Ansys Multiphysics (v. 12) tutorial for electrostatic finite element analysis on spur gear teeth

ANDREAS NIKOLAKAKIS

Athens 2012
Contents

1. INTRODUCTION .................................................................................................................. 5

2. ANSYS MULTIPHYSICS - ELECTROSTATIC ANALYSIS ...................................................... 6
   2.1. Finite Element Modeling - Stages .................................................................................. 6
   2.2. Importing the geometry of the specimen .................................................................. 7
   2.3. Defining the type of elements and properties of the material ................................ 7
   2.4. Creating the finite element models - Meshing. ....................................................... 9
   2.5. Applying loads and solving the problem. ................................................................. 11
   2.6. Postprocessing the results ...................................................................................... 13
   2.7. Conclusions ............................................................................................................ 15

REFERENCES .......................................................................................................................... 17
1. INTRODUCTION

The finite element method (FEM) is the dominant discretization technique in structural mechanics. The basic concept in the physical interpretation of the FEM is the subdivision of the mathematical model into disjoint (non-overlapping) components of simple geometry called finite elements or elements for short. The response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function, or functions, at a set of nodal points.

The response of the mathematical model is then considered to be approximated by that of the discrete model obtained by connecting or assembling the collection of all elements.

The three-dimensional FEA programs can be a useful tool in investigating design parameters for spur gears. The computational effort can be simplified by considering single tooth models. Such models are widely used and accepted in the literature [1-2].
2. ANSYS MULTIPHYSICS - ELECTROSTATIC ANALYSIS [3]

2.1. Problem Description

Constant current is applied between two electrodes and the electric potential is measured by two other measuring electrodes, which were placed in selected positions over the gauge area of a spur gear tooth. Their readings are associated with the actual position of the crack tip [4] using FEA. The optimum position of the electrodes is found performing rigorous Electrostatic Field Analysis Simulations.

This tutorial describes the steps of the Electrostatic Analysis on spur gear teeth.
2.2. Finite Element Modeling - Stages

The separation of work into discrete stages is necessary for the finite element modeling of a structure:

1st Stage  →  Design of the geometry.

2nd Stage  →  Defining material properties and the type of elements.

3rd Stage  →  Creating the finite element models - meshing.

4th Stage  →  Applying loads and boundary conditions.

5th Stage  →  Solving the problem.

6th Stage  →  Postprocessing the results (listing, plotting).

2.3. Importing the geometry of the specimen.

- The software «Ansys APDL» is launched.

The geometry of the spur gear teeth is imported from Solidworks to Ansys. The type of these files is *.sat.

Given the specific location of the files at the hard drive:

- File → Import → SAT…→ the appropriate *.sat file is selected.

2.4. Defining the type of elements and properties of the material.

The type of finite elements is defined at the pre-processor. This selection depends on the type of the Finite Element Analysis, which will be performed [Fig. 2.1].

- Pre-processor → Element Type → Add/Edit/Delete → Add…

- Left section → Elec Conduction

  Right section → Brick 8node 69
Figure 2.1: Library of Element Types.

Figure 2.2: Defining the value of electrical resistivity.
The electrical resistivity of the material id defined [Fig. 2.2]:

- Preprocessor → Material Props → Material Models → Electromagnetics → Resistivity → Orthotropic.

- RSVX, RSVY, RSVZ → 1.43e-7 (this is the value for steel material; the value of the electrical resistance is different for other materials) → OK.

Afterwards, the coordinates of the points, on which the electrodes of current’s application and electric potential’s measurement will be placed, are imported. These points are designed as hardpoints, namely their location does not change after the creation of the mesh. The points of electrical potential’s measurement must be equidistant from the points of the constant current’s application [5].

- Pre-processor → Modeling → Create → Keypoints → Hard PT on area → Hard PT by coordinates.

- Then, the area, on which the hardpoints will be designed, is selected [Fig. 2.3] → The coordinates of the 4 points are imported (two points for the current’s application and two for the measurement of the electric potential).

### 2.5. Creating the finite element models - Meshing.

The next step is the creation of the mesh [Fig. 2.4]. Most of the meshing operations can be done within the MeshTool. When global attributes are set, they are used for all elements on the model. Nonetheless, it is possible to assign different attributes to different geometric entities in the model.

- Pre-processor → Meshing → Mesh Tool:

  Element attributes: Global → Smart Size: checked → Mesh quality: 5 (1→ Fine, 10→ Coarse) → Shape: Tet, Free → Mesh: Volumes → Pick all.
Figure 2.3: Selection of the area, on which the hardpoints are designed.

Figure 2.4: Creation of mesh.
2.6. Applying loads and solving the problem.

- Solution → Analysis Type → New Analysis → Steady-State → OK [Fig. 2.5].

In order to find the serial number of each hardpoint [Fig. 2.6]:

- Utility menu → List → Keypoint → Hard Points.

Given the serial number of each hard point, the constant current is applied on the two, already designed, hard points.

Solution → Define Loads → Apply → Electric → Excitation → Current → On Keypoints →
Insert the serial number of the hardpoints → OK → Constant Value → VALUE Load AMPS value = 0.005 [Fig. 2.7].

- The same procedure is followed for the second hardpoint. The only difference is that
the current’s value is, now, negative:

- … → VALUE Load AMPS value = -0.005.

In order to submit the model to Ansys for solving:

- Solution → Solve → Current LS → OK.
Figure 2.6: List of the already designed hardpoints.

Figure 2.7: Application of constant current of 5mA.
2.7. Post-processing the results.

The General Post-processor is used to look at the results over the whole model at one point in time.

The potential drop can be found following the next steps:

- The serial number of the nodes, from which the electric potential is measured, is located in the same way, that the current’s application hardpoints [Fig. 2.6]:
  - Utility menu List → Keypoint → Hard Points → Nodes section.

- General Postproc → List Results → Nodal Solution → DOF Solution → Electric potential → File (*.txt)

This *.txt file provides the electric potential’s value of each node. Hence, potential drop between the two points can be calculated.

The equipotential lines can be plotted:

- General Postproc → Plot Results → Contour Plot → Nodal Solu → DOF Solution → Electric potential → OK.

The number of contours, as well as, the range of each contour can be modified by right-clicking on the colour scale [Fig. 2.8].

Also, the current density vector can be plotted via the next procedure [Fig. 2.9]:

- General Postproc → Plot Results → Vector Plot → Predefined → Vector item to be plotted:
  - Left section → Current Density

  Right section → Cpl’d Source JS

  Scale factor Multiplier → 0.1

  Vector scaling will be → Uniform
Figure 2.8: Modification of the contour's number and each contour's range.

Figure 2.9: Plotting the current density vector field.
2.8. Conclusions

This tutorial presents a step-by-step procedure of electrostatic analysis on spur gear teeth in Ansys Multiphysics.

The above procedure can be modified in relation to the type of each technical problem. Hence, all the necessary parameters and characteristics of the problem must be taken into account, before performing the analysis.


