FIFTH INTERNATIONAL CONGRESS ON SOUND AND VIBRATION
DECEMBER 15-18, 1997
ADELAIDE, SOUTH AUSTRALIA

FINITE ELEMENT ANALYSIS OF GEARS IN MESH

By
Seney Sirichai, Ian Howard, Laurie Morgan, and Kian Teh
School of Mechanical Engineering
Curtin University of Technology
Perth, Australia
howard@vesta.curtin.edu.au

ABSTRACT

This paper presents a finite element analysis of two spur gears in mesh. The model predicts the torsional mesh stiffness of the two gears in mesh when one of the gears is restrained from rotating, with the other gear having a torque input load. The mesh stiffness of two gears in mesh varies with the meshing position as the teeth rotate within the mesh cycle and the resulting torsional stiffness decreases and increases dramatically as the meshing of the teeth change from the double pair in contact to the single pair of teeth in contact and vice-versa. The accuracy of finite element modelling of contact problems depends on the choice of the penalty parameter for the contact element. For modelling of gear teeth in mesh, the penalty parameter also varies as the gear teeth rotate through the cyclic mesh. This paper presents a simple strategy of how to determine an appropriate value of the penalty parameter as the gears rotate through the mesh cycle. The resulting torsional stiffness is predicted as a function of the position of the mesh point in the mesh cycle. The results which are presented show evidence of a change in the torsional stiffness during the double pair of teeth in contact.

Keyword: torsional stiffness, contact stiffness, contact element, penalty parameter, spur gear.

INTRODUCTION

Gears are one of the most critical components in industrial rotating machinery. In recent years, many different procedures have been developed to model the behaviour of gears in mesh [1-10]. One of the many factors which can be investigated is the torsional mesh stiffness variation as gear teeth rotate through the mesh cycle. The torsional stiffness of gears is the ratio between the input torsional load and the total angular rotation of the gear. Total angular rotation is defined as the angle through which a gear turns due to bending of the gear teeth when in loaded mesh with a fixed mating gear [1]. In order to model contact problems with the finite element method, the stiffness relationship between the two contact bodies must be established, otherwise the two bodies will pass through each other. One of the means of solving this problem is through the use of a contact element. The convergence and accuracy of contact problems depends on the choice of the penalty parameter for the contact element which must be defined by the user [2]. Some of the previously published work [3,4,5,6,7,8] uses analytical techniques such as the variational inequality approach, which tend to be complicated and difficult to use. This paper presents a simple method for choosing an appropriate value of the penalty parameter. The method is verified using two cylinders in contact and the result is compared with Hertzian theory. The same strategy is then used to determine an appropriate value of the penalty parameter of gears in cyclic mesh.

FINITE ELEMENT MODEL

The finite element model of spur gears in mesh is based upon test gears which have been used in an experimental investigation [1], and the test gear parameters are shown in Table 1. The test gears have a ratio of 1:1. The involute and fillet tooth profile equations used in the finite element model have been introduced by references [9,10]. A customised AutoCAD program was used to generate the profile of the teeth and the profile was then transferred from AutoCAD to ANSYS [11], to generate the finite element model of gears in mesh.

Table 1 Test Gear Parameter

• gear type	standard involute, full-depth teeth
• material	aluminum
• modulus of elasticity, E	69 Gpa,
• Poisson's ratio, ν	0.33
• face width	0.015 m.
• module, m	6 mm.
• number of teeth	23
• pressure angle	20 degrees
• theoretical contact ratio	1.59
• theoretical angle of meshing cycle	24.912 degrees
• addendum	1.00m
• dedendum	1.25m

Most of the previously published finite element analysis with gears has involved only partial teeth models. In an investigation of gear transmission errors using factors such as variational torsional mesh stiffness, the whole body of the gears in mesh must be modelled, because the contact stiffness should account for the flexibility of the entire body of the gear not just the local stiffness at the contact point. A finite element model of the spur gears in mesh used in this investigation is shown in Figure 1. The gears were modelled by the quadratic 2D plane strain four-noded element. The contact effect was modelled using 2D point-to-surface general contact elements which included elastic coulomb frictional effects. The contact element was generated as a symmetric contact.

The torsional mesh stiffness of gears in mesh at particular positions throughout the mesh cycle was generated by rotating both solid gears then creating a finite element model in that particular position. In order to develop representative results, a large number of finite element models at different meshing positions were undertaken for this investigation. One of the most important criteria for each model was that the first potential contact nodes of both surfaces must be created exactly on the intersection point between the pressure line and the involute curve for that particular tooth. The additional problem of determining the penalty parameter (contact stiffness) at each contact position, must be overome so that it can be defined in the finite element model. To overcome these difficulties, a simple technique was developed to estimate the penalty parameter at each meshing position as shown below.

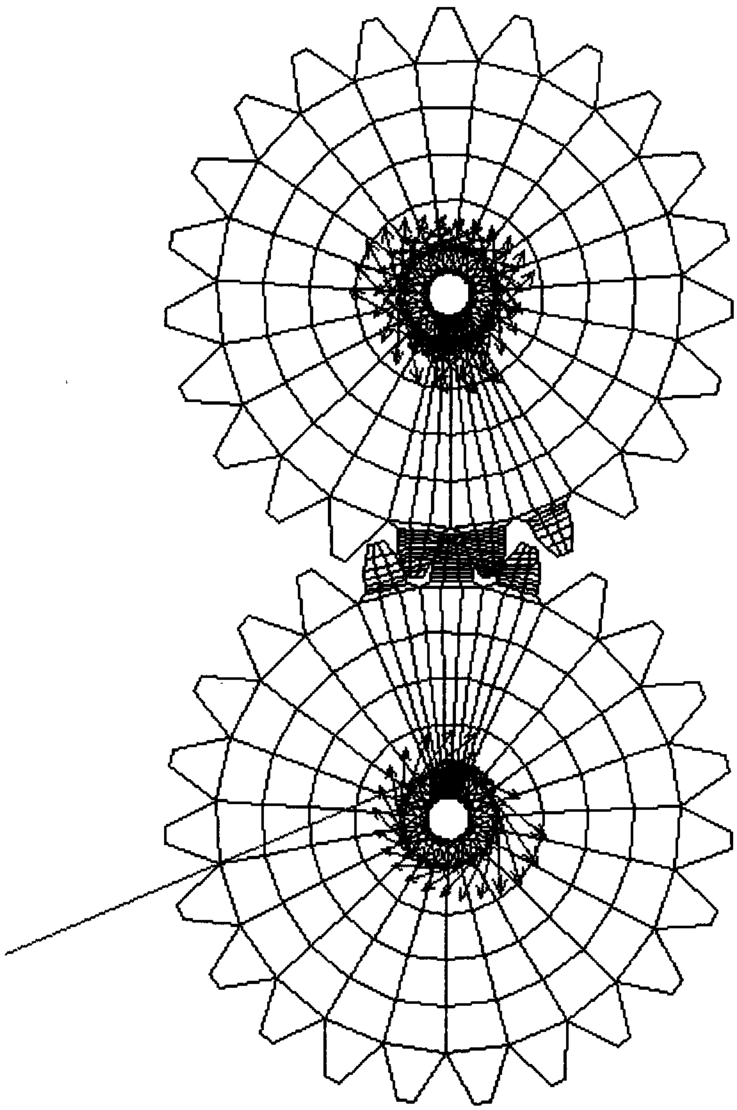


Figure 1. Finite Element Model of the spur gears in mesh with the load and boundary conditions.

PROBLEM DESCRIPTION

In order to handle a contact problems with the finite element method, the stiffness relationship between the two contact areas can be established through a spring (contact element) that is placed in between the two contacting areas where contact occurs. The stiffness of a spring or contact element is called the contact stiffness, Kn. This method of enforcing contact compatibility is called the penalty parameter method and is used in this paper where the contact stiffness is the penalty parameter. The accuracy of the solution which is obtained depends on the choice of the contact stiffness. If the contact stiffness is too small, the surface penetration may be too large which may cause unacceptable errors. On the other hand, if the contact stiffness is too large, the contact stiffness may produce severe numerical problems in the solution process or simply make a solution impossible to solve because of ill-conditioning with the stiffness matrix as some terms of the matrix are very large compared to others.

There are other methods for establishing the relationship between the two contacting areas, such as the Lagrange multiplier method which controls surface penetration by the user-defined addition of another force to push the two surfaces apart. Other methods involve coupling the two surfaces together or applying a force to push the two areas apart so that the two surfaces just touch. The main strategy behind the determination of an appropriate value of the contact stiffness (normal, Kn, and sticking or tangential contact stiffness, Kt) is that the value of contact stiffness should be computed from the local compliance of the component. The method by which this can be done depends on the kind of FEA models as given below.

1. For bulky solid models, the contact stiffness accounts for the flexibility of only the *local stiffness* of the materials at the point of contact.
2. For flexible models, such as beams, shells, and gears, the contact stiffness accounts for the flexibility of the *entire component* and not just the local stiffness of the material at the point of contact.

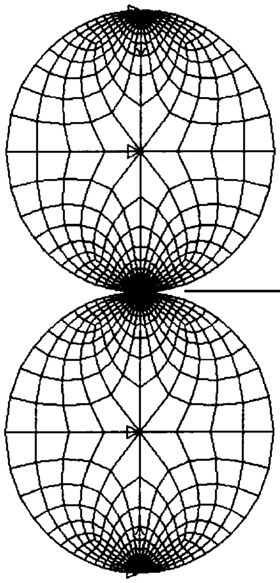
For both cases, the user-defined contact stiffness of the contact element can be seen as being equivalent to each component stiffness connected in series as given by,

$$Kn = \frac{f(K_1 * K_2)}{(K_1 + K_2)}, \tag{1}$$

where K_1 and K_2 are the stiffness of each component, f is the correction factor (usually between 1 to 3), which accounts for variations in contact problems such as mesh size, amount of real nodes in contact, frictional effects, accuracy of contact nodes position, etc. In case of translational stiffness, $K_{1,2}$ is equivalent to translation load divide by translation deflection in the same load direction. For the case of torsional stiffness of gear in mesh, $K_{1,2}$ will be equal to torsional load divide by angular deflection.

VERIFICATION AND IMPLEMENTATION

The purpose of the verification is to confirm that the techniques which have been developed for determining the contact stiffness can be applied to general contact problems and to assess the accuracy of the results. In order to verify the validity of the procedure, the frictional contact between two identical cylinders, [2], is further analysed as shown in Figure 2. The elastic properties of the cylinder are given as E=69 GPa., and ν= 0.33, with diameter 0.138 m. The coefficient of friction was taken as μ=0.4. The cylinders were modelled using the quadratic 2D plain strain four-node elements with an increased number of elements at the candidate contact surface to 100 μm mesh size.



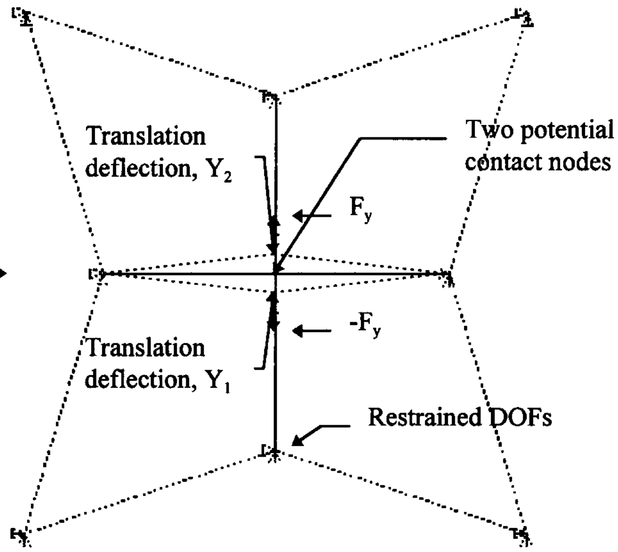


Figure 2. FEM of two cylinders in contact. Figure 3. The potential contact nodes.

The contacting area was modelled using 2D point-to-surface general contact elements with a symmetric contact. As the two cylinders in contact are bulky solids, the contact elements should be modelled by using only the local stiffness of the material at the point of contact. This can be achieved as shown in Figure 3, where only the first two potential contact nodes with two elements on each cylinder are considered. By applying a concentrated load (Fy=1000000 N) to both potential contact nodes in a single-iteration (linear) analysis where the surrounding nodes are restrained, the contact stiffness, Kn can be computed. From the applied load and the resulting deflections, equation (1) can be seen to give, using 1.0 for f,

$$\begin{aligned} Kn &= f\,(K_1 * K_2)\,/\,(K_1 + K_2), \\ &= f.\,(Fy)\,/\,(Y_1 + Y_2), \qquad (2) \\ &= 1000000/(1.60773\text{ e-5} + 1.60773\text{ e-5}), \\ &= 3.11\text{ e10} \quad \text{N/m} \quad \approx \quad 0.45E \quad \text{N/m}. \end{aligned}$$

The comparison of the deflection results of the centre of the cylinders for various values of the normal contact stiffness, with Hertzian theory are shown in Figure 4, [2]. It can be clearly seen that the gradient curve between the compressive force and the approach deflection starts to converge when Kn is equal to 0.5E. Any further increase in the value of Kn yields the same gradient, showing that the normal contact stiffness Kn is approximately equal to 0.5E, with less than 3 % difference from the Hertzian theory. The technique for finding the normal contact stiffness for bulky solids as used above, (Kn $\approx$ 0.45E), therefore appears to be correct.

The application of contact stiffness between two spur gears in mesh involves flexible bodies and so the contact stiffness must include the flexibility of the entire gear body, not just the local stiffness of the material at the contact tooth. The technique by which the flexible contact stiffness is obtained involves restraining four potential contact nodes (only two potential contact nodes for a single pair of teeth in contact) and applying a torsional load to both gear hubs in a single-iteration (linear) analysis. The contact stiffness is thus computed from the applied torque and the corresponding angle of rotation of the gear hub as shown in equation (3),

$$Kn = \frac{f(T_1 * T_2)}{(T_2 * \theta_1 + T_1 * \theta_2)}, \tag{3}$$

where $T_{1,2}$ and $\theta_{1,2}$ are the applied torque and the corresponding angle of rotation of the hub of gear 1 and 2 respectively.

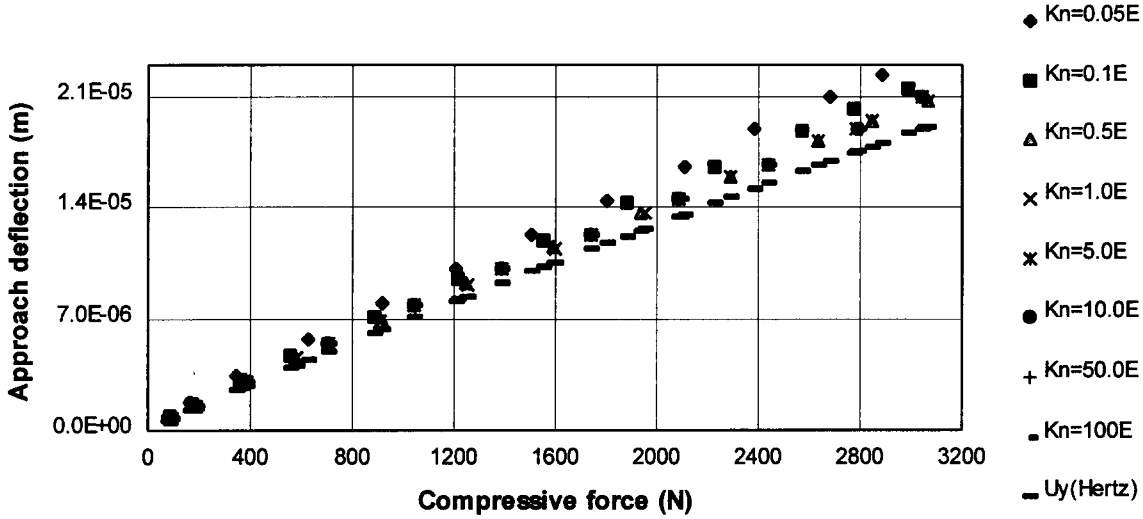


Fig.4 The effect of varying contact stiffness Kn

As the mesh geometry of the gear teeth changes over each mesh cycle, the contact stiffness must also vary within each meshing cycle. The value of Kn as formulated in equation (3) must therefore be computed at each meshing position. By using a value of 2.0 for f to account for mesh size, the addition of frictional effects, and the number of actual nodes in contact, the contact stiffness, Kn, was computed for the full completed meshing cycle as shown in Figure 5. Having computed the contact stiffness at each meshing position, the actual torsional stiffness of the gears in mesh can be determined.

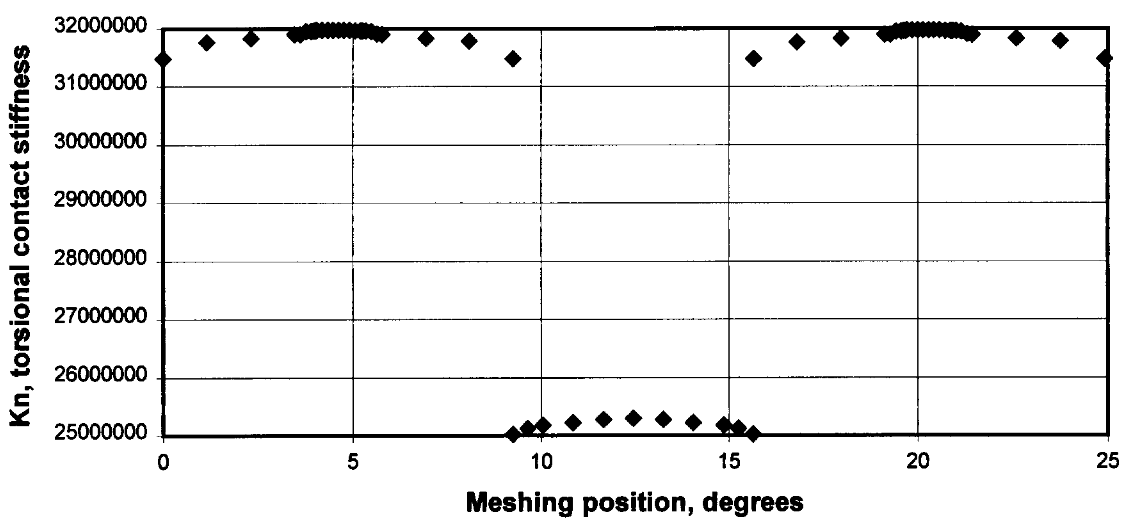


Figure 5. The variation of torsional contact stiffness, Kn over the mesh cycle.

RESULT AND DISCUSSION

The torsional stiffness of the gears in mesh was obtained by introducing a prescribed rotation of the coupled nodes of the input gear hub, while restraining all nodes of the output gear hub. The computed value of the contact stiffness at each particular meshing position as shown in Figure 5, was used in the FE model. The result of the torsional stiffness of the gear mesh at particular meshing positions was obtained by calculating the gradient of the linear relationship between torque and angular displacement and in this manner the torsional stiffness of the entire meshing cycle was obtained, as shown in Figure 6.

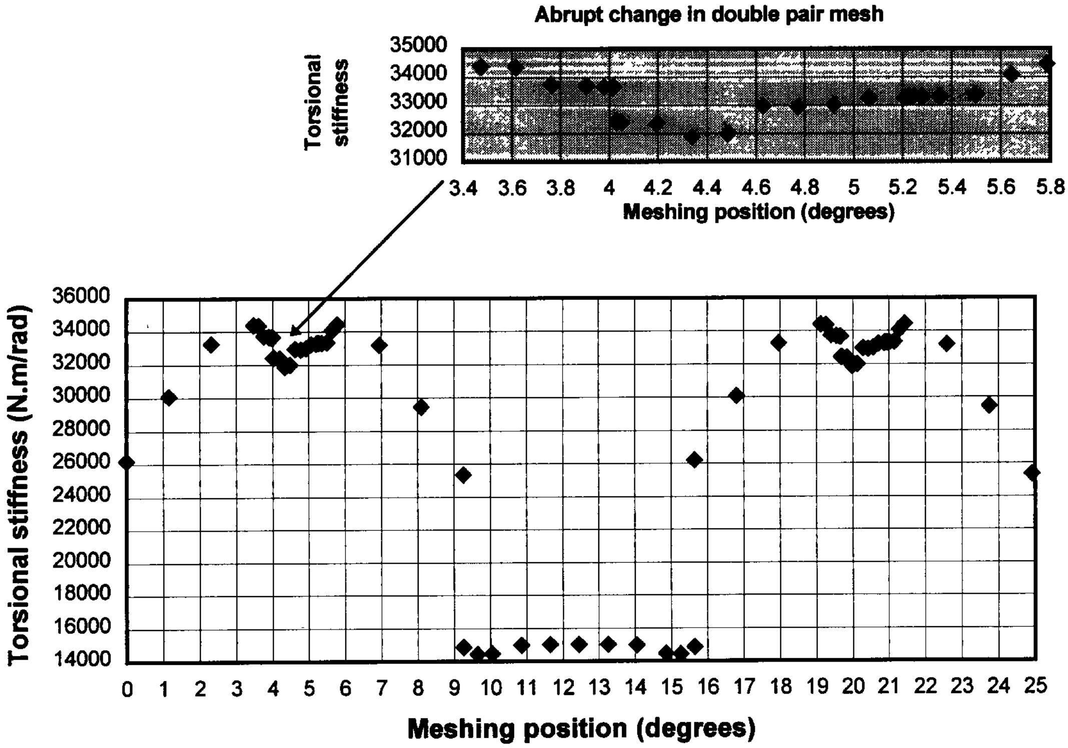


Figure 6. Torsional stiffness (N.m/rad) of one complete mesh cycle

As expected, the resulting torsional stiffness of the gears in mesh varies dramatically throughout the meshing cycle, and it appears to be periodic with the tooth position. As the number of teeth in mesh changes from two to one and then back to two teeth in contact, the torsional stiffness increases and decreases [10,12,13]. At the middle of the double pair of teeth in contact, an abrupt change in the torsional stiffness also appears and is the subject of ongoing investigation.

CONCLUSION

In this paper, a new strategy for choosing an appropriate value of contact stiffness has been presented. The method has been demonstrated to work well for both bulky solids and for flexible components by analysis of two cylinders in contact and of two spur gears in mesh. The method involves the creation of first potential contact nodes and keypoints. The keypoints are used for specifying the edge lengths of the elements and mesh size at the candidate contact surface. The first potential contact node is restrained and the contact stiffness at each particular position of mesh is then computed.

The results for the stiffness of two cylinders in contact have been shown to agree very closely with Hertzian theory. The application of the technique to two gears in mesh appears to have been successful, with the resulting torsional stiffness being periodic over each mesh cycle. Experimental verification is currently being investigated. It is hoped that the technique which has been developed for determining the contact stiffness of mechanical components in contact will be particularly useful for all kinds of contact problems, not just the examples which have been considered.

REFERENCES

1. Sirichai, S.(1996); Morgan, L.; Howard, I.; Teh, K., The Measurement Of Static Torsional Stiffness Of Gears In Mesh ISMTII'96, Hayama Kanayawa Pref., JAPAN, pp 251-258.
2. Sirichai, S.(1997); Howard, I.; Morgan, L.; Teh, K., A Synthetic Verification of 2D Contact Stiffness 2nd NUMETe, UPM, Selangor, MALAYSIA, pp 261-271.
3. M. H. Refaat and S. A. Meguid(1994) On The Elastic Solution Of Frictional Contact Problems Using Variational Inequalities Int. J. Mech. Sci., Vol. 36, No. 4, pp. 329-342.
4. M. H. Refaat and S. A. Meguid(1995) On The Contact Stress Analysis Of Spur Gears Using Variational Inequalities Computers & Structures., Vol. 57, No. 5, pp. 871-882.
5. M. H. Refaat and S. A. Meguid(1996) A Novel Finite Element Approach To Frictional Contact Problems International Journal for Numerical Methods in engineering., Vol. 39, No. 5, pp. 3889-3902.
6. M. H. Refaat and S. A. Meguid(1997) Update Lagrangian Formulation Of Contact Problems Using Variational Inequalities International Journal for Numerical Methods in engineering., Vol. 40: pp 1-19., 1997 by John Wiley & Son, Ltd.
7. W. R. C. Underhill, M. A. Dokainish, G. E. Oravas(1997) A Method For Contact Problems Using Virtual Elements Compit. Methods. Appl. Mech. Engrg., Vol. 143, pp. 229-247.
8. Piotr Kowalczyk(1994) Finite-Deformation Interface Formulation For Frictionless Contact Problems Communications in Numerical Methods in Engineering., Vol. 10, pp. 879-893.
9. Litvin, F.L.(1989) Theory of Gearing NASA Reference Publication 1212.
10. Kuang, J.H., and Yang, Y.T.(1992) An Estimate Of Mesh Stiffness And Load Sharing Ratio Of A Spur Gear Pair International Power Transmission and Gearing Conference, Vol. 1, ASME, pp. 1-9.
11. ANSYS, Inc., Revision 5.2 (1995) User's Manual Procedures Volume 1
12. R.G.Munro(1997) The Use Of Optical Gratings In Gear Metrology Third International Conference On Laser Metrology And Machine Performance, Huddersfield, UK, Computational Mechanics Publications, pp. 244-252.
13. Umezawa, K., Sato, T., and Ishikawa, J., "Simulation on rotational vibration of spur gears". Bulletin of JSME, Vol.27, No.223, pp. 102-109 (Jan. 1984).