Thermal Stress Analysis of D C Motor using Finite Element Method

Anandkumar S Malipatil
Department of Thermal Power engg.
Visvesvaraya Technological University
Gulbarga, India

P Ravinder Reddy
Department of Mechanical engg.
CBIT
Hyderabad, India

Abstract :- **The rating of electrical devices such as machines and transformers is often determined by Mechanical and thermal considerations. For example the maximum winding current is typically determined by the maximum temperature, which the insulation can withstand without damage as excessive loss of life. Similarly the maximum speed of a motor or generator is typically determined by mechanical considerations related to the structural integrity of the rotor or performance of the bearings. In converting from electrical to mechanical energy, the wastage of energy is inevitable, some of energy being degraded into heat. In dynamo electrical machinery, loss of energy occurs in electrical circuits and those portions of magnetic circuits that are subjected to varying magnetization. The temperature rise resulting across the sections named as rotor and stator is therefore a major factor in the rating of an electric motor. In order to keep the operating temperature within the limits, the electric motor should dissipate the heat at the same rate as it was produced. As long as the temperature rise does not exceed a specified value, the actual thermal condition of the motor mainly influence how long the motor will lost, because the life of the motor highly depends on the life of the insulation. A sustained 10^0c increase temperature reduces the insulation life approximately 50%. After detailed analysis, insulation thickness of 0.22mm is predicted to provide best temperature distribution for rotor.**

A part of the motor is to be simulated with reasonable assumption. The analysis is carried for two cases those are with varying insulation thickness and varying heat transfer coefficients. In the present work thermal as well as structural analysis was carried out for a fan cooled D.C. motor and also to show how a commercially available software ANSYS can be used for such analysis, because the temperature distribution inside the motor is essentially a diffusion process and it is very difficult to analyze it precisely because of three dimensional effect and imponderable parameters such as the thermal contact resistance between the materials. The average temperature rise and various losses of different parts of a D.C. Motor has been provided by *Integrated electrical company, Bangalore*.

Keywords*: *Rotor, Stator, Hypertherm, Von misses.

NOMENCLATURE

C_v	Specific heat at constant volume
K	Stiffness matrix
M	Volume integral
S	Cooling surface in m^2
T	Temperature
c	Cooling coefficient
e	Thermal energy
P	Heat developed in W
P_d	Heat conduction per m^2
q_a	Applied heat flux
q_c	Conductive heat flux
q_j	Heat flux
q_s	Heat sources
q_v	Radiative Heat flux
u	Velocity
ρ	Density
μ	Dynamic viscosity
ρ_i	Resistivity of materials
λ	Specific heat dissipation
Θ	Temperature difference
Φ	Dissipation term

1. INTRODUCTION

Heat transfer is as important as electro-magnetic and mechanical design of an electric motor. The analysis of heat transfer and fluid flow in motors is actually more complex, more nonlinear, and more difficult than the electro-magnetic behavior. It is often dealt with by means of simplified equivalent circuits, and rarely receives the detailed analysis lavished on electro-magnetic aspects.

Perhaps there is some justification for using approximate methods for heat transfer, when exact methods are pursued for Electro-magnetic design. The Electro-magnetic design determines the geometry of laminations, which are cut to fine tolerances. Their geometry and thickness, together with their materials properties and the design of the winding, determines whether or not the motor will deliver the required torque. They also determine the precise voltage and currents that will be experienced by the power semiconductors in the controllers. All of these items critically affect the manufacturing cost. By contrast, as long as the temperature rise does not exceed a nominal or specified value, the actual thermal condition of the motor mainly influence how long the motor will last, and has only a marginal influence on whether the torque can be delivered. Further more, the motor designer often has little control over the ultimate thermal environment of the motor, so there may be little point in attempting exact thermal analysis.

There are two major aspects to the thermal problem. One is heat removal, and the other is temperature distribution within the motor.

In most motors a mixture of air convection, conduction to the frame mountings, and radiation removes heat. In highly rated machines direct cooling by oil mist or even liquid coolants can be used to achieve the power density.

The temperature distribution within the motor is essentially a diffusion problem. It is difficult to analyze precisely, because of three dimensional effect and 'imponderable' parameters such as thermal contact resistance between, say, a bunch of copper conductors and a slot liner.

The most important aspect of the temperature distribution problem is finding the hottest temperature in the motor, given a certain distribution of losses and a known rate of heat removal. The steady-state temperature distribution can be very different from the transient distribution, and different methods of analysis may be needed for the two cases. The main reasons for limiting the temperature rise winding and frame of a motor are:

1. To preserve the life of the insulation and bearings,
2. To prevent excessive heating of the surroundings, and
3. To prevent injury caused by touching hot surfaces.

The life of the electrical insulation can be predicted only by statistical methods, but in broad terms the life is inversely related to the temperature, and the relationship is exponential, so that a sustained 10°c increase in temperature reduces the insulation life by approximately 50%.[1]

The increase in winding temperature increases the resistivity of the winding: a 50°c rise by 20%, and a 135°c rise by 53%, increasing the I^2R losses by the same amount if the current remains the same. The increase in resistance is used in test procedures to determine the actual temperature rise of the winding, but this obviously an average temperature; hot-spot temperature can be 10-20°c higher. This project deals with finite element method to predict the temperature distribution in an electric motor subject to generalized thermal loading, which consists of uniformly distributed heat sources and convection boundary condition. Here we have a motor of known geometry and thermal loadings. We also know by experiment, the maximum temperature rise in the motor when it runs. Here air is forced through the motor for cooling purpose. Though it is an electrical machine we consider only mechanical aspect of the motor and we have given more stress on the rotating portion (rotor) of the motor, as it is the critical part of a motor

With this attempt of analysis heat dissipation rate is enhanced by reducing its insulation thickness and increasing its airflow rate inside the holes. By that we may arrest the deterioration of the insulating material and may increase the life of insulating material. When the life of insulating material is improved, the performance of the motor may be improved. Its durability is also improved by some extent.

The objective of this work was to study the temperature distribution inside a fan cooled D.C. Motor and also to show how commercially available software ANSYS can be used for this analysis. The present work is an attempt to specify the heat transfer coefficient inside the motor and to find out the temperature distribution if the insulation thickness inside the motor is changed and also if air flow rate changed.

2. COMPUTATIONAL MODEL

Geometric modeling is concerned with the use of a CAD system to develop a mathematical description of the geometry of an object. The mathematical description, called a model, is contained in computer memory. This permits the user of the CAD system to display an image of the model on a graphics terminal and to perform certain operations on the model. These operations include creating new geometric models from basic building blocks available in the system, moving the images around on the screen, zooming in on certain features of the image and so on. These capabilities permit the designer to construct a model of new product (or its components) or to modify an existing model.

Three various types of geometric models used in computer aided design, one classification distinguishes between two-dimensional and three-dimensional models. Two-dimensional models are best utilized for design problems in two dimensions, such as flat objects layouts of buildings. In the first CAD systems developed in the early 1970's two-dimensional systems were used principally as automated drafting systems. They were often used for three-dimensional objects, and it was left to the designer or drafts man to properly construct the various views of the object. Three-dimensional CAD systems are capable of modeling an object in three dimensions. The operations and transformations on the model are done by the system according to user instructions in three-dimensions. This is helpful in conceptualization of the object since the true three-dimensional model can be displayed in various views and from different angles. [12]

2.1 Governing equation:

The governing equation of two dimensional steady state heat conduction with heat generation is given by

$$\frac{\partial q_x}{\partial x}+\frac{\partial q_y}{\partial y}-Q=0$$

By substituting q_x = $k\frac{\partial T}{\partial X}$ $and\ q_y - k\frac{\partial T}{\partial Y}$ in the above equation we get,

$$\frac{\partial}{\partial X}\left[k\frac{\partial T}{\partial X}\right]+\frac{\partial}{\partial Y}\left[k\frac{\partial T}{\partial Y}\right]+Q=0$$

With the boundary conditions

$T = T_o$ on ST, q_n = qo on S_q and $q_n = h\,(T - T\infty)$

Where T_0 is the specified temperature, q_n is the specified heat flux and q_n is the conversion heat flux. In Galarkin's we seek an approximate solution of T such that,

$$\iint_A\left[\frac{\partial}{\partial X}\left(k\frac{\partial T}{\partial X}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)\right]dA\iint_A \phi QdA=O$$

For every ϕ (x, y) constructed from the same basis function as those used for T and satisfying ϕ - 0 on S_T. After integration,

$$\iint_A \left\{ \frac{\partial}{\partial x}\left(\phi k \frac{\partial T}{\partial X} \right) + \frac{\partial}{\partial y}\left(\phi k \frac{\partial T}{\partial Y} \right) - \left[k \frac{\partial \phi}{\partial X}\frac{\partial T}{\partial X} + k \frac{\partial \phi}{\partial Y}\frac{\partial T}{\partial Y} \right] \right\} dA + \iint_A \phi Q dA = 0$$

Since $S=S_T+S_Q+S_C$, $\phi=0$ on S_T, $q_n = q_0$ on S_q and $q_n = h(T-T_\infty)$ on S_C and equn. 4 becomes,

$$\int_{Sq} \phi q_O \, d S - \int_{Sc} \phi h (T - T\infty) dS \int \int_A \left[k \frac{\partial \phi}{\partial X}\frac{\partial T}{\partial X} + k \frac{\partial \phi}{\partial Y}\frac{\partial T}{\partial Y} \right] dA + \iint_A \phi Q dA = 0$$

Now, we introduce the isoperimetric relations for the quadrilateral elements such as $T = NT^e$. Further, we denote the global virtual temperature vector as ψ whose direction equals number of nodes in the finite element model. The virtual temperature distribution within each element is interpolated as,

$$\Phi = N\psi$$

In matrix form the above equation becomes

$$KT = R$$

3. GEOMETRY AND BOUNDARY CONDITIONS

Construct a geometric model of the motor. For the rotating part a 90-degree sector was taken and for the stator part a 45-degree sector, since the respective sections are symmetric. As this is a very complicated geometry and there are number of different materials in close vicinity at the edge of the rotor section the grid should be entered very carefully. First by taking a partial annulus command draw one annulus (r1=165, r2=65, θ1=90, θ=180) and draw circles (d=16) at respective positions. The angular positions of circles can be calculated manually. Then for slot section, it is not in a proper shape so modified as rectangle as shown in figure 3.2. Which will not affect the analysis much. The rectangle positions are given in the appendix1. Slot consists of epoxy, insulation and copper and the dimensions of the slot section as shown in figure 3.1 with the dimensions draw rectangles and overlap them by using different commands. But as the thickness of insulation is too small (0.3 mm), its orientation is also modified as shown in figure 3.3. After drawing all rectangles, the final model will be obtained. Figure 3.4 shows the final model of the rotor section. Similarly the stator section is modeled as shown in figure 3.6. After generating model, then select a proper element (PLANE77) for meshing.

Initial Condition:

Initial values of variables may be specified for the overall computational domain or parts of it. For our case, the variable is temperature and it is assumed to be atmospheric (31 ^{0}C.)

Direchlet Boundary Condition:

The specified or prescribed (Dirichlet type boundary condition) values of variables may be given at nodes. Boundary condition for velocity, temperature, stream function or velocity potential may be applied depending on the formulation. Nodal or elemental heat sources can also be applied. For our case we put only elemental heat generation as prescribed boundary condition.

Numan Boundary condition:

If gradient of a variable on the surface of the computational domain is given, it is called Numan boundary condition. In our case we put convective heat transfer as a Numan boun

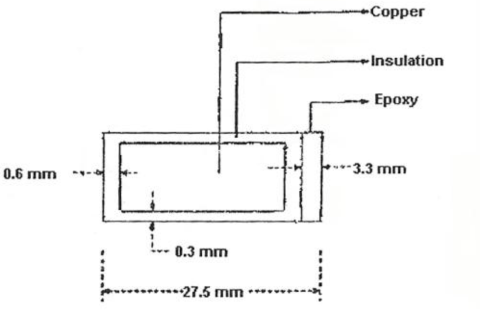


Fig. 3.1 Geometry of Typical slot

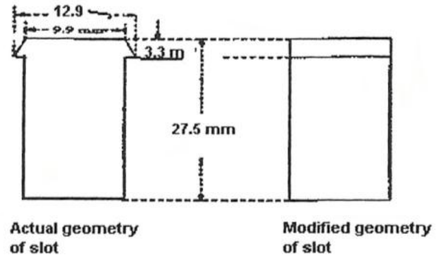


Fig.3.2 Geometry of typical slot

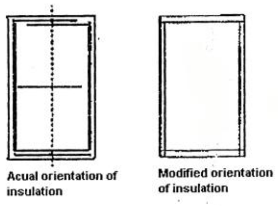


Fig.3.3 Modified orientation of insulation

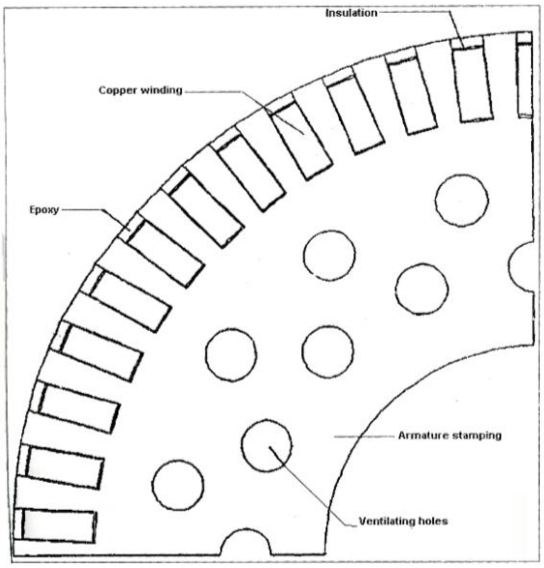


Fig.3.4 Geometry of a 90-degree rotor section

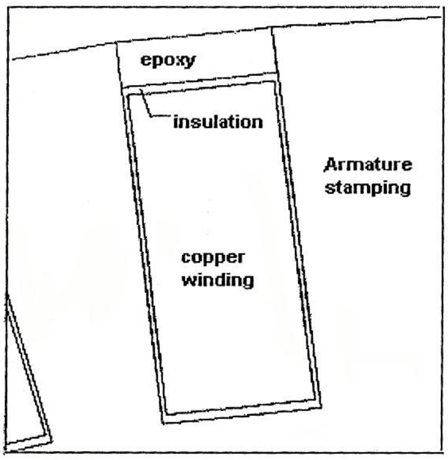


Fig.3.5 Enlarge view of slot

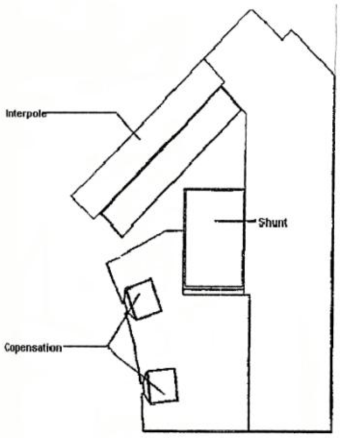


Fig.3.6 Geometry of a 45-degree section stator

4. SOFTWARES USED

Finite Element Analysis (FEA) has become the most popular choice of practicing engineers to solve real life problems in stress, vibration, heat flow and other fields of analysis. ANSYS is general purpose Finite Element Analysis software, which enables engineers to perform the following tasks.

- Build computer models or transfer CAD models of structures, products, components, or systems.
- Apply operating loads or other design performance conditions.
- Study the physical responses, such as stress levels, temperature distributions, or the impact of electromagnetic fields.
- Optimize a design early in the development process to reduce production costs.

A typical ANSYS has three distinct steps:

1. Pre-processor(build a model)
2. Solution(apply the loads and obtain the solution)
3. Postprocessor(review the results)

Temperature distribution Analysis

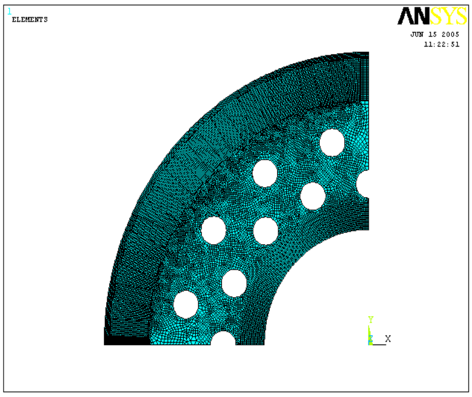


Fig.4.1 Finite element mesh of rotor section

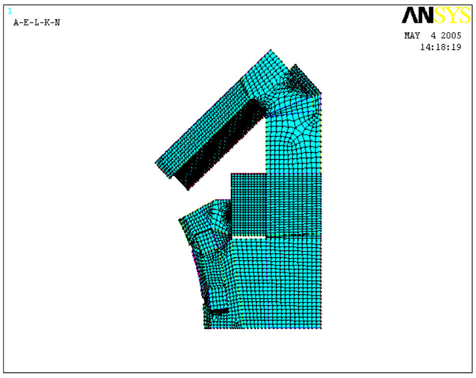


Fig.4.2 Finite element mesh of Stator section

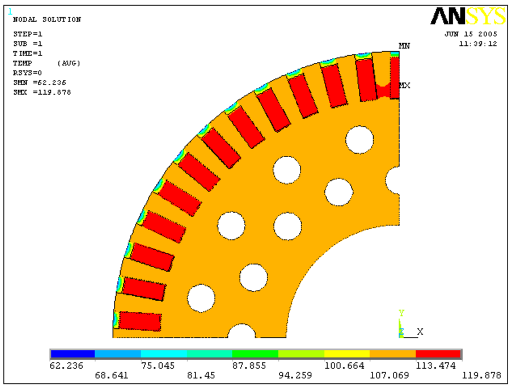


Fig.4.3 Temperature distribution inside the rotor for both hi=50 W/m^{2}K and ho=200 W/m^{2}K; t=0.3mm

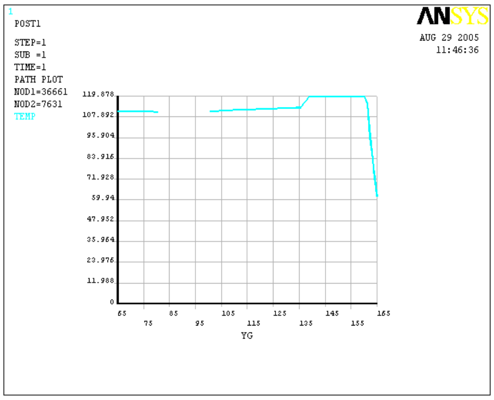


Fig.4.4 Temperature distribution along y-axis for both hi=50 W/m^{2}K and ho=200 W/m^{2}K; t=0.3mm

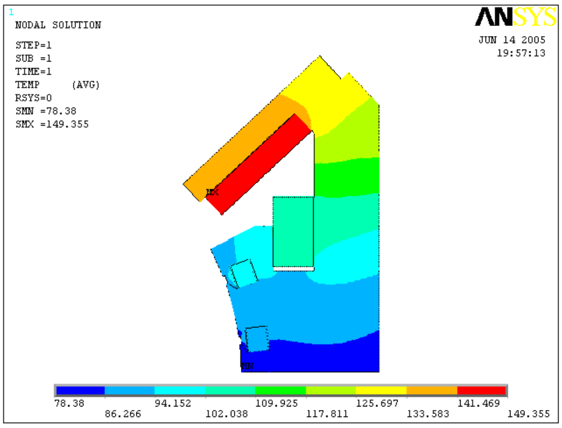


Fig.4.5 Temperature distribution of the stator for both hi=75 W/m^{2}K and ho=10 W/m^{2}K

After a successful run of ANSYS file the output file is obtained. This file gives the temperature contour by applying different commands in ANSYS. This contour shows the temperature distribution throughout the stator. For other values of outside 'h', the temperature variation values are given in a table. Also the experimental values are given in the following table.

Temp of I.P.	**Temp of Compensation**	**Temp of Shunt**
165	118	123

Table S: Observed value of Temp. Inside the stator

Inside h W/m^2 ^{0}C	**Temp of I.P.**	**Temp. of Compensation**	**Temp. of Shunt**
50	184	125	133.5
75	149.355	102.038	94.152
100	140	93	99

Table S Temperature distribution inside the stator for outside h=10 W/m^2 K

Inside h W/m^2 ^{0}C	**Temp. of I.P.**	**Temp. of Compensation**	**Temp. of Shunt**
50	182	120	130
75	153.3	94.5	106
100	139.5	92	98

Table S: temperature distribution inside the stator for outside h =15W/m^2 K

5. STRUCTURAL ANALYSIS

In many cases, engineers follow a thermal analysis with a stress analysis to calculate thermal stresses (that is stresses caused by thermal expansions or contractions). The structural analysis is to be carried out by switching the element from thermal to structural and giving the boundary conditions as like in thermal analysis.

1. Iron:

Modulus of elasticity = 211×10^9 N/m^2
Density = 7897 Kg/m^3
Thermal coefficient of expansion = 12×10^{-6}
Poisson's ration = 0.3

2. Copper:

Modulus of elasticity = 1.172×10^{11}N/m^2
Density = 8954 Kg/m^3
Thermal coefficient of expansion = 16.56×10^{-6}

Poisson's ration = 0.3

3. Modulus of elasticity of hypertherm = 150 N/mm^2

4. Modulus of elasticity of epoxy = 3000 N/mm^2

5. Modulus of elasticity of double glass mica = 5.516×10^{10} N/m^2

6. Modulus of elasticity of semicatherm = 4×10^4 N/mm^2

3.Rotor:

In the analysis of a rotor, the boundary condition applied are given below

- Displacement is given symmetry on lines at the edges.
- Temperature from thermal analysis by a result file.

After a successful run of ANSYS file the output file is obtained. This file gives the stress distribution by applying different commands in ANSYS. This contour shows the stress variation throughout the rotor. For other values of outside 'h', the stress distribution values are given in the table 1. Figure 1 shows the stress (von misses) distribution for outside h = 200 and inside h=50.

Inside value of h W/m^2 ^{0}C.	Outside value of h W/m^2 ^{0}C.	Rotor with an insulation (0.3mm)	Rotor with an insulation (0.15mm)	Rotor without insulation and epoxy
100	100	0.429-881.182	0.122-880.526	0.640-1184
50	200	0.0787-929.157	0.136-975.269	0.644-1222
15	300	0.0762-966.021	0.076-986.01	0.644-1222

Table 1: Stress (Von misses) distribution for different values of h

Stator:

In the analysis of a stator, the boundary conditions applied are given below

- Displacement is given symmetry on lines at the edges.
- Temperature from thermal analysis by a result file.

Results:

After a successful run of ANSYS file the output file is obtained. This file gives the stress contour by applying different commands in ANSYS. This contour shows the stress distribution throughout the stator. Figure 2 shows the stress (von-misses) distribution of stator for inside h=75W/m^{2}k and outside h=10W/m^2 k.

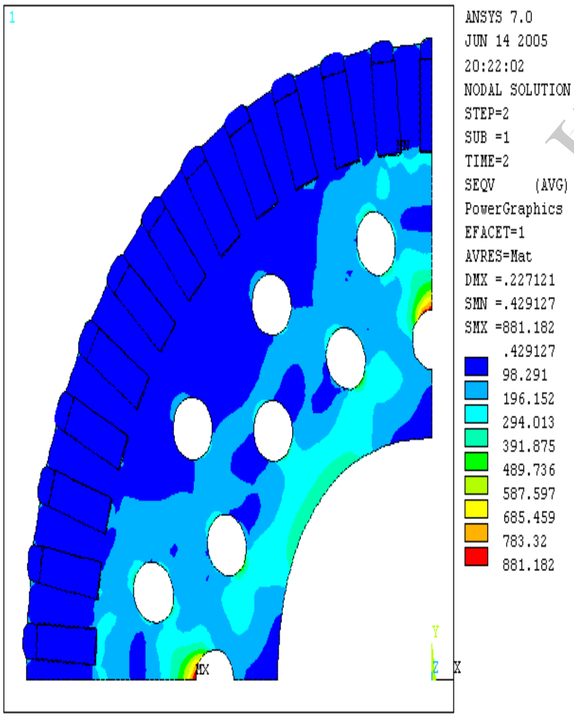


Fig.1 Stress distribution inside the rotor for both hi=50 W/m^2 K and ho=200 W/m^2 K; t=0.3m

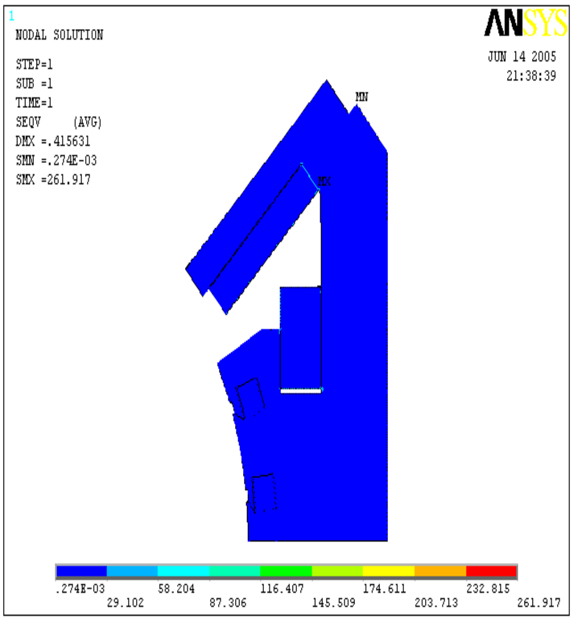


Fig.2 Stress distribution inside the Stator for both hi=75 W/m^2 K and ho=10 W/m^2 K; t=0.3m

6. RESULTS AND DISCUSSION

Temperature Distribution:

Heat distribution & Heat dissipation is one of the major factor while designing the D.C. Motor. For effective heat dissipation it is necessary to evaluate performance of insulation.. In the present analysis the study of full insulation thickness, half insulation thickness & without insulation & epoxy has been carried out.

Case No.	**h insight w/m^2 ^{0}c**	**h outside w/m^2 ^{0}c**	**Average Temp. from analysis in ^{0}c for rotor with full insulation thickness 0.3**	**Standard**	**% Difference**
I	100	100	95.547	90	6.16
II	50	200	91.057	90	1.17
III	15	300	88.651	90	-1.49

Table 6.1: For full Thickness Insulation (0.3mm)

Case No.	**h insight w/m^2 ^{0}c**	**H outside W/m^2 ^{0}c**	**Average Temp. from analysis in ^{0}c for rotor**	**Standard**	**% Difference**
I	100	100	95.3685	90	5.9
II	50	200	90.0925	90	0.102
III	15	300	87.1035	90	-3.2

Table 6.2: For Half Thickness Insulation (0.15mm)

Case No.	h insight w/m^2 ^{0}c	h outside w/m^2 ^{0}c	Average Temp. from analysis in ^{0}c for rotor	Standard	% Difference
I	100	100	109.513	90	21.68
II	50	200	117.18	90	30.2
III	15	300	118.252	90	31.39

Table 6.3: For Zero Thickness Insulation

These conditions for h inside and outside are standard for the D.C. Motor, as motor operates in between these cases so for the analysis of D.C. Motor, we can benchmark it.

After reviewing & comparing the results of average temperature with standard temperature as shown in above three tables i.e. 6.1, 6.2 & 6.3. There are two cases of h for half thickness where we have got % difference very less at specified heat transfer coefficient. We have plotted graph of average temperature Vs insulation thickness at specified heat transfer coefficient as shown in fig 4. After calculating minimum for three curves separately with use of successive quadratic estimation method (point estimation method), we got 0.22 mm as minimum insulation thickness so we recommend it as the best case for rotor. The table 6.5 shows that the values of temperature of various parts of stator are within the range and in good agreement with validated results [12].

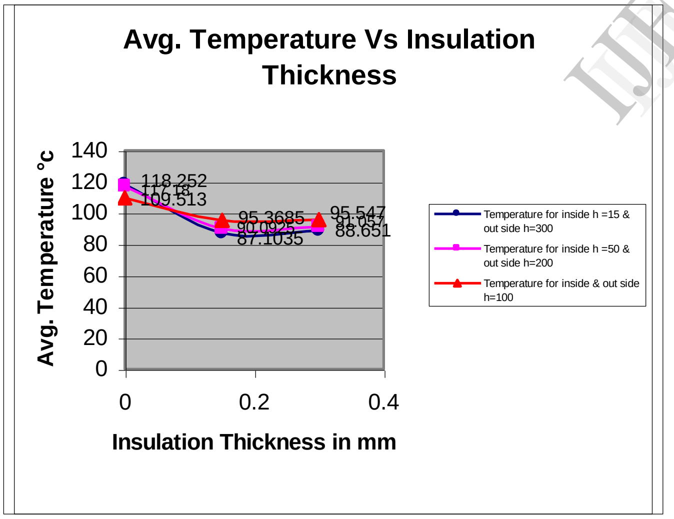
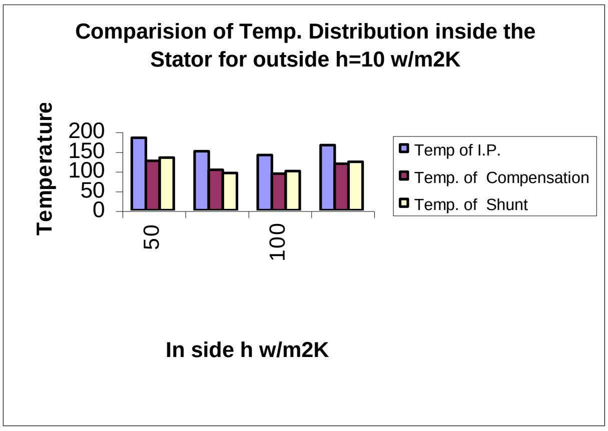
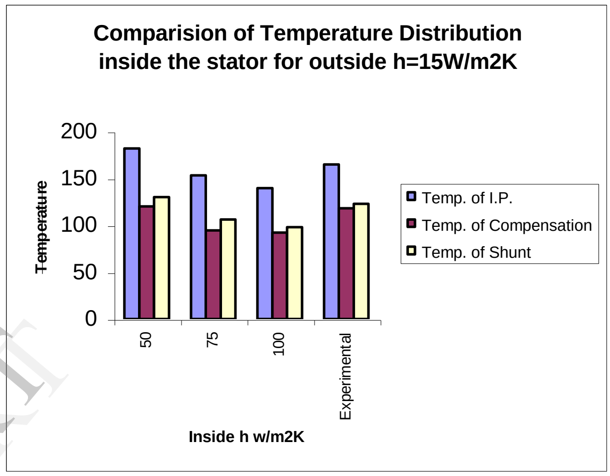


Figure 4: Graph for Avg. Temperature Vs Insulation Thickness for the three cases of h.





Stress analysis:

The high temperature induces thermal stresses. These thermal stresses are responsible for deformation of D.C. motor. For rotor thermal stresses induced has less effect on the deformation this is due to the high dissipation of heat in rotor which leads to lower the temperature and low thermal stresses. In case of stator, thermal stresses induced are higher due to stationery stator the deformation due to this is considerable. The von-mises stress developed is within range .It is less than the yield stress of material used for stator.

7. CONCLUSIONS

For this work, different section of the motor is simulated according to some reasonable assumptions. The following conclusions are obtained from the present investigation.

1. To specify the value of heat transfer coefficient in rotor section, several analysis were carried out for the different value of 'h' at both inside and outside boundary. From this analysis it is clear that for some particular value of h, the temperature inside the copper winding which is obtained by analysis has matched with the measured value. From these values of 'h', the inside heat transfer coefficient as 50

W/m^2 ^{0}C and outside heat transfer coefficient as 200 W/m^2 ^{0}C are taken.

2. From the results, by reducing insulation thickness did not help to reduce the temperature inside the rotor.
3. The only parameter that can reduce the temperature inside the rotor is the higher air flow rate so that heat transfer coefficient value becomes higher.
4. The temperature can be reduced only be increasing airflow rate or by increasing the heat transfer area, i.e., the size of the ventilating holes.
5. Insulation thickness of 0.22 mm was predicted to provide best temperature distribution for rotor.
6. For the stator section the exact measured temperature rise are not obtained. The reason for that is the non-uniform airflow around the stator. Also the insulation thickness around the shunt is about two times larger than the insulation around the compensation where as the conductivity of the insulating material is of same order. So it is quite possible that the temperature of shunt will be higher than the temperature of the compensation.
7. High temperature in motor induces the thermal stresses in motor . The intensity of thermal stress is low for both rotor as well as stator.

8. REFERENCES

1. Phil Coley, Motor Reliability Conference 2004,PDMA Corpn "Fault Zone Analysis of Stator".
2. A. E Clayton & N N Hancock, "The Performance and Design of Direct Current Machines" C B S Publishers & Distributors, Third Edition,2002
3. A. E. Fitzgerald Charles Kingsle, Jr. Stephen D. Umans "Electrical machinery" Tata Mc Graw-Hill, Sixth Edition, 2004.
4. Dr. P. S. Bimbhra "Electrical Machinery", Khanna Publishers, Seventh Edition,2004.
5. B.L. Theraja, A.K.Theraja, "A Textbook of Electrical Technology", Volume-2, S. Chand & Co. Ltd. 2001.
6. V N Mittle, "Basic Electrical Engineering", Tata McGraw-Hill Publishing Co. Ltd. ,2000.
7. Belegundu, Ashok. D.Chandupatla, Tirupathi.R., 'Introduction to Finite element in Engineering'.
8. Singiresu S. Rao "The Finite Element Method in Engineering", Butterworth Heineman Publishers, Third Edition, 2001.
9. ZeinKiewicz. O.C., "The Finite Element Method ", Third Edition,McGraw –hill Book Company, 1997
10. Klaus-Jurgen Bathe, "Introduction to Finite Element Method" Van Nostrand Reinhold, New York, 1972.
11. Desai/Abel, "Introduction to the Finite Element Method", C B S Publishers & Distributors,2002.
12. Mikell P. Groover, "Automation, Production systems, and computer-integrated manufacturing", Prentice Hall India, 1997.
13. Optimization for Engineering Design by Kalyanmoy Deb Prentice Hall, 5th edition 1999. New Delhi.