of energy needs large sizes of plant resulting in increased cost of delivered energy.

b) Intermittency and lack of dependability are the main disadvantages of renewable energy sources.

c) Low energy density also results in lower operating temperatures and hence low efficiencies.

d) Although renewable are essentially free, there is definite cost effectiveness associated with its conversion and utilization.

e) Much of the construction materials used for renewable energy devices are themselves very energy intensive.

f) The low efficiency of these plants can result in large heat rejections and hence thermal pollution.

g) The renewable energy plants use larger land masses.

I. New Sources of Energy

The new sources of energy are available for local exploitation. In many cases, autonomous and small power plants can be built to avoid transmission losses. Most prominent new sources of energy are tidal energy, ocean waves, OTEC, peat, tar sand, oil shales, coal tar, geo thermal energy, draught animals, agricultural residues etc., The total energy production in India is 14559×1015 joules. 93% of India's requirement of commercial energy is being met by fossil fuels, with coal contributing 56%, and oil and natural gas contributing 37%. Water power and nuclear power contributing only 7% of total energy production. Comparing

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 2, Issue 7, January 2013

the total energy production in India from commercial sources with that of world, it is only 3.5% of total world production.

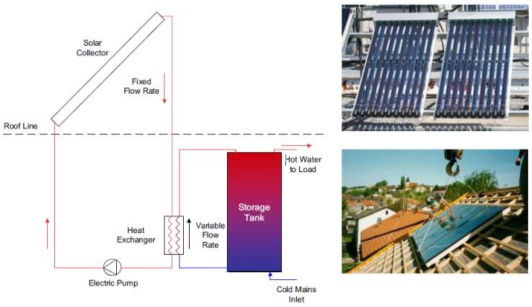

Fig 5: Photovoltaic Systems

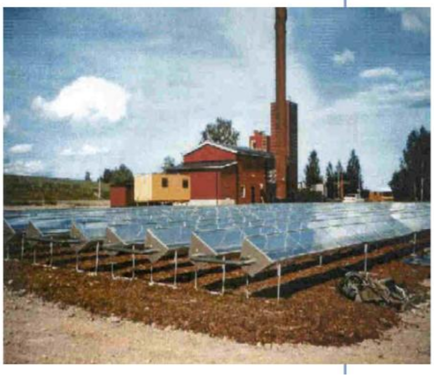

Fig 6: Solar collectors array

1. The advantages of photovoltaic solar energy conversion.

a) Absence of moving parts.

b) Direct conversion of light to electricity at room temperature.

c) Can function unattended for long time.

d) Low maintenance cost.

e) No environmental pollution.

f) Very long life.

g) Highly reliable.

h) Solar energy is free and no fuel required.

i) Can be started easily as no starting time is involved.

j) Easy to fabricate.

k) These have high power-to-weight ratio, therefore very useful for space application.

l) Decentralized or dispersed power generation at the point of power consumption can save power transmission and distribution costs.

m) These can be used with or without sun tracking.

J. The Limitations of Photovoltaic Solar Energy Conversion

a) Manufacture of silicon crystals is labor and energy intensive.

b) The principle limitation is high cost.

c) The insulation is unreliable and therefore storage batteries are needed.

d) Solar power plants require very large land areas.

e) Electrical generation cost is very high.

f) The energy spent in the manufacture of solar cells is very high.

g) The initial cost of the plant is very high and still requires a long gasification period.

K. Support Structures

Implementation of large area solar arrays is considered to be a necessity. Several design approaches of the supporting structures have been presented in order to achieve the maximum overall efficiency. They are loaded mainly by aerodynamic forces. International regulations as well as the competition between industries define that they must withstand the enormous loads that result from air velocities over 120 km/h. Furthermore, they must have a life expectancy of more than 20 years. Nowadays the demand for clean, renewable energy sources is increasing. In order to collect solar power effectively, it is necessary to use large areas of solar panels properly aligned to the sun. A wide variety of design solutions is suggested so as to achieve maximum efficiency. In this paper the analysis of two different design approaches are presented: 1.A fixed system that is mounted to a certain position as shown in Figure 1. The orientation of the solar panel array is adapted to the installation site so that the efficiency of the system is optimized.

L. Loads on the Solar Panels

Mainly wind is the main load for the structure. Critical position need to be identified and sections should be designed based on the requirements. Again further structural weight is another load region. Seismic effects also need to be analyzed for structural safety of the members.

Fig 7: Arrangement 1

Fig 8: Arrangement 2

II. METHODS AND METHODOLOGY

A. Finite Element Method

Most often the mathematical models results in algebraic, differential or integral equations or combinations thereof.

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 2, Issue 7, January 2013

Seldom these equations can be solved in closed form (Exact form), and hence numerical methods are used to obtain solutions. Finite difference method (FDM) is a classical method that provides approximate solutions to differential equations with reasonable engineering accuracy. There are other methods of solving mathematical equations that are taught in traditional numerical methods courses. Finite Element Method is one of the numerical methods of solving differential equations. The FEM originated in the area of structural mechanics, and has been extended to other areas of solid mechanics and later to other fields such as heat transfer, fluid dynamics and electromagnetic devices. In fact FEM has been recognized as a powerful tool for solving partial differential equations and integral-differential equations, and in the near future it may become the numerical method of choice in many engineering and applied science areas. One of the reasons for FEM's popularity is that the method results in computer programs versatile in nature that can be used to solve many practical problems with least amount of training. Obviously there is a danger in using computer programs without proper understanding of the theory behind them, and that is one of the reasons to have a thorough understanding of the theory behind the finite element method.

1. Procedure of Finite Element Method.

The general description of the finite element method can be detailed in a step-by-step procedure. This sequence of steps describes the actual solution process that is followed in setting up and solving any equilibrium problem.

1. Descretisation of the Continuum.

The continuum is the physical body, structure or solid being analyzed. Discretisation may be simply described as the process in which the given body is subdivided into an equivalent system of finite elements. The finite elements may be triangles or quadrilaterals for a 2-D continuum. For a 3-D continuum the finite elements may be either tetrahedral or hexahedral in shape and the elements could be linear quadratic or cubic in order.

2. Selection of the Displacement Modes.

The assumed displacement functions or models represent only approximately the actual or the exact distribution of the displacements, for example: The displacement function is commonly assumed in a polynomial form, and practical conditions limit the number of terms that can be retained in the polynomial. The simplest displacement model that is commonly employed is a linear polynomial. There are three interrelated factors that influence the selection of the displacement model.

- First the type and the degree of the displacement model must be chosen.
- Second the particular displacement magnitudes that describe the model must be selected.
- Third the model should satisfy certain requirements that ensure that the numerical results approach the correct solutions.

3. Element Stiffness Matrix Using - Variation Principle.

The stiffness matrix consists of the coefficients of the equilibrium equations derived from the material and geometric properties of an element and obtained by the use of principle of minimum potential energy. The stiffness relates the displacements at the nodal points to the applied forces at the nodal points. The distributed forces applied to the structure are converted into equivalent concentrated forces at the nodes. The equilibrium relation between the stiffness matrix [k], the nodal force vector {Q}, and the nodal displacement vector {q}, is expressed as a set of simultaneous linear algebraic equations.

$$[K] \{q\} = \{Q\}$$

4. Assembly of the algebraic equations for the overall descretised continuum.

This process includes the assembly of the overall or the global stiffness matrix for the entire body from the individual element stiffness matrices, and the overall global force or load vector from the element nodal force vectors. In general, the basis for an assembly method is that the nodal interconnections require that the displacements at the node to be the same for all elements adjacent to that node. The overall equilibrium relations between the total stiffness matrix [K], the total load vector {R} and the nodal displacement vector for the entire body {r} will again be expressed as a set of simultaneous equations.

$$[K] \{r\} = \{R\}$$

5. Solutions for the Unknown Displacements.

The algebraic equations assembled are solved for the unknown displacements. In linear equilibrium problems, this is a relatively straightforward application of matrix algebra techniques. However, for non-linear problems the desired solutions are obtained by a sequence of steps, each step involving a modification of stiffness matrix and /or load vector.

6. Computation of the Element Strains and Stresses from the Nodal Displacements.

In certain cases the magnitudes of the primary unknowns that is the nodal displacements, will be all that are required for an engineering solution. More often the quantities derived from the primary unknowns, such as strains and / or stresses, must be computed.

7. Types of Elements

Few of the important FEM elements are as follows, TRUSS: Slender element (length>>area) which supports only tension or compression along its length; essentially a 1D spring. BEAM: Slender element whose length is much greater that its transverse dimension which supports lateral loads, which cause flexural bending. TORSION: Same as truss but supports torsion. 2D SOLID: Element whose geometry definition lies in a plane and applied loads also lie in the same plane. Plane stress occurs for structures with small thickness compared with its in plane dimension - stress components associated with the out of plane coordinate are zero. Plane strain occurs for structures where the thickness becomes large

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 2, Issue 7, January 2013

compared to its in plane dimension - strain component associated with the out of plane coordinate are zero. PLATES: Element whose geometry lies in the plane with loads acting out of the plane which cause flexural bending and with both in plane dimensions large in comparison to its thickness - two dimensional state of stress exists similar to plane stress except that there is a variation of tension to compression through the thickness. SHELLS: Element similar in character to a plate but typically used on curved surface and supports both in plane and out of planeloads – numerous formulations exist. 3D SOLID: Element classification that covers all elements - element obeys the strain displacement and stress strain relationships.

III. HYPERMESH

A. Introduction

Altair® Hyper Mesh is a high-performance finite element pre- and post-processor for major finite element solvers, allowing engineers to analyze design conditions in a highly interactive and visual environment. Hyper Mesh's graphical user interfaces easy to learn and supports the direct use of CAD geometry and existing finite element models, reducing redundancy. Advanced post-processing tools that ensure complex simulations are readily visualized and easily understood. Hyper Mesh offers unparalleled speed, flexibility and customization.

B. Hyper mesh Tools

Hyper Mesh provides a variety of tools That enables the integration of engineering process.. The following tools are available to enhance productivity of the system.

- ✓ **Basic macros**: Users can create Macros that automate a process or Series of steps
- ✓ **Custom utilities**: Users can take the Advantage of the power within the Tcl/Tk toolkit, which can be used for Building custom solutions that are Fully integrated with Hyper Mesh
- ✓ **Configure the Hyper Mesh interface**: Redefine the layout of HyperMesh's menu system through an easy to use interface
- ✓ **Export templates**: Export templates allow the Hyper Mesh database to be written out to formats other solvers and programs can read
- ✓ **Input translators**: Extend HyperMesh's interface to support by adding users input translators for reading different analysis data decks
- ✓ **Results translators**: One can create his own specialized results translators to convert analysis specific results into HyperMesh results format
- ✓ Geometry Interfacing and Cleanup

Hyper Mesh provides import/export access to a variety of industry leading CAD data formats for generating finite element models. Within Hyper Mesh there are a series of tools for cleaning up or' mending' imported geometry entities. Imported geometry can contain surfaces with gaps, overlaps and misalignments that may prevent an auto mesher from creating quality meshes. By eliminating misalignments and holes and by suppressing the boundaries between adjacent surfaces you can mesh across larger, more logical regions of the model and improve overall meshing speed and quality. Model Building and Editing. For building and editing models,HyperMesh presents users with a sophisticated suite of easy-to-use tools. For 2D and 3D model creation, users have access to a variety of mesh generation panels as well as HyperMesh's powerful auto meshing module. The surface auto meshing module in HyperMesh provides users with robust tool for mesh generation a well as gives users the ability to interactively adjust a variety of mesh parameters per surface (or surface edge) including element density, element biasing, mesh algorithm and more. HyperMesh can also quickly automesh a closed region with high-quality first or second order tetrahedral elements. The tetra automesh module uses the powerful AFLR algorithm. Users can control element growth options for structural and CFD modeling requirements, select floatable or fixed boundary tria elements and re-mesh local regions.

- ✓ Solver Interfaces (Import and Export of FEA codes).

HyperMesh supports a host o different solver formats for both import and export. Along with fully–supported solvers, Hyper Mesh also provides the flexibility to support additional solvers via a complete export template language and C libraries for development of input translators.

- ✓ Post-Processing.

HyperMesh provides a complete suite of post-processing features that enable. You to easily and accurately understand and interpret complex simulation results. HyperMesh presents a complete suite of visualization tools to view results using is surface, deformed, contour, transient, vector plot and cutting plane with contour displays. Also supported are deformed, linear, complex model, and transient animation displays. These features, combined with a user friendly interface, allow you to quickly identify problem areas and help to shorten results evaluation.

C. ANSYS

1 Introduction

ANSYS is a general purpose finite element program that can be applied to varied range of engineering problems varying from linear to nonlinear to transient. ANSYS capabilities are briefed here.

2 Pre-processing

ANSYS directly interfaces with the pre-processing module of 3-D interactive color graphics macro (PREP7) with extensive modeling capabilities for finite element model generation and problem definition. Highlights of the capabilities are.

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 2, Issue 7, January 2013

- ✓ CAD/CAM interface, directly from geometry database or through IGES format. Both command and menu driven modes with online help.
- ✓ 3-D geometric modeling including points, lines, arcs, curves, surfaces and solids as well as surface intersections.
- ✓ Geometric transformations including translation, rotation, scaling, mirror, imaging and dragging the curve along an arbitrary 3-d path.
- ✓ 3-D interactive finite elements mesh generation including automatic element generation.
- ✓ Mesh grading with uniform or non-uniform spacing.
- ✓ Merging separate models into a larger one.
- ✓ Definitions of element attributes including material and geometric properties.
- ✓ Specification of loading and boundary conditions.
- ✓ Extensive model editing capabilities.
- ✓ Extensive plotting options including boundary line, hidden line removal and shrink element plots for selected elements or regions.
- ✓ Color shading and light effects.
- ✓ Model checking including calculation of element areas, volumes, normal and distortion index.

IV. PROBLEM DEFINITION AND FINITE ELEMENT MODEL DEVELOPMENT

A. Problem Definition

Design, cad modeling and Analysis of the Solar panel system based on design requirements through design principles is the main definition of the problem. Here objectives include

- ✓ Wind load estimations
- ✓ Inertia torque estimations
- ✓ Drive torque estimations
- ✓ Structural member design
- ✓ Panel sections design
- ✓ Ball screw and power calculations
- ✓ Meshing and analysis of the problem for design safety

Requirement of the problem:

- ✓ Solar panels are essential components of present day solar collectors. A non conventional energy tapping system. Most popular in all the solar devices along with wind mills, wind turbines etc.
- ✓ It is required to check structural safety of the equipment.
- ✓ Life of solar panels depends on structural stability for all the load types.
- ✓ Solar panels can tap more energy by turning towards sun direction.

B. Design Methodology

- ✓ Adaptive Design of the Members
- ✓ Theoretical Checking of individual members for structural safety
- ✓ Modeling and Analysis of the panel structure for Strength
- ✓ Optimization of Panel members strength
- ✓ PLC controls for drive mechanism
- ✓ Final Report generation

C. Material Properties Q235B

- ✓ Minimum yield strength = 235Mpa
- ✓ Ultimate tension strength Us=430Mpa
- ✓ Design strength py=215Mpa if t<=16mm
- ✓ Shear strength =0.6py
- ✓ Axial strength=Pa=405Mpa
- ✓ Young's modulus=Est=2e11Mpa
- ✓ Density=7850kg/m^3.
- ✓ Poison's ratio=0.3
- ✓ Coefficient of thermal expansion α=1.2e-5/^{0}c.
- ✓ Aluminum (6063T5)
- ✓ Tension strength fu=160Mpa
- ✓ 0.2% tension proof stress f0.2=110MPa
- ✓ Bending limiting stress =90Mpa
- ✓ Shear limiting stress =55Mpa
- ✓ Bearing limiting stress=185Mpa
- ✓ Modulus of Elasticity E=70Gpa
- ✓ Density=2800kg/m3.
- ✓ Poison's ratio=0.33
- ✓ Coefficient of thermal expansion α=2.35e-5
- ✓ Material factor =1.2
- ✓ Bolts & Nuts: A2-70
- ✓ Tension strength =280Mpa
- ✓ Shear strength=265Mpa
- ✓ Bearing strength =280Mpa
- ✓ Limiting deflection :
- ✓ For Alum rails = Span /200

D. Wind Load Calculations

- ✓ Wind Load =150 KMPH (41.66m/sec)
  Code: BS-6399-2 [6]
- ✓ Wind load q_s= $(150/3.6)^2$*0.613Pa=1.064Kpa
- ✓ Size effect factor for external pressure=c_a=0.952
- ✓ The net pressure overall coefficient=C_{pe}=1.8
- ✓ Pressure P_e=c_a*c_{pe}*qs=1.823 Kpa
  (182kg/m^2)=.0018Mpa
  Or general formulae for wind pressure
- ✓ Wind load q_s= $0.6V^2$ where v=55.55m/sec [7]
- ✓ considering sizing factors, net pressure coefficient
- ✓ Pressure=0.0018Mpa

E. Load Calculations

- ✓ Seismic design load:
- ✓ Seismic fortification intensity:7
- ✓ Design basic acceleration of ground motion: 0.15g
- ✓ Dynamic coefficient βE=5 αmax=0.15
- ✓ Weight of modules: wm=245N
- ✓ Area of Module A=1.93 m^2

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 2, Issue 7, January 2013

- ✓ Seismic design load

  Es= βE* max*wm=5*0.15*245=183.75N

Check for aluminum clamps:

The effect of wind suction was the most critical. So design load was taken wind suction only

- ✓ The shear force by wind suction

  Ft=1.2*pc*la*Lb=29184N
- ✓ Projected area=6.7*2.42=16.214m^2
- ✓ Design strength of aluminium clamp

  for shear = 82.2 N/mm^2.
- ✓ Thickness of clamp t=5mm
- ✓ Width of clamp b =40mm
- ✓ Number of clamp n=12
- ✓ Load on each clamp F1=29184.12=2432N
- ✓ Shear stress on each clamp

  τ =2432/(40*5)=12.16N/mm^2.

  Clamp is safe for shear
- ✓ Checking for bending:
- ✓ Maximum bending moment on structure:

  M max=wl^2/2
- ✓ Mmax=0.018*$(2424)^2$/2=5288N-mm
- ✓ Bending stress in the structure
- ✓ Stress = 5288/(40*5^2)/6=31.728 N/mm^2. This stress is also less than the allowable stress of the member

Checking for bolt strength:

- ✓ For M6 bolt effect area =20.1mm^2.
- ✓ Shear strength =265Mpa
- ✓ Tension strength =280Mpa
- ✓ Tension Strength capacity =As*pt=5628N
- ✓ Shear strength capacity=As*ps=20.1*265=5326.5N
- ✓ No of bolts: 8
- ✓ So total tension capacity of the bolts

  =8*5628=45024N which is higher than the total wind load

  So bolts are safe for the design.

F. Methodology

Step 1: Load calculation

- ✓ Wind load calculation
- ✓ Inertial calculation
- ✓ Torque calculations

Step 2: Design

- ✓ Sizing
- ✓ Strength parameters
- ✓ Gear calculations

Step 3: Modeling

- ✓ 3-D modeling using CATIA V5

Step 4: Analysis

- ✓ Analyzing by FEA using ANSYS

G. Inertia Torque Calculations

- ✓ Inertia Torque = Iα
- ✓ Here I=Mass moment of Inertia = 300*r^2
- ✓ Here = r= 1.1m
- ✓ I=300*1.1^2=363kg-m^2.
- ✓ Velocity 0.25^0/sec.=0.25*π/180=0.004rad/sec
- ✓ Acceleration =(Vf-vi)/t = 0.004rad/sec^2
- ✓ Inertia torque Ti=363*.004=1.452N-m

H. Drive Torque Calculations

- ➢ Total pressure area=16.2m^2
- ➢ Due to symmetry pressure load for torque calculation

  F=8.1*0.0018*106=14580N
- ➢ So maximum drive torque requirements 1.435*14580 N-mm considering extreme condition.
- ➢ Td=1.435*14580=20922.3N-m
- ➢ So total torque requirements

  Tt=20922.3+2.471=20924.771N-m

  Considering the frictional torque as 10% total torque
- ➢ Total torque requirement

  Tt=20.924.771+0.1*20924.771=~23017N-m
- ➢ Load acting on the ball screw Fd=14580

  Considering allowable stress for ball screw material as 140N/mm2.
- ➢ Minimum section required

  A=14580/140=104.14mm2.
- ➢ So diameter of ball screw required

  d=πd2/4=104.14mm2
- ➢ Minimum diameter of ball screw Required d= 11.51mm

So a standard size of 12mm can be selected. The two drafting models show dimensional views of the problem. All the dimensions are represented in mm. Major parts, moving plate, sector gear and base frame represented. Due to preliminary stages of design remaining parts are cad modeled to find the required dimensions after analysis.

3 Dimensional view of the model

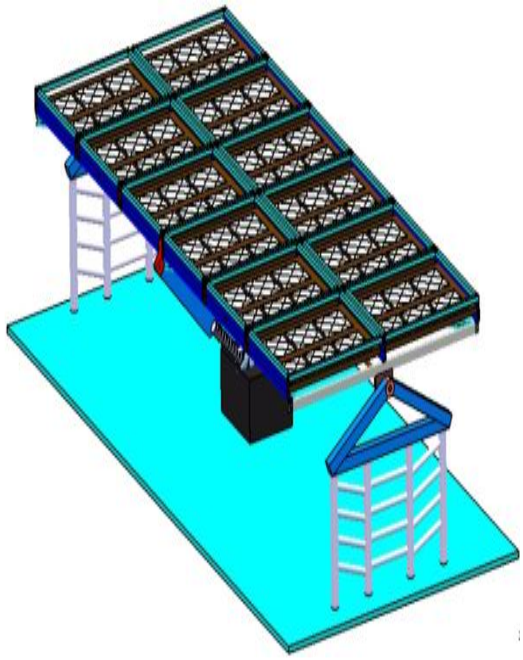

Fig: 9 Dimensional Model of Motion Simulator Meshed Model

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 2, Issue 7, January 2013

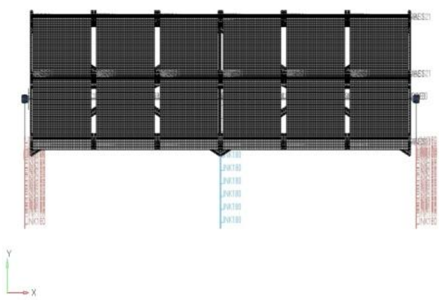

Fig 10 : Meshed Model of the Required Components

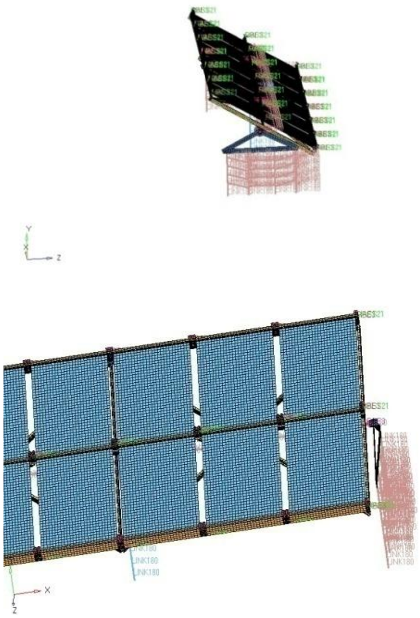

Fig 11 : Meshed Model

Assumptions:

- ✓ The material is assumed to be isotropic and homogenous
- ✓ The analysis is done with in elastic limits
- ✓ All approximations applicable to Finite element methods are applicable for the problem.
- ✓ 4 noded tetrahedral elements are used for representation.
- ✓ Coupling constraints are used for connection between the members.

VI. RESULTS & DISCUSSION

The solar panel supporting structure is analyzed for given and calculated loads. The results are as follows.

Initial Structural Analysis: The figures shows developed maximum deformation in the structure. Maximum deformation is around 2.43mm. The red region shows maximum deformation side in the problem. The base is completely constrained. Maximum loading condition is considered for analysis. 30 ^{0} inclination is considered for analysis as projected area is more in this view.

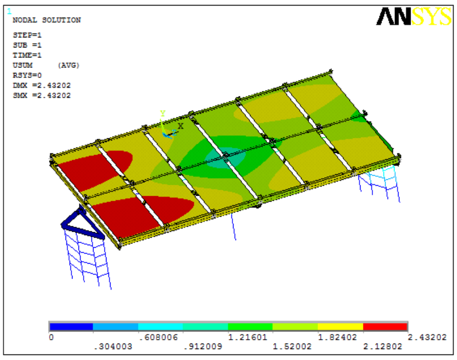

Fig 12: Deformation Due To the Loads

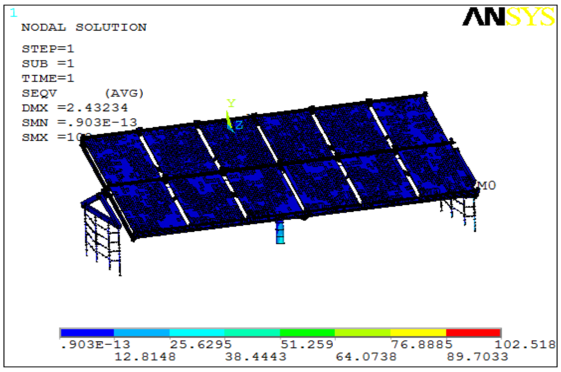

Fig 13: Vonmises Stress in the structure

The figure shows vonmises stress in the structure. Maximum vonmises stress is around 102.518Mpa taking place at the column panel intersection. Remaining sections are less than the allowable stress. Even the stress of 102.518 Mpa is less than the allowable stress of 140Mpa. So structure can be optimized using ANSYS design optimizer. The stresses in different components are as follows.

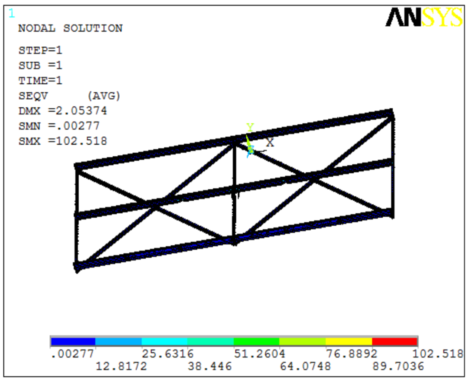

Fig 14: Frame Stress

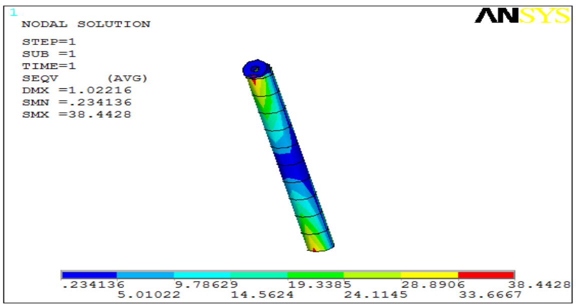

Fig 15: Central Colum Stress

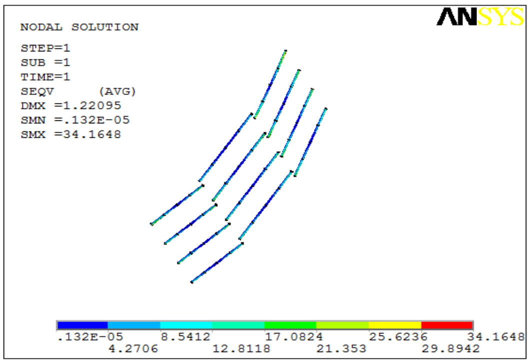

Fig 16: Horizontal Supports stress

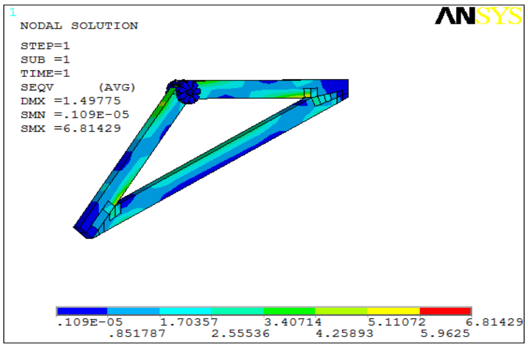

Fig 17: Side Support Stress

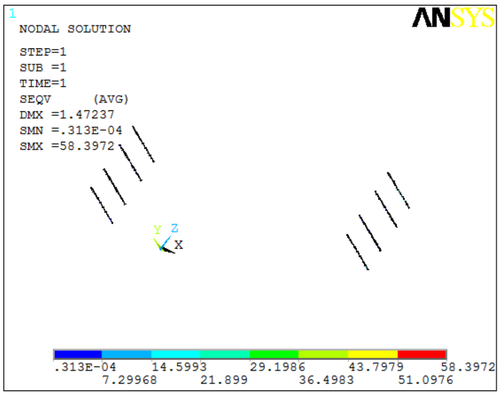

Fig 18: Vertical column stress

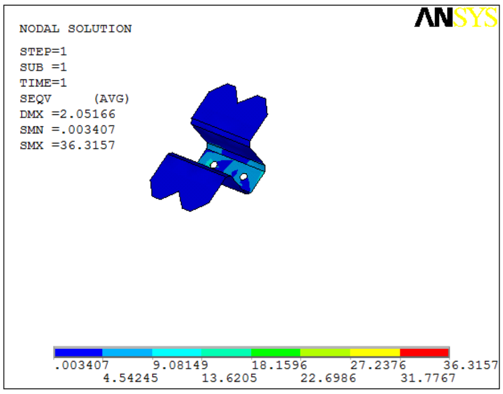

Fig 19: Stresses in Clamps

Design Parameters: Totally 6 design variables are considered for design optimization based on the developed stresses in the members. Design optimizer requires geometrical variables as design variables and stage variables (Stress & Deflection) and an objective objection. In the problem, weight is considered as objective function.

The Design Optimizer Results Are As Follows.

Variables/ Feasible sets	SET 1	SET 2	SET 3	SET 4
MAXD(SV)	2.4320	3.1348	2.8518	3.3705
MAXS(SV)	55.255	97.520	84.764	100.35
T1(DV)	50.000	43.887	41.007	25.497
T2(DV)	5.0000	3.3716	4.1222	3.3716
T3(DV)	15	8.8799	5.9675	8.8774
T4(DV)	16.5	12.655	11.242	12.966
T5(DV)	50	31.502	25.341	14.47
T6(DV)	15	10.415	3.7152	4.9697
WT(OBJ)	4508	4213.9	4216	4042.3
Variables/ Feasible sets	**SET 5**	**SET 6**	**SET 7**	**SET 8**
MAXD(SV)	3.933	4.451	4.6643	4.7108
MAXS(SV)	108.24	126.48	137.89	140.05
T1(DV)	13.779	10.872	10.28	10.13
T2(DV)	3.2172	2.8934	2.7329	2.705
T3(DV)	8.8753	8.841	8.8283	8.8244
T4(DV)	12.963	15.663	16.299	16.433
T5(DV)	11.233	10.367	10.158	10.098
T6(DV)	3.5111	3.1561	3.0598	3.0337
WT(OBJ)	3955.3	3907.3	3888.6	3885

Variables/ Feasible sets	SET 9	SET 10	*SET 11*
MAXD(SV)	4.7029	4.6927	4.6802
MAXS(SV)	140.04	140.11	140.12
T1(DV)	10.182	10.268	10.363
T2(DV)	2.705	2.705	2.705
T3(DV)	12.161	6.0218	5.3442
T4(DV)	15.539	16.425	16.415
T5(DV)	10.094	10.084	10.082
T6(DV)	3.0315	3.0266	3.025
WT(OBJ)	3889.6	3881.4	3880.9

Best set results are as follows.

Variables/ Feasible sets	SET 11
MAXD(SV)	4.6802
MAXS(SV)	140.12
T1(DV)	10.363
T2(DV)	2.705
T3(DV)	5.3442
T4(DV)	16.415
T5(DV)	10.082
T6(DV)	3.025
WT(OBJ)	3880.9

Optimized Set

Final set results are as follows.

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 2, Issue 7, January 2013

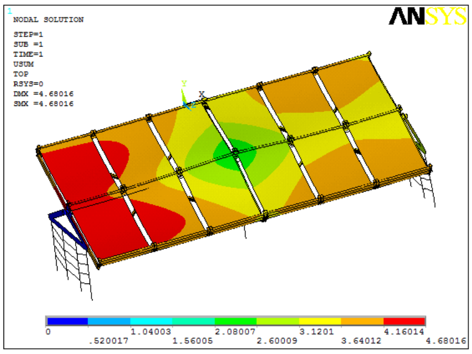

Fig 20: Final Deformation in the Problem

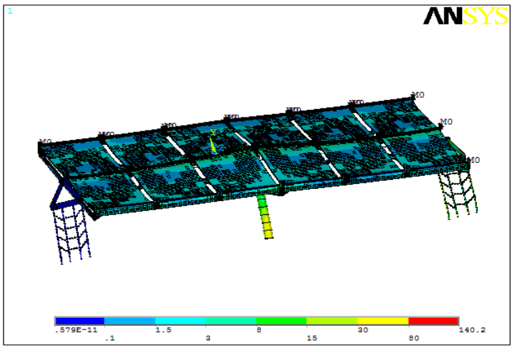

Fig 21 : Final Stress in the Structure

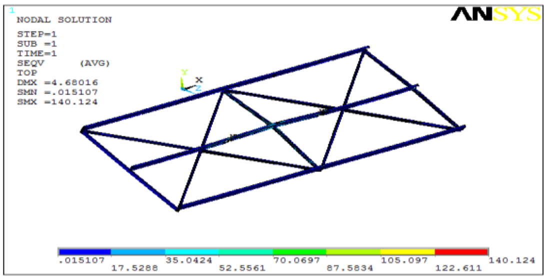

Fig 22: Final Stress in the Frame

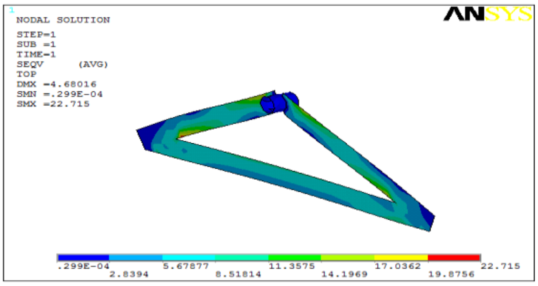

Fig 23 : Side Frame Stress

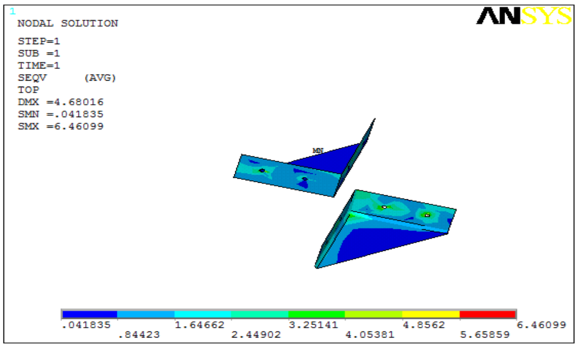

Fig 24 : Stresses in Clamps

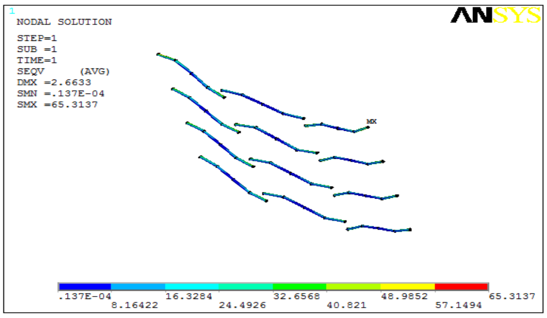

Fig 25 : Stresses in Horizontal members

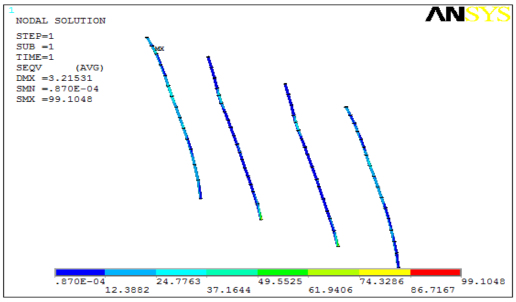

Fig 26 : Stresses in Vertical Members

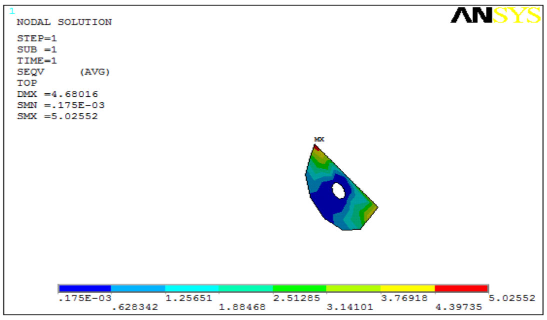

Fig 27 : Support Member Stress

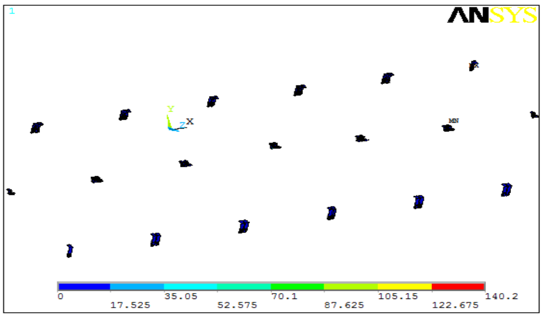

Fig 28 : L-Clamps Members' Stress

Discussion

Design optimization is carried out using ANSYS optimizer tool for better stress distribution and reduction in weight. Initial analysis results shows stress and deflections are well below the allowable deflection (6mm) and stress (140Mpa) of the structure. So the structure is based on selecting suitable design parameters and state variables. Weight is considered as the objective function of the problem. Total of 6 design variables and 2 state variables are selected based on the stress and deflections on the structure based on the initial analysis. Total of 11 feasible sets are obtained using design optimization tool. Final weight of the structure is reduced from initial 4508kgs to 3880.9kg. So a reduction of 13.9% is obtained using the design tool. The graphs shows deflection

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 2, Issue 7, January 2013

and stresses are increasing with the weight reduction. This is clear from the fact that, reduction in cross section reduces moment of inertia and in-turn increases both deflection and stress in the structure. Also parameter variation with the design tool also represented. So Finite element based design tools are useful in obtaining structural optimization which will reduce the cost of the structures.

VI. CONCLUSIONS & FURTHER SCOPE

A. Conclusion

The solar panel support structure is design optimized using computer based Finite element software ANSYS. The results summary is as follows.

- ✓ Initial wind load calculations are carried out based on the given wind speed.
- ✓ Initial calculations are carried out to find critical position of loading. Calculations are based on projected area of the problem.
- ✓ Calculations are done for structural members based on allowable stresses of the structure.
- ✓ The three modelling is carried out using modelling software CATIA and meshing is done using Hyper mesh software. The meshed file is imported to ANSYS in 'imp' file format for structural analysis.
- ✓ Initial analysis results shows stresses and deflections with in allowable range. So Optimization is carried out by identifying possible regions of optimization.
- ✓ Totally 6 design variables and 2 state variables are selected along weight as objective function; design optimization is carried out for the problem.
- ✓ The design optimization process shows 11 feasible sets satisfying the stress and deflection requirements for structural safety.
- ✓ A weight reduction of 13.9% can be observed in the problem. This reduction is also significant as the production costs, manufacturing costs and maintenance costs will reduce with less requirement of the material. Also dynamic effects in the structure also will reduce.
- ✓ The graphs are represented showing variation of weight, stress and deflection with iterations. Weight reduction with iterations can be observed along with increase in stress and deflection with reduction in thickness.

B. Further Scope

- ✓ Dynamic Analysis can be carried out
- ✓ Detailed design analysis can be carried out
- ✓ Possible thermal expansion problems can be accommodated in the analysis
- ✓ Possible electrical interference in the problem can be studied.
- ✓ Possible spectrum load analysis can be carried out.
- ✓ Topology optimization can be carried out.
- ✓ Buckling analysis can be carried out

REFERENCES

[1] G.S.G. Beveridge and R.S. Schechter, Optimization : theory and practice, McGraw-Hill, New York , 1970

[2] J.L. Kuester and J.H. Mize, Optimization techniques with FORTRAN, McGraw-Hill, New York, 1973.

[3] M.J. Panik, Classical Optimization: foundations and extensions, North-Holland Publishing Co., Amsterdam, 1976.

[4] D. Koo, "Elements of Optimization" Springer-Verlag, New York, 1977.

[5] D.G. Carmichael, Structural modeling and Optimization, Ellis Horwood Chichester, 1981.

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

[7] Y. M. Xie and G. P. Steven, "A simple evolutionary procedure for structural Optimization," Comp.Struct. Vol. 49, pp. 885-896, 1993.

[8] Concepts and Application of Finite Element Analysis Rober D. Cook, David S – JohWiely& Sons Pte. Ltd. Fourth Edition 2003.

[9] Finite Elements in Engineering – Tirupathi R. Chandrupatla, Ashok D. Belegundu, Prentice – Hall of India Pvt. Ltd, 2003.

[10] Arrora, J. "Introduction to Optimum design", 2nd ed., Academic Press, 2004

[11] Ravindran A, eklaitis G.V. Ragsdell K.M., "Engineering Optimization: Methods and Applications", John Wiley Sons, UK,2006.

[12] 12 .BS6399: Part 2: British Standard Code of Practice for Wind Loads, BSI, 1997.

[13] Prof. K. Lingaiah, Machine Design Data Hand Book Suma Publishers, Second Edition, 1989.

[14] www.google.com.

[15] www.matweb.com.

AUTHOR BIOGRAPHY

Mr. Ravindra Naik is currently working as assistant professor in Dept. of Mechanical Engineering, AGMR Engineering and technology, Varur, Hubli, and Karnataka, India. Ten Year industrial experience and one international journal published and one national conference given.

Mr. Vinayakumar B. Melmari is currently working as assistant professor in Dept. of Mechanical Engineering, Shaikh college of Engineering and technology, Belgaum, Karnataka, India. Two Year industrial experience and one international journal published and two national conferences given.

Mr. Adarsh Adeppa is currently working as assistant professor in Dept. of Mechanical Engineering, Bheemanna Khandre institute of technology, Belgaum, Karnataka, India. One international journal published and one national conference given.