**AN ABSTRACT OF THE THESIS OF**

Brenda E. Thompson for the degree of Master of Science in Electrical and Computer Engineering presented on January 17, 1995. Title: Three-Dimensional Finite Element Design Procedure for the Brushless Doubly Fed Machine.

Redacted for Privacy

Abstract approved: ______________________

Dr. Alan K. Wallace

Brushless Doubly Fed Machines (BDFM) have potential advantages in variable speed generation and adjustable speed drive applications. The most significant of these advantages is a reduction in the power electronic converter rating, and therefore a reduction in overall system cost. Presently, efforts are being directed at optimizing the design of the BDFM and investigating areas of commercial feasibility. One possible aid in the investigation of design alternatives is finite element analysis.

Finite element analysis is a numerical method for determining the field distribution in a dimensional model. Finite element techniques have been successfully used for some time in the design of induction, reluctance and permanent magnet machines. However, the characteristics of the BDFM require adjustment of the finite element design procedure used for conventional singly-fed induction machines. In this thesis, a three-dimensional finite element design procedure for modeling the BDFM has

been developed. This design procedure avoids the difficulties previously associated with finite element modeling of the BDFM.

. The three-dimensional finite element design procedure developed in this thesis was used to model the 6/2 pole 5 horsepower BDFM laboratory machine. From the simulation results, the induced currents in the BDFM rotor bars were calculated.

In the course of investigating three-dimensional finite element analysis for the BDFM, two different commercially available finite element analysis software packages were examined and tested. The first was Maxwell 3D Field Simulator produced by Ansoft Corporation, and the second was MSC/EMAS (Electromagnetic Analysis System) and MSC/XL by MacNeal-Schwendler Corporation. These two software packages are compared and their advantages and disadvantages/limitations are discussed.

A tutorial for setting up and solving a three-dimensional BDFM model using MSC/XL and MSC/EMAS is presented. This goal of this tutorial is to guide a new user of MSC/XL and MSC/EMAS through the creation, setup, simulation, and analysis of a BDFM model. This tutorial contains condensed information included in the MSC/XL and MSC/EMAS program documentation provided by MacNeal-Schwendler. In addition, modeling techniques particular to the BDFM, which are not included in the program documentation, are described. This tutorial is applicable only to those individuals interested in learning how to use MSC/XL and MSC/EMAS in order to simulate a BDFM model.

# Three-Dimensional Finite Element Design Procedure for the Brushless Doubly Fed Machine

by

Brenda E. Thompson

A THESIS

submitted to

Oregon State University

in partial fulfillment of
the requirements for the
degree of

Master of Science

Completed January 17, 1995
Commencement June 1995

Master of Science thesis of Brenda E. Thompson presented on January 17, 1995

APPROVED:

Redacted for Privacy

Major Professor, representing Electrical and Computer Engineering

Redacted for Privacy

Chair of Department of Electrical and Computer Engineering

Redacted for Privacy

Dean of Graduate School

I understand that my thesis will become part of the permanent collection of Oregon State University libraries. My signature below authorizes release of my thesis to any reader upon request.

Redacted for Privacy

Brenda E. Thompson, Author

## ACKNOWLEDGMENTS

I would like to thank my major professor, Dr. Alan Wallace, for the amount of time and effort he has put into guiding me towards the completion of my thesis, for first suggesting that I get involved in working with finite element analysis of the BDFM, and for his willingness to help when I had questions about the BDFM or my finite element results. I would also like to thank Dr. Rene Spee and Dr. G. C. Alexander for their helpful suggestions about my finite element work and other topics. Special thanks to Dr. Alan Wallace, Dr. Molly Shor, Dr. G. C. Alexander, and Dr. David Butler, Graduate Council Representative, for being on my defense committee.

I would like to thank all the student members of the BDFM research and design group for their helpful suggestions about my work, during BDFM meetings and other times: Shibashis Bhowmik, Michael Boger, Bhanu Gorti, Tim Lewis, Sreekumar Natarajan, Arif Salim, Ernesto Weidenbrug, and Donsheng Zhou. I would like to thank Arif Salim for introducing me to finite element analysis and teaching me how to use Maxwell 2D Field Simulator by Ansoft Corporation.

I would like to thank James Neuner and Pat Lamers, members of the Macneal-Schwendler customer support staff, for their willingness to answer questions about Macneal-Schwendler's software in a timely manner. Their help was very valuable in learning how to use this software.

I would like to thank Tom Lieuallan for his help in setting up the HP workstation, installing the finite element software, and for his willingness to provide help whenever I had a computer question or problem.

I would like to thank Ed Lake for helping me to proofread my thesis, and for his help instructing me in the creation of the figures contained in this thesis with AutoCad.

Finally, I would like to thank my parents and close friends for their love, support and encouragement during my years at Oregon State University.

# Table of Contents

## List of Figures

## List of Tables

# List of Appendix Figures

## List of Appendix Tables

# Three-Dimensional Finite Element Design Procedure for the Brushless Doubly Fed Machine

## 1. Introduction

Due to recent improvements in power electronics, adjustable speed drives (ASDs) are being installed in increasing numbers in industrial applications. However, the majority of all industrial and commercial motors still operate at fixed speed. Similarly, fixed speed generators provide the bulk of the world's power supply, although many power sources could be more efficiently converted if variable speed generation (VSG) were used.

The transition from fixed speed systems to ASD and VSG systems has been delayed by the fact that the speed of AC machines is linked to their frequency. Although frequency control by power electronics has made significant advances in recent years, it still has two major obstacles preventing its more widespread application. First, the cost of electronic power converters is many times higher than the cost of the machines they control. Second, the electronic power converters pollute the power supply system with harmonics of voltage and current. Possible solutions to the harmonic problem serve to increase the cost of the system. Therefore, it is important to investigate possible methods for minimizing the ratings, and hence costs, of power electronic converters.

Ongoing studies at Oregon State University have shown the potential for many advantages by using a doubly-fed connection of the self-cascaded induction machine in ASD and VSG applications [1-5]. The use of a self-cascaded, or brushless doubly-fed machine (BDFM), in combination with a power electronic converter, can offer a number of advantages over conventional induction machines in ASD and VSG systems. These advantages include tolerance to power converter failure, controllable power factor, and reduced harmonic pollution. Most importantly, depending on the requirements of the application, a reduction in the power electronic converter rating, and therefore cost, can be achieved.

Presently, efforts are being directed at optimizing the design of the BDFM and investigating areas of commercial feasibility. One possible aid in the investigation of design alternatives is finite element analysis. Finite element analysis is a numerical method for determining the field distribution in a dimensional model. This thesis will emphasize only the electromagnetic field distribution in a three-dimensional model geometry. Finite element techniques have been successfully used for some time in the design of induction, reluctance and permanent magnet machines. From a finite element solution, important design quantities such as flux distribution, flux density, winding inductance, eddy currents, hysteresis losses, force, torque and losses can be calculated. Applying the finite element technique to the BDFM, however, has posed a number of difficulties. This thesis will present a design method for modeling the BDFM using finite element techniques.

# 2. Brushless Doubly Fed Machine

## 2.1 BDFM Characteristics

The stator winding connection of the BDFM is based on the work of L. J. Hunt [6] and later developments by Creedy [7]. The BDFM stator, as shown in Figure 2.1, consists of two sets of three phase stator windings of different pole numbers and

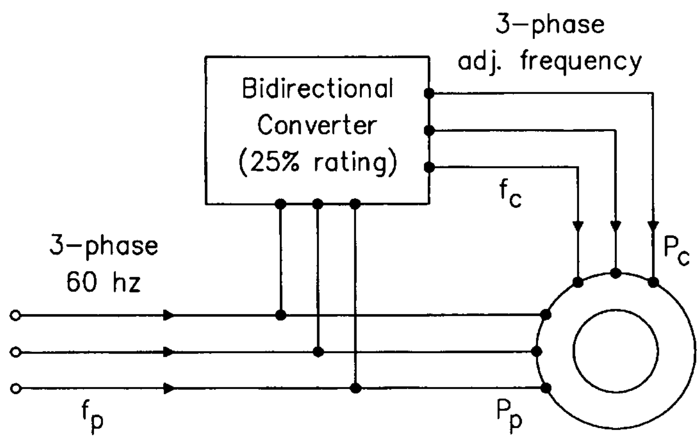


Figure 2.1: BDFM Stator Structure

different frequencies. These two separate sets of windings are wound on the same stator frame and share the same slots. One set of windings is the power winding, which is connected directly to the power supply system and which supplies the bulk of the machine power. The second set is the control winding, which supplies a fraction of the machine power through a power electronic converter. The advantage of the BDFM system over more conventional motors and generators is that most of the power flows directly between the machine and the power system. Therefore, the rating of the required converter should be a fraction of that required to process all of the machine power, thus reducing cost and induced harmonics of the power electronics.

The rotor design of the BDFM is based on work by Creedy and Broadway [7,8]. As shown in Figure 2.2, the BDFM rotor is an unique, cage type rotor with nested loops. Unlike the squirrel cage rotor of an induction motor, which has rings to short all the rotor bars on both ends, the BDFM rotor has a ring to short all the rotor bars only on one end while at the other end the bars are selectively shorted together to form distinct loops. The rotor design is mechanically simple enough to be die-cast, while at the same time having the capability of supporting two fields of different pole numbers and different frequencies from the stator.

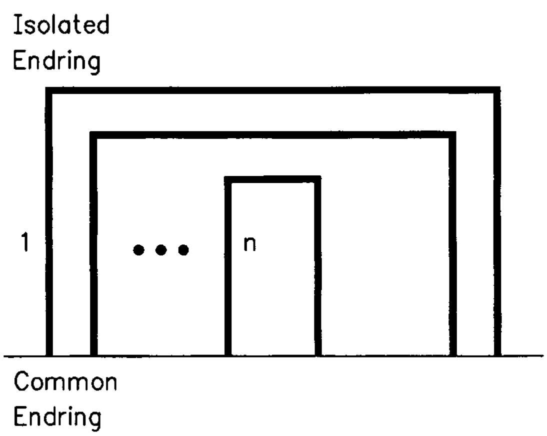


Figure 2.2: BDFM Rotor Structure

## 2.2 Basic Performance Equations

The number of loop groupings or rotor nests is determined by the sum of the pole-pairs of the power and control windings:

$$\text{Number of Nests} = P_p + P_c \qquad \text{(Equation 2.1)}$$

where $P_p$ is the number of pole-pairs of the power winding, and $P_c$ is the number of pole-pairs of the control winding.

The BDFM has all the robust, maintenance-free features of a squirrel-cage induction machine. In order to operate successfully, the BDFM must switch from operation as two induction motors in the same magnetic circuit (the "double-induction" mode) to a mode where the rotor field induced by one of the stator windings is locked together with the stator field of the other stator winding, and vice versa (the "synchronous mode").

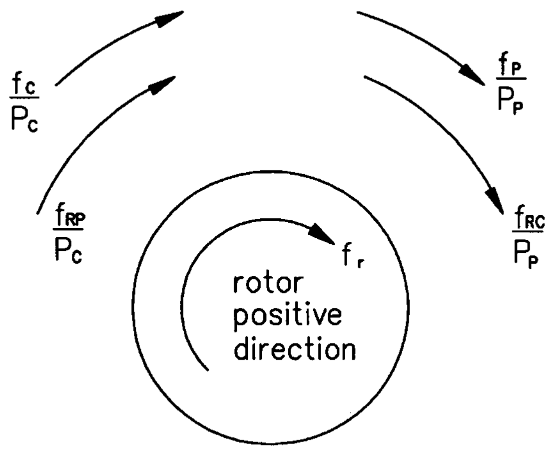


Figure 2.3: Velocities of Interacting Fields

In the synchronous mode the field interaction and the mechanical speed of the rotor, as shown in Figure 2.3, are related by [9]

$$\frac{f_p}{P_p} = f_r + \frac{f_{RC}}{P_p} \qquad \text{(Equation 2.2)}$$

and

$$\frac{f_c}{P_c} = f_r + \frac{f_{RP}}{P_c} \qquad \text{(Equation 2.3)}$$

where $f_p$ and $f_c$ are the frequencies applied to the power and control windings, respectively; $f_{RP}$ and $f_{RC}$ are the rotor frequencies induced by interaction with the fields of the power and control windings; $f_r$ is the mechanical rotational frequency.

For synchronous operation to be attained, it is required that

$$f_{RC} = -f_{RP} \qquad \text{(Equation 2.4)}$$

with the result that

$$f_r = \frac{f_p \pm f_c}{P_p + P_c} \qquad \text{(Equation 2.5)}$$

The control frequency, $f_c$, can either be positive (same sequence as $f_p$), or negative (opposite sequence to $f_p$). The electrical frequency of rotor currents in synchronous operation can be related to the system frequencies as [9]

$$f_{r,el} = f_p - P_p f_r = P_c f_r \mp f_c \qquad \text{(Equation 2.6)}$$

## 2.3 Applications of the BDFM

The BDFM can be used in place of commercial and industrial squirrel-cage AC induction machines. It is particularly suited for potential niche applications in ASD or VSG systems including, but not limited to, pump drives, wind power generation, and automotive alternators.

# 3. Finite Element Analysis Method

## 3.1 Definition and Concept

Finite Element Analysis (FEA) is a numerical method that is widely used to solve many engineering problems. One application of FEA is to solve for the electromagnetic fields in electrical devices. Electromagnetic fields represent the foundation of all electrical engineering. Maxwell's equations, a system of four coupled partial differential equations, serve as the basis for electromagnetic field calculations. The solution of these equations, however, is a very difficult task.

Often engineers approximate field behavior through abstract concepts. Much insight can be gained from analytic techniques and approximations. However, such techniques are useful only in relatively simple devices, and at some point approximations will fail. More often engineers require accurate solutions involving complicated materials, geometries, and loading conditions. For this reason, engineers are turning to numerical methods for answers to real life problems.

FEA is one numerical method of solving Maxwell's differential equations. There are several steps that make up the finite element method.

## 3.2 Finite Element Model

The first step in FEA is to specify a finite element model. The model geometry describes the size and shape of the device to be analyzed. The geometry is divided into subregions called finite elements. Elements may be irregular so that the modeling of complicated geometries is both easier and more accurate. Points where elements join are referred to as grid points. Material properties, excitations, and boundary conditions are applied to the finite element model. Material properties associated with elements represent the permittivity, conductivity, and permeability properties of the various materials in different regions of the model. Excitations such as currents are applied to the model. Boundary conditions are used to simulate physical behavior outside the model boundaries.

### 3.3 Solution of Maxwell's Equations

Maxwell's equations are the basis for electromagnetic field calculations. These four partial differential equations relate the space and time variation of electric and magnetic fields to material properties, and to excitations. They describe a broad range of behavior, including electrostatics, magnetostatics, eddy currents, waveguides, antennas, etc. Thus, Maxwell's equations form the basis for the analysis of virtually every electromagnetic device, from computer microcircuitry, to large power generators and transformers. Maxwell's equations [10] are traditionally written as:

$$\nabla \cdot \vec{D} = \rho_{free} \qquad \text{(Equation 3.1)}$$

$$\nabla \cdot \vec{B} = 0 \qquad \text{(Equation 3.2)}$$

$$\nabla \times \vec{E} = -\dot{\vec{B}} \qquad \text{(Equation 3.3)}$$

$$\nabla \times \vec{H} = \vec{J}_{cond} + \dot{\vec{D}} \qquad \text{(Equation 3.4)}$$

These four equations state the following:

- Gauss's law: the sources of $\vec{D}$ are free charge
- $\vec{B}$ has no sources
- Faraday's law: electric fields are induced by time-varying magnetic fields
- Ampere's law: the sources of $\vec{H}$ are conduction current plus the displacement current

The fields (electric field $\vec{E}$ and magnetic field $\vec{B}$) are the primary unknown quantities of interest in electromagnetic field analysis. However, there are three disadvantages to solving directly for the unknown vectors $\vec{E}$ and $\vec{B}$. First, the six unknown components of these two fields in three-dimensional space cannot be chosen arbitrarily because they are related through Maxwell's equations. Thus, the number of unknowns is larger than is actually needed. The second disadvantage is related to discontinuities in material properties. There are two well-known boundary conditions that must be met at such interfaces: (1) the normal component of $\vec{D}$ must be continuous across the interface; (2) the tangent component of $\vec{H}$ must be continuous. Any solution strategy that involves $\vec{E}$ and $\vec{B}$ must enforce these conditions at every interface. This requirement puts an unnecessary burden on numerical computations. The third disadvantage is that $\vec{E}$ and $\vec{B}$ may be infinite at sharp corners of certain materials. These infinite solutions cause numerical difficulties in computers.

Therefore, electromagnetic potential functions are introduced to eliminate the disadvantages of dealing with $\vec{E}$ and $\vec{B}$ directly. These potential functions are the magnetic vector potential $\vec{A}$, and a time-integrated electric scalar potential, $\Psi$. In terms of these potential functions, the electric and magnetic fields are given by [10]:

$$\vec{B} = \nabla \times \vec{A} \qquad \text{(Equation 3.5)}$$

$$\vec{E} = -\nabla \dot{\Psi} - \dot{\vec{A}} \qquad \text{(Equation 3.6)}$$

Maxwell's equations are rewritten in terms of these potential functions. The values of $\vec{A}$ and $\Psi$ at the model grid points are called degrees of freedom (DOFs). There are four DOFs at each grid point: three components of the vector potential and one component of the scalar potential.

The principle of virtual work is now used to formulate the overall energy stored in the solution region according to the following energy relationships:

$$W_H = \frac{1}{2}\vec{H} \cdot \vec{B} \qquad \text{(Equation 3.7)}$$

$$W_E = \frac{1}{2}\vec{E} \cdot \vec{D} \qquad \text{(Equation 3.8)}$$

The objective is to solve for the unknown potentials $\vec{A}$ and $\Psi$ by minimization of the energy function [10]. The problem volume is divided into finite elements. The energy associated with each element is computed in terms of the potential degrees of freedom; the results are then summed over the elements to represent the energy of the entire problem volume. When the energy function is set to zero, a single equation is obtained.

This equation is entirely equivalent to Maxwell's equations in their complete and general form. This equation is [10]:

$$[\varepsilon]\{\ddot{u}\} + [\sigma]\{\dot{u}\} + \left[\frac{1}{\mu}\right]\{u\} = \{\vec{J}\} \qquad \text{(Equation 3.9)}$$

where the vector {u} represents the four DOFs per grid point, the matrix [ε] represents permittivity, the matrix [σ] represents conductivity, the matrix $\left[\frac{1}{\mu}\right]$ represents permeability, and $\{\vec{J}\}$ is an excitation vector which represents the contributions of all model excitations.

The associated initial condition is:

$$[\varepsilon]\{\dot{u}_i\} = \{\vec{J}_i\} \qquad \text{(Equation 3.10)}$$

These matrix equations, which are equivalent to Maxwell's equations in their complete and general form, are solved using a formal series of matrix operations for the unknown potentials:

$$\{u\} = \begin{Bmatrix} \vec{A} \\ \Psi \end{Bmatrix} \qquad \text{(Equation 3.11)}$$

The numerical methods used to solve Equations 3.7 and 3.8 are specified by solution sequences [10]. Each sequence represents a particular mathematical technique. Thus, a particular application may be analyzed using several techniques, such as magnetostatic analysis, frequency response analysis, transient analysis, or eigenvalue analysis.

## 3.4 Data Recovery

Once a solution for the potential DOFs at each grid point have been obtained, the fields $\vec{E}$ and $\vec{B}$ are recovered within each element. Other quantities, such as electromagnetic energies, induced conduction currents, power losses, etc., can also be determined.

# 4. Finite Element Design Procedure for the BDFM

## 4.1 Methods of Modeling Induction Motors

Finite element analysis techniques have been used successfully for some time in the design of induction, reluctance and permanent magnet machines. Neglecting end effects, these machines can easily be investigated using two-dimensional finite element analysis.

In three-phase ac squirrel-cage induction motors, the rotor current distribution is one of the main unknown quantities of interest. One goal of finite element analysis for induction machines is to calculate the induced or eddy current distribution in the rotor conductors, as well as the total resulting magnetic field. This can be accomplished for an induction motor by doing an ac analysis of a two-dimensional cross-section of the machine.

### 4.1.1 Solution Frequency

In an ac induction machine analysis, the frequency selected as the solution frequency is the slip frequency, or frequency seen by the machine rotor [11]. For example, to model the machine at start up, the solution frequency would be 60 Hz. At high speed, a low frequency as seen by the rotor is used. The slip frequency is appropriate to use for an ac solution because currents are induced in the rotor conductors at the frequency seen by the rotor.

### 4.1.2 Periodic Boundary Conditions

Finite element simulations of ac induction machines, as well as other types of electric machines, have shown that machines have an identical magnetic field distribution on a pole by pole basis [11]. The magnetic field patterns show that only one pole pitch needs to be modeled in a machine with identical poles. Thus, the number of elements and grid points in a finite element model can greatly be reduced if symmetry can be used and only one pole of the machine modeled. This is advantageous because a model with fewer elements and grid points will have a faster solution time and require less resources such as disk space to solve.

In an induction machine having identical poles, each pole boundary has periodic boundary conditions. For a two-dimensional model, the periodic boundary conditions are expressed in polar $(r,\theta)$ coordinates as [12]:

$$A(r,\theta_0 + p) = -A(r,\theta_0) \qquad \text{(Equation 4.1)}$$

where $\vec{A}$ is vector potential, $\theta_0$ is the angle of one radial boundary, and p the pole pitch angle. This boundary condition is called an alternating periodic boundary condition. If the geometry requires modeling two poles, then the vector potentials on the boundary are set equal with no negative sign. This is referred to as a repeating periodic boundary condition. Generally, an odd number of poles requires alternating and an even number repeating boundary conditions.

## 4.2 FE Modeling of Doubly Fed Characteristics

Applying the techniques of induction machine analysis described above to the BDFM is difficult due to the following considerations:

### 4.2.1 Complications of the Nested Rotor Structure

Because of the nested loop rotor structure, (the absence of a solid endring on one side), the BDFM analysis problem is three-dimensional in nature. The nested loops impose electrical constraints on the model that cannot be properly represented with a two-dimensional analysis.

### 4.2.2 Inclusion of Two Excitation Frequencies

The presence of two stator windings carrying currents of differing frequencies requires the consideration of two frequencies at any time. This poses a problem because the ac solution method requires that a single solution frequency be specified.

### 4.2.3 Boundary Conditions More Difficult to Determine

The symmetry of the magnetic field distribution in the BDFM is not as simple to determine as that of an induction machine, because of the presence of the two stator windings of different pole numbers. Thus, determining the section of the BDFM that should be modeled to properly represent the entire machine, and the selection of the appropriate boundary conditions, requires some consideration.

This chapter discusses how these difficulties have been dealt with and a finite element procedure for modeling the BDFM has been developed.

## 4.3 Three-Dimensional Simulation of the BDFM

Because the BDFM analysis problem is three-dimensional in nature, three-dimensional finite element modeling of the BDFM has been investigated. A three-dimensional analysis avoids the approximations involved in developing and/or combining two-dimensional models, and allows accurate representation of the nested rotor structure. For this three-dimensional analysis, a commercial software package produced by MacNeal-Schwendler Corporation (MSC), called MSC/XL and MSC/EMAS (Electromagnetic Analysis System) was used. The work was done on a Hewlett Packard workstation, model 715/50, with 48 MB of RAM and approximately 2.5 GB of hard disk space.

Several three-dimensional BDFM models will be presented, along with the materials, excitations, and boundary conditions used in the setup of the models. The results of these BDFM model simulations will also be presented. Each of the BDFM models presented is based on the prototype 5 hp laboratory machine, which has a 6 pole power winding and a 2 pole control winding configuration. Analysis techniques used to model BDFM will also be discussed.

### 4.3.1 Modeling in the Rotor Reference Frame vs. the Stator Reference Frame

In finite element simulation of ac induction machines, the solution frequency used in an ac analysis is the frequency seen by the rotor, or the slip frequency. The frequency seen by the rotor is not as simple to imagine for the BDFM, because of the presence of two sets of stator windings operating at different frequencies.

A way of obtaining the frequency observed by the BDFM rotor for use as the solution frequency for an ac analysis is to model the BDFM in the rotor reference frame. The rotor reference frame frequency, or frequency of the rotor currents during synchronous operation of the machine, is determined from Equations 2.5 and 2.6, which are restated here:

$$f_r = \frac{f_p \pm f_c}{P_p + P_c} \qquad \text{(Equation 4.2)}$$

$$f_{r,el} = f_p - P_p f_r = P_c f_r \mp f_c \qquad \text{(Equation 4.3)}$$

Equation 4.3 determines the frequency seen by the BDFM rotor during synchronous operation. Since during synchronous operation the fields induced in the rotor by the power and the control windings are locked together at the same frequency, only this one rotor frequency needs to be specified in the ac analysis. If it is desired to determine the induced rotor currents during synchronous operation of the machine, Equation 4.3 can be used to determine the solution frequency to be used in an ac analysis. Thus, modeling the BDFM in the rotor reference frame eliminates the need for two frequencies to be included in the simulation at once.

If it is desired to model the BDFM during startup, or during other conditions when it is not operating in synchronous mode, the fields induced in the rotor would not be locked together at one frequency. Therefore, it is not be possible to simulate the BDFM under dynamic conditions with the ac analysis module, because the ac solution method requires that a single solution frequency be specified. Possibilities exist to overcome this difficulty by using the transient analysis module, which allows waveforms of different types and/or frequencies to be included in the analysis simultaneously.

### 4.3.2 Symmetry of the BDFM Model

In induction machine analysis, it has been shown that only one pole pitch of the machine needs to be modeled because of the symmetry of the magnetic field distribution by pole pitch. The magnetic field symmetry of the BDFM model is not as simple to determine because of the presence of two stator windings of different pole numbers.

The finite element model of the BDFM presented in this thesis is based on the 5 horsepower BDFM laboratory machine. The laboratory machine has a 6 pole power winding and 2 pole control winding configuration.

Theoretically, the alternating periodic boundary conditions used to model induction machines should be appropriate for a 180 degree model of a 6/2 pole BDFM. This is because one pole of the two-pole winding and three poles of the six-pole winding are present in a 180 degree model. Therefore, an odd number of poles is modeled for both windings. Since alternating periodic boundary conditions are required for an odd number of poles, theoretically alternating periodic boundary conditions should be the appropriate boundary conditions for a 6/2 pole 180 degree BDFM model. Results of the BDFM model simulations presented in this chapter verify this idea.

The MSC/EMAS Modeling Guide [13] suggests that the best way to determine the correct boundary conditions is to make a coarse finite element model of the entire device and observe the relationship obeyed by the vector potential $\vec{A}$ at grid points one pole pitch, or other distance, apart. Then these constraints are applied to a fine finite element model of that portion of the device.

## 4.4 Results of BDFM Model Simulations

Several three-dimensional BDFM models were constructed and analyzed using MSC/EMAS. First, results from a coarse 360 degree model of the BDFM are presented. Next, results from a 180 degree model, with a finite element mesh identical to the 360 degree model, and with alternating periodic boundary conditions applied along the symmetry plane, are presented. Comparison of the full model and half model results verifies that the alternating periodic boundary conditions are correct. Finally, results from a more detailed 180 degree BDFM model are presented. For each analysis, synchronous operation of the BDFM is assumed and the ac analysis module is used. It should be noted that the ac analysis module is a linear analysis module that does not take into account the $\vec{B}$ - $\vec{H}$ curve of magnetic materials.

### 4.4.1 Device Geometry

Each of the three-dimensional finite element BDFM models was based on the 5 horsepower laboratory machine. The dimensions of the laboratory machine stator laminations are shown in Figure 4.1 (shown on the following page). The stator windings consist of a 6 pole power winding and a 2 pole control winding. The stator contains 36 slots and the stator stack length is 100 mm.

The dimensions of the laboratory machine rotor laminations are shown in Figure 4.2. The rotor laminations were custom made to provide an air gap of 0.7 mm. The rotor was designed with 40 slots. For the 6/2 pole BDFM, the required 4 nests have 5 loops each, and no common cage.

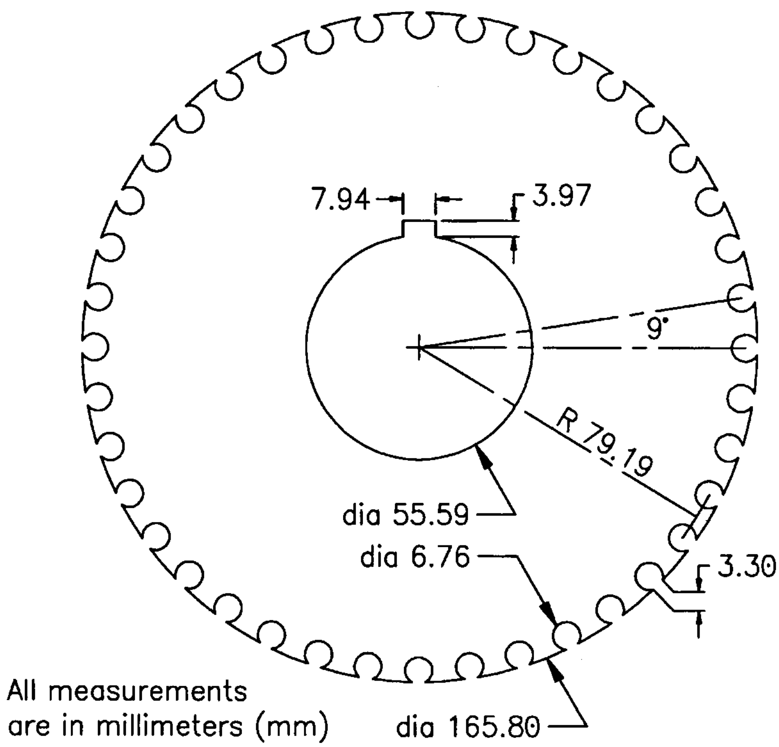


Figure 4.2: Rotor Lamination

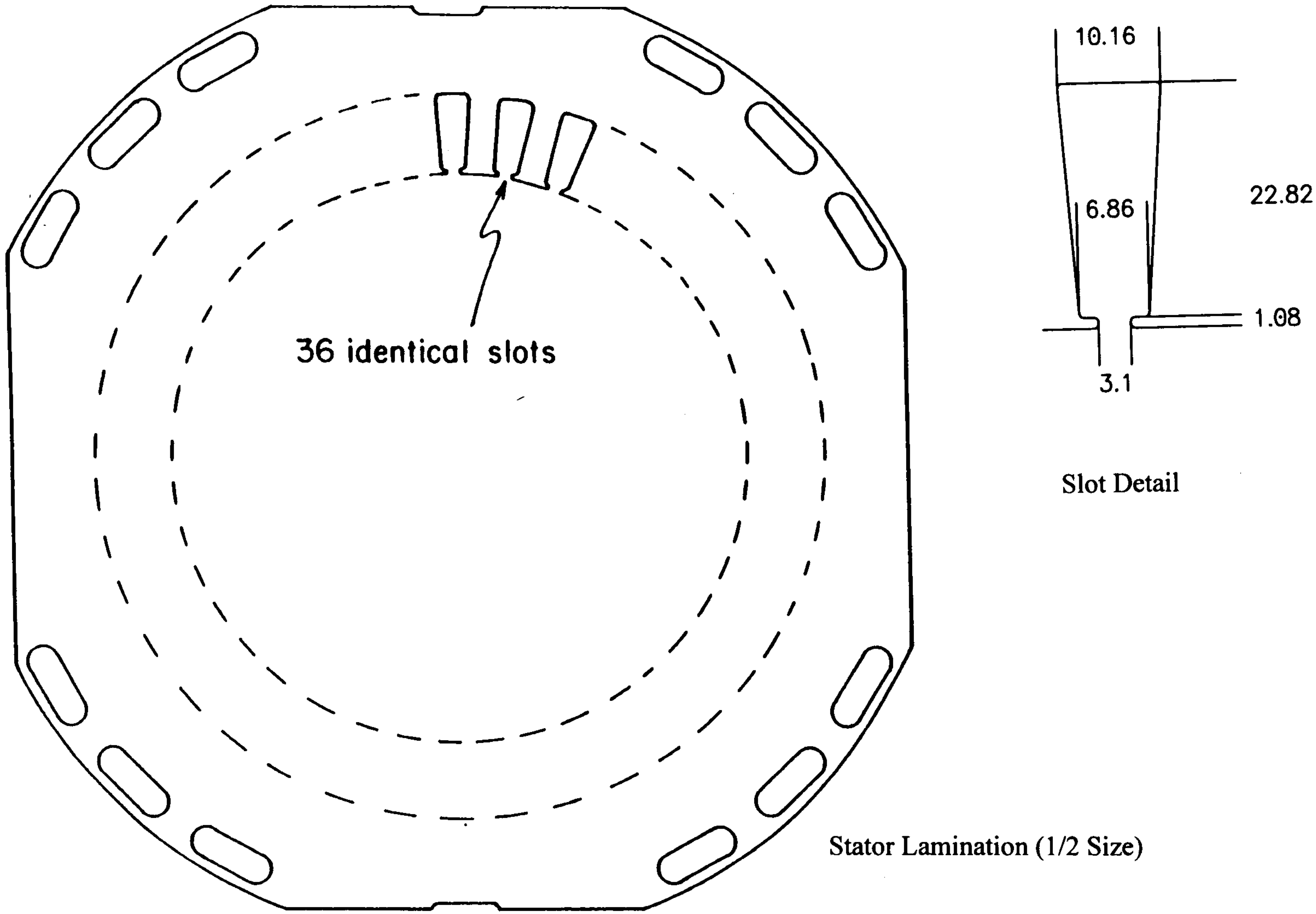


Figure 4.1: Stator Lamination

### 4.4.2 A Coarse 360 Degree BDFM Model

A coarse 360 degree three-dimensional finite element model of the BDFM was constructed. The main goal in construction of this model was to determine if magnetic field symmetry exists within the BDFM and to verify what portion of the device can be modeled using appropriate boundary conditions to accurately represent the whole machine.

The 360 degree model had a rather coarse finite element mesh consisting of 7776 hexahedron and pentahedron elements and 8325 grid points. The finite element model is shown in Figure 4.3.

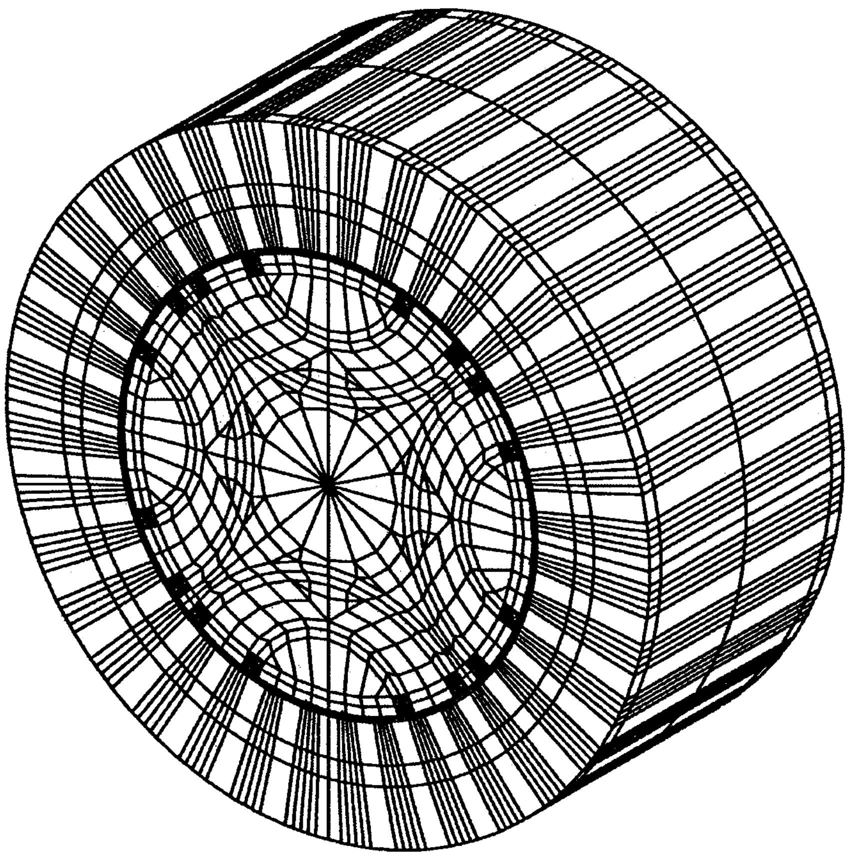

Figure 4.3: Coarse Three-Dimensional 360 Degree BDFM Finite Element Model

In constructing the 360 degree model of the BDFM, a simplifying assumption was made about the laboratory machine rotor. The rotor was modeled with only 16 slots (instead of the actual 40 slots) and only the first and third loops of each nest were modeled, or two loops for each of the 4 nests (instead of the actual five loops for each of the 4 nests). This simplification was made in order to reduce the complexity of the finite element mesh, and hence reduce the amount of disk space required to generate a solution.

The body of the machine was modeled with two layers of three-dimensional elements, each 50 mm long, as shown in Figure 4.3. The nested loops of the rotor, as well as the common endring, are modeled with one layer of three-dimensional elements extending 6.76 mm beyond the machine body on opposite sides. Ideally, several more layers of elements should be used to obtain a more accurate representation of the machine. The configuration described was used because of disk space limitations.

#### 4.4.2.1 Materials

Table 4.1 lists the material properties that were assigned to the objects that make up the BDFM model.

Note that although the stator slot actually contain copper windings, they are modeled as air since a current density excitation is used to specify the exact current flowing in the windings. To model the stator windings as copper would cause the program to induce additional eddy currents in the stator windings.

| Object | Material | Relative Permeability | Relative Permittivity | Electrical Conductivity (siemens/meter) |
|---|---|---|---|---|
| Shaft | Air | 1 | 1 | 0 |
| Rotor | Lam. Steel | 2000 | 1 | 0 |
| Rotor Bars | Copper | 1 | 1 | 5.8E+07 |
| Air Gap | Air | 1 | 1 | 0 |
| 6 pole Stator Slot | Air | 1 | 1 | 0 |
| 2 pole Stator Slot | Air | 1 | 1 | 0 |
| Stator | Lam. Steel | 2000 | 1 | 0 |
| Nested Loops | Copper | 1 | 1 | 5.8E+07 |
| End Ring | Copper | 1 | 1 | 5.8E+07 |

Table 4.1: Material Properties used in the 360 Degree BDFM Model

#### 4.4.2.2 Excitations

For the 360 degree BDFM model, an equivalent surface current density is used to establish a current flow of 100 amp-turns peak in each of the 6 pole and 2 pole slots. Figure 4.4 shows the 6 pole and 2 pole stator winding excitation locations and directions and specified for use in the ac analysis.

In Figure 4.4, for the 6 pole winding, $Phase_a = 0^\circ$, $Phase_b = 240^\circ$, and $Phase_c = 120^\circ$. For the 2 pole winding, $Phase_a = 0^\circ$, $Phase_b = 120^\circ$, and $Phase_c = 240^\circ$.

#### 4.4.2.3 Boundary Conditions

Figure 4.5 shows the applied boundary conditions for the 360 degree BDFM model. A cylindrical coordinate system is used to define boundary directions. Along

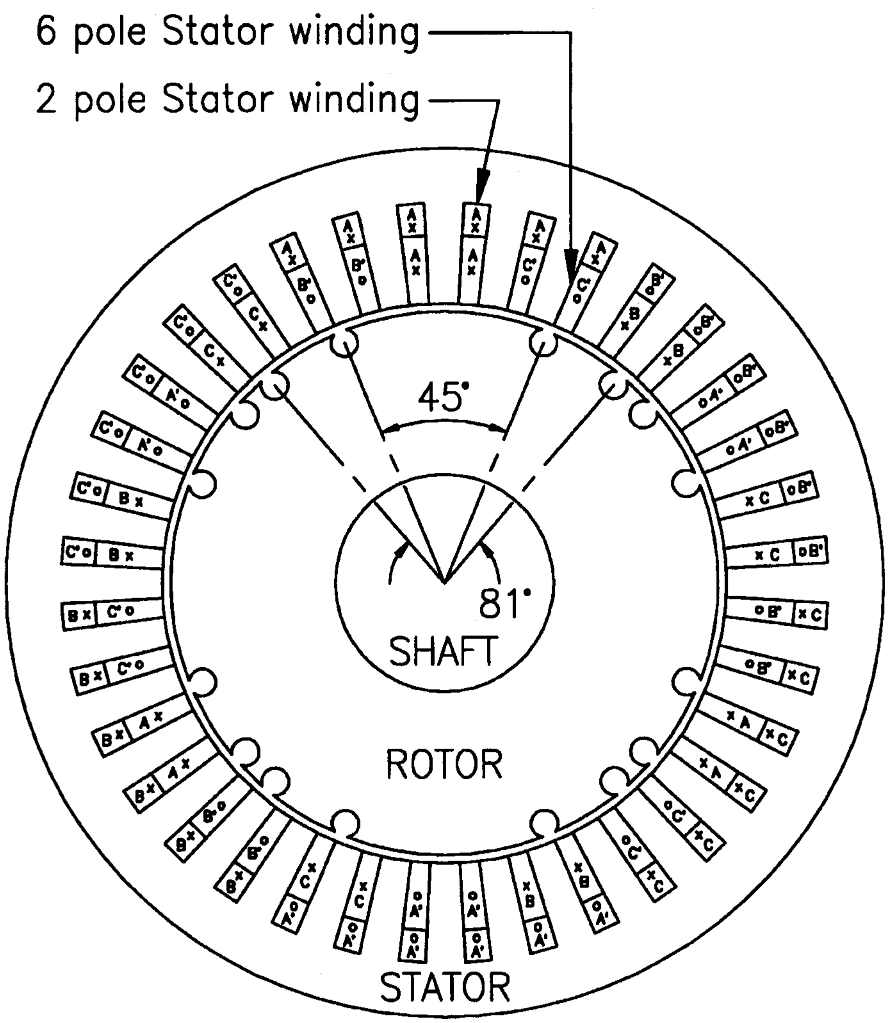


Figure 4.4: Stator Excitations for the 360 Degree BDFM Model

the outer radius of the model, the tangential components of the magnetic vector potential $\vec{A}$ are set to zero $(A_\theta, A_z)$. The tangential components of $\vec{A}$ are also set to zero along the motor ends $(A_r, A_\theta)$. Setting the tangential components of $\vec{A}$ to zero along the outer boundaries of the machine constrains the magnetic fields to remain within the machine outer boundaries.

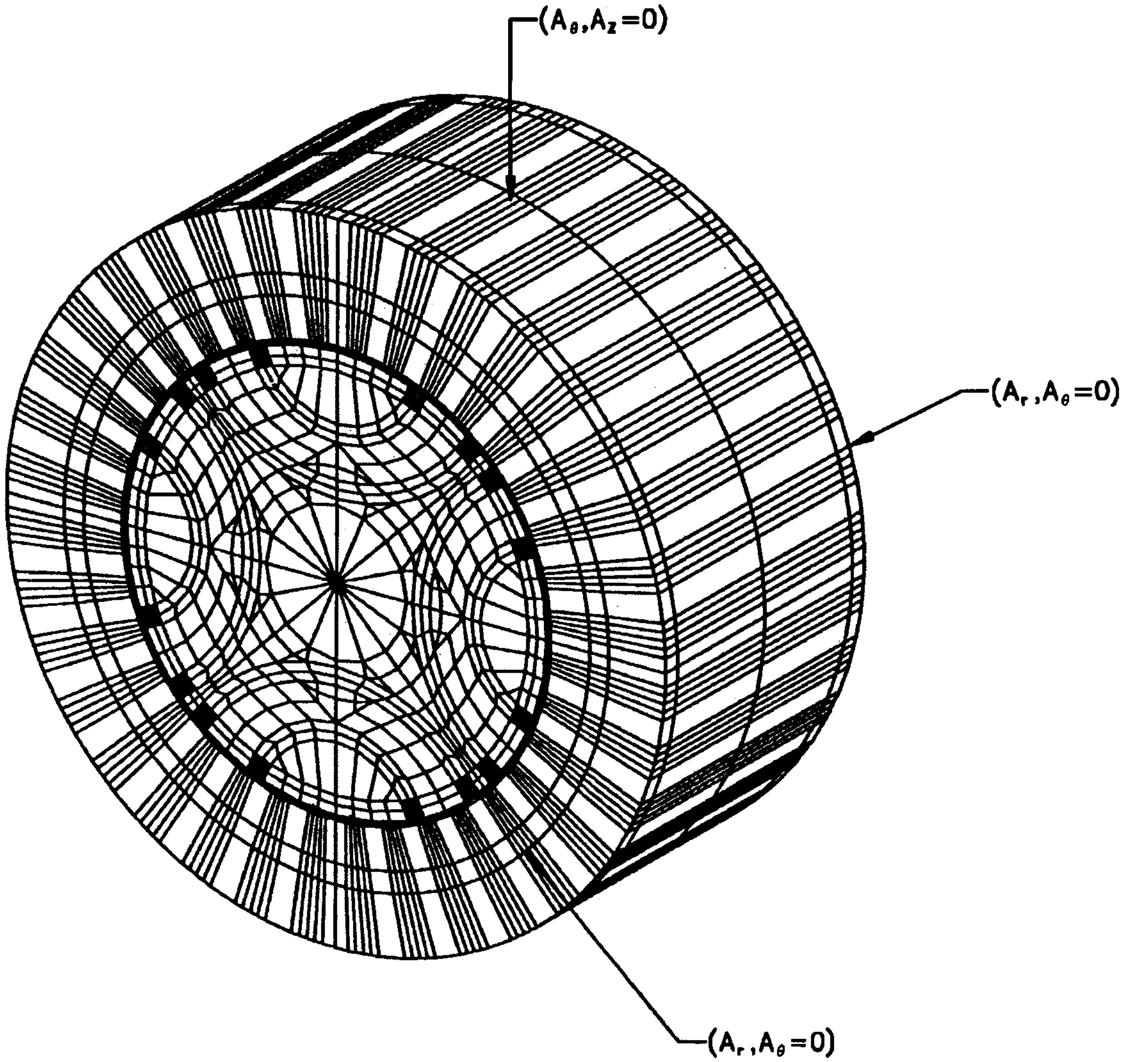


Figure 4.5: Boundary Conditions for the 360 degree BDFM Model

#### 4.4.2.4 Solution Frequency for an AC Analysis

An ac analysis was done for this 360 degree BDFM model utilizing the linear ac module. The solution frequency used was determined from Equation 4.4, assuming synchronous machine operation. A power winding frequency of 60 Hz and a control winding frequency of -20 Hz (opposing sequence) were assumed. This results in a rotor reference frame frequency of 30 Hz, which was specified as the solution frequency in the ac analysis. This corresponds to a machine speed of 600 rpm.

In the examination of the results that follow, the reader should note that the stator solution is in the rotor reference frame.

#### 4.4.2.5 Results

##### 4.4.2.5.1 Contour Plot of Magnetic Vector Potential

A contour plot of magnetic vector potential, $\vec{A}$, along a cross-section of the BDFM located at the machine center is shown in Figure 4.6.

A line of constant magnitude of vector potential $\vec{A}$ is called a magnetic flux line. Observing the pattern of the magnetic flux lines along a planar cross section of the machine shows the symmetry present in the machine's magnetic field distribution. Figure 4.6 shows that the BDFM exhibits 180 degree symmetry.

##### 4.4.2.5.2 Arrow Plot of Magnetic Flux Density

An arrow plot of magnetic flux density, $\vec{B}$, along a cross-section of the BDFM located at the machine center is shown in Figure 4.7.

An arrow plot of magnetic flux density represents the direction and magnitude of the magnetic flux density with colored arrows. The colors of the arrows indicate the magnitude of the magnetic flux density, in tesla, at every location along the machine cross-section. This allows the user to identify areas where the machine may be

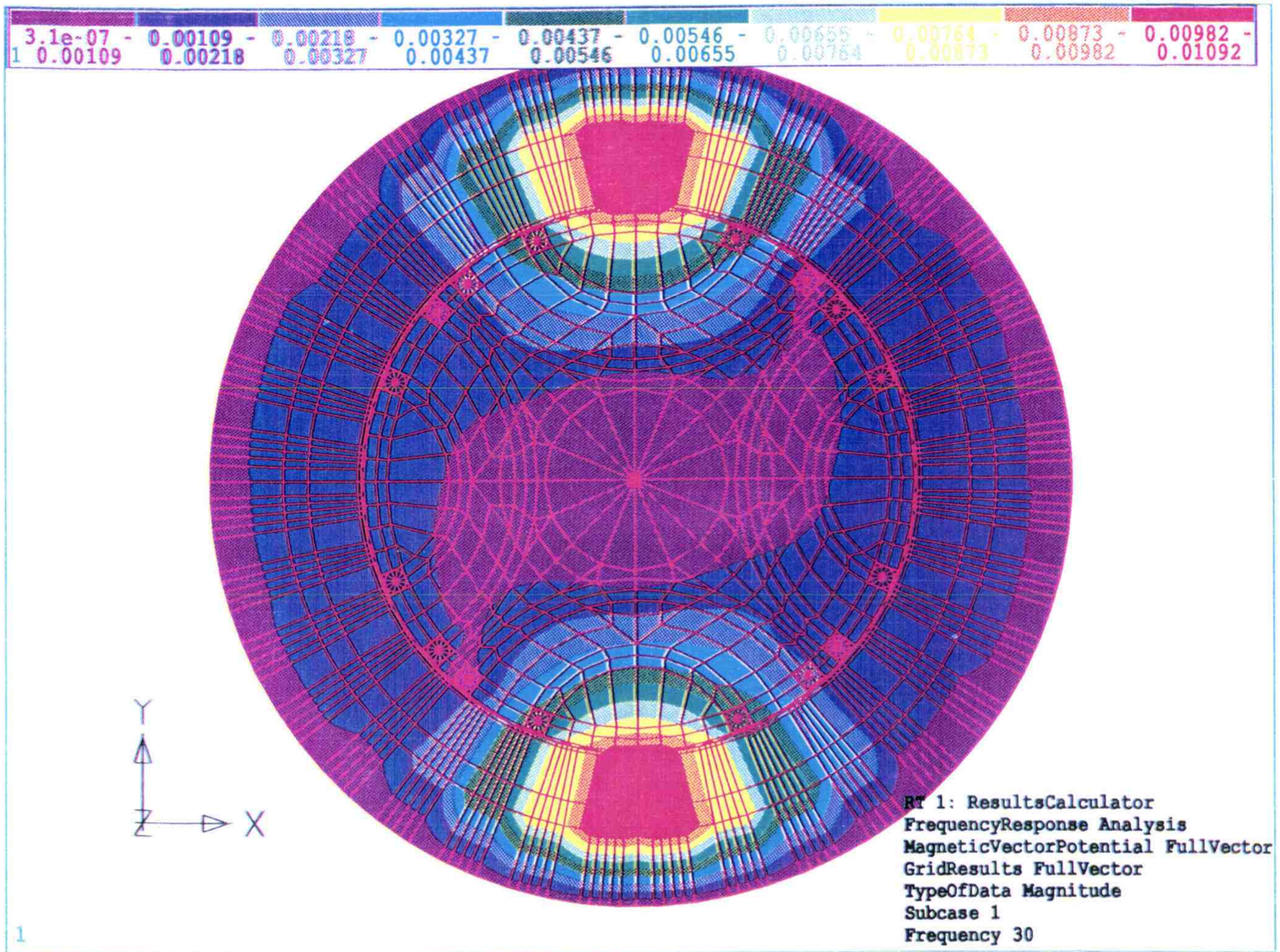


Figure 4.6: Contour Plot of Magnetic Vector Potential along a Center Cross-Section of the 360 Degree BDFM Model

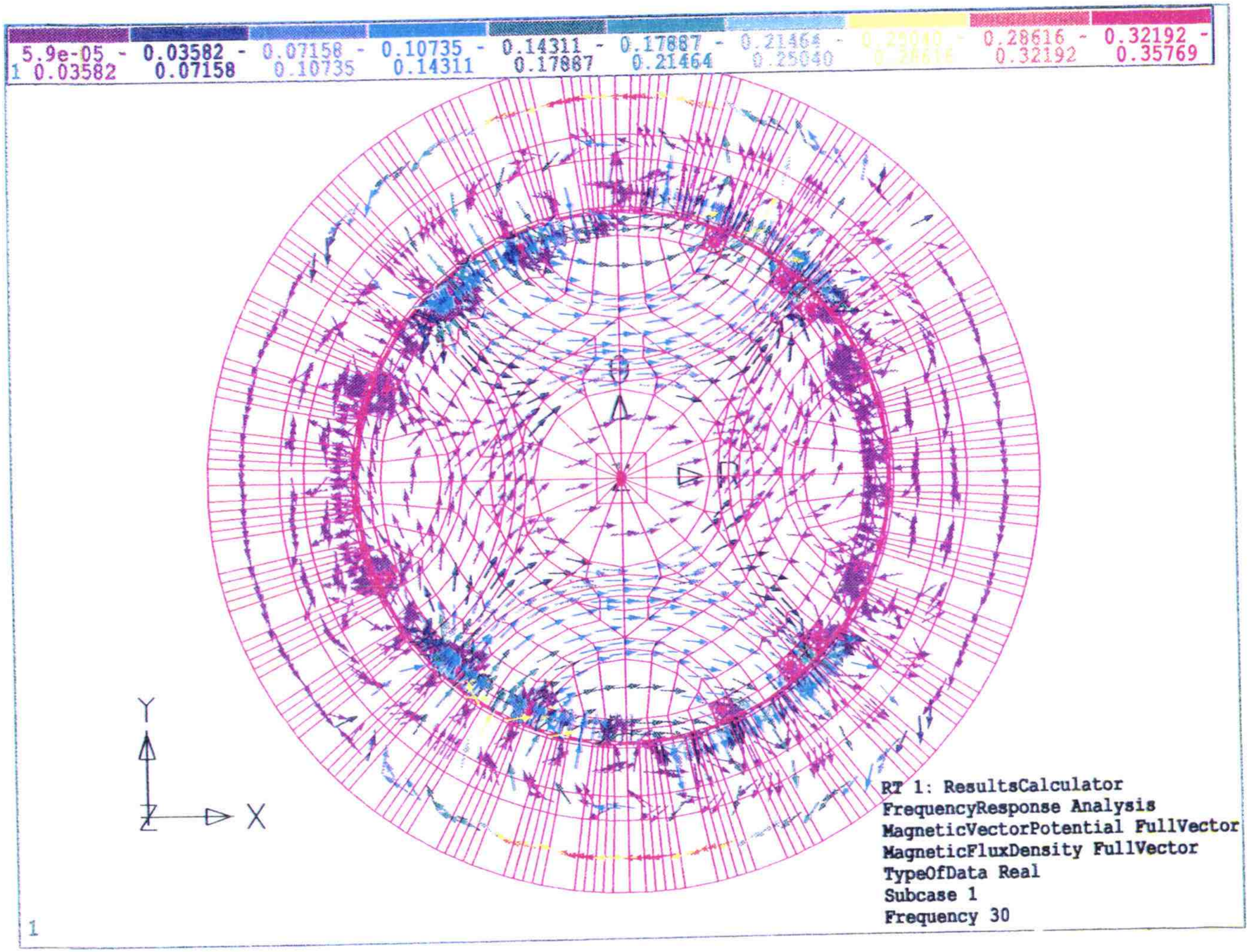


Figure 4.7: Arrow Plot of Magnetic Flux Density along a Center Cross-Section of the 360 Degree BDFM Model

saturating. The flux density values for this BDFM model (0.35925 tesla maximum) show that the machine is not saturating, for the previously specified excitation level of 100 amp/turns peak per slot.

An arrow plot of magnetic flux density can also be used to observe the field symmetry pattern of the machine. The arrow plot of Figure 4.7 also shows that the field pattern exhibits 180 degree symmetry, giving credibility to this approach.

#### 4.4.2.5.3 Currents in the Rotor Bars

The currents flowing in every rotor bar of the BDFM rotor were determined using MSC/XL's solution calculator. The currents were calculated by specifying a plane perpendicular to the rotor bar direction (the xy plane) and intersecting this plane with one rotor bar at a time. The program then calculates the total current flowing in the rotor bar in the positive z-direction from the conduction current density by the following formula:

$$I = \int (\vec{J}_c \cdot ds) \qquad \text{(Equation 4.4)}$$

where $\vec{J}_c$ is the conduction current density and ds is the integration surface.

The currents were calculated at several axial (z) positions along each rotor bar, and identical results were obtained. The calculated total currents in each rotor bar are presented in Table 4.2, and the rotor bar labels are identified in Figure 4.8.

Table 4.2 shows that the rotor bars currents are equal in magnitude and 180 degree out of phase for rotor bars connected by a loop. In other words, the rotor currents

| Bar Number | Real, Imaginary (Amperes) | Magnitude, Phase (Amperes, Degrees) |
|---|---|---|
| 1 | 8.26766, 25.5532 | 26.8574, 72.0712 |
| 2 | -133.203, -33.6687 | 137.392, -165.815 |
| 3 | 30.8736, -186.89 | 189.423, -80.6197 |
| 4 | -7.13813, -87.3815 | 87.6725, -94.6701 |
| 5 | 7.13833, 87.385 | 87.676, 85.33 |
| 6 | -30.8734, 186.89 | 189.423, 99.3803 |
| 7 | -133.272, -33.6829 | 137.463, -165.816 |
| 8 | 8.28426, 25.5654 | 26.8741, 72.0455 |
| 9 | -8.28503, -25.565 | 26.874, -107.956 |
| 10 | 133.271,33.6827 | 137.461, 14.1839 |
| 11 | -30.7961, 186.89 | 189.41, 99.3572 |
| 12 | 7.18307, 87.3974 | 87.6921, 85.3015 |
| 13 | -7.18318, -87.3998 | 87.6945, -94.6984 |
| 14 | 30.796, 186.889 | 189.409, -80.6428 |
| 15 | 133.203, 33.668 | 137.393, 14.1851 |
| 16 | -8.6676, -25.5533 | 26.8572, -107.927 |

Table 4.2: Total Calculated Currents in the Rotor Bars for the 360 Degree BDFM Model

are equal in magnitude but flowing in opposite directions for connected bars, as is expected. Also, the magnitude of the rotor bar currents in the first loop are greater than the currents in the third loop, as is expected by consideration of the laboratory machine. Thus, the rotor bar currents indicate that the BDFM model has 180 degree symmetry.

### 4.4.3 A Coarse 180 Degree BDFM Model

A three-dimensional 180 degree model of the BDFM, with a finite element mesh identical to the 360 degree BDFM model, was set up. The material properties, excitations, and outer boundary conditions were identical to those used for the 360 degree model. An ac solution was generated at a rotor reference frame frequency of 30 Hz, as in the 360 degree model. Alternating periodic boundary conditions were used

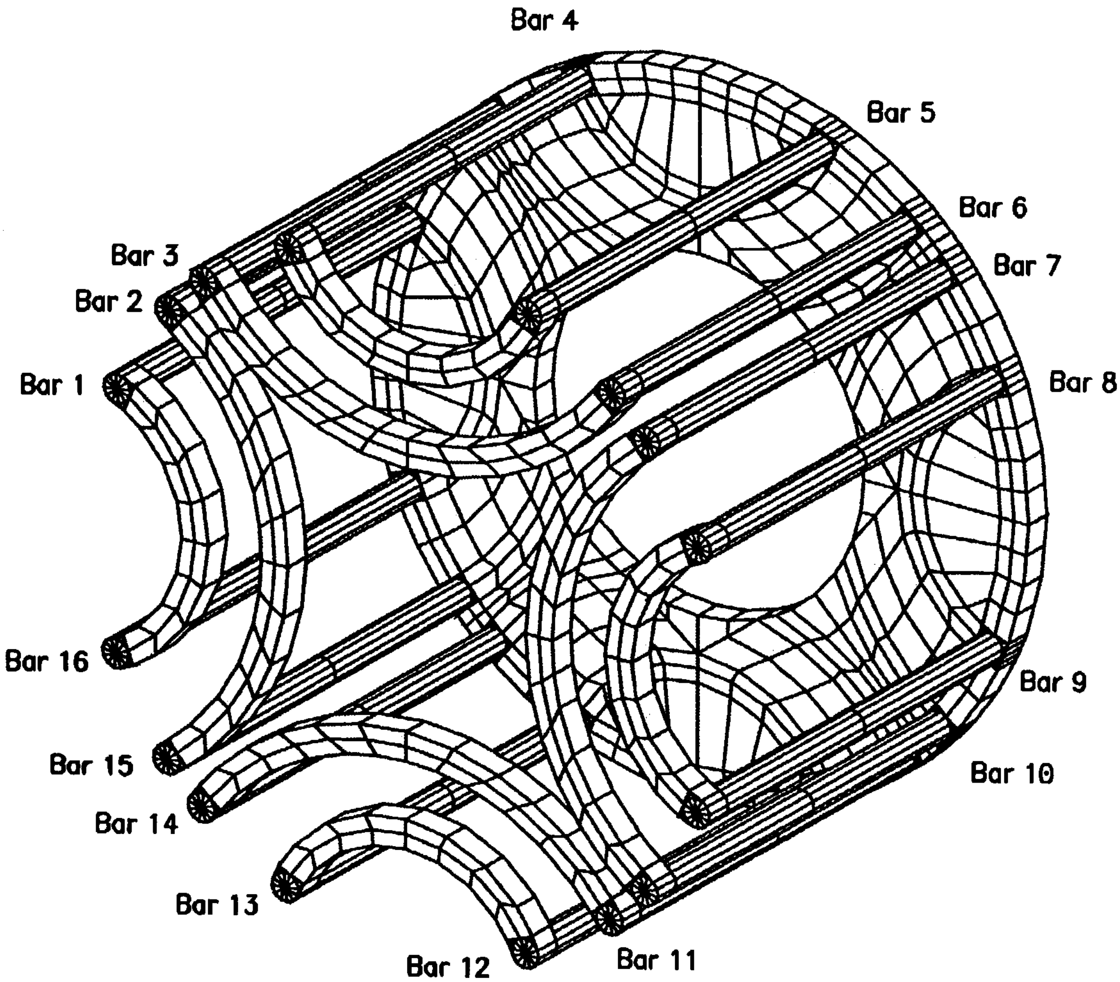


Figure 4.8: Rotor Bars Labels for the 360 Degree BDFM Model

along the symmetry plane of the 180 degree model. The 180 degree finite element model is shown in Figure 4.9.

This 180 degree model had a finite element mesh consisting of 3888 hexahedron and pentahedron elements and 4280 grid points (basically half the number of elements and grid points utilized in the coarse 360 degree model).

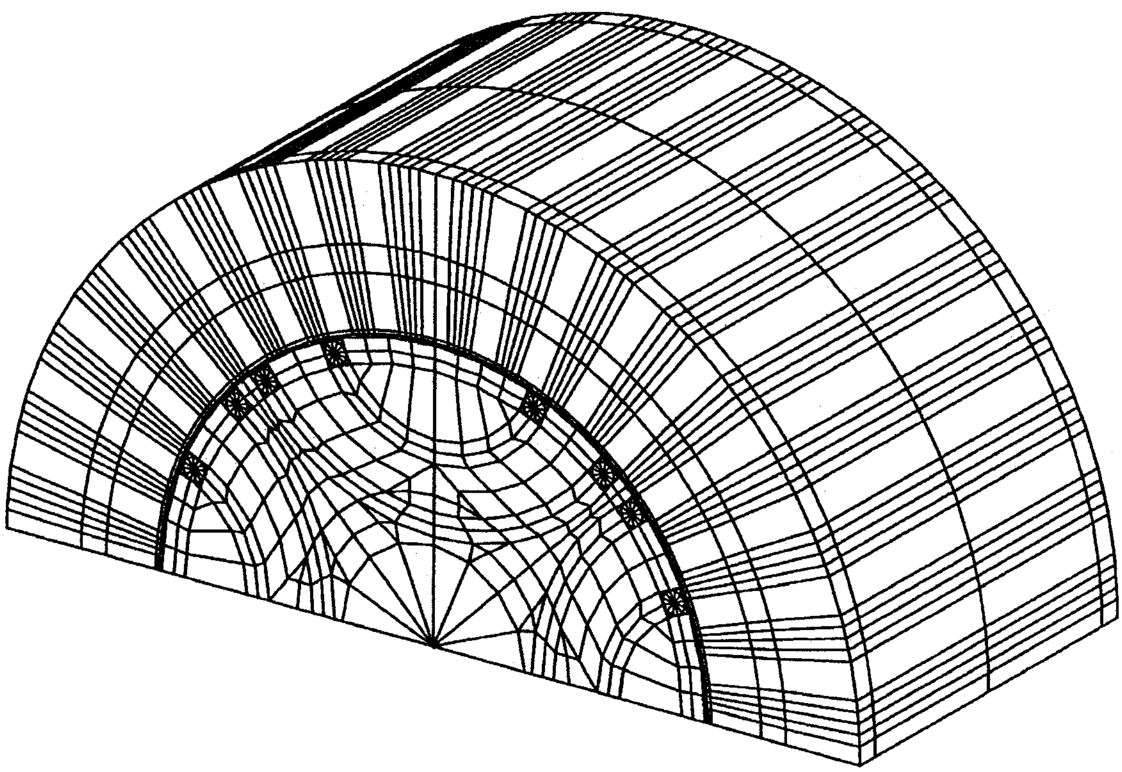

Figure 4.9: Coarse Three-Dimensional 180 degree BDFM Finite Element Model

The purpose of this coarse 180 degree model simulation was to verify that the use of alternating periodic boundary conditions on a 180 degree BDFM model produces results consistent with those obtained for a 360 degree model.

#### 4.4.3.1 Boundary Conditions

Figure 4.10 shows the applied boundary conditions for the 180 degree BDFM model. A cylindrical coordinate system is used to define the boundary directions. The tangential components of the magnetic vector potential, $\vec{A}$, are set to zero along the outer radius of the model and along the motor ends, as in the 360 degree model. Alternating periodic boundary conditions were applied along the two radial faces of the symmetry plane, at $\theta = 0$ and $\theta = 180$ degrees. This boundary condition forces every degree of freedom (three $\vec{A}$ components and $\Psi$) to be equal in magnitude but opposite in direction as follows:

$$A_r(r,\theta_0 + p, z) = -A_r(r,\theta_0, z) \qquad \text{(Equation 4.5)}$$

$$A_\theta(r,\theta_0 + p, z) = -A_\theta(r,\theta_0, z) \qquad \text{(Equation 4.6)}$$

$$A_z(r,\theta_0 + p, z) = -A_z(r,\theta_0, z) \qquad \text{(Equation 4.7)}$$

$$\Psi(r,\theta_0 + p, z) = -\Psi(r,\theta_0, z) \qquad \text{(Equation 4.8)}$$

where $\vec{A}$ is vector potential, $\theta_0$ is the angle of one radial boundary, and p the pole pitch angle.

#### 4.4.3.2 Results

Examination of the results obtained from the coarse 180 degree BDFM model are in close agreement with the results obtained from the 360 degree model, verifying the symmetry of the BDFM model and the use of alternating periodic boundary conditions.

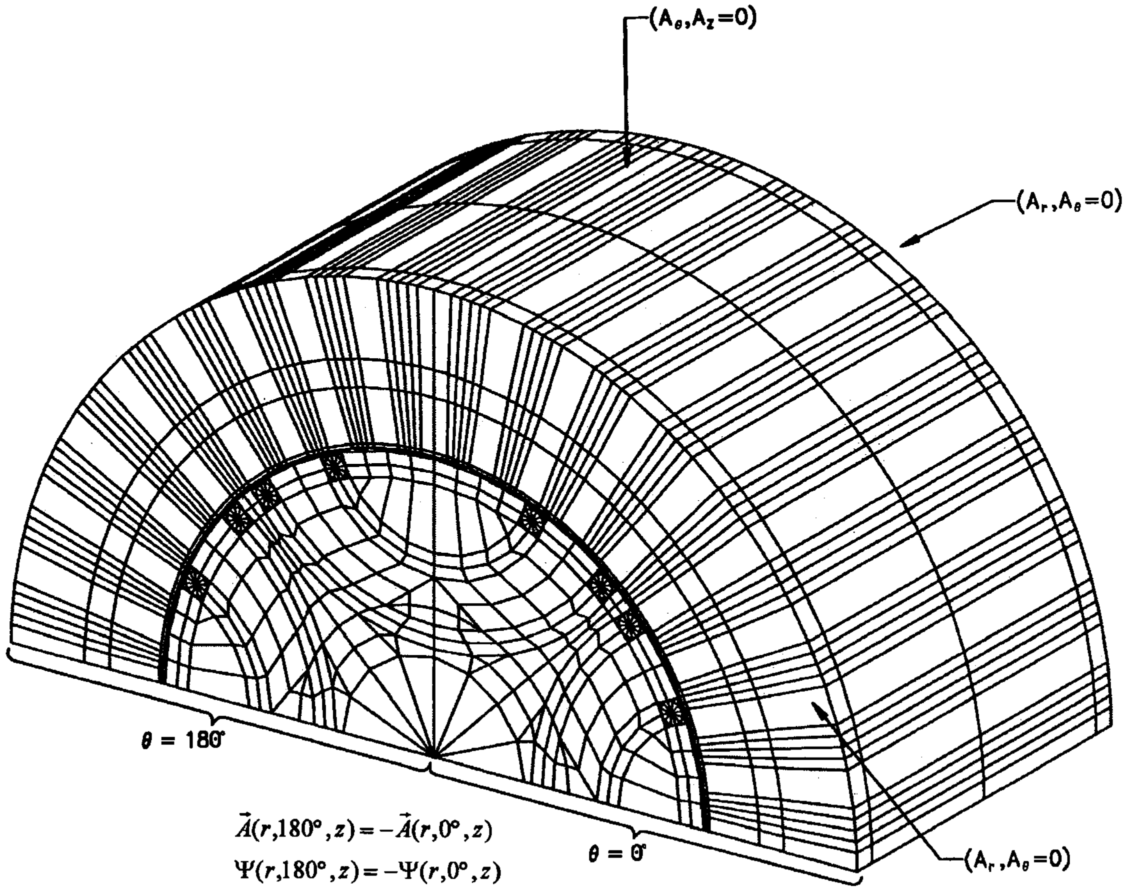


Figure 4.10: Boundary Conditions for the Coarse 180 Degree BDFM Model

#### 4.4.3.2.1 Contour Plot of Magnetic Vector Potential

A contour plot of magnetic vector potential, $\vec{A}$, along a cross-section of the BDFM located at the machine center is shown in Figure 4.11.

This contour plot of magnetic vector potential is identical to Figure 4.6, the contour plot of magnetic vector potential for the 360 degree model. This contour plot is one verification that the alternating periodic boundary conditions are correct.

#### 4.4.3.2.2 Arrow Plot of Magnetic Flux Density

An arrow plot of magnetic flux density, $\vec{B}$, along a cross-section of the BDFM located at the machine center is shown in Figure 4.12.

This arrow plot of magnetic flux density is almost exactly identical to Figure 4.7, the arrow plot of magnetic flux density for the 360 degree model. This arrow plot is another verification that the selected alternating periodic boundary conditions are correct.

#### 4.4.3.2.3 Currents in the Rotor Bars

The total current flowing in each rotor bar of the 180 degree model was calculated using the same method used for the 360 degree model.

The currents were again calculated at several axial (z) positions along each rotor bar, and identical results were obtained. The calculated total currents in each rotor bar are presented in Table 4.3, and the rotor bar labels are identified in Figure 4.13.

| Bar Number | Real, Imaginary (Amperes) | Magnitude, Phase (Amperes, Degrees) |
|---|---|---|
| 1 | 8.29692, 25.0857 | 26.4222, 71.6987 |
| 2 | -132.462, -33.4865 | 136.629, -165.813 |
| 3 | 30.8917, -186.883 | 189.419, -80.6139 |
| 4 | -7.13535, -87.3859 | 87.6767, -94.668 |
| 5 | 7.13555, 87.3894 | 87.6802, 85.332 |
| 6 | -30.8915, 186.883 | 189.419, 99.386 |
| 7 | -132.463, -33.4866 | 136.63, -165.813 |
| 8 | 8.29609, 25.0859 | 26.4221, 71.7006 |

Table 4.3: Total Calculated Currents in the Rotor Bars for the Coarse 180 Degree BDFM Model

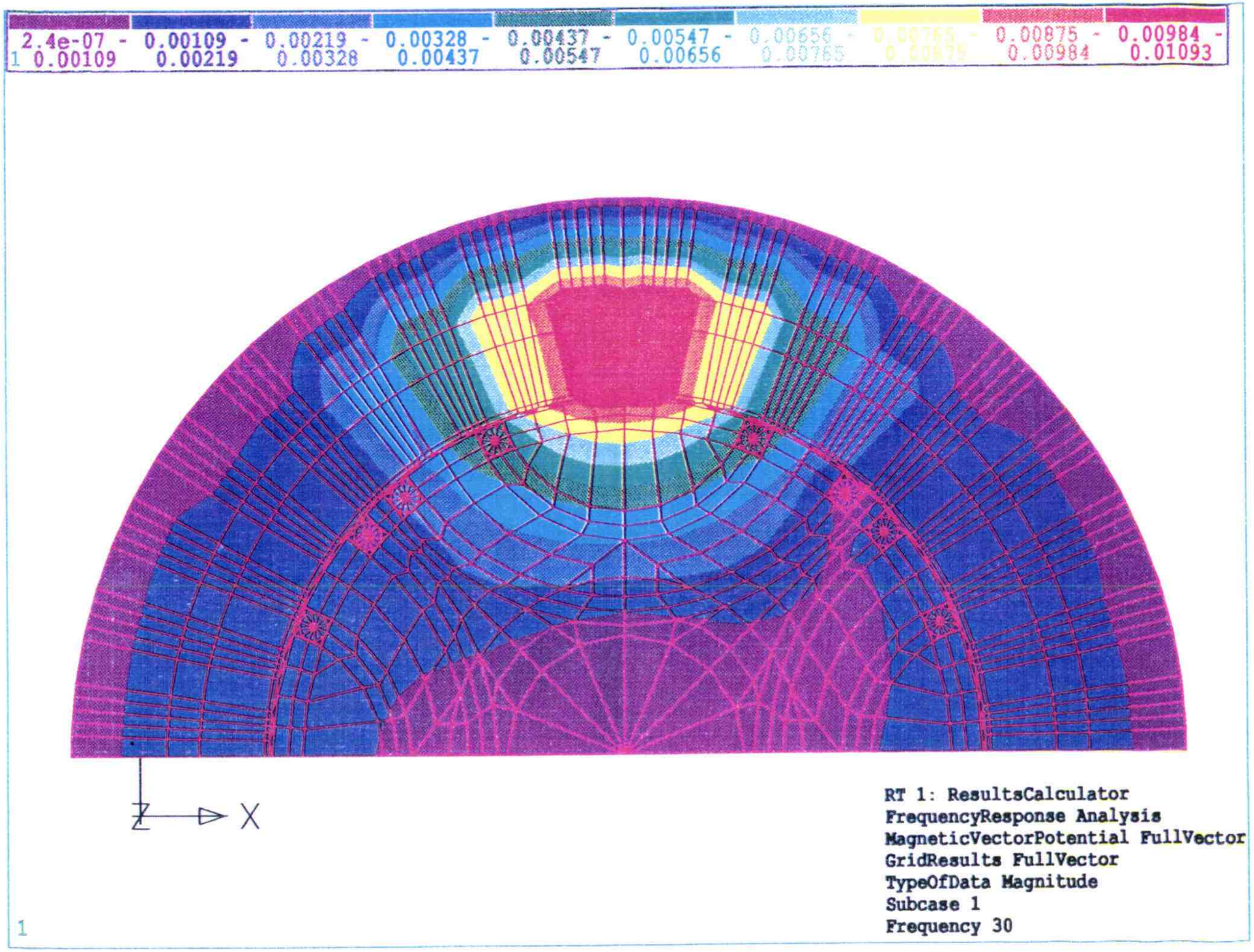


Figure 4.11: Contour Plot of Magnetic Vector Potential along a Center Cross-Section of the Coarse 180 Degree BDFM Model

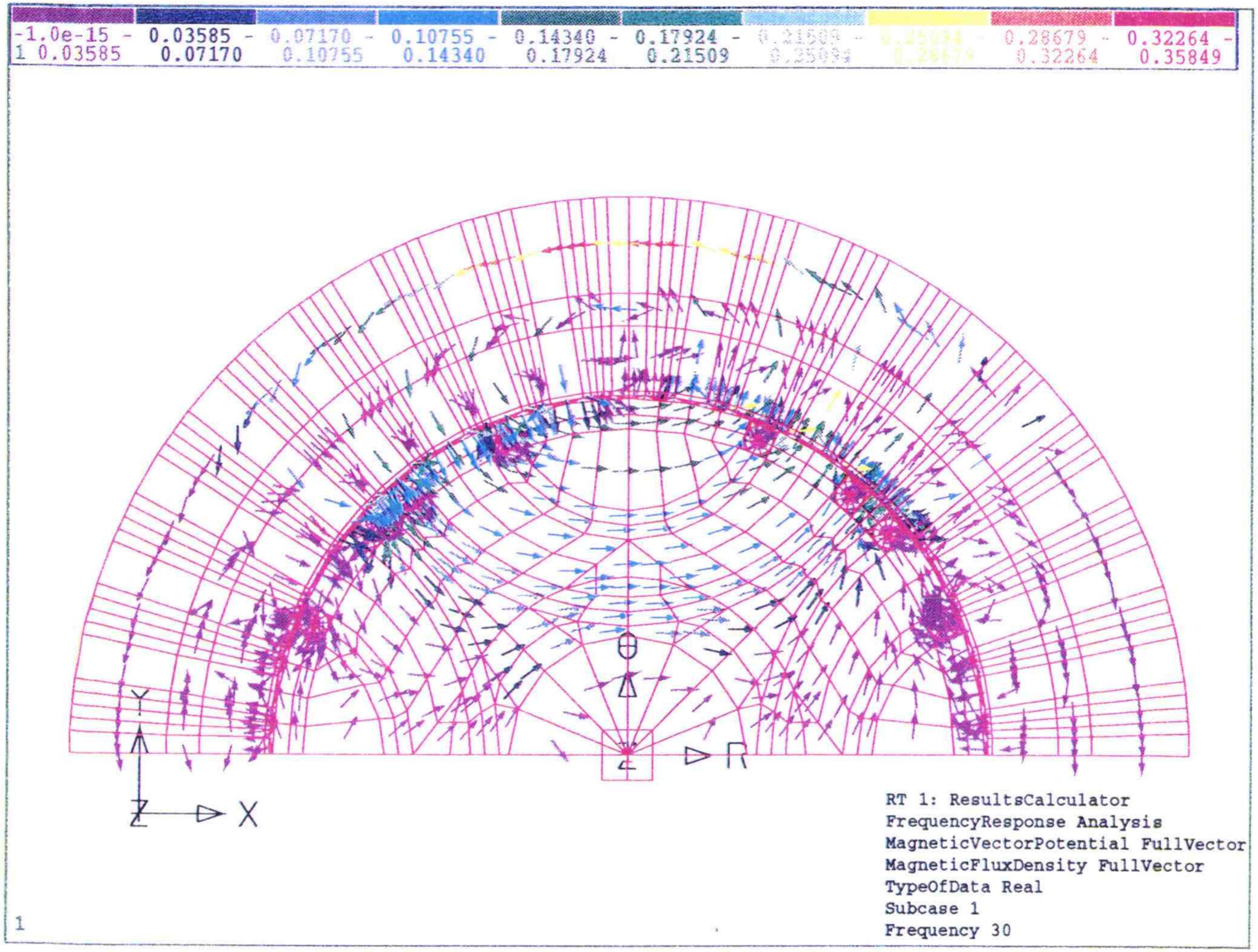


Figure 4.12: Arrow Plot of Magnetic Flux Density along a Center Cross-Section of the Coarse 180 Degree BDFM Model

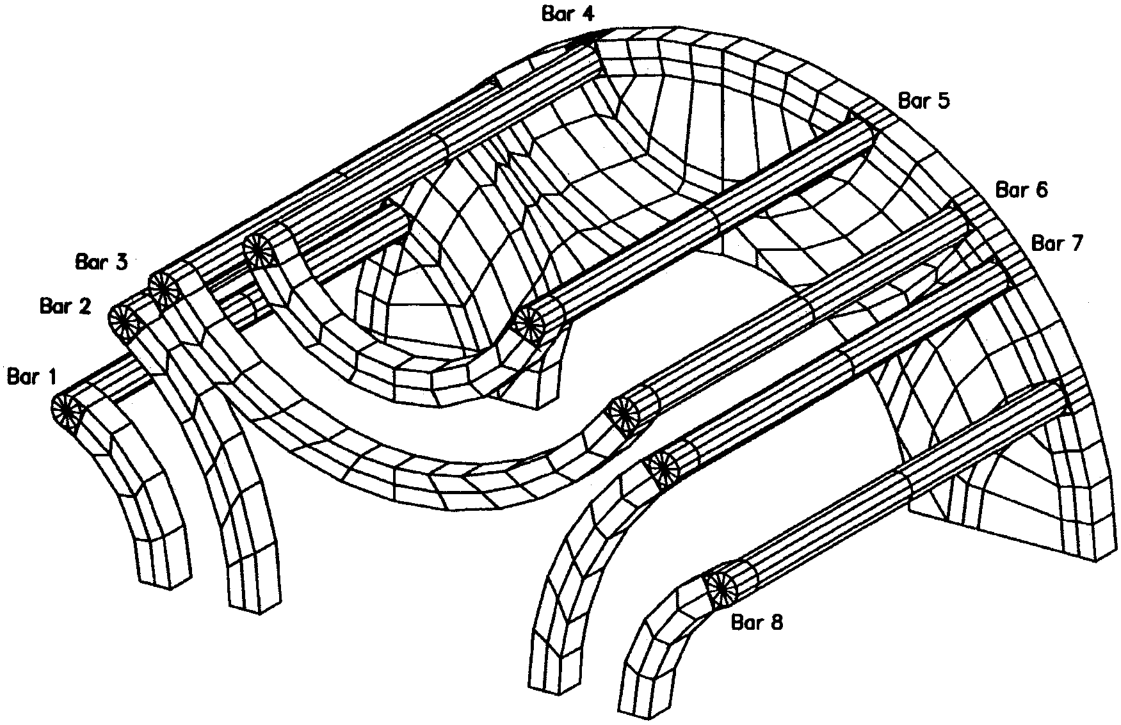


Figure 4.13: Rotor Bars Labels for the Coarse 180 Degree BDFM Model

These total currents calculated for the 180 degree model agree within about 2 % with the total currents calculated for the 360 degree model. The 2 % error may be due in part to limitations within MSC/XL for applying the boundary conditions, that do not allow the boundary conditions to be applied such that the 360 degree model is exactly represented. Since the currents are in close agreement, the alternating periodic boundary conditions are verified.

### 4.4.4 A Detailed 180 Degree BDFM Model

A finer and more detailed 180 degree model of the BDFM was constructed. The goal of constructing this model was to obtain a more accurate representation of the 5 horsepower BDFM laboratory machine.

This detailed model included all 40 of the rotor bars present in the laboratory machine, as well as all 5 of the loops per each of the 4 nests. A finer finite element mesh was used, especially in the rotor conductors and the air gap, where field gradients change most rapidly. The finite element mesh consisted of 8976 hexahedron and pentahedron elements and 9510 grid points. The finite element model is shown in Figure 4.14.

The body of the machine was modeled with two layers of three-dimensional elements, each 50 mm long, as shown in Figure 4.14. The nested loops of the rotor, as well as the common endring, are modeled with one layer of three-dimensional elements extending 6.76 mm beyond the machine body on opposite sides.

Identical excitations and boundary conditions, and similar material properties to those used in the coarse 180 degree BDFM model were used in for the setup of this detailed 180 degree model. An ac solution was generated, again using a rotor reference frame frequency of 30 Hz.

#### 4.4.4.1 Results

##### 4.4.4.1.1 Currents in the Rotor Bars

The total current flowing in each rotor bar of the detailed 180 degree model was calculated using the same method discussed previously.

The currents were again calculated at several axial (z) positions along each rotor bar, and identical results were obtained. The calculated total currents in each rotor bar are presented in Table 4.4, and the rotor bar labels are identified in Figure 4.15.

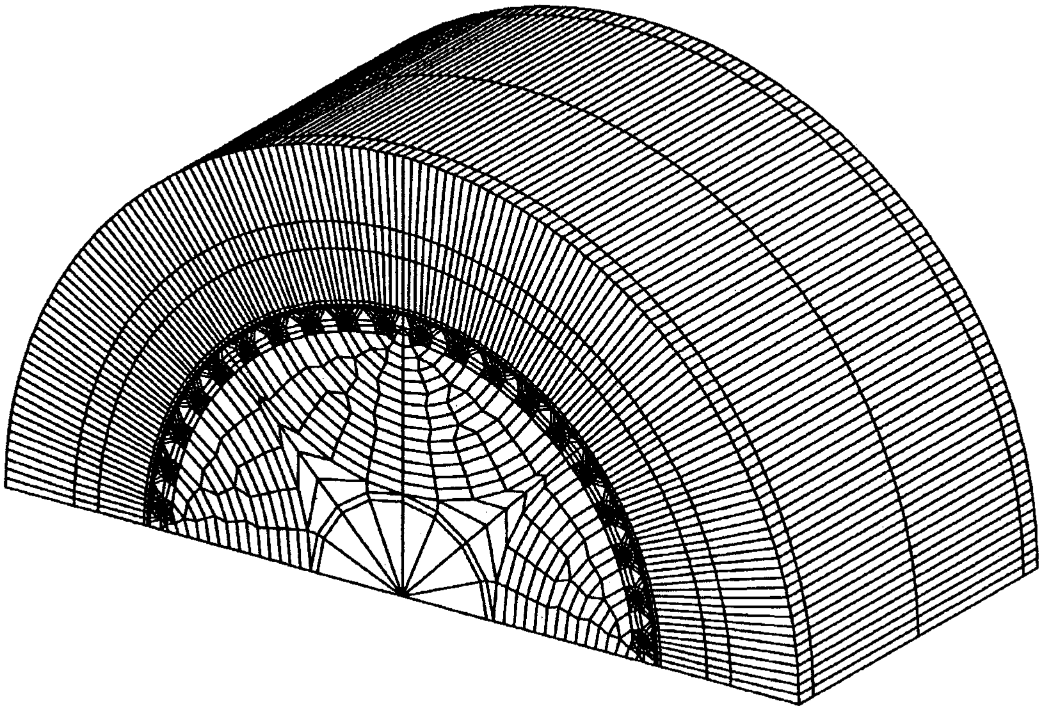

Figure 4.14: Detailed Three-Dimensional 180 degree BDFM Finite Element Model

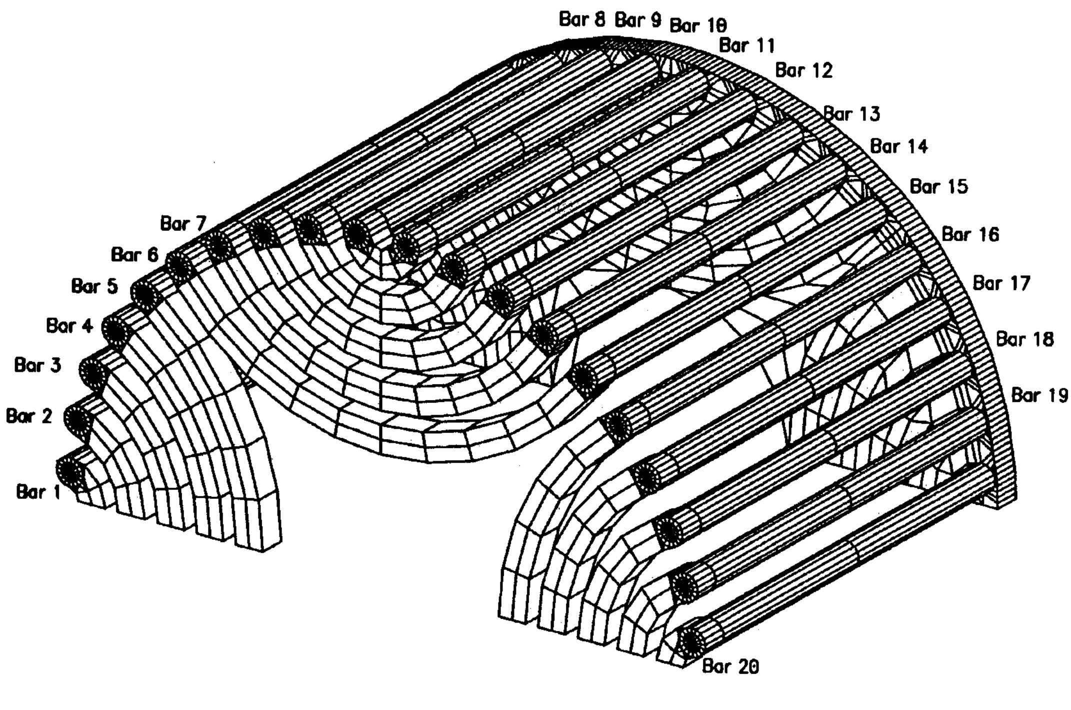


Figure 4.15: Rotor Bars Labels for the Detailed 180 Degree BDFM Model

The rotor bar currents are equal in magnitude and 180 degrees out of phase for rotor bars connected by a loop.

The magnitude of the rotor bar currents becomes greater as the slot span of the loops becomes larger (moving from the inner to outer loops). Figure 4.16 is a graph of slot span vs. rotor bar current magnitude, which shows that the current magnitude increases from the inner to outer loops.

| **Bar Number** | **Real, Imaginary** (Amperes) | **Magnitude, Phase** (Amperes, Degrees) |
|---|---|---|
| 1 | 0.0271658, -0.372618 | 0.373607, -85.8302 |
| 2 | 3.10542, 0.175448 | 3.11038, 3.23362 |
| 3 | 11.1839, 12.7192 | 16.9368, 48.675 |
| 4 | -16.7531, 21.1527 | 26.9833, 128.379 |
| 5 | 115.5, -40.2878 | 122.325, -160.771 |
| 6 | 30.2753, -140.761 | 143.98, -77.8615 |
| 7 | -8.09887, -75.7353 | 76.1671, -96.1038 |
| 8 | -8.56011,-39.4474 | 40.3655, -102.243 |
| 9 | 1.19354, -28.9307 | 28.9553, -87.6376 |
| 10 | -4.10281,-3.87263 | 5.64184, -136.653 |
| 11 | 4.10331, 3.86986 | 5.6403, 43.3229 |
| 12 | -1.19341, 28.932 | 28.9566, 92.362 |
| 13 | -1.19341, 39.4493 | 40.3674, 77.7569 |
| 14 | 8.09896, 75.7372 | 76.169, 83.8963 |
| 15 | -30.2749, 140.76 | 142.979, 102.138 |
| 16 | -115.502, -40.2885 | 122.326, -160.771 |
| 17 | -16.7533, 21.1523 | 26.9832, 128.38 |
| 18 | 11.1836, 12.7189 | 16.9364. 48.6752 |
| 19 | 3.10525, 0.175529 | 3.1102, 3.23529 |
| 20 | 0.0270078, -0.372556 | 0.373534, -85.8537 |

Table 4.4: Total Calculated Currents in the Rotor Bars for the Detailed 180 Degree BDFM Model

Information of this type, on slot span vs. rotor bar current magnitude for the range of operating frequencies, should be used to design a more effective rotor for the

BDFM. A grading of bar size within the loops of each nest, with the outer loops having the largest conductor size, should be investigated as a possible means of equalizing or improving current distribution within the loops, or equalizing loss distribution to minimize thermal problems.

#### 4.4.4.1.2 Distribution of Conduction Current Density Within the Rotor Bars

Distribution of conduction current density within each rotor bars was also examined for the detailed 180 degree model. Plots of conduction current density across each rotor bar from bottom to top and also from right to left were made in order to observe how the current is distributed within the rotor bars.

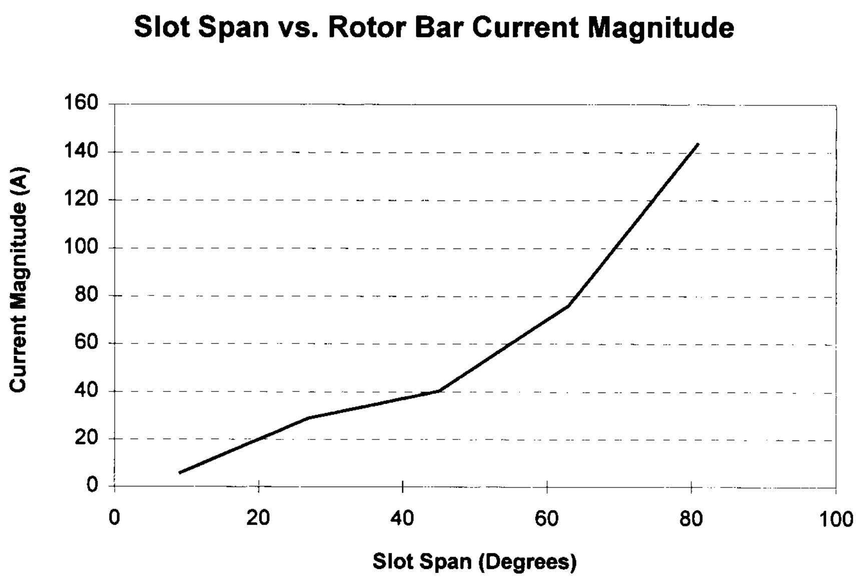


Figure 4.16: Slot Span vs. Rotor Bar Current Magnitude

Figure 4.17 shows the path along which the conduction current density was plotted from bottom to top. Figure 4.18 is a plot of conduction current density in bar 6 along the path indicated in Figure 4.17. Figure 4.19 is a plot of conduction current density in bar 7 along the path indicated in Figure 4.17.

These figures show that the conduction current density varies across the rotor bars from bottom to top, but this variation is not great.

Figure 4.20 shows the path along which the conduction current density was plotted from right to left. Figure 4.21 is a plot of conduction current density in bar 6 along the path indicated in Figure 4.20. Figure 4.22 is a plot of conduction current density in bar 7 along the path indicated in Figure 4.20.

These figures show the conduction current density varies across the rotor bars from right to left. This variation is greater than the variation observed from bottom to top. This variation is due probably to the magnetic field distribution surrounding the rotor bars, but is not of a sufficient magnitude to cause concern.

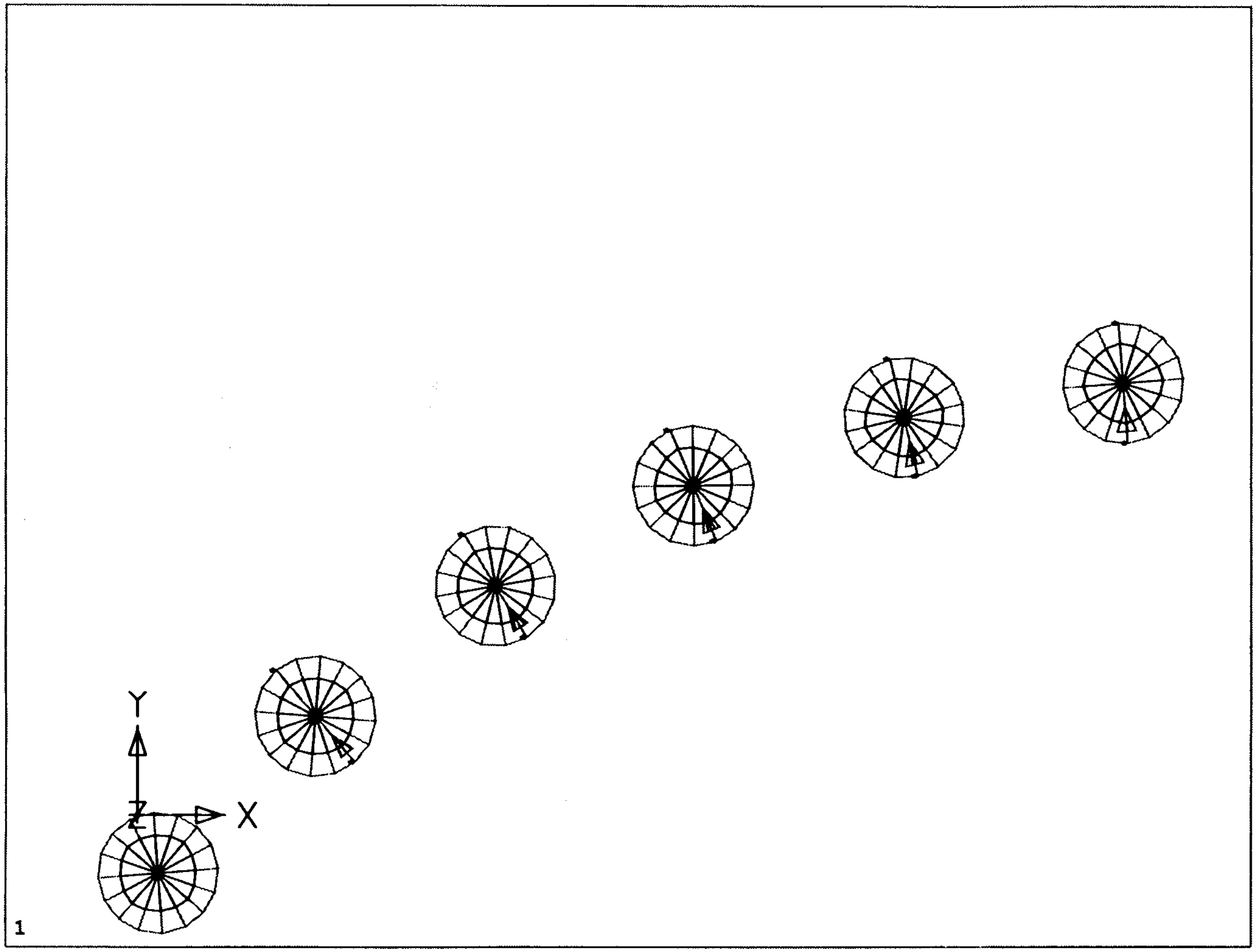


Figure 4.17: Path along which Conduction Current Density was Plotted from Bottom to Top

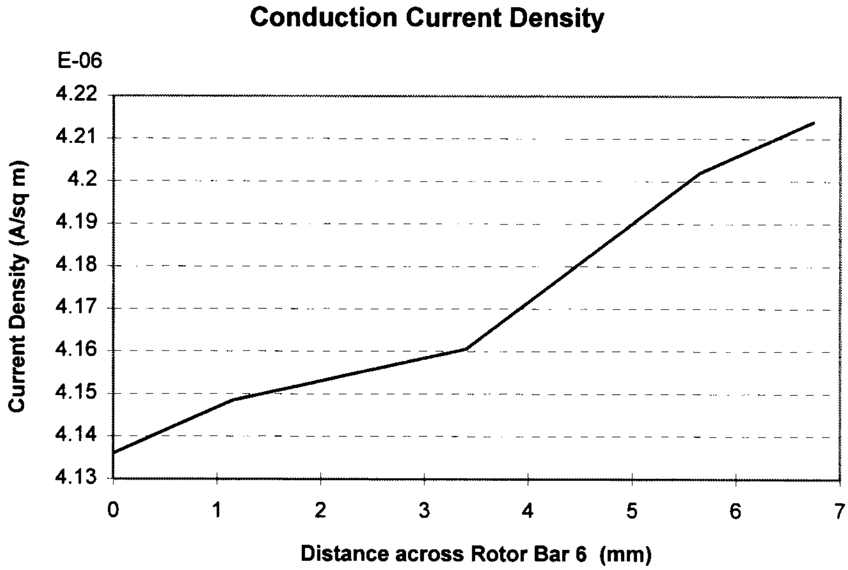


Figure 4.18: Conduction Current Density in Bar 6 (in reference to Figure 4.17)

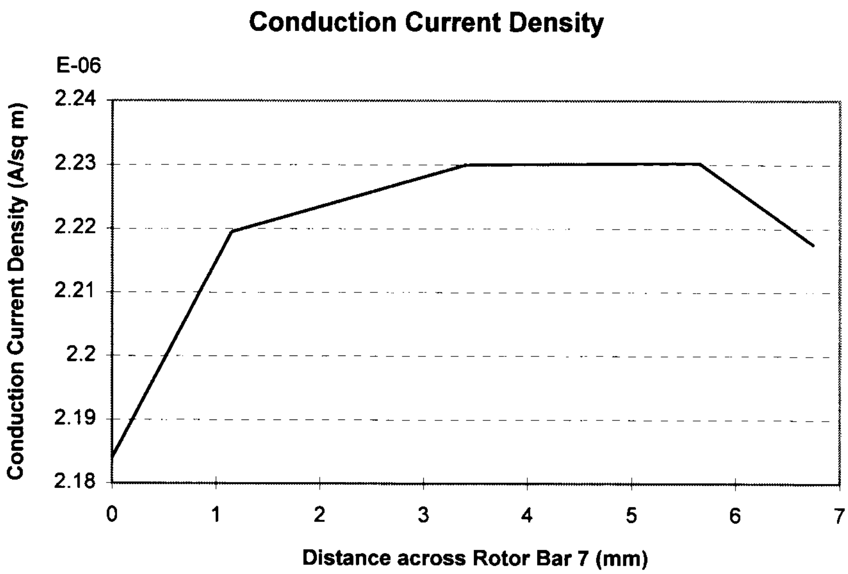


Figure 4.19: Conduction Current Density in Bar 7 (in reference to Figure 4.17)

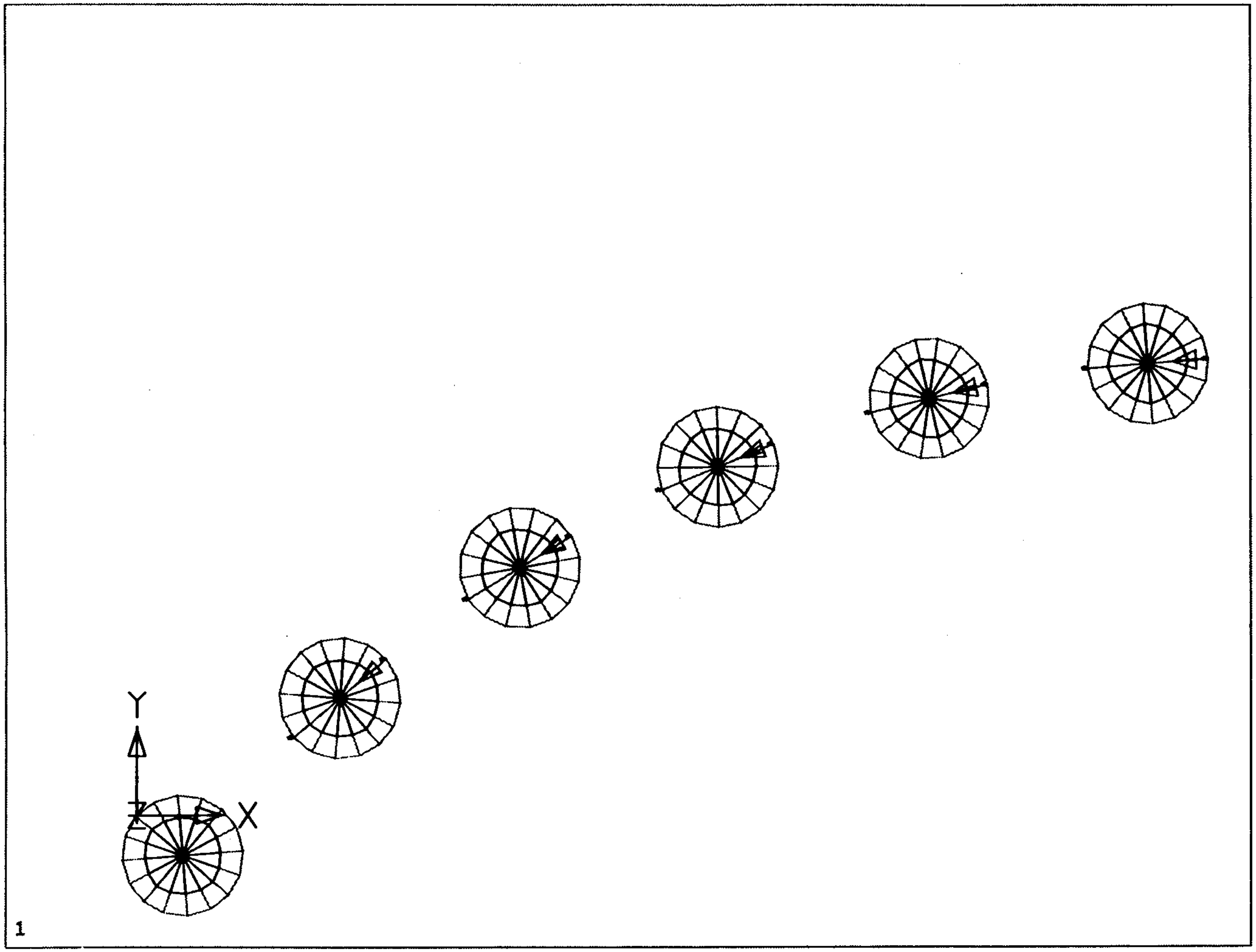


Figure 4.20: Path along which Conduction Current Density was Plotted from Right to Left

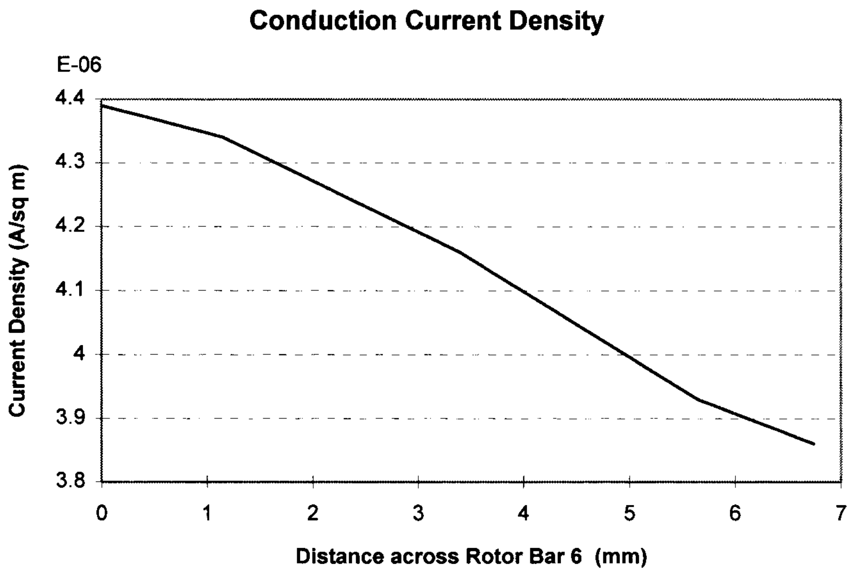


Figure 4.21: Conduction Current Density in Bar 6 (in reference to Figure 4.20)

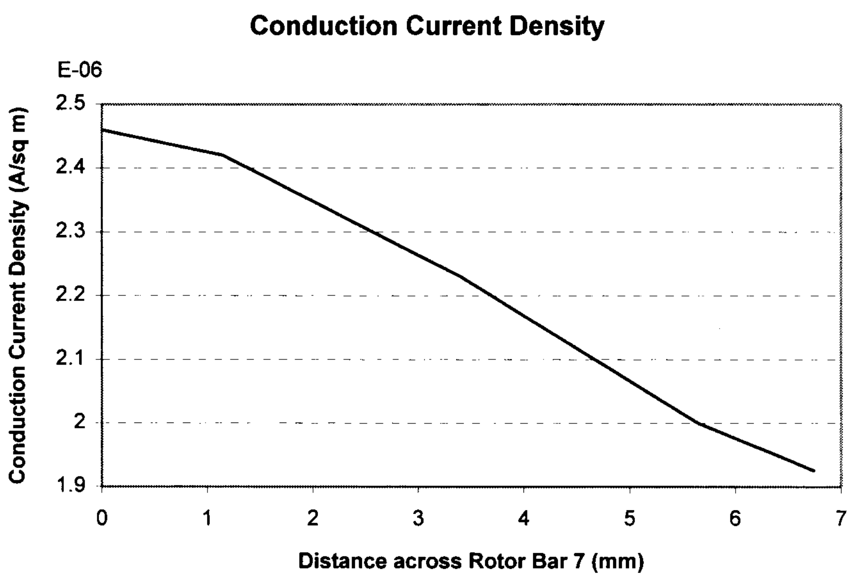


Figure 4.22: Conduction Current Density in Bar 7 (in reference to Figure 4.20)

# 5. Comparison of Two Three-Dimensional Finite Element Analysis Software Packages

## 5.1 Introduction

In the course of investigating three-dimensional finite element analysis for the BDFM, two different commercially available finite element analysis software packages were examined and tested. The first was Maxwell 3D Field Simulator produced by Ansoft Corporation [14], and the second was MSC/EMAS (Electromagnetic Analysis System) [10] and MSC/XL [15] by MacNeal-Schwendler Corporation (MSC). This chapter will compare these two software packages and discuss their advantages and disadvantages/limitations.

## 5.2 Maxwell 3D Field Simulator by Ansoft Corporation

### 5.2.1 Advantages

The main advantages of Maxwell 3D Field Simulator by Ansoft Corporation are its solid modeling procedure, step-by-step design process, and automated meshing technique.

#### 5.2.1.1 Solid Modeling Procedure

In a solid modeler, the finite element model is defined by the device structure or geometry, which consists of a group of "solid" objects. Throughout the modeling process, the solid objects that define the model are manipulated by referring to their names. For example, as the device structure or geometry is being created, solid objects such as cylinders, blocks, spheres, etc., are each assigned a name. Objects can then be rotated, copied, displayed or removed from the display, and have material properties assigned to them by referring to their names. This is a very convenient and easy way of manipulating the device geometry.

The main advantage of a solid modeling procedure is that the finite element model is defined by its geometry, a set of solid objects, and not by the finite element mesh itself. This allows the mesh to be easily modified if necessary, without having to redefine that entire model. The mesh can be refined throughout the model, or only in a particular object by specifying the object name.

The solid modeling procedure is one major advantage that Maxwell 3D Field Simulator has over MSC/XL. MSC/XL uses a wireframe modeling procedure, in which the finite element model is defined by the actual finite elements and grid points. A wireframe modeler is more difficult to use than a solid modeler. A wireframe model is manipulated by referring to collections of finite elements that make up the geometry. Therefore, the user must keep track of which element identification numbers belong to what part of the geometry. If the model is large, this can be quite a task. Also, in a wireframe modeler, once the finite element model is completed, it is not possible to change or refine the mesh without creating the whole model over again, which requires a substantial investment of time.

With Maxwell 3D Field Simulator's solid modeling procedure, analysis results can be calculated and displayed for a particular object by selecting its name. The solid modeling procedure makes creation, viewing, refinement of the mesh, and results analysis easy for the user.

#### 5.2.1.2 Step-by-Step Design Procedure

Maxwell 3D Field Simulator uses a step-by-step design procedure, which makes the program easy for users unfamiliar with finite element analysis to learn and use. When the program is started, the Maxwell 3D Field Simulator main menu appears, listing the general procedure steps for the user to follow. These general procedure steps are [14]:

Select Solver Type

Draw Geometric Model

Setup Materials

Setup Boundary Conditions

Setup Executive Parameters

Setup Solution Parameters

Solve

View Fields

The program displays a check mark next to each step after it has been successfully completed. In general, the steps must be chosen in the sequence listed above. For example, the "Setup Materials" step is operable only after a geometric model has been created using the "Draw Geometric Model" step. This step-by-step procedure is helpful for new users because it makes sure that each design step is completed, and in the appropriate order.

#### 5.2.1.3 Automated Meshing Technique

The Maxwell 3D Field Simulator uses an automated meshing technique. The program automatically generates an initial finite element mesh when "Setup Materials" is chosen from the main menu. If desired, the user has the option of refining the mesh in selected areas once the initial mesh in complete, by choosing the object to be refined. The program then automatically adds a specified number of additional elements to the selected object.

The automated meshing procedure used by Maxwell 3D Field Simulator has the advantage of being faster and much easier to use than a manual meshing technique, such as the one used by MSC/XL. Manual meshing is slow and requires a lot of attention from the user, as it is prone to user errors. An automatic meshing procedure is very helpful for users unfamiliar with finite element analysis, who may not know how to design an effective finite element mesh.

The automated meshing procedure used by Maxwell 3D Field Simulator does not give the user control over the exact size, shape, and position of each individual element, as the manual meshing technique used by MSC/XL does. However, this type of user control is probably not necessary. Automated meshing is faster, much easier for the user, and reduces the chance of user error. An automated meshing technique is definitely a benefit for new users.

### 5.2.2 Disadvantages/Limitations

#### 5.2.2.1 Only Two Analysis Modules Available

At the time of this evaluation, Maxwell 3D Field Simulator included only two analysis modules (or solution methods) that were available as full releases. These were the Electrostat module and the Magnetostat module. The Electrostat module is used to compute static electric fields due to voltage distributions and charges. It has no use for the BDFM analysis problem. The Magnetostat module is used to compute static magnetic fields, due to DC currents, static external magnetic fields, and permanent magnets. It has some use for the BDFM problem if the rotor bar currents are already known from lab data and the magnetic field distribution and magnitude needs to be calculated. It cannot be used to calculate induced currents in the rotor conductors, one of the main quantities of interest in the BDFM analysis problem.

The Maxwell 3D Field Simulator analysis module that was used predominately was the Eddy Current module. This module can be used to calculate time-varying magnetic fields due to AC currents and oscillating external magnetic fields. It should be used for the BDFM problem to calculate currents induced in the rotor bars by the AC three-phase stator windings. However, at the time of evaluation, the Eddy Current

Module was still a "beta test version", which can explain the many glitches and problems that were encountered in working with it.

#### 5.2.2.2 Solution Parameters have to be Re-entered each Time a Modification is Made

In the setup procedure used by Maxwell 3D Field Simulator, all of the material property and excitation setup parameters for the entire model have to be re-entered each time any modification is made. Likewise, each time a modification is made to the model boundary conditions, all of the boundary conditions must be re-entered. In finite element modeling, it is often informative to observe the effect of changing one model property at a time. For example, the material property of one object in the model may be changed, the magnitude of the excitations may be varied, or a particular boundary condition may be changed. Having to re-enter all of the material properties and excitations or re-enter every boundary condition each time a small modification is made is not very convenient.

#### 5.2.2.3 Very Poor Program Diagnostics

The main problem with Maxwell 3D Field Simulator was its very poor program diagnostics. No error messages are provided by the program to help the user identify problems during the solution process.

#### 5.2.2.3.1 Program Continues to Execute and Status is not Available to the User

While Maxwell 3D Field Simulator was solving a finite element problem, the program would continue to execute without issuing error or information messages. The only information provided to the user about the solution status was a shaded bar, ranging from 0 to 100 percent, that would appear on the screen supposedly to allow the user to monitor the progress of the system in completing the solution. This presented the problem that sometimes the program would continue to "solve", sometimes for over one week, with the shaded bar still at 0 percent and no other information available. The program did not display any error or information messages (such as disk full, more swap space needed, more memory needed, etc.) to let the user know what was happening or what problems had been encountered. The user was unaware whether to continue to let the program solve, or to choose to terminate and make modifications.

Occasionally, the solution process would cause the system to "crash". No error or information messages were available to the user about what caused such a "crash".

#### 5.2.2.4 Poor Customer Support

In addition to having no error messages available, poor customer support was provided for Maxwell 3D Field Simulator. Customer support was unable to identify exactly why the program was unable to solve the problem. The only information provided by customer support was that the problem size was too large, and more swap space was needed. However, they were unable to tell how much swap space was needed. Also, customer support would often not even answer the questions e-mailed to them. This lack of customer support made it difficult for the user to effectively use the program.

#### 5.2.2.5 No results due to Problem Encountered

Due to lack of program diagnostics and customer support, no useful results where obtained by the Maxwell 3D Field Simulator for the BDFM problem. Attempts were made to obtain a solution by reducing the size of the BDFM model by reducing the number of rotor bars and nested loops, reducing the number of stator windings, and reducing the length of the machine. The program was finally able to generate a solution using a much simplified model. However, the program indicated that the solution had 120 percent error due to having a finite element mesh that was too coarse. When the finite element mesh was refined, the program was again unable to obtain a solution to the problem.

## 5.3 MSC/XL and MSC/EMAS by MacNeal-Schwendler Corporation

### 5.3.1 Advantages

#### 5.3.1.1 Many Modeling Modules Available for a Variety of Problems

MSC/EMAS includes a large number of analysis modules (solution methods), so a large variety of electromagnetic problems can be modeled. These solution methods include electrostatic, current flow, magnetostatic, magnetostatic with current flow, nonlinear magnetostatic, nonlinear magnetostatic with current flow, AC, modal AC, transient, modal transient, nonlinear transient, real eigenvalue, complex eigenvalue, and modal complex eigenvalue. The MSC/EMAS documentation includes a selection tree and other suggestions to help users select the appropriate solution method for their particular problem.

#### 5.3.1.2 Well Documented Program Diagnostics

MSC/EMAS errors are well documented. There are literally thousands of different fatal error, warning and information messages that can be issued by the program. These error messages each are assigned a number. Errors found during the solution process are listed in the <filename.f06> file, one of the output files of MSC/EMAS. Each message includes a brief explanation. More detailed explanations are contained in Chapter 6 of the MSC/EMAS User's Manual [10]. If the program is unable to solve the problem for some reason, the solution is automatically terminated within about one hour at maximum, and an explanatory message is issued in the <filename.f06> file.

#### 5.3.1.3 Setup Parameters are Saved and only have to be Entered Once

In MSC/XL, the setup parameters of the finite element model (the material properties, excitations, and boundary conditions) are each assigned an identifying number, and are saved as they are created so that they only have to be entered once. During the analysis preparation step, excitations and boundary conditions to be included in the analysis are selected by identification number. All of the excitation and boundary conditions can be included, or only selected ones. If a modification needs to be made, the appropriate material property or excitation identification number is selected and changed, with no need to re-enter all of the setup parameters. In the setup procedure used by Maxwell 3D Field Simulator, all of the setup parameters have to be re-entered each time any modification is made. Since the setup procedure is time-consuming and painstaking, the setup procedure used by MSC/XL is much more convenient.

### 5.3.2 Disadvantages/Limitations

#### 5.3.2.1 Wireframe Modeler

MSC/XL, the graphical pre- and post-processor designed for use in conjunction with MSC/EMAS, is a wireframe modeler. In a wireframe modeler, a device geometry is created by specifying points, curves, and surfaces along the dimensions of the model. The geometric surfaces serve as templates for creation of finite elements and grid points. The geometric entities are no longer needed once the finite element mesh has been completed, because the finite element mesh and grid points define the model. Material properties, excitations, and boundary conditions are applied directly to elements by specifying their identification number.

A wireframe modeler has two main disadvantages. First, a wireframe modeler is more difficult to use than a solid modeler. In a model with several thousand elements, keeping track of element identification numbers is more painstaking than simply manipulating objects by name. Second, in a wireframe modeler the model is defined by elements and grid points instead of by objects, therefore it is not possible to easily refine the mesh as with a solid modeler.

A solid modeler similar to that of Maxwell 3D Field Simulator would be beneficial for MacNeal-Schwendler users. MSC made available in September 1994 a new graphical pre- and post-processor called MSC/ARIES, meant to replace MSC/XL, which uses a solid modeling procedure. MSC/ARIES was not used for the work done in this thesis due to its late arrival.

#### 5.3.2.2 No Step-by-Step Procedure Menu

MSC/XL does not contain a main step-by-step procedure menu. New users who are not familiar with the basic steps involved in finite element modeling would probably find MacNeal-Schwendler's programs difficult to learn to use. A step-by-step main menu similar to that of Maxwell 3D Field Simulator would make MSC/XL easier and more intuitive to use.

## 6. Conclusions and Recommendations

The characteristics of the BDFM require adjustment of the finite element design procedure used for conventional, singly-fed induction machines. In this thesis, a three-dimensional finite element design procedure for modeling the BDFM has been developed.

This thesis has shown how the difficulties previously associated with finite element modeling of the BDFM have been overcome. Three-dimensional finite element analysis of the BDFM was investigated, which allows the nested loop structure of the BDFM rotor to be modeled. It has been shown how an ac analysis of the BDFM using the rotor reference frame frequency as the solution frequency can be used to calculate the induced or eddy currents in the rotor conductors. It has also been shown that the electromagnetic fields present in the 6/2 pole BDFM exhibit 180 degree symmetry. Therefore, the simulation of a 180 degree model of the BDFM with alternating periodic boundary conditions applied along the symmetry plane can be used in place of a full machine simulation, for 6/2 pole machine. For a BDFM with a different number of power and/or control winding poles, the symmetry of the electromagnetic fields would need to be reexamined in order to determine what portion of the machine to model.

The three-dimensional finite element design procedure developed in this thesis was used to model the 6/2 pole 5 horsepower BDFM laboratory machine. From the simulation results, the induced currents flowing in the BDFM rotor bars were calculated. These calculations indicated that an uneven current distribution exists within the nested rotor loops. The total current flowing in the rotor bars increases as one moves from the inner loop to the outer loop.

Potential exists to improve the rotor design of the BDFM using the three-dimensional finite element design procedure. A grading of bar size within the loops should be investigated as a possibility of improving current or loss distribution. The outer loops should contain the largest conductor sizes since the largest currents are induced in the outer loops. Variations of slot span for the loops should be investigated. Since the inner rotor loop carries a very small current compared to the outer loops, perhaps the inner loop could be eliminated entirely.

BDFM models should be set up with more realistic values for currents in the 6 and 2 pole stator windings. The current magnitude of 100 amp/turns peak per slot that was used in the excitation setup of the BDFM models as described in Chapter 4 was chosen for convenience. The actual currents flowing in the 6 and 2 pole stator windings are larger in magnitude, and not necessarily equal in magnitude. A finite element field solution using more realistic values for the 6/2 pole stator winding currents should be generated. The magnetic flux density values throughout the model should then be examined to determine areas where the stator or rotor cores may be saturating.

If the stator and/or rotor cores are found to be saturating when more realistic values for the stator winding currents are applied, then this saturation effect should be investigated further, taking into account the nonlinearity of the laminated steel. This could be accomplished by using the Nonlinear Magnetostatic module included with MSC/EMAS, which allows a B-H curve for laminated steel to be included in the analysis. The induced rotor currents calculated from an AC analysis, as well as the stator winding currents, would be used as input excitations. The field distribution and flux density values could then be examined when the nonlinearity of laminated steel stator and rotor is taken into account.

Other useful motor design parameters can be determined for the BDFM from finite element field solutions. These parameters include, but are not limited to, power loss, steady-state torque, and winding inductances. MSC/XL includes a built-in calculation for determining power loss in any group of elements in the model. Calculation of steady-state torque and winding inductance would require more in-depth consideration.

In the investigation of three-dimensional finite element analysis of the BDFM, two different finite element software packages were investigated: Maxwell 3D Field Simulator by Ansoft Corporation and MSC/EMAS and MSC/XL by MacNeal-Schwendler Corporation. An evaluation and comparison of these two software packages was presented in this thesis. MSC/EMAS and MSC/XL were found to be far superior to Maxwell 3D Field Simulator. MSC/EMAS and MSC/XL shown good potential for being an effective design tool for the BDFM, and it is recommended that this software continue to be used.

In September 1994, MacNeal-Schwendler released a new user interface, MSC/ARIES, which is meant to replace MSC/XL, the older user interface described in this thesis. MSC/ARIES uses a solid modeling procedure instead of the wireframe modeling procedure used by MSC/XL. MSC/ARIES was not examined in depth because of its late arrival. A brief overview of the MSC/ARIES manuals shows that it has potential for being an easier to use, faster, and more versatile user interface than MSC/XL. However, since no actual experimentation with MSC/ARIES was done, it is unknown if it contains program flaws typical for first releases. It is recommended that MSC/ARIES be investigated, however, if program flaws are found to exist, the modeling procedure presented in this thesis for MSC/XL can be used.

Additional disk space is also needed for the HP 715/50 workstation. Ideally, a 4 GB hard disk should be purchased. This 4 GB hard disk should be set up to contain both solution output files and the MSC/EMAS scratch directory which contains scratch files created during the solution process. Additional disk space is necessary because the HP 715/50 workstation contains only a 1 GB internal hard drive, which is insufficient. The work presented in this thesis was made possible by the loan of two additional external hard disks: a 500 MB hard disk and a 1 GB hard disk. Even this additional disk space was insufficient. Several BDFM models that were created did not run because of lack of hard disk space. MSC/EMAS was created such that the solution output files and the scratch temporary files reside within the same directory on the same drive therefore requiring that a large hard disk be used.

**BIBLIOGRAPHY**

[1] A.K. Wallace, R. Spee and H.K. Lauw, "The Potential of Brushless Doubly-Fed Machines for Adjustable Speed Drives," *Conference Record, IEEE IAS Pulp and Paper Industry Conference*, 1990, pp. 45-50.

[2] H.K. Lauw, "Characteristics and Analysis of the Brushless Doubly-Fed Machine," Final Report U.S. Department of Energy, Contract No. 79-85BP24332, June 1989.

[3] R. Spee, A.K. Wallace and H.K. Lauw, "Simulation of Brushless Doubly-Fed Adjustable Speed Drives," *IEEE IAS Annual Meeting Conference Record*, 1989, pp. 738-743.

[4] A.K. Wallace, R. Spee and H.K. Lauw, "Dynamic Modeling of Brushless Doubly-Fed Machines," *IEEE IAS Annual Meeting Conference Record*, 1989, pp. 329-334.

[5] R. Li, R. Spee, and A.K. Wallace, "Synchronous Drive Performance of Brushless Doubly-Fed Motors," *IEEE IAS Annual Meeting Conference Record*, 1992, pp. 631-638.

[6] L.J. Hunt, "A New Type of Induction Motor," *Journal IEE* (London), 1907, vol. 39. pp. 648-677.

[7] F. Creedy, "Some Developments in Multispeed Cascade Induction Motors," *Journal IEE* (London), 1927, vol. 59, pp. 511-532.

[8] A.R.W. Broadway and L. Burbridge, "Self-Cascaded Machine: A Low Speed Motor or High Frequency Brushless Alternator," *Proceedings IEE* (London), 1970, vol. 117, pp. 1277-1290.

[9] M.A. Salim, and R. Spee, "High Frequency Cage Rotor Designs," *IEEE IAS Annual Meeting Conference Record*, 1992, pp. 18-24.

[10] J.R. Brauer, B.E. MacNeal, and J.C. Neuner, "MSC/EMAS User's Manual," The MacNeal-Schwendler Corporation, July 1994.

[11] J.R. Brauer, A. Frenkel, and M.A. Gockel, "Complex Periodic Boundary Conditions for AC Finite Element Models," The MacNeal-Schwendler Corporation, 1994.

[12] J.R. Brauer, "What Every Engineer Should Know About Finite Element Analysis," The MacNeal-Schwendler Corporation, 1990.

[13] B.E. MacNeal, "MSC/EMAS Modeling Guide," The MacNeal-Schwendler Corporation, 1991.

[14] "Maxwell 3D Field Simulator User's Reference," Ansoft Corporation, 1993.

[15] Ken Peterson, "MSC/XL User's Manual," The MacNeal-Schwendler Corporation, April 1993.

[16] "MSC/EMAS AC Analysis User Interface Guide," The MacNeal-Schwendler Corporation, April 1993.

[17] "MSC/EMAS Installation Procedures for HP 9000/700 (HP-UX)," The MacNeal-Schwendler Corporation, August 1993.

# APPENDIX

# A. Tutorial for Setting up and Solving a 3D BDFM Model using MSC/XL and MSC/EMAS

## A.1 Introduction

The goal of this tutorial is to guide a new user of MSC/XL and MSC/EMAS through the creation, setup, simulation, and analysis of a BDFM model. The 5 horsepower BDFM lab machine discussed in Chapter 4 is used as an example. The step-by-step creation of the detailed 180 degree model presented in Chapter 4 is described in this chapter. This tutorial will not attempt to give a full description of all the commands and options available in MSC/XL and MSC/EMAS. An overview of basic commands used in MSC/XL and one approach to creating a BDFM model is presented. Topics important to creating an effective finite element model will be discussed.

## A.2 An Overview of MSC/XL

MSC/XL is a graphical pre- and post-processor application designed for use in conjunction with MacNeal-Schwendler's finite element analysis software [15]. The two products that MSC/XL currently supports are MSC/NASTRAN (mechanical engineering problems) and MSC/EMAS (electromagnetic field analysis). With MSC/XL, users can build complete, ready-to-run finite element models, then analyze them with MSC/EMAS or MSC/NASTRAN without leaving the MSC/XL environment. Analysis results can also be displayed by MSC/XL. This combination of software tools provides users with a complete finite element modeling, analysis, and results processing package.

### A.2.1 Screen Layout

The screen layout for MSC/XL is shown in Figure A.1.

In MSC/XL, there are two methods of entering commands. Commands can be entered by typing them in the blue bar (command line) or they can be entered by picking from the cascading/pop-up menus. When a command is entered from picks on the cascading or pop-up menus, the equivalent typed command version is displayed in the history tile. The two methods of entering commands in MSC/XL can be used interchangeably. MSC/XL provides the user with control over all visual contents, including labels, visibility, colors, titles, multiple views with different orientations, multiple data displays, etc.

### A.2.2 Using the Mouse

In MSC/XL, each of the three mouse buttons has a distinct function.

- Pick Button: The left mouse button is used to pick items from the cascading menus, the quick access menu (QAM), the graphics tile, or the history tile.
- Change Button: The middle mouse button is used to toggle items in the QAM, activate pop-up choices in the QAM, activate blue bar input in pop-up menus, and act as a return key for blue bar input.

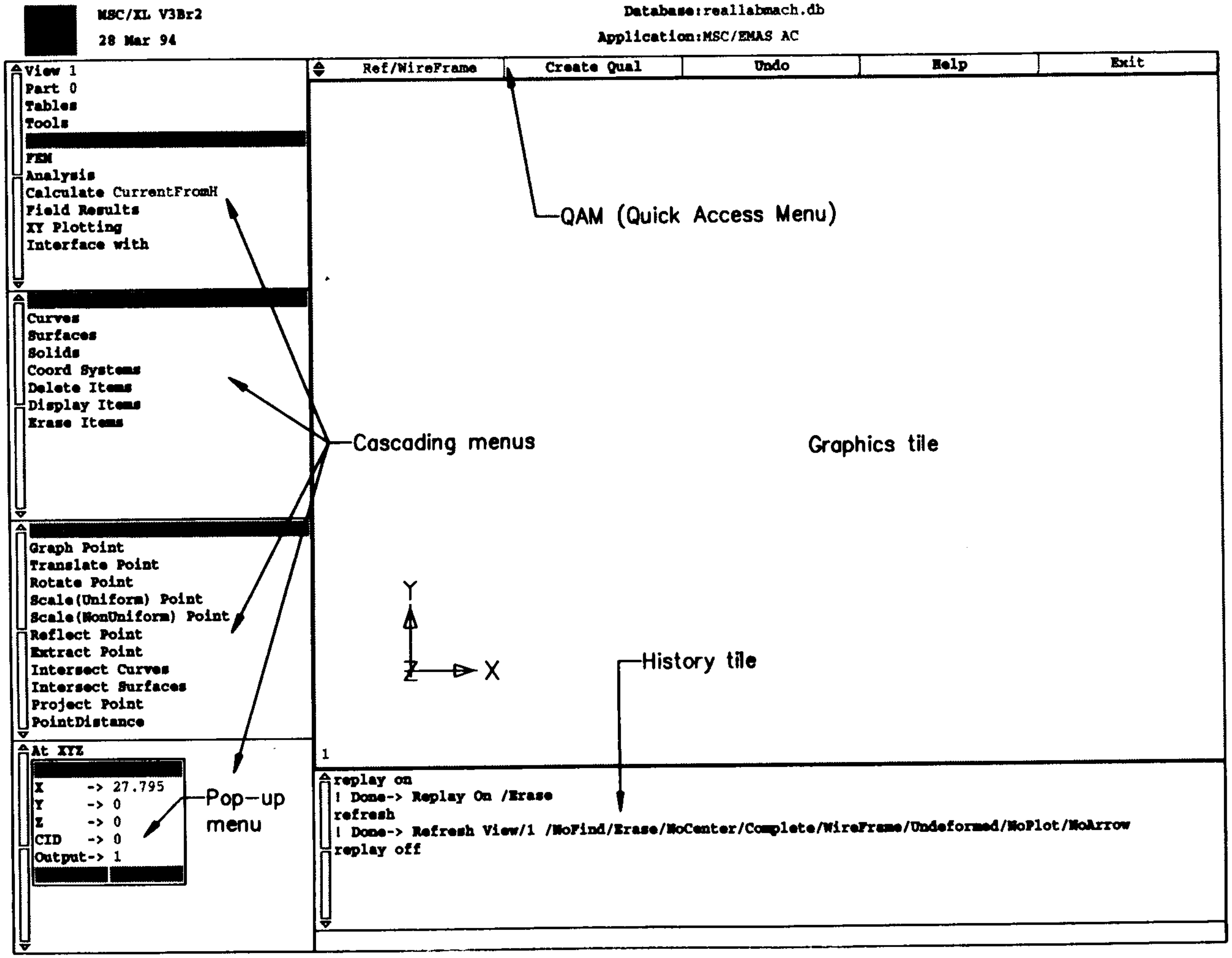


Figure A.1: MSC/XL Screen Layout

- Cancel Button: The right mouse button is used to close a pop-up menu or cancel blue bar input in a pop-up menu.

### A.2.3 Capabilities

MSC/XL supports all aspects of model building. Model geometry is specified by defining various geometric entities, including points, curves, surfaces, solids, and coordinate systems. Meshing procedures are then used to subdivide geometric entities into meshes containing finite elements connected to grid points. Excitations, boundary constraints, and material properties can then be added using forms unique to field analysis. MSC/XL also has a number of model checking features. For standard types of analysis, it is possible to set up the entire input file, run an MSC/EMAS analysis, and look at results without leaving MSC/XL. For advanced applications, a small amount of hand editing in the input file may be needed.

### A.2.4 Data Files

MSC/XL has access to several data files, as shown in Figure A.2. MSC/XL has its own database, <filename.db>, where it stores model data and data tables in binary form. During each MSC/XL session, all the typed commands the appear in the history tile are stored in an ASCII file called <filename.hist>. MSC/XL produces an ASCII input file, <filename.dat>, which is read as input to the solver, MSC/EMAS. Results from MSC/EMAS are contained in a binary data base, <filename.xdb>. MSC/XL can access multiple <filename.xdb> files generated from a single model geometry (same elements and grid points) in order to compare results from different runs. External ASCII data can also be brought in and plotted using MSC/XL's XY-plotting capabilities.

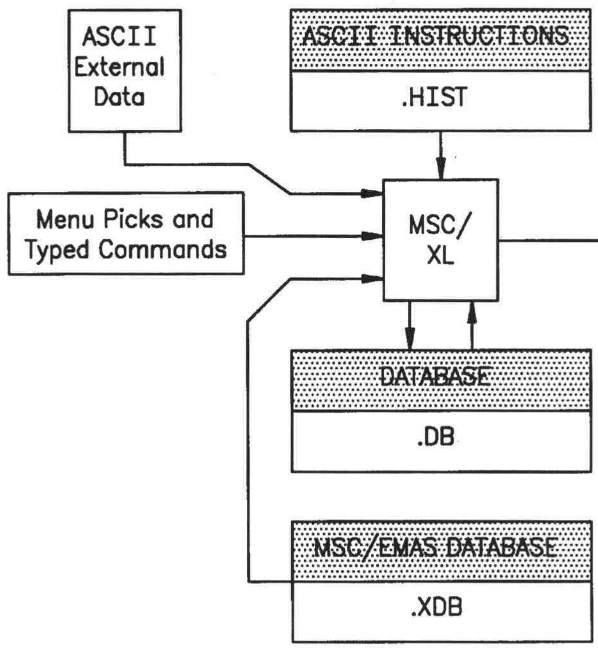


Figure A.2: Data Flow in MSC/XL

Output data can be represented in various forms. Options include three-dimensional arrow plots, three-dimensional contour plots, arrow or contour plots through a cross section of the model, or XY plots along a specified path through the model. MSC/XL can plot results calculated by MSC/EMAS using the Results Database or can calculate new results based on database quantities using the Results Calculator.

## A.3 An Overview of MSC/EMAS

MSC/EMAS (Electromagnetic Analysis System) is a general purpose finite element program for analyzing electromagnetic fields. Analysis methods are based on a vector potential formulation of electromagnetism. This formulation results in a single matrix equation which represents Maxwell's equations in their complete and general form, as discussed in Chapter 3. Solutions to this equation are obtained using a formal series of matrix operations. Though MSC/EMAS has enough input and output capabilities to stand alone as a field analysis program, it is generally used as the "solver" along with the graphics pre- and post-processor, MSC/XL.

The matrix equations representing Maxwell's equations are solved through a series of formal matrix operation (e.g., multiplication, decomposition, eigenvalue extraction), called solution sequences. Each operation in the sequence is carried out by an independent group of subroutines called a module.

Matrix operations are specified in a unique internal programming language, DMAP (Direct Matrix Abstraction Programming). MSC/EMAS comes with a number of standard solution sequences, which implement common forms of analysis, e.g., static analysis, frequency response (AC) analysis, transient analysis, and eigenvalue analysis Users can select any of the standard sequences, or make up DMAP programs of their own.

MSC/EMAS creates/uses several data files, as shown in Figure A.3. These include files are described briefly as follows:

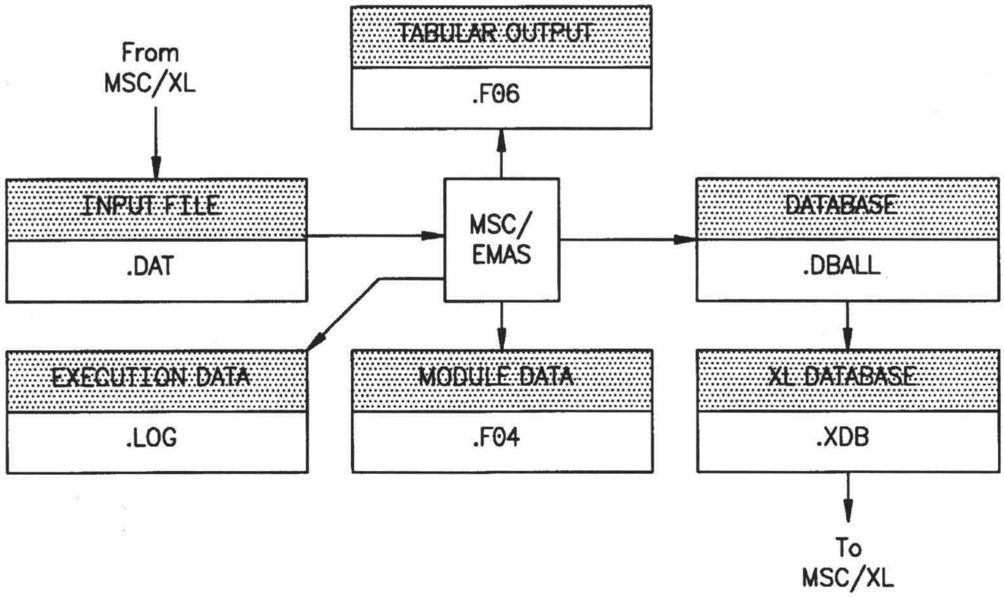


Figure A.3: Data Flow in MSC/EMAS

- <filename.dat> (ASCII): This is the input data file for MSC/EMAS, which can be written automatically by MSC/XL for standard analysis methods.
- <filename.f06> (ASCII): This file is the main, user-oriented program output file. It contains echoes of the input data, information from various numerical modules, warning and error messages, and tabulated output.
- <filename.f04> (ASCII): This file contains information on the execution of modules, including clock time, CPU time, and I/O usage.
- <filename.log> (ASCII): This small file contains information on the configuration of the computer at execution time, and final accounting statistics for the job.

- <filename.dball> (Binary): This large file contains the database for MSC/EMAS. It contains model information, internal data tables, and output information.
- <filename.xdb> (Binary): This file contains model data and results to be used by MSC/XL.
- Scratch Files (Binary): Temporary scratch space is needed during the solution process.

These output files from MSC/EMAS contain much useful information, especially when trying to correct model errors.

### A.4 Modeling Tasks

Finite element modeling requires the following major analysis tasks:

- Produce the model
  - Create geometry
  - Generate the finite element mesh
  - Apply material properties, excitations, and boundary conditions
- Solve the matrix problem
- Validate the solution

Each of these major analysis tasks will be discussed in relation to the BDFM motor model.

### A.4.1 Planning the MSC Session

Several topics should be thought out before actually sitting down at the computer to create the model. Planning beforehand can save lots of time correcting errors later.

#### A.4.1.1 Deciding on Units

All the machine's geometric dimensions must be entered in the same units, e.g., inches, millimeters, meters, etc. It is not necessary to specify the units being used until just before the model is run, but one set of geometry units must be chosen at the beginning and used consistently when creating geometric entities. The device dimensions may be in any set of length units, as long as the same units are consistently used.

All other input quantities (material properties and excitations) must also be entered in a consistent set of units, and all output quantities must be interpreted in these same units. MSC/EMAS uses the MKS system of units as its default system due to its wide acceptance. The user can elect to use any other consistent set of units. Table A.1 shows common input/output quantities and the corresponding MKS units.

| Quantity | Symbol | MKS Units |
|---|---|---|
| Mass | m | kg |
| Length | L | m |
| Time | t | sec |
| Charge | Q | coul |
| Energy | W | $kg\text{-}m^2/sec^2$ |
| Frequency | f | $sec^{-1}$ |
| Capacitance | C | farad |
| Conductivity | $\sigma$ | mho |
| Current | I | amperes |
| Current Density | $\vec{J}$ | $amps/m^2$ |
| Displacement | $\vec{D}$ | $coul/m^2$ |
| Electric Field | $\vec{E}$ | volt/m |
| Electric Potential | $\phi$ | volts |
| Time Integrated Electric Potential | $\Psi$ | webers |
| Inductance | L | henrys |
| Magnetic Field Strength | $\vec{H}$ | amps/m |
| Magnetic Flux Density | $\vec{B}$ | $webers/\ m^2$ |
| Permeability | $\mu$ | henrys/m |
| Permittivity | $\varepsilon$ | farads/m |
| Reluctivity | $\upsilon$ | m/henry |
| Resistance | R | ohms |
| Resistivity | $\rho$ | ohm-m |
| Vector Potential | $\vec{A}$ | webers/m |

Table A.1: MKS Units

#### A.4.1.2 Drawing a symmetry "Wedge"

A hand sketch of a radial cross section of the BDFM model should be drawn to prepare for geometry creation. Figure A.4 shows an example sketch of the BDFM lab machine.

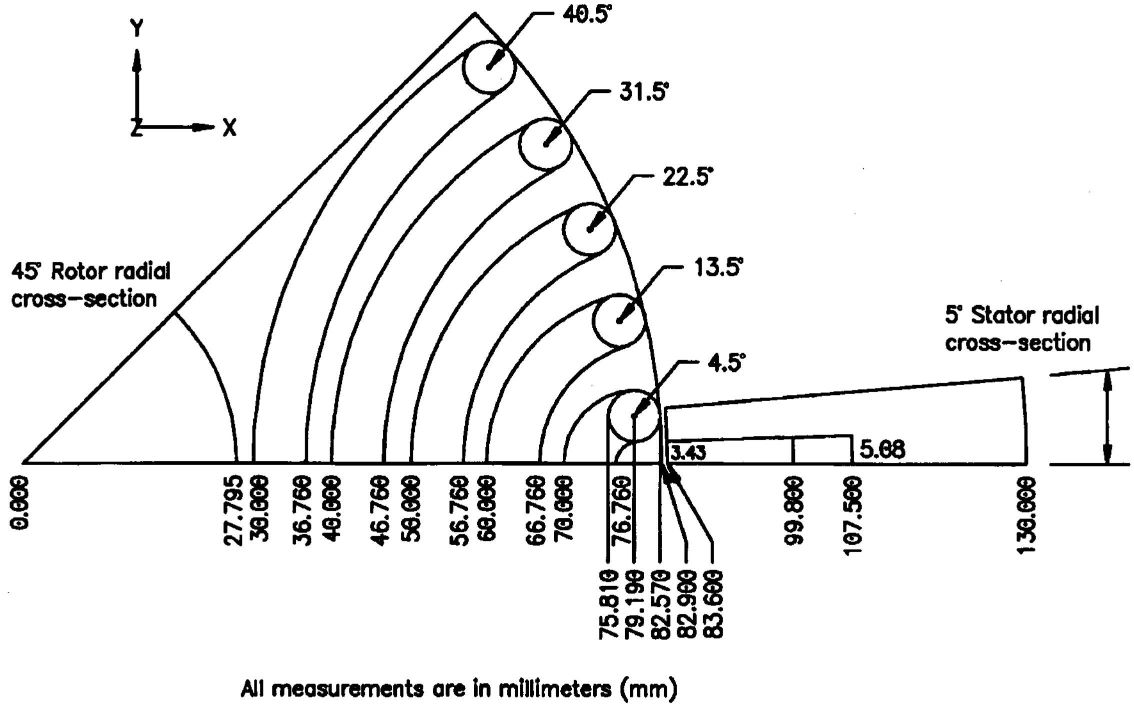


Figure A.4: Radial Cross-Section of the BDFM Model

The hand sketch should show the location of each point and curve that will be used to construct the geometry. The intersection of each curve with the x-axis should be labeled with a location, as each of these points will be the start of the geometry.

A three-dimensional finite element mesh is constructed in MSC/XL by "extruding" or sweeping selected elements of a two-dimensional mesh over a specified distance. Therefore, the two-dimensional mesh, which will be created first, must include an element pattern that can be extruded to create a three-dimensional mesh. In other words, for the BDFM problem, the outline of the endring and nested loops must appear in the two-dimensional mesh pattern.

Observe the radial symmetry of the machine cross section. In other words, notice what smallest radial section of the machine cross section geometry can be rotated and/or reflected to form the entire section of the machine being modeled. The BDFM lab machine geometry shown in Figure A.4 exhibits 45 degree radial symmetry for the rotor, and 5 degree symmetry for the stator.

Since the rotor and stator structure exhibit different degrees of radial symmetry, two-dimensional rotor and a stator radial wedges are created and meshed separately from each other. First, the 45 degree rotor wedge is created and meshed. Then the 5 degree stator wedge is created, meshed, and reflected and rotated to create a 45 degree wedge. The rotor and stator meshes are then connected by meshing the air gap. Once a 45 degree model of the rotor, stator, and air gap is completed, this section can be reflected and rotated to create a two-dimensional 180 degree model. Finally, selected sections of the two-dimensional mesh are extruded to form a three-dimensional 180 degree model.

#### A.4.1.3 Entering MSC/XL

To begin an MSC/XL session, follow these steps:

1. Type xl at the system prompt.
2. In the Choose a Database File pop-up menu, pick an existing file or pick New and enter the new filename in the blue bar.
3. In the Choose Application pop-up menu, pick MSC/EMAS.
4. In the Choose Subapplication pop-up menu, pick the subapplication to be used for this analysis. The subapplication can easily be changed later by clicking on Application: MSC/EMAS (Subapplication) in the logo tile.

### A.4.2 Creating Geometry

The first step in building a finite element model with MSC/XL is to create the geometry that defines the model's shape. Geometry is actually any number of points, curves, surfaces and solid figures that act as templates for the finite element mesh. It is the geometry that normally fixes the exact size and position of object in the model. The purpose of the geometry is only to serve as a template for the finite element mesh, consequently, it can be deleted once the mesh has been created.

The general procedure to be followed in creating BDFM model geometry is to use the hand sketch drawn earlier to create two radial cross sections:

1. Create a geometric model of the 45 degree rotor radial cross section.
2. Create a geometric model of the 5 degree stator radial cross section.

#### A.4.2.1 "Undo" Command and "Delete Item" Option

The best way to learn how to create a model geometry and finite element mesh in MSC/XL is to experiment and observe the results. Two helpful options in this process are the Undo command and the Delete Item option. The Undo command is located in the quick access menu. Undesired results can quickly be reversed by using the Undo command, which reverses the last command performed. Only one command can be reversed. If a mistake is several commands behind, the Delete Item option removes unwanted points, curves, surfaces, elements, or grid points. Items can be deleted with the Delete Item option as follows:

1. Pick Geometry → Delete Item from the cascading menus.
2. Choose the type of item to be deleted. The items can then be deleted by picking them directly from the graphics tile.

#### A.4.2.2 Defining a New Coordinate System

One coordinate system is defined automatically in MSC/XL. This is the basic Cartesian coordinate system, which has identifier 0, and is centered at the origin (0,0,0). The direction of the basic coordinate system axes are shown in the MSC/XL graphics tile. The basic coordinate system provides the frame of reference for all entities (including other coordinate systems). It is possible for the user to define new coordinate systems relative to the basic coordinate system, or some other previously defined coordinate system. Use of alternate coordinate systems can make geometry creation easier.

MSC/XL allows the user to define three types of coordinate systems: rectangular, cylindrical, and spherical. A cylindrical coordinate system is useful in geometry creation and problem setup. A basic cylindrical coordinate system with center at the origin can be created as follows:

1. Pick Geometry → Coord System → Define Coord System → By Origin/Angles from the cascading menus.
2. In the pop-up menu, enter the following parameters:
   - Origin X,Y,Z: 0,0,0 (Defines the origin of the new coordinate system).

- Angle 1,2,3: 0,0,0 (Defines the rotation of the new coordinate system from an existing coordinate system about Axis 1,2,3).
- Axis 1,2,3: X,Y,Z (Defines how the coordinate axes are specified in relation to an existing coordinate system).
- Type: Cylindrical (The type of new coordinate system).
- CID: 0 (The identification number of the existing coordinate system on which the new coordinate system is based).
- Output: 1 (The identification number of the new coordinate system).

#### A.4.2.3 Creating Points

Geometry is created by first defining points. Points are created along the x-axis of the basic coordinate system, at the x-coordinate positions labeled in Figure A.4. Points are created along the x-axis by using the Define Point option.

##### A.4.2.3.1 Define Point

Points can be created using the Define Point option as follows:

1. Pick Geometry → Points → Define Point → At XYZ from the cascading menus.
2. In the pop-up menu, enter the following parameters:
   - X,Y,Z: Defines the coordinates of the point. Enter the distance from the origin as the x-coordinate.

- CID: The identification number of the coordinate system used to define the point.
- Output: Each point is given an identification number as it is created. Points are numbered in the order that they are created.

#### A.4.2.4 Creating Curves

Once points have been specified along the x-axis, the next step is to create curves. A curve will be created for every curve present in Figure A.4. Curves can be created by connecting points, sweeping points, reflecting curves, rotating curves, and defining curves. Each of these operations is described here.

##### A.4.2.4.1 Connect Points

Points along the x-axis are connected together to form straight line curves with the Connect Points option as follows:

1. Pick Geometry → Curves → Connect Points from the cascading menus.
2. Two points can easily be connected by picking them directly from the graphics tile. Alternately, enter the two point identification numbers in the pop-up menu.

#### A.4.2.4.2 Sweep Point

Circular curves can be created by sweeping points through an angle around an axis of rotation. The sweep point option is useful for creating the circular curves forming the outline of the rotor bars. The curves outlining one-half of one rotor bar should be formed by sweeping the points defining the edges of the rotor bar around the rotor bar center. Curves are created with the Sweep Point option as follows:

1. Pick Geometry → Curves → Sweep Point from the cascading menus.
2. In the pop-up menu, enter the following parameters:
   - Iterate: The number of times the point(s) will be swept.
   - Point: The identification number of the point(s) to be swept.
   - From X,Y,Z: Defines the center of rotation around which the point(s) will be swept.
   - To X,Y,Z: Together with From X,Y,Z, defines the axis around which the point(s) will be swept.
   - Angle: Angle, in degrees, through which the point(s) will be swept. Maximum value is 180 degrees.
   - Offset: The initial angle offset before the sweeping begins.
   - CID: The coordinate system in which the From X,Y,Z and To X,Y,Z coordinates are specified, and in which the sweeping occurs.
   - Output: The identification number of the new curve(s). The identification numbers of curves is arbitrary.

### A.4.2.4.3 Reflect Curve

New curves can be created by reflecting existing curves across a plane. For example, the entire rotor bar outline can be created by reflecting curves defining half of the rotor bar outline around the Y = 0 plane. Curves can be reflected around an axis using the Reflect Curve option as follows:

1. Choose Geometry → Curves → Reflect Curve → Coordinate System from the cascading menus.
2. Enter the following parameters in the pop-up menu:
   - Create/Modify: Specifies if new curves are to be created or existing ones modified.
   - Exist ID: The identification number of the curve(s) to be reflected.
   - CID: The identification number of the coordinate system in which the reflection plane is specified.
   - Plane: Defines the plane across which the curve is to be reflected.
   - Output: The identification number of the new curve.

The Reflect Curve option also automatically creates new points at the ends of new curves.

### A.4.2.4.4 Rotate Curve

New curves can also be created by rotating existing curves. For example, curves outlining the other four rotor bars can be created by rotating the curves defining the first rotor bar around the origin using the z-axis as the axis of rotation. The spacing angle between the rotor bars is specified as the angle and the number of rotor bars to be created is specified for iterate. Curves can be rotated using the Rotate Curve option as follows:

1. Pick Geometry → Curves → Rotate Curve from the cascading menus.
2. In the pop-up menu, enter the following parameters:
   - Create/Modify: Specifies if new curves are to be created or existing ones modified.
   - Iterate: The number of times that the curve(s) are to be rotated.
   - Exist ID: The identification number of the curve(s) to be rotated.
   - From X,Y,Z: Defines the point around which the curve(s) is to be rotated.
   - To X,Y,Z: Together with From X,Y,Z, defines the axis of rotation.
   - Angle: Defines the angle through which the curve(s) is to be rotated.
   - Offset: The initial offset angle before the rotation is started.
   - CID: The identification number of the coordinate system in which the rotation is defined.
   - Output: The identification number of the new curve.

The Rotate Curve option automatically creates new points at the ends of new curves.

#### A.4.2.4.5 Define Curve

The define curve option creates a curve by specifying the two endpoints and a center rotation point. The define curve option is useful for creating curves defining the outlines of the nested loops. The point along the x-axis specifying one end of the loop and a point along the periphery of the rotor bar are connected together by a curve. Curves can be created with the Define Curve option as follows:

1. Pick Geometry → Curves → Define Curves → Center 2 Points from the cascading menus.
2. The center rotation point, followed by the two points to be connected, can be picked directly from the graphics tile. Alternately, the point identification numbers can be entered in the pop-up menu.

Curves defining an outline of the model are created using a combination of the connect point, sweep point, reflect curve, rotate curve, and define curve options, or other options available in MSC/XL.

When creating curves, it is important to realize that surfaces, which will be defined next, are created by connecting a maximum of four curves. Therefore, the curves must be laid out such that a surface can be created everywhere in the model by connecting at most four curves.

#### A.4.2.5 Creating Surfaces

After curves have been created, the next step in creating geometry is to specify surfaces. If the curves have been properly defined throughout the radial wedge such that surfaces can be defined by connecting at most four curves, the creation of surfaces is an easy step. Surfaces can be created using the Define Surfaces option as follows:

1. Pick Geometry → Surfaces → Define Surface → By (#) Edges from the cascading menus. Specify whether the surface will be created by connecting 3 or 4 edges (curves).
2. Curves defining the edges of the surface can be picked directly from the graphics tile. Alternately, the identification numbers of the curves can be entered in the pop-up menu.

When picking curves from the graphics tile, curves should be picked consistently in either a clockwise or a counterclockwise direction around the surface boundary. If curves are picked in a random order a deformed surface will result. If a mistake is made, delete the surface using Undo and try again.

When surfaces have been defined for all areas in the radial cross-section, the geometry creation is complete. The finite element mesh can now be created.

### A.4.3 Generating the Finite Elements

After geometry creation, the next step in producing the model is to generate a finite element mesh. A finite element mesh is a collection of connected grid points and elements that subdivide the geometry and represent the problem volume. The ability of a model to represent the actual problem is determined in part by the quality of the finite element mesh. Before discussing how the mesh is actually created in MSC/XL, several key factors to consider in meshing will be discussed.

### A.4.3.1 Finite Element Terminology

The two important components of the finite element mesh, grid points and elements, should first be defined.

#### A.4.3.1.1 Grid Points

Grid points (also called nodes) play a central role in finite element modeling. Grid points are the connection points between elements, and are created along with elements as part of the meshing process. Grid points are not the same as geometric points. Grid points that exist at element corners are called corner nodes. Grid points that lie along the element faces are called midedge nodes.

In the solution process, potential values are directly calculated only at grid points. Four potential degrees of freedom (DOF) (three $\vec{A}$ components and $\Psi$ ) are defined at each grid point. Boundary conditions are applied to potential DOFs at grid points.

#### A.4.3.1.2 Elements

An element is the connection between grid points that embodies the spatial properties of the model. Elements are assigned material properties of permeability, permittivity, and conductivity. Excitations are also applied to elements.

The user must decide to construct either first order or second order elements. In first order elements with only corner nodes, potentials vary linearly along the element edges. Interpolation of field values inside first order elements involves low-order polynomials. Second order elements contain midedge nodes between corner nodes, and potentials vary quadratically along the element edges. Figure A.5 illustrates the concept of grid points, elements, corner nodes, and midedge nodes.

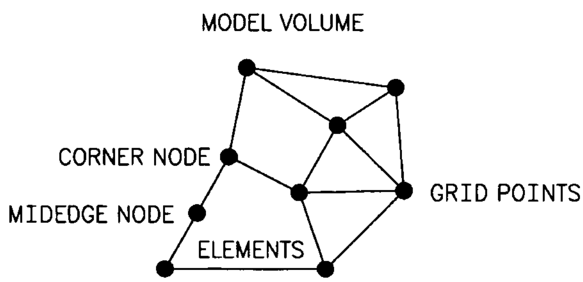


Figure A.5: Finite Elements and Grid Points

### A.4.3.2 Key Factors in Meshing

Key factors to consider in constructing the finite element mesh include:

- Element Choice - the different types of elements available in MSC/XL will be discussed

- Element Connections - several rules for connection elements should be observed
- Density - Element and grid point density determines overall solution accuracy
- Order - The polynomial order (or degree) used for interpolation within each element affects solution accuracy
- Distortion - Elements may be somewhat distorted relative to their nominal geometries to accommodate geometry; but such distortions affect accuracy.

The best mesh results from a tradeoff between model requirements, accuracy, and system requirements (such as disk space available).

#### A.4.3.2.1 Element Choice

Available elements in MSC/XL are grouped into categories based on element dimensionality. Different types within each category have different nominal geometries. The basic element categories are three-dimensional, two-dimensional, axisymmetric, one-dimensional, open-boundary, circuit elements, and scalar elements. Choice of elements depends on model requirements.

Since a three-dimensional BDFM model is to be constructed, three-dimensional elements will be used. These three-dimensional elements will be created by extruding two-dimensional elements. Two and three-dimensional elements available in MSC/XL are shown in Figure A.6.

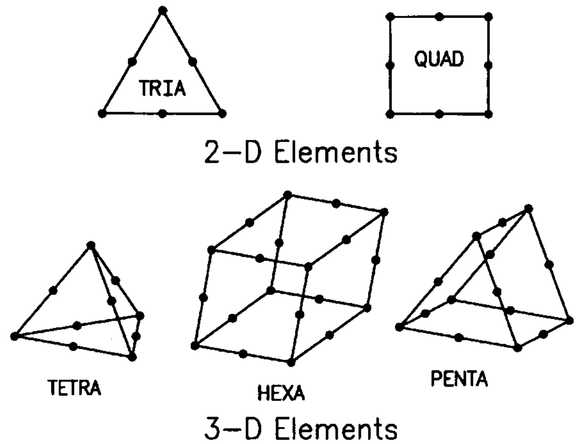


Figure A.6: Available Two and Three-Dimensional Elements

Extruding a TRIA element results in a PENTA element; extruding a QUAD element results in a HEXA element. TETRA elements are formed by methods other than extrusion. Since extrusion is the easiest method for creating a three-dimensional BDFM model, only PENTA and HEXA elements are used in the three-dimensional BDFM finite element model.

Generally, the most cost-efficient results are obtained using QUAD and HEXA elements, with nominal rectangular geometries. These should be used throughout the mesh wherever the geometry allows.

#### A.4.3.2.2 Element Connections

In connecting elements the following rules should be observed:

- Discontinuities in material properties or excitations should coincide with element boundaries. In other words, a transition between a steel rotor and a copper rotor bar must occur at element boundaries, not within an element. If the geometry has been created properly, this should not be a problem.

- Elements must share common grid points along boundaries. In other words, elements must match up along boundaries, not be randomly connected to grid points.

#### A.4.3.2.3 Mesh Density

Element density is a tradeoff between accuracy and solution cost. Solution accuracy is determined by how fast field values change with position (field gradients) and the number of elements per unit length. Mesh density may also be dictated by fine geometric detail.

Users must use engineering judgment and knowledge of field distributions to identify regions of high field gradients that require fine mesh density. In the BDFM model, high field gradients will usually be found in and surrounding the air gap and rotor bars. These areas should be meshed more finely than the other areas, such as the rotor and stator.

#### A.4.3.2.4 Mesh Order

When a second order mesh is used with midedge nodes present, potential functions are interpolated using second-order polynomials instead of first order polynomials. This provides a larger set of functions and in general improves accuracy. However, the number of DOFs is also increased, as well as the solution time and system requirements. Although midedge nodes do provide more accurate answers, the user must access the accuracy per unit cost.

For all the work done in this thesis, first order meshes containing only corner nodes were used because of disk space limitations.

#### A.4.3.2.5 Element Distortion

The finite elements in MSC/XL are intended to be close to the following nominal geometries:

| | |
|---|---|
| TRIA | Equilateral triangle or right triangle |
| QUAD | Square |
| TETRA | Equilateral Tetrahedron |
| PENTA | Equilateral Right Prism |
| HEXA | Cube |

Severe element distortion from these intended shapes can produce inaccurate results. MSC/XL will allow distorted elements to be created, so it is up to the user to monitor this. Severely distorted elements cause user warning messages to appear in the <filename.f06> output file.

The following guidelines are offered concerning element distortion:

- It is recommended that midedge nodes be collinear with the associated corner nodes, and be located at the middle of the edge. Midedge nodes located outside of the center half of the edge cause gross errors.

- High aspect ratios may not be a problem in rectangular elements unless field gradients are too large.
- Keep the taper angles on trapezoidal QUAD elements less than fifteen degrees.

Figure A.7 illustrates these concepts.

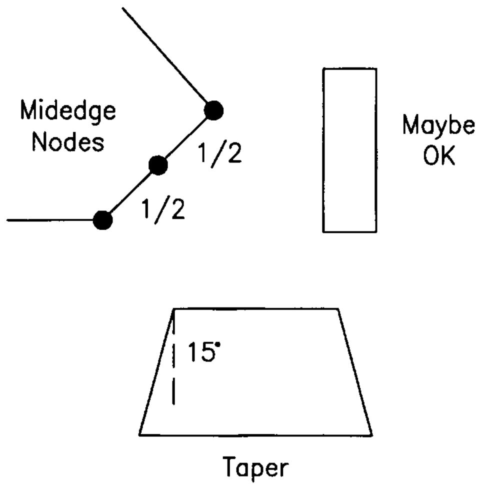


Figure A.7: Distortion of Midedge Nodes, Aspect Ratios, and Taper Angles

### A.4.3.3 Preparation for Meshing - Assigning PIDs

In preparation for meshing the geometry in MSC/XL, it should be determined how property identification numbers, or PIDs, will be assigned to elements. PIDs are assigned to elements as they are created, and later used to assign material properties to elements, and to collect sets of elements together in groups. In creation of the two-dimensional mesh, the important consideration is how objects can be assigned PIDs so they can be collected as groups (since 2-D elements will not be assigned material properties in the final model this consideration is not important now). These groups of elements will later be selectively extruded to form three-dimensional elements representing "objects". Tables A.2 and A.3 help to explain this concept. Table A.2 shows how PIDs can be assigned to groups of two-dimensional elements that form a

| Two-Dimensional groups of elements representing "objects" | PIDs assigned to two-dimensional elements |
|---|---|
| Shaft | 1 |
| Rotor | 2 |
| Rotor Bars | 3 |
| Nest/Rotor/Endring | 4 |
| Rotor/Endring | 5 |
| Air gap | 6 |
| 6 pole stator slot | 7 |
| 2 pole stator slot | 8 |
| Stator | 9 |

Table A.2: Assignment of PIDs to Two-Dimensional Elements

| Three-Dimensional groups of elements representing "objects" | Formed by extruding two-dimensional elements with PIDs | PIDs of three-dimensional elements |
|---|---|---|
| Shaft | 1 | 10 |
| Rotor | 2,4,5 | 11 |
| Rotor Bars | 3 | 12 |
| Air gap | 6 | 13 |
| 6 pole stator slot | 7 | 14 |
| 2 pole stator slot | 8 | 15 |
| Stator | 9 | 16 |
| Nested loops | 3,4 | 17 |
| Endring | 3,4,5 | 18 |
| Air surrounding nested loops | 1,2,5,6,7,8,9 | 19 |
| Air surrounding endring | 1,2,7,8,9 | 19 |

Table A.3: Assignment of PIDs to Three-Dimensional Elements particular "object" in the two-dimensional BDFM cross-sectional model. Table A.3 shows how these two-dimensional groups of elements can be selectively extruded to form three-dimensional elements that define "objects" by choosing certain PIDs of the two-dimensional elements.

#### A.4.3.4 Meshing in MSC/XL

Once a plan for PID assignment has been made, the finite element mesh is ready to be created. A general procedure should be followed when creating the mesh for the BDFM model as follows:

1. Create a two-dimensional mesh for the 45 degree rotor geometry radial cross-section.
2. Create a two-dimensional mesh for the 5 degree stator geometry radial cross-section.
3. Reflect and rotate the 5 degree stator section to obtain a 45 degree stator section.
4. Connect the rotor and stator meshes together by connecting grid points in the air gap.
5. Reflect and rotate the complete 45 degree section to obtain a 180 degree two-dimensional model.
6. Extrude selected elements of the two-dimensional mesh to form a three dimensional mesh.

#### A.4.3.4.1 Parametric Meshing

The technique found to be the easiest for two-dimensional mesh creation of the BDFM problem is parametric meshing of surfaces, (although other techniques are available). In this procedure, the number of elements to be created along the edges of a surfaces are specified. Elements are created within the surfaces according to specifications and grid points are automatically created at element corners. The parametric meshing procedure can be used as follows:

1. Pick FEM → Mesh → Parametric Mesh → Surfaces from the cascading menus.
2. In the pop-up menu, enter the following parameters:
   - NoMidNode/Midnode: Specifies whether the elements will contain midnodes (quadratic elements).
   - Surface: The identification number of the surfaces to be meshed.
   - Type: The type of element to be created. In created the two-dimensional mesh, QUAD should always be selected here, as QUAD elements are more desirable than TRIA elements. If the shape of the surface dictates, TRIA elements will automatically be created by the meshing routine even though QUAD was selected.
   - U,V: Defines the number of elements to be created along the edges of the surface. U and V directions depend on how the surface was created.

- Pattern: The element orientation corresponding to different element types. The default 1 is acceptable.
- PID: The property identification number to be assigned to the elements created.
- Uspace, Vspace: The spacing between the generated grid points for the surface's U and V parametric directions.
- GridIds: The identification numbers of the new grid points.
- Output: The identification numbers of the new elements.

Another useful parametric meshing procedure is parametric meshing of an already created element. Parametric meshing of an element can be accomplished by:

1. Picking FEM → Mesh → Parametric Mesh → 4 grids from the cascading menus.
2. The four grid points surrounding the element to be meshed can be picked directly from the graphics tile. Alternately, the grid point identification numbers can be entered in the pop-up menu.
3. The old element should be deleted after the new elements are created.

#### A.4.3.4.2 Connect Grids

Connecting Grids is useful meshing procedure to use in areas of transition between fine and coarse elements. The grid points that have been created by the parametric meshing procedure can be connected by picking them directly. This technique is used to create the air gap in the BDFM model once the rotor and stator 45 degree radial section have been completed. Grid points along the outer edge of the rotor are connected to grid points along the inner edge of the stator by using the Connect Grids option. The air gap can contain QUAD or TRIA elements, or usually both. Elements are created within the air gap by connecting grids until the entire air gap is closed.

Use the Connect Grids option as follows:

1. Pick FEM → Elements → Connect Grids → Create (Type of Element) from the cascading menus. It is important to choose the correct type of element to be created. MSC/XL will create a TRIA element even if QUAD is chosen for the type of element (and vice versa). This error is not detectable with the mesh checking procedures, but will result in an user fatal error during the solution process.
2. In the pop-up menu, specify the PID for the elements being created. The grid points to be connected can then be picked directly from the graphics tile.

#### A.4.3.4.3 Reflecting and Rotating the Mesh

Portions of the mesh that are symmetrical to an already created portion can be created by reflecting and rotating the existing portion. For example, the full 45 degree section of the stator mesh can be created by reflecting and rotating the 5 degree stator radial wedge. First the 5 degree section is reflected about the $\theta = 5$ degree plane, then the resulting 10 degree section is rotated 4 times about the $\theta = 10$ degree plane. This will result in a 50 degree section. The excess elements can easily be deleted to form the 45 degree section.

Also, once the rotor, stator, and air gap have been entirely meshed to create a 45 degree radial machine section, the full 180 degree model can be created by reflecting and rotating the existing 45 degree section. First the 45 degree section is reflected about the $\theta = 45$ degree plane, then the resulting 90 degree section is rotated once about the $\theta = 90$ degree plane.

Elements can be reflected in MSC/XL as follows:

1. Pick FEM → Element → Reflect Element → Coordinate System from the cascading menus.
2. In the pop-up menu, enter the following parameters:
   - Create/Modify: Specifies if existing elements are to be modified or new ones created.
   - ExistID: Identification numbers of elements to be reflected.
   - CID: The identification number of the coordinate system in which the reflection will occur.
   - Plane: The plane across which the elements will be reflected.
   - Output: The identification numbers of the new elements.

Similarly, elements can be rotated in MSC/XL as follows:

1. Pick FEM → Element → Rotate Element from the cascading menus.
2. In the pop-up menu, enter the following parameters:

- Create/Modify: Specifies if existing elements are to be modified or new ones created.
- Iterate: The number of times that the rotation is to be performed.
- ExistID: Identification numbers of elements to be rotated.
- From X,Y,Z: Define the center of rotation.
- To X,Y,Z: Together with From X,Y,Z, define the axis of rotation.
- Angle: The angle through which the elements are to be rotated.
- Offset: The initial offset angle before the rotation begins.
- CID: The identification number of the coordinate system in which the reflection will occur.
- Output: The identification numbers of the new elements.

#### A.4.3.5 Mesh Checking Procedures

Any mesh produced by MSC/XL will initially contain errors. Some errors are present by default because of the way MSC/XL is designed; other errors are user errors. It is a good idea to use the mesh checking procedures included in MSC/XL often to check the two-dimensional mesh as it is being created. The sooner errors are identified, the easier they are to correct. The mesh should always be checked before it is reflected or rotated. Also, after the two-dimensional mesh is complete, and before the three dimensional mesh is created, it is necessary to check for common errors associated with meshing. Mesh errors can be much more easily identified and corrected in two dimensions than in three dimensions. Unidentified mesh errors will produce fatal errors or warning messages during the solution process. Typical mesh errors are duplicate grid points, unconnected grid points, duplicate elements, free edges or free faces, and element voids. Several examples of these are illustrated in Figure A.8.

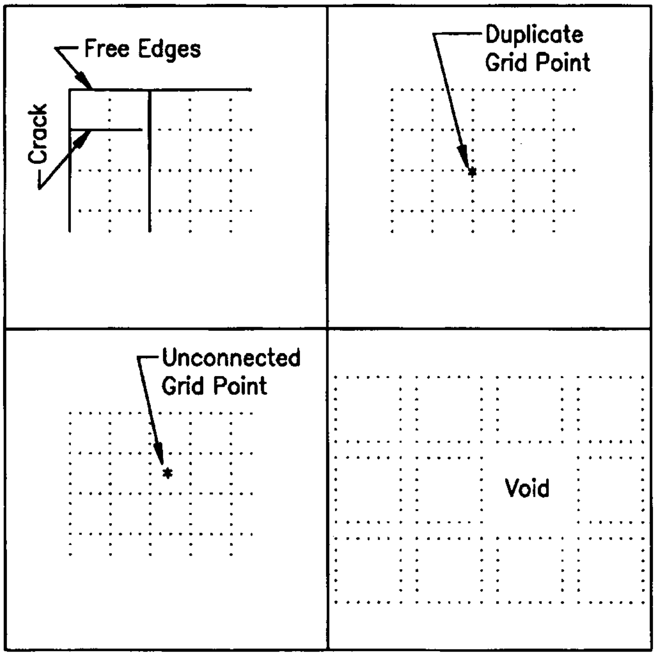


Figure A.8: Typical Mesh Errors

#### A.4.3.5.1 Duplicate Grid Points

Duplicate grid points occur along the shared edges between surfaces (and along the shared faces of solids) when a finite element mesh is created. Duplicate grid points will almost always be present in the model, since they are generated by the meshing process. MSC/XL has an automatic check for duplicate grid points. The duplicate grid points must be deleted by selecting FEM → Check FEM → Duplicate Grids → Find & Equivalence from the cascading menus. Duplicate grids should be eliminated first before checking the mesh for other errors, since duplicate grids can be the be the cause of other mesh errors, free edges and free faces. Duplicate grid points do not usually produce fatal errors.

#### A.4.3.5.2 Unconnected Grid Points

An unconnected grid point usually produces a warning message when MSC/EMAS tries to assemble system matrices. They are sometimes quite difficult to identify visually. Unconnected grid points are easily eliminated in MSC/XL using the typed command:

*Delete Grid/All*

MSC/XL will only delete grids that are not connected to elements.

#### A.4.3.5.3 Duplicate Elements

Duplicate elements can occur when geometric entities are meshed more than once. It's a good idea to check for duplicate elements by selecting FEM → Check FEM → Duplicate Elements → Find & Equivalence from the cascading menus.

#### A.4.3.5.4 Free Edges or Free Faces

A free edge (in a 2-D mesh) is any element edge that is not shared by two elements. Similarly, a free face (in a 3-D mesh) is any element face that is not shared by two elements. Free edges and free faces should only occur at the intended boundary of the model. When they occur in the interior, it usually means that the elements have not been properly connected, resulting in a "crack" or "seam" in the model. Cracks usually represent a modeling error that must be corrected.

Duplicate grid points within the model will always produce free edges and free faces. Duplicate grids should therefore be eliminated before trying to find free edges and faces. Cracks in the interior of the mesh, with causes other than duplicate grids, can then be isolated and identified.

It is important to identify and correct all free edges in the two-dimensional mesh before extruding to create a three dimensional mesh. Free faces are usually the result of extruding a two-dimensional mesh that has unidentified free edges. The possibility of having a three dimensional model with free faces can almost be eliminated if free edges in the two-dimensional model are corrected. Also, free edges are much easier to locate visually.

The easiest way to check for free edges (or free faces) that occur interior to the model boundaries is to have MSC/XL plot free edges (or free faces). This can be done be picking FEM → Check FEM → Free Edges (or Free Faces) → Find. It is helpful to enter the command *Refresh/AxesOnly* in the Blue Bar before finding the free edges (or free faces), because then only the free edges or faces are displayed (not the entire finite element model) when a subsequent Find is chosen. Free edges or free faces can be found over the entire model or between Parts, PIDs or MIDs. The model option is the most helpful in location interior model cracks. With the model option, the only free edges (or free faces) that are displayed for a model without errors are the model boundary. The PID option is a useful technique for checking that the correct PID was assigned to each element. When the PID option is used, all of the boundaries between objects or PIDs should be observed.

If free edges or free faces exist in the interior of the model, those areas must be investigated for missing elements, improperly connected elements, or other discontinuities. Deleting the free edges (or free faces) by picking FEM → Check FEM → Free Edges → Delete from the cascading menus does not fix the cracks, it only removes the free edges or free faces from the display screen.

#### A.4.3.5.5 Element Voids

Element voids, or missing elements, occasionally occur if an element is deleted and are sometimes difficult to detect. Element voids will show up as a local grouping of free edges or faces in the model interior. If a void is suspected, it is easily confirmed by displaying a "shrink plot" of the model. A shrink factor other than zero (default) will shrink the display of an element toward its center by the fraction amount. For example, a shrink factor of 0.2 will cause finite elements to be drawn 80 percent of their actual size. The shrink factor can be changed by selecting Tables → Display → FE Visual → Shrink from the cascading menus. Voids are most easily repaired by editing in the missing element by hand.

#### A.4.3.6 Mesh Organization

When the two-dimensional mesh is completed, it will contain many elements and will regenerate on the screen rather slowly. Rather than having to always remember element identification numbers when manipulating groups of elements and always having to display all the elements on the screen at once, it is helpful to use two mesh organization techniques: groups and parts.

##### A.4.3.6.1 Using Groups

A group collects several elements together by some common criterion. The group is assigned a name, so that the elements can be manipulated by name rather than having to always remember and enter their identification numbers. A useful criterion for creating groups are property identification numbers (PIDs). A PID was assigned to every element as it was created. It is helpful to collect elements in groups according to their PIDs. One group is created for each PID, with a name corresponding to the object that group of elements represents.

Groups can be created in MSC/XL as follows (some common settings used when grouping elements are included):

1. Pick Tools → Group → Define from the cascading menus.
2. In the pop-up menu, enter the following parameters:
   - Name: Assigns a name to the group.
   - Type: Element (Select the type of entity to be grouped).

- IDList: All (The elements to be considered in the grouping selection).
- Criterion: PID = PID # (The criterion to be used to group the elements).
- ViewID: 1 (The identification number for the view from which the items are to be selected).
- PartIdList: 0 (The identification number for the parts from which the items are to be selected).
- Windowmode: View (The type of window to use in the grouping. View is the entire screen view).
- CollectMode: Inside (Whether items to be grouped lie inside or outside the window).
- BoundaryMode: Include (Whether items intersecting the window boundary are to be included or excluded from consideration for group selection).

#### A.4.3.6.2 Using Parts

A part is a method of grouping elements such that they can selectively be displayed or removed from the screen. Use of parts allows sections of the model to be viewed separately on the screen, and is a useful model checking tool.

One part, with identifier 0, is automatically defined in MSC/XL. All entities are placed in part 0 by default unless new parts are created. It is not really necessary to place elements of the two-dimensional mesh into separate parts, however, it is useful for checking purposes to create additional parts before the three dimensional mesh is created. It is helpful to create a part for every "object" present in the three dimensional model (such as the rotor, stator, air gap, rotor bars, etc.). Elements are then placed in selected parts as they are created. This later allows each "object" making up the three dimensional model to be viewed separately from the rest of the model.

New parts are created in MSC/XL simply by picking Part, selecting New and giving the new part an identification number.

Before a group of two-dimensional elements is extruded to form three dimensional elements, the appropriate part number should be selected by choosing its number. All entities such as elements are placed in the currently selected part as they are created.

#### A.4.3.7 Extruding the Two-Dimensional Mesh to Make a Three-Dimensional Mesh

Once a 180 degree two-dimensional finite element mesh of the BDFM cross-section has been completed, a three dimensional mesh of the BDFM model can be created. The three dimensional mesh will be created by extruding sections of the two-dimensional mesh to create each of the three dimensional BDFM model "objects" as previously discussed in the section on assigning PIDs.

The two-dimensional mesh lies in the Z = 0 plane, unless otherwise defined. The origin (0,0,0) will be the center of the machine model, and three dimensional elements will be formed by extrusion in both the positive and negative Z directions.

The three dimensional BDFM finite element mesh should consist of several lengthwise "sections". These sections are the machine body, the nested loop end connections, the endring, and two layers of "air" elements on each end of the machine.

The machine body includes the shaft, rotor, rotor bars, air gap, stator slots, and stator. It is recommended that the machine body be divided in three to four lengthwise sets of elements. The nested loop end connections and the endring are sets of elements that extend beyond the machine body on opposite ends of the model. It is recommended that the nested loop end connections and the endring be divided into two to three lengthwise sets of elements. The two layers of "air" elements on each end of the machine are included to account for leakage effects on the end of the machine rather than causing the fields to be abruptly chopped off at the machine ends. It is recommended that one set of lengthwise "air" elements be included on each machine end.

The detailed 180 degree model of the BDFM lab machine was constructed with two sets of lengthwise elements along the machine body, one set of elements each for the nested loop end connections and the endring, and without "air" elements on each end, to reduce disk space requirements for solving the model.

Two-dimensional elements can be extruded to form three dimensional elements in MSC/XL by using the Extrude Element 2D option as follows:

1. Select the part identification number for the part where the newly created three dimensional elements are to be placed.
2. Choose FEM → Mesh → Extrude Element 2D from the cascading menus.

3. In the pop-up menu, enter the following parameters:
   - NoMidNode/Midnode: Specifies whether the elements will contain midnodes (quadratic elements).
   - Iterate: The number of times that the two-dimensional elements are to be extruded.
   - Element: The identification numbers or the group names of the elements to be extruded.
   - Delta X,Y,Z: The length in the geometry units of the new three dimensional elements.
   - Offset X,Y,Z: The initial offset distance before the elements are extruded.
   - CID: The identification number of the coordinate system in which the extrusion occurs.
   - PID: The property identification number of the new extruded element.
   - GridOut: The identification numbers of the newly created grid points.
   - Output: The identification numbers of the newly created elements.

The machine body for the detailed 180 degree BDFM model was formed by extruding two-dimensional element sets 50 mm in both the positive and negative Z directions, using an iterate value of 1. The nested loop end connections and the endring were formed by extruding two-dimensional element sets A.76 mm with an offset of 50 mm, and -A.76 mm with an offset of -50 mm, respectively, using an iterate value of 1.

Once the entire three dimensional mesh has been completed, it is important to check it using the mesh checking procedure previously discussed. It is also helpful in checking the three dimensional mesh to display each "object" separately by part identification number. This can easily be done if elements making up each "object" have been placed in separate parts. Parts can be posted or unposted from the display screen by choosing View → Contents and choosing the part identification numbers of the parts to be posted or unposted. By selecting one part number at a time, the user can check to see that the mesh was properly created.

Once the three dimensional finite element mesh has been completely created and checked, material properties, excitations, and boundary conditions can be assigned to the model.

### A.4.4 Problem Setup

#### A.4.4.1 Material Properties

Three properties are needed to describe materials in electromagnetic problems. These three electromagnetic material properties are permittivity (or dielectric constant), electrical conductivity, and permeability. Material properties are assigned to groups of elements in MSC/XL by specifying values for relative permittivity, absolute electrical conductivity, and relative permeability. Before the process for setting material properties in MSC/XL is discussed, a brief background of these three properties is given.

### A.4.4.1.1 Permittivity

Permittivity $[\varepsilon]$ is defined to satisfy the equation:

$$\vec{D} = \varepsilon_0 \, [\varepsilon] \, \vec{E} \qquad \text{(Equation A.1)}$$

where $\vec{D}$ is a electric displacement (coulombs/meter$^2$ in MKS) , $\vec{E}$ is the electric field (volts/meter in MKS), and $\varepsilon_0$ is the permittivity of free space (8.854E-12 Farads/meter in MKS). The 3x3 relative dielectric tensor $[\varepsilon]$ (relative to free space, dimensionless) is

$$[\varepsilon] = \begin{bmatrix} \varepsilon_{11} & \varepsilon_{12} & \varepsilon_{13} \\ \varepsilon_{21} & \varepsilon_{22} & \varepsilon_{23} \\ \varepsilon_{31} & \varepsilon_{23} & \varepsilon_{33} \end{bmatrix} \qquad \text{(Equation A.2)}$$

#### A.4.4.1.1.1 Isotropic Permittivity

If $\vec{D}$ and $\vec{E}$ fall in the same direction, then the permittivity is isotropic, and its permittivity matrix $[\varepsilon]$ has zero off-diagonal elements and the same values for the on-diagonal elements. Thus, in the isotropic case only one number is needed to fully describe permittivity (the scalar relative permittivity, $\varepsilon$ ). Most materials have a constant isotropic permittivity because their relationship between $\vec{D}$ and $\vec{E}$ is linear and the same in all directions.

#### A.4.4.1.1.2 Anisotropic (Symmetric) Permittivity

In anisotropic dielectrics, permittivity changes with direction relative to the material, and the polarization is no longer always parallel to the electric field. This property is represented by specifying off-diagonal as well as diagonal terms for the relative dielectric tensor $[\varepsilon]$. The terms can be specified in Cartesian, cylindrical, or spherical coordinate systems.

#### A.4.4.1.1.3 Unsymmetric Permittivity

There exist materials called ferroelectrics in which $\vec{D}$ is a nonlinear function of $\vec{E}$. Although they are not very commonly used, MSC/XL allows unsymmetric permittivities to be specified.

#### A.4.4.1.2 Conductivity

Conductivity $\sigma$ defines how currents are proportional to electric fields (Ohm's law):

$$\vec{J} = [\sigma]\,\vec{E} \qquad \text{(Equation A.3)}$$

where $\vec{J}$ is the current density (amps/meter$^2$ in MKS), $\vec{E}$ is electric field (volts/meter in MKS), and $[\sigma]$ is the conductivity tensor ((Ohm-meter)$^{-1}$ in MKS). The tensor conductivity matrix $[\sigma]$ is:

$$[\sigma] = \begin{bmatrix} \sigma_{11} & \sigma_{12} & \sigma_{13} \\ \sigma_{21} & \sigma_{22} & \sigma_{23} \\ \sigma_{31} & \sigma_{32} & \sigma_{33} \end{bmatrix} \quad \text{(Equation A.4)}$$

Similarly to permittivity, this tensor becomes isotropic if $\vec{J}$ and $\vec{E}$ are in the same direction, in which case its only nonzero terms are diagonal entries of the same conductivity. In anisotropic materials, $\vec{J}$ and $\vec{E}$ are not parallel, so the full conductivity tensor is used.

The conductivity of air, vacuum, or other materials is zero because they conduct no current density $\vec{J}$ unless $\vec{E}$ is high enough to cause arcing. The most common conductor, copper, has $\sigma$ = 5.8E+07 siemens/m.

Steel and iron have a conductivity in the range of about 1.0E+06 to 1.0E+07 siemens/m. If $\vec{B}$ changes with time, then to reduce losses in machines the steel is often laminated. The lamination lowers conductivity in the direction across the laminations and hence lowers the losses.

#### A.4.4.1.3 Permeability

Permeability $\mu$ is defined by the equation:

$$\vec{B} = \mu_0 \, [\mu] \, \vec{H} \quad \text{(Equation A.5)}$$

where $\vec{B}$ is magnetic flux density (webers/m$^2$, or teslas, in MKS), $\vec{H}$ is magnetic field strength (amps/m in MKS), and $\mu_0$ is the permeability of vacuum (12.7E-7 henries/m in MKS). The tensor relative permeability matrix (dimensionless) is:

$$[\mu]=\begin{bmatrix}\mu_{11} & \mu_{12} & \mu_{13}\\ \mu_{21} & \mu_{22} & \mu_{23}\\ \mu_{31} & \mu_{32} & \mu_{33}\end{bmatrix}$$ (Equation A.6)

This tensor becomes isotropic if $\vec{B}$ and $\vec{H}$ are in the same direction, in which case its only nonzero terms are diagonal entries of the same permeability. Linear isotropic magnetic materials are characterized by a single scalar parameter, the relative permeability $\mu$. Linear anisotropic materials are characterized by entering off-diagonal as well as diagonal terms for the relative permeability tensor, $[\mu]$.

Most materials have a relative permeability very close to 1. Ferromagnetic material, such as iron, steel, nickel, and cobalt, have relative permeability much higher than 1. Iron or steel is commonly used in magnetic devices because it has a relative permeability of several thousand and is inexpensive. Ferromagnetic materials, however, have a highly nonlinear $\vec{B}$-$\vec{H}$ curve. MSC/XL allows the user to specify a $\vec{B}$-$\vec{H}$ curve for a material in its nonlinear analysis modules.

#### A.4.4.1.4 Setting Material Properties in MSC/XL

MSC/XL provides default material properties for fifteen materials via the supplied Materials.EMAS file located in the XL_PATH directory. These default material properties all assume linear, isotropic materials. Table A.4 shows the default values for the relative permeability, absolute electrical conductivity, and relative permittivity of the supplied materials:

| Material | Relative Permeability | Absolute Conductivity | Relative Permittivity |
|---|---|---|---|
| Air or Vacuum | 1.0 | 0.0 | 1.0 |
| Aluminum | 1.0 | 3.54E+07 | 1.0 |
| Bakelite | 1.0 | 1.0E-09 | 4.74 |
| Copper | 1.0 | 5.8E+07 | 1.0 |
| Freshwater | 1.0 | 1.0E-03 | 81.0 |
| Gold | 1.0 | 4.1E+07 | 1.0 |
| Laminated Steel | 2000.0 | 0.0 | 1.0 |
| Plexiglass | 1.0 | 0.0 | 3.45-i0.04 |
| Polyethylene | 1.0 | 0.0 | 2.26 + i0.0002 |
| Polystyrene | 1.0 | 0.0 | 2.55 - i0.0005 |
| Porcelain | 1.0 | 1.0E-10 | A.0 |
| Rubber | 1.0 | 0.0 | 2.75 - i0.002 |
| Seawater | 1.0 | 5.0 | 81.0 |
| Silicon | 1.0 | 0.0 | 11.8 |
| Steel | 2000.0 | 5.0E+06 | 1.0 |

Table A.4: Supplied Materials in MSC/XL

The Materials.EMAS file can be edited to include other materials. Different materials can also be defined within MSC/XL by choosing New from the materials pop-up menu. The Edit option provides a pop-up for entering the desired relative permeability, absolute electrical conductivity, and relative permittivity.

Material properties are defined in MSC/XL as follows:

1. Choose FEM → MaterialProperty from the cascading menus. A material property identifier must be entered. The material property identifier is a PID belonging to elements in the model. A material property needs to be created for every PID in the model.

2. After choosing a material property identifier corresponding to one of the model PIDs, choose one of the materials from the FEM → MaterialProperty → Material pop-up, or pick New to specify the permittivity, conductivity, and permeability of a different material other than the defaults. Toggling the FEM → MaterialProperty → Material → Edit option allows values for the permittivity, conductivity, and permeability tensors to be edited by hand.

MSC/XL allows isotropic, anisotropic (symmetric), and unsymmetric materials to be specified. This selection is made by toggling TypeOfMat. The material tensors can also be either real or complex. This type is selected by toggling TypeOfData.

The CID option under FEM → MaterialProperty selects the coordinate system to be used to enter anisotropic or unsymmetric material tensors.

The Thickness and Area fields are unnecessary for a 3-D model (these fields should be left blank). A thickness is entered for 2-D QUAD and TRIA elements, while an area is entered for 1-D LINE elements, but these elements are not present in a 3-D model.

Once a material has been selected for each PID in the model, the material property setup is complete.

#### A.4.4.1.5 Materials for the BDFM Model

Table A.5 shows typical material properties used in the material property setup of a BDFM model. All materials used for the BDFM model are default materials included in MSC/XL.

| 3-D "Objects" | Material | Relative Permeability | Relative Permittivity | Electrical Conductivity (siemens/meter) |
|---|---|---|---|---|
| Shaft | Air | 1 | 1 | 0 |
| Rotor | Lam. Steel | 2000 | 1 | 0 |
| Rotor Bars | Copper | 1 | 1 | 5.8E+07 |
| Air gap | Air | 1 | 1 | 0 |
| 6 pole stator slot | Air | 1 | 1 | 0 |
| 2 pole stator slot | Air | 1 | 1 | 0 |
| Stator | Lam. Steel | 2000 | 1 | 0 |
| Nested loops | Copper | 1 | 1 | 5.8E+07 |
| Endring | Copper | 1 | 1 | 5.8E+07 |
| Air surrounding end connections | Air | 1 | 1 | 0 |

Table A.5: Materials Used in the Setup of the BDFM Model

Note the stator windings are assigned the material property of air even though the windings actually consist of copper. This is because a current density excitation will later be applied to the stator winding elements. Since the exact current density will be specified in the windings, a conductivity is not necessary for these elements. To specify a conductivity for elements to which an excitation is applied will cause additional eddy currents to be induced in these elements. Therefore, the stator windings should be specified in the material setup as having material property air, which has identical permittivity and permeability to copper, but zero conductivity.

#### A.4.4.2 Excitations

Excitations are included in a finite element model to represent either applied electrical currents or permanent magnets. For the BDFM model, current excitations are included to represent the three phase stator windings. Methods for setting up current excitations in the AC module will be discussed.

#### A.4.4.2.1 Available Excitations

The following excitations are available in the AC analysis module: current density, current region, edge $\vec{H}$ field, edge $\vec{J}$ field, line current, permanent magnetization, point current, point current axisymmetric, point current scalar, surface $\vec{H}$ field, surface $\vec{J}$ field, and volume current source.

#### A.4.4.2.2 Applying Excitations to the BDFM Model

The appropriate excitation to use to model the three phase stator windings is the current density excitation. The current density excitation can be applied to the elements representing the three phase stator windings by first grouping the elements to be excited, and then applying the current density excitation.

| | | | | | | | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| A | C’ | C’ | B | B | A’ | A’ | C | C | B’ | B’ | A | A | C’ | C’ | B | B | A’ | |
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | |
| X | . | . | X | X | . | . | X | X | . | . | X | X | . | . | X | X | . | |

| | | | | | | | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | A’ | C | C | B’ | B’ | A | A | C’ | C’ | B | B | A’ | A’ | C | C | B’ | B’ | A |
| | 19 | 20 | 21 | 22 | 23 | 24 | 25 | 26 | 27 | 28 | 29 | 30 | 31 | 32 | 33 | 34 | 35 | 36 |
| | . | X | X | . | . | X | X | . | . | X | X | . | . | X | X | . | . | X |

where, $Phase_a = 0^0$, $Phase_b = 240^0$, $Phase_c = 120^0$

Table A.6: Six Pole Stator Winding Layout

| | | | | | | | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| A | A | A | B' | B' | B' | B' | B' | B' | C | C | C | C | C | C | A' | A' | A' | |
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | |
| X | X | X | . | . | . | . | . | . | X | X | X | X | X | X | . | . | . | |
| | A' | A' | A' | B | B | B | B | B | B | C' | C' | C' | C' | C' | C' | A | A | A |
| | 19 | 20 | 21 | 22 | 23 | 24 | 25 | 26 | 27 | 28 | 29 | 30 | 31 | 32 | 33 | 34 | 35 | 36 |
| | . | . | . | X | X | X | X | X | X | . | . | . | . | . | . | X | X | X |

where, $Phase_a = 0^0$, $Phase_b = 120^0$, $Phase_c = 240^0$

Table A.7: Two Pole Stator Winding Layout

Tables A.6 and A.7 show how the stator source currents for the 6-pole and 2-pole windings were specified, respectively. Single layer windings are assumed. For both the 6-pole and 2-pole windings, 100 amp-turns peak per slot is assumed.

First the elements to be excited should be grouped together in logical sets. This is done by using the Tools menu to define a group. One procedure is to define each individual slot of the 6 pole winding as a group, and each set of 2 pole slots with the same phase angle as a group. This can be done by the following steps:

1. Choose Default View 1
2. Display only the elements associated with the 6 pole or 2 pole winding (one winding at a time, the one being grouped)
3. Pick Tools → Group → Define from the cascading menus
4. In the pop-up menu, define a name for the group and choose polygon window
5. Pick Do It and use the polygon window to select the elements to be part of that group.

Once the elements that make up the three phase stator windings have been logically defined in groups, the current density excitation can be applied. A current density excitation will be created for every stator slot group that was defined as follows:

1. Pick FEM → Excitation from the cascading menus, and give the excitation an identification number.
2. Choose FEM → Excitation → Edit and toggle to Current Density.
3. In the pop-up menu, enter the following parameters:
    - ElementIds: The name of the group to which the excitation is to be applied
    - Current Density: The desired current density (J) in amperes/$m^2$. The current density can be calculated by dividing the total peak current flowing the slot (100 amperes) by the total cross sectional area of the slot being excited.
    - CID: The chosen coordinate system. It is easy to use the default Cartesian coordinate system to define current density excitations.
    - Dir1,2,3: Specify the direction of the excitation (Dir1, Dir2, Dir3 correspond to X,Y,Z in Cartesian coordinates). If the length of the machine has been defined in the Z direction, the current density will have a positive or negative Z direction only, as in Tables A.6 and A.7.

It is important to visually check each excitation as it is being applied. Each current density excitation is represented visually by an arrow which indicates the direction of the excitation. The current density excitation label includes the excitation

ID, followed by a comma, followed by the current density magnitude. Displaying a 3-D view of only the stator windings while the excitations are being applied makes it easy to see where an excitation has been applied.

In an AC analysis, a phase angle must be assigned for each excitation. A multi-phase analysis can be achieved by defining multiple excitations, each with differing phase angles. A phase angle is assigned to each excitation group as shown in Tables A.6 and A.7 for the BDFM. To enter a phase angle for an AC excitation choose FEM → Excitation → Phase from the cascading menus and enter a phase angle (degrees) for the chosen Excitation ID.

#### A.4.4.3 Boundary Conditions

Electric and magnetic fields need to be constrained appropriately along the outer boundaries of the finite element model. These boundary conditions, or constraints on the electric and magnetic fields, are on the three components of magnetic vector potential $\vec{A}$ and electric scalar potential, $\Psi$ (also referred to as degrees of freedom, or DOFs). A constraint fixes a particular DOF to a specified value throughout all of the solution process. Constrained DOFs are removed from the problem before it is solved, and constrained values are substituted wherever they appear in the equations.

There are two main types of constraints in MSC/EMAS. In a single-point constraint, or SPC, a single DOF is assigned a fixed value. In a multipoint constraint, or

MPC, a specified DOF is equal to some linear combination of any number of other DOFs. Such constraints have a number of uses, including periodic boundary conditions.

#### A.4.4.3.1 Fixed Boundary Conditions

For the purposes of applying boundary conditions at surfaces, DOFs are divided into three classes:

- $\vec{A}$ DOFs tangent to the boundary
- $\vec{A}$ DOFs normal to the boundary
- $\Psi$ DOFs

When one of the whole classes of DOFs listed above are constrained to zero along entire surfaces, the following "gross" magnetic field boundary conditions are produced:

| Constraint | "Gross" Field Condition |
|---|---|
| $\vec{A}_{\tan gent} = 0$ | $\vec{B}_{normal} = 0$ |
| $\vec{A}_{normal} = 0$ | $\vec{B}_{\tan gent} = 0$ |
| $\Psi = 0$ | $\vec{E}_{\tan gent} = 0$ |

The effect of the fixed boundary conditions listed above is illustrated in Figure A.9.

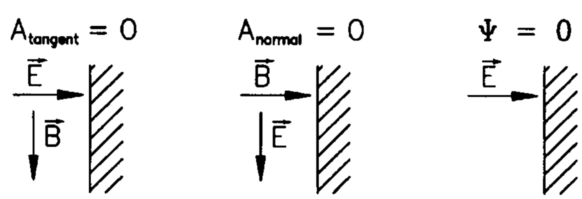


Figure A.9: Fixed Boundary Conditions

In two-dimensional problems, a line of constant normal magnetic vector potential $\vec{A}$ is called a magnetic flux line. For most electric machines, $\vec{B}$ flows in the plane of the steel laminations and the flux is assumed to be confined to the steel outer boundary. Flux lines along such a boundary (not crossing it) are enforced by setting $\vec{A} = 0$ along the boundary.

The existence of three vector components of $\vec{A}$ in three-dimensional electromagnetic problems makes boundary conditions more complicated than in two-dimensional problems. The three components of $\vec{A}$ may be $A_x$, $A_y$, and $A_z$, or in cylindrical devices such as motors, $\vec{A}$ is conveniently expressed in the three cylindrical components $A_r$, $A_\theta$, $A_z$.

Although plots of contours of constant $\vec{A}$ in two-dimensional problems are flux line plots, in three dimensions flux line plots are not rigorously defined. Contours of constant total magnitude of $\vec{A}$ are sometimes analogous to planar flux plots.

Boundary conditions of $\vec{A}$ in three dimensions are governed by the curl of $\vec{A}$, for example, in cylindrical coordinates:

$$\vec{B} = \left(\frac{\partial A_z}{\partial \theta} - \frac{\partial A_\theta}{\partial z}\right)\vec{u}_r + \left(\frac{\partial A_r}{\partial z} - \frac{\partial A_z}{\partial r}\right)\vec{u}_\theta + \left(\frac{\partial A_\theta}{\partial r} - \frac{\partial A_r}{\partial \theta}\right)\vec{u}_z \qquad \text{(Equation A.7)}$$

Thus, it can be seen that one way of enforcing $B_z = 0$ is to set $A_r = A_\theta = 0$. In other words, $\vec{B}$ can be prevented from crossing a boundary surface by setting the tangential components of $\vec{A}$ to zero.

### A.4.4.3.2 Periodic Boundary Conditions

Many electric machines have identical poles. The number of grid points contained in a finite element model can be greatly reduced if the mesh only needs to contain one pole. In any electric machine having identical poles, each pole boundary has periodic boundary conditions. For three-dimensional machine models, periodic boundary conditions are expressed in cylindrical (r,θ,z) coordinates by Equation 4.5 - 4.8. This periodic boundary condition is implemented with a multipoint constraint (MPC) and is called an alternating boundary condition. If the geometry requires modeling two poles, then the $\vec{A}$'s on the boundary are set equal instead of opposite, which is referred to as a repeating boundary condition. Generally, an odd number of poles requires alternating periodic boundaries and an even number requires repeating periodic boundaries. Figure A.10 illustrates the alternating periodic boundary conditions.

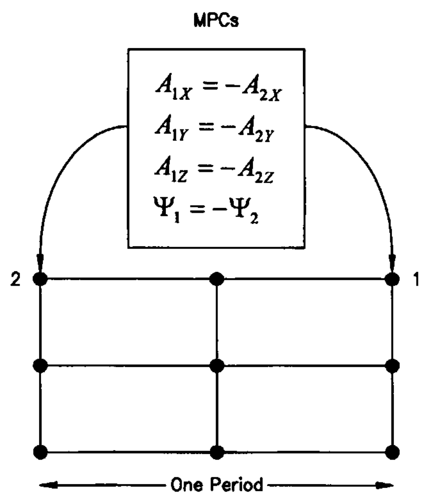


Figure A.10: Alternating Periodic Boundary Conditions

Applying alternating periodic boundary conditions to a 180 degree model of a 6/2 pole BDFM provides results consistent will a 360 degree model simulation, as discussed in Chapter 4.

### A.4.4.3.3 Applying Boundary conditions to the BDFM Model

#### A.4.4.3.3.1 Outer Boundaries

For the BDFM model, we can constrain $\vec{B}$ to remain within the machine boundaries by constraining the tangential components of $\vec{A}$ to be zero along the outer circumference of the machine, and also on the two machine ends. If the machine has been drawn with a cylindrical coordinate system, and the z coordinate lies along the axis of the machine, the tangential components of $\vec{A}$ along the radial boundary are $A_\theta$, $A_z$ and on the two machine ends the tangential components of $\vec{A}$ are $A_r$, $A_\theta$.

In MSC/XL, setting the tangential components of $\vec{A}$ to zero (or assigning any value to a DOF) is called a fixed potential boundary condition (also referred to as a single-point constraint, or SPC). The easiest way to set the tangential components of $\vec{A}$ = 0 in MSC/XL is to use the By CSPlane option as follows:

1. Choose FEM → BoundaryCondition and choose an arbitrary boundary condition identification number
2. Choose FEM → BoundaryCondition → Edit (Fixed Potential) → By CSPlane

3. In the pop-up menu, fix the desired potential in the chosen coordinate system (CID), and enter the value for the chosen potential. Together the CID and Plane pop-up entries define the constant plane to which the Fixed potential(s) will be applied. CID represents the coordinate system in which the plane is constant. The Plane options will be x,y,z for a Cartesian system, r,θ,z for a cylindrical system, and r,θ, ϕ for a spherical system.

#### A.4.4.3.3.2 Periodic Boundaries for the 180 Degree Model

Alternating periodic boundary conditions are set up along the symmetry plane of the 180 degree model. Periodic boundary conditions are a type of dependent potential or multi-point constraint (MPC). MPCs relate a potential of one grid point to a potential of another through a constant coefficient.

The easiest way to apply alternating periodic boundary conditions to the BDFM 180 degree model is to use the Periodic option as follows:

1. Choose FEM → BoundaryCondition and choose an arbitrary boundary condition identification number
2. Choose FEM → BoundaryCondition → Edit/DependentPotential → Periodic
3. In the pop-up menu, choose Alternating to force all of the grid point potentials on the two planes to be equal in value but opposite in sign. Together the CID1 and Plane1 pop-up entries define the dependent constant plane of grid points. Similarly, CID2 and Plane2 defines the independent

constant plane of grid points. CID represents the coordinate system in which the plane is constant.

For the BDFM half model, choose the defined cylindrical coordinate system for both CID1 and CID2, the choose $\theta = 0$ for Plane1 and $\theta = 180$ for Plane2. This will relate all potentials ($A_r$, $A_\theta$, $A_z$, $\psi$) of the $\theta = 0$ plane to all potentials of the $\theta = 180$ plane.

It is important to note that a single grid point cannot simultaneously be constrained by a SPC and a MPC. To do so causes a fatal error in MSC/EMAS. When SPCs are applied using the ByCSPlane option, and MPCs are applied using the periodic option, SPCs and MPCs will both be present along all four edges of the symmetry plane. The SPCs must be removed from these edges by choosing FEM → BoundaryCondition/(ID of the SPC) → Delete and picking the SPCs along the edges from the graphics tile.

#### A.4.4.4 AC Analysis Preparation

Once all material properties, excitations, and boundary conditions have been applied to the finite element model, it can be prepared for analysis. This is done by selecting Analysis → Edit from the cascading menus and entering the appropriate values in the pop-up menu.

#### A.4.4.4.1 Control Section

Select the excitations and boundary conditions to be included in the analysis by choosing their identification numbers. A separate analysis is required to vary material properties, excitations, and/or boundary conditions. Enter the desired solution frequency for the AC analysis, which is the rotor reference frame frequency.

#### A.4.4.4.2 Unit Section

Choose the units for the geometry along with a factor to divide those units by (if necessary). For example: Meters/10 is equivalent to decimeters. Select the units for time.

#### A.4.4.4.3 Degrees of Freedom

A degree of freedom can be removed for every grid point in the model by toggling the component to Inactive. For the 3-D BDFM model, all degrees of freedom are necessary and should be left active.

## A.4.5 Solving the Problem

### A.4.5.1 Invoking MSC/EMAS

Once the Analysis → Edit pop-up has been completed, the analysis to ready to be run. Choosing Analysis → Write from the cascading menus creates the <filename.dat>

file (the input file for MSC/EMAS). Choosing Analysis → Run then starts the analysis. The Write and Run steps are usually only done separately if the <filename.dat> file needs to be edited by hand. In most cases, it is easier to combine the two steps by choosing Analysis → Write&Run and entering the following in the pop-up menu:

1. Enter a new file name for the output files, or choose an existing file. If chosen, an existing file will be overwritten.
2. Choose whether results should be printed to the .f06 file (Printed Results). Printing results in the .f06 file will increase the amount of disk space required for the output files. Setting Printed Results to Off does not affect the ability to display results graphically (the .xdb file is not affected), so if having enough disk space is a consideration, Printed Results should be Off.
3. Select the output method for the field results in the finite elements. The field results ($\vec{B}$ and $\vec{E}$) will be recovered from the calculated vector potentials at a different number of locations depending on which output method is chosen. The output methods are defined as follows:
   - Center: The field results will be defined at the element centroids only.
   - CenterCorner: The field results will be defined at the element centroid and at each of the corner grid points.
   - CenterCornerMidside: The field results will be defined at the element centroid, and at each of the corner grid points (and midside grid points for any quadratic elements).

The selection of either of the last two options will result in the creation of additional sets of ElementResults in the ResultsTable known as element nodal results or GPFields.

The MSC/EMAS User's Manual describes the use of GPFields in post-processing calculations as being more accurate. However, for the BDFM models, the selection of CenterCorner and the use of GPFields caused problems in the calculation of total rotor bar currents and was found to produce inaccurate results. Therefore, the selection of Center as the output method for field quantities is recommended.

4. Toggle Restart to On to do a restart of a previous analysis. Select an existing .MASTER file for the restart.

The above procedure will automatically invoke MSC/EMAS and start the solution process.

#### A.4.5.2 System Requirements

System requirements are important to keep in mind while constructing the finite element mesh and before the finite element model is run. The work presented in this thesis was done on a Hewlett Packard 715/50 workstation with 48 MB of RAM and approximately 2.5 GB of hard disk space.

The system requirement of most concern for the models presented in this thesis was disk space, since MSC/EMAS creates very large output files and scratch files during the solution process. The disk space requirements of MSC/EMAS vary depending on the number of grid points in the model, the number of degrees of freedom, the chosen solution sequence, output requests, and other factors.

A graph estimating the amount of disk space required vs. the number of grid points contained in a finite element model is included in the MSC/EMAS Installation Procedures for HP 9000/7000 (HP-UX) Guide [17]. This graph is reproduced for convenience in Figure A.11.

The data plotted in Figure A.11 represents two sets of problems. The lower line represents typical two-dimensional engineering problems (the model is a flat plate). The upper line represents a "highly connected" model (the model is a cubical solid). The data was generated for a static analysis using MSC/NASTRAN solution Sequence 101 (this would be similar to running a static analysis in MSC/EMAS) [17].

The MSC/EMAS AC BDFM model simulations presented in this thesis were found to approximately follow the cellular cube line of Figure A.11. The largest BDFM model that ran to completion with the existing system configuration was the detailed 180 degree BDFM model which contained 9510 grid points.

#### A.4.5.3 Solution Time

The solution time for the BDFM models presented in this thesis was not a major consideration. The time that MSC/EMAS takes to generate a solution varies depending on the number of grid points in the model, the number of degrees of freedom, the chosen solution sequence, output requests, and other factors.

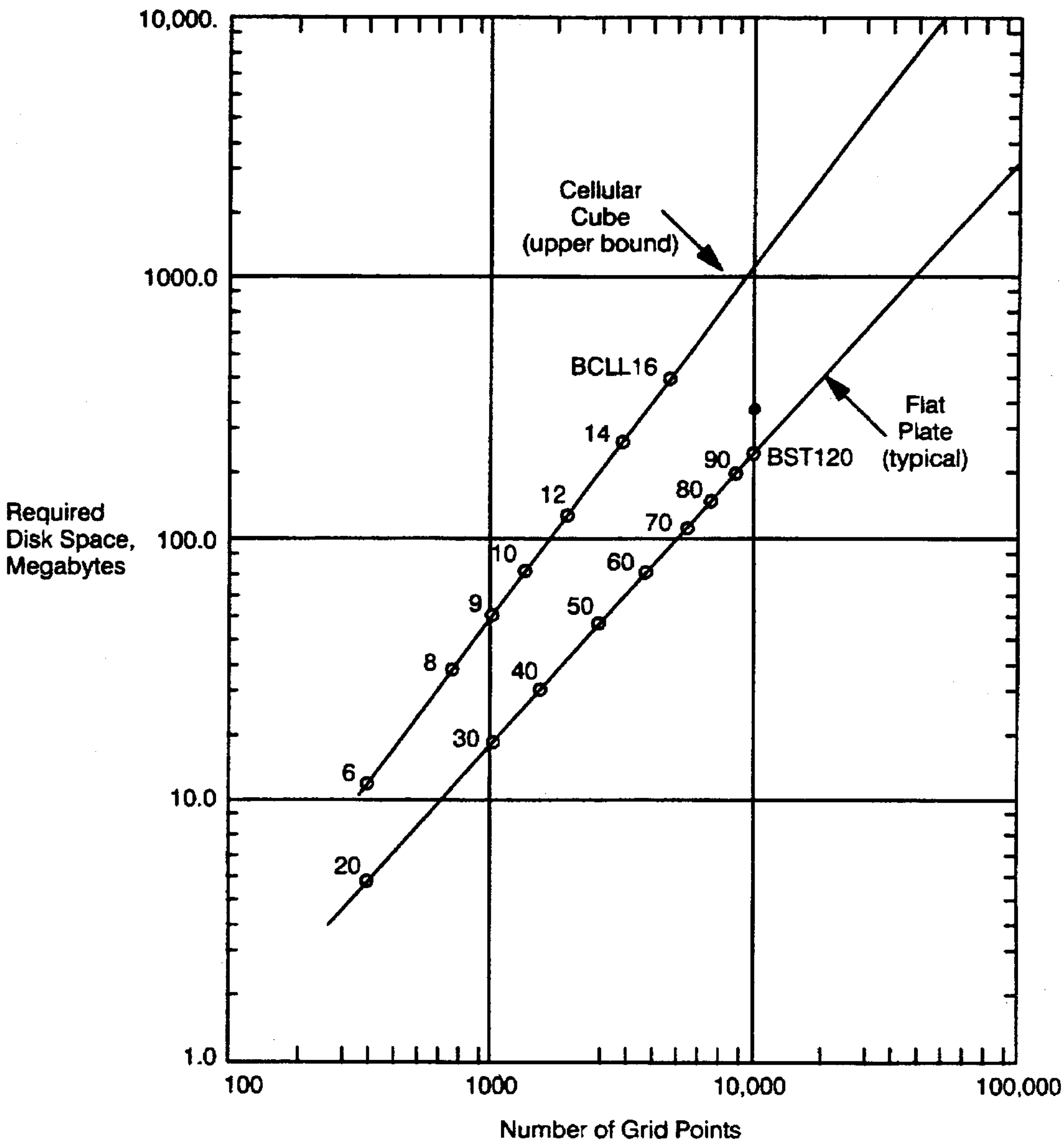


Figure A.11: Disk space requirements for MSC/EMAS based on the cellular cube and flat plate for a MSC/NASTRAN solution 101 run [17]

The solution time for an AC analysis of the detailed 180 degree BDFM model, which contained 9510 grid points, was approximately 5 hours. The solution time for an AC analysis of the coarse 360 degree BDFM model, which contained 8325 grid points, was slightly faster at approximately 4.5 hours. The coarse 180 degree BDFM model, which contained 4280 grid points, ran surprisingly quickly with a solution time of approximately 30 minutes. The solution time required therefore seems to increase exponentially as the number of model grid points increases.

### A.4.6 Results and Validation

The first thing to do when the MSC/EMAS run finishes is to read the <filename.f06> file to check for error messages. If for some reason the <filename.f06> file contains error or warning messages, these should be investigated. Chapter 6 of the MSC/EMAS User's Manual [10] contains error message numbers and explanations.

If no error messages are found in the <filename.f06> file, the user can then proceed to the results processing functions in MSC/XL to view the results graphically.

#### A.4.6.1 Accessing MSC/EMAS Results

The first step in results processing is to establish the connection between a Results Table and a file containing results data (filename.xdb). The filename.xdb file contains the model and the results from the MSC/EMAS analysis. The filename.xdb file should be read into the filename.db file from which the results were generated. This can be done as follows:

1. Make sure that the current filename.db file loaded in MSC/XL is the database file from which the results to be processed were generated.

2. Choose Field Results → ResultsTable from the cascading menus.
3. Select the Type of results to be processed. Analysis Results refers to results from an MSC/EMAS analysis. ImportResults refers to results quantities in an external file format.
4. Choose the filename.xdb File to be accessed.

The results contained in the filename.xdb file are now available for processing in MSC/XL.

#### A.4.6.2 Producing Contour Plots

A "contour plot" is a plot in which MSC/EMAS result quantities are displayed as colored lines or bands. These contours are lines or colored regions representing constant values. Contours can be displayed as colored lines or as colors which fill an element. For the BDFM analysis problem, line contour plots of magnetic vector potential $\vec{A}$ show the distribution of magnetic flux lines within the machine.

To make a contour plot of magnetic vector potential like the ones presented in Chapter 4, use the following procedure:

1. Choose Field Results → Contours from the cascading menus.
2. Select a Style of contouring from one of the three choices. LineContour was chosen for the contour plots presented in Chapter 4.
3. Choose the AverageMethod that MSC/XL will use to average results. Default is acceptable.

4. Use the QuickEditRT option to select the following:
   - Select the Results quantity for the contour plot. To obtain a contour plot of magnetic vector potential, for Grid Results choose Magnetic Vector Potential and for Element Results choose GridResults. Only element results are used for contour plots, so if grid results are to be plotted, the Element Results must be set to GridResults.
   - Select the desired components (VectorResult) of the result. Full Vector was chosen for the contour plots presented in Chapter 4.
   - Select the TypeOfData. Magnitude was selected for the contour plots presented in Chapter 4.
5. Display the contours in the chosen view by picking Plot in View.

#### A.4.6.3 Producing Arrow Plots

Arrow plots are displays of vector fields. An arrow plot displays arrows at the grid points or at the element centers. The ColorRangeTable values show the magnitude of the field quantity and the arrows show the field direction.

For the BDFM analysis problem, arrow plots of magnetic flux density $\vec{B}$ show the magnitude and direction of the $\vec{B}$ field in tesla. This allows the user to examine if the flux density values are in appropriate ranges, and if not, where the problem areas exist.

To make an arrow plot of magnetic flux density like the ones presented in Chapter 4, use the following procedure:

1. Choose Field Results → Arrows from the cascading menus.
2. Vary the arrow size with the vector magnitude by toggling AutoSize to on or produce plots with all arrows of the same size with AutoSize = Off.
3. To make all arrows one color turn AutoColor Off.
4. Choose the AverageMethod for MSC/XL to use to average the results. Default is acceptable to use.
5. Since magnetic flux density is an element result, toggle Grid Arrow to Invisible and Elt Arrows to Visible.
6. Use the QuickEditRT option to select the following:
   - Select the Results quantity for which an arrow plot is to be obtained. To obtain an arrow plot of magnetic flux density, choose Magnetic Flux Density for Element Result.
   - Select the desired components (VectorResult) of the result. Full Vector was chosen for the arrow plots presented in Chapter 4.
   - Select the TypeOfData. Do not select TypeofData = Magnitude or Phase, as these results do not have physical meaning while using arrow plots.
7. Pick Plot in View to graphically display the arrows.

### A.4.6.3 Results Plots on Cut Surfaces

When examining field distributions, it helpful to view plots of various field quantities along two-dimensional surfaces within the three-dimensional model. Although it is possible in MSC/XL to view three-dimensional plots of field quantities, often two-dimensional plots at different positions within the model are easier to view and to interpret. For the BDFM model, cross-sectional contour plots of vector potential and arrow plots of magnetic flux density are helpful to view. Several of these plots were presented in Chapter 4.

To make a plot of a field quantity along a two-dimensional surface, first a plane called a Cutsurface must be defined. A Cutsurface can be defined in MSC/XL as follows:

1. Choose Tools → CutSurface → Edit from the cascading menus.
2. In the pop-up menu, specify values for the following parameters:
   - CID: The identification number of the coordinate system in which the cutsurface is to be defined.
   - From X,Y,Z: Defines the position of the cutsurface.
   - To X,Y,Z: Together with From X,Y,Z, defines the positive direction of the cutsurface.
3. Choose Tools → CutSurface → Intersect from the cascading menus.
4. In the pop-up menu, specify the element identification numbers or group names with which the cutsurface is to be intersected.

Once a Cutsurface is defined and intersected with the finite element model, MSC/XL will automatically extract the results defined in the results table from the elements the Cutsurface passes through.

To obtain a contour plot or an arrow plot on the Cutsurface only, use the following procedure:

1. Choose View 1 for displaying Cutsurface plots.
2. Type unpost part/all in the command line. This will remove all of the parts from the display screen, so that only the outline of the Cutsurface will be displayed.
3. Type refresh/linecontour or refresh arrow in the command line to produce the results plot on the Cutsurface.

#### A.4.6.5 XY Plotting Along CutPaths

XY plots of field quantities along a path through the finite element model can be created in MSC/XL. The plots of conduction current density across the BDFM rotor bars presented in Chapter 4 were examples of such plots. To make an XY plot of a field quantity along a path through the finite element model, first a path called a Cutpath must be defined. A Cutpath can be defined in MSC/XL as follows:

1. Create a geometric curve within the finite element model defining the path over which the field quantity is to be plotted.
2. Choose Tools → CutPath → Define CurvebyCurve from the cascading menus.

3. In the pop-up menu, enter the identification number of the geometric curve to be specified as a Cutpath.
4. Choose Tools → CutPath → Intersect from the cascading menus.
5. In the pop-up menu, specify the element identification numbers or group names with which the Cutpath is to be intersected.

Once a Cutpath is defined and intersected with the finite element model, MSC/XL can create XY plots of field quantities along the Cutpath.

To obtain a XY plot of a field quantity along the Cutpath, use the following procedure:

1. Pick XY Plotting → Graph → Type Results from the cascading menus.
2. In the fourth cascading menu, choose the following parameters:
   - X Data: Path
   - Y Data: Instantaneous Results
   - CutPathID: The identification number of the appropriate Cutpath
   - NumOfPoints: The number of points to be selected by the program along the Cutpath for plotting purposes
3. Use the QuickEditRT option in the fourth cascading menu to select the appropriate field quantity to be plotted along the Cutpath.
4. Select XY Plotting → Load Data from the cascading menus. MSC/XL loads the appropriate field data along the Cutpath.
5. Pick XY Plotting → Plot in View <ID> from the cascading menus to plot the graph.

#### A.4.6.6 Calculations

Several calculations are automatically defined in MSC/XL for post-processing of AC results. The calculation that was used to calculate the total induced current in each of the BDFM rotor bars will be presented here. For a complete list of calculations available in MSC/XL, refer to the MSC/EMAS User Interface Guide for AC Analysis [16].

The total current in the BDFM rotor bars was calculated using the Current from $\vec{J}_c$ calculation included in MSC/XL. This calculation is defined as follows:

$$I = \int(\vec{J}_c \cdot ds) \qquad \text{(Equation A.8)}$$

where $\vec{J}_c$ is the conduction current density and ds is the integration surface.

Total currents are calculated using the Current from $\vec{J}_c$ calculation as follows:

1. Define a Cutsurface and intersect it with the elements over which the integration will occur.
   - Define the Cutsurface by choosing Tools → CutSurface → Edit from the cascading menus. Enter values in the pop-up menu that define the Cutsurface location and direction.
   - Intersect the Cutsurface with the elements over which the integration will occur by choosing Tools → Cutsurface → Intersect from the cascading menus. Enter the identification numbers or group name of the elements to be intersected with the Cutsurface.
2. Choose Calculate and choose CurrentFrom$J_c$ from the list of options.

3. Choose the AverageMethod for MSC/XL to use to average the results. Default is an acceptable choice.
4. Choose Calculate → Calculate from the cascading menus.
5. In the pop-up menu, choose CutSurface as the Surface Type, and choose the appropriate Cutsurface identification number.
6. The calculation results will appear in MSC/XL's history tile.

#### A.4.6.7 Generating Hardcopy Files

To set up the specification for hardcopy plots it is necessary to edit the Hardcopy Table by choosing Table → Hardcopy from the cascading menus and entering appropriate specifications.

Hardcopy files of the graphics tile only can be generated by typing the refresh/plot command in the command line. This command creates a hardcopy of the current display. One of the following options can also be added on the refresh/plot command to obtain various different hardcopy plots:

| | | |
|---|---|---|
| refresh/plot | /wireframe | (creates a wireframe model hardcopy plot) |
| | /hiddenline | (creates a hiddenline model hardcopy plot) |
| | /linecontour | (creates a linecontour hardcopy plot) |
| | /arrow | (creates an arrow hardcopy plot) |

To generate a hardcopy of the entire MSC/XL screen, type replay on, create the desired picture, then type replay off. All refreshes after the replay on command is issued are written to the plot file. Refer to Chapter 5 of the MSC/XL User's Manual [15] for more information about generating hardcopy files.