International Journal of Modern Trends in Engineering and Research
www.ijmter.com
e-ISSN No.:2349-9745, Date: 2-4 July, 2015

A CONTACT STRESS ANALYSIS OF SPUR GEAR TO OPTIMIZE MASS OR WEIGHT USING FEA TECHNIQUES

Mr Suhas Alkunte[1], Dr. S.Y.Gajjal[2]

Mechanical Department, Navsahyadri Group of institutions, FOE,suhasalkunte@rediffmail.com
Mechanical Department, NBN Sinhagad school of Engineering, shekhar.gajjal@sinhgad.edu

Abstract— The Gears while transmitting the power develop stresses at the mating positions. The main objective of this study is to add circular holes to reduce weight or mass of the gear. This paper represents that contact stress analysis between two spur gear teeth was considered at different contact positions during rotation. In this study Pinion rotation from first contact point to last contact point to produce 4 cases. Each case shows that sequence position of contact stress between these two teeth. As Finite Element Method (FEM) is the easy and accurate technique for stress analysis, FEA is done in finite element software ABAQUS. The results obtained showed that difference between magnitude of the stresses for a given configuration of a spur gear and Hertz contact stress, while transmitting power also trying to find ways for reducing weight or Mass of the gear. This paper is focused towards minimization of stress as well as mass. Even a slight reduction in the stress and mass results in great increase in the fatigue life of a gear.

Keywords- Contact stress, FEA, ABAQUS, Hertz Contact, Spur Gear, Mass

I. INTRODUCTION

Gears are used for a wide range of industrial applications. They have varied application starting from textile looms to aviation industries. They are the most common means of transmitting power. They change the rate of rotation of machinery shaft and also the axis of rotation. For high speed machinery, such as an automobile transmission, they are the optimal medium for low energy loss and high accuracy. Their function is to convert input provided by prime mover into an output with lower speed and corresponding higher torque. Toothed gears are used to transmit the power with high velocity ratio. During this phase, they encounter high stress at the point of contact. A pair of teeth in action is generally subjected to two types of cyclic stresses:

i) Bending stresses inducing bending fatigue

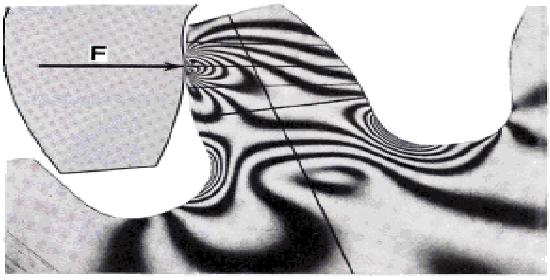
ii) Contact stress causing contact fatigue.

Both these types of stresses may not attain their maximum values at the same point of contact. However, combined action of both of them is the reason of failure of gear tooth leading to fracture at the root of a tooth under bending fatigue and surface failure, due to contact fatigue. When loads are applied to the bodies, their surfaces deform elastically near the point of contact. Stresses developed by Normal force in a photo-elastic model of gear tooth are shown in the Fig.1 The highest stresses exist at regions where the lines are bunched closest together.

The highest stress occurs at two locations:

A. At contact point where the force F acts

B. At the fillet region near the base of the tooth

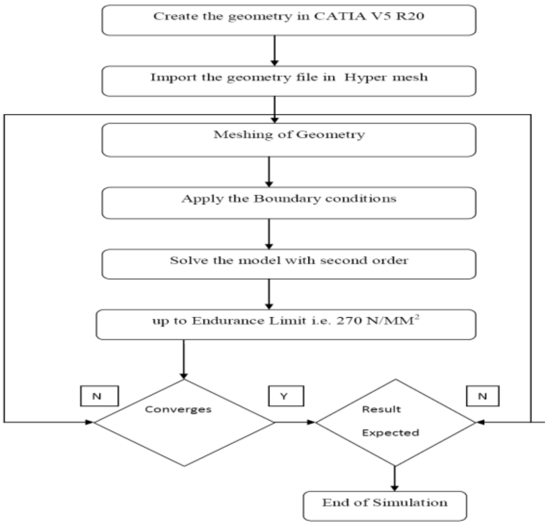


"*Figure 1.Photo Elastic Model of gear tooth*"

[4]The surface failures occurring mainly due to contact fatigue are pitting and scoring. It is a phenomenon in which small particles are removed from the surface of the tooth due to the high contact stresses that are present between mating teeth. Pitting is actually the fatigue failure of the tooth surface. Hardness is the primary property of the gear tooth that provides resistance to pitting. In other words, pitting is a surface fatigue failure due to many repetitions of high contact stress, which occurs on gear tooth surfaces when a pair of teeth is transmitting power. Gear teeth failure due to contact fatigue is a common phenomenon observed. Even a slight reduction in the stress at root results in great increase in the fatigue life of a gear. For many years, gear design has been improved by using improved material, hardening surfaces with heat treatment and carburization, and shotpeening to improve surface finish etc. Few more efforts have been made to improve the durability and strength by altering the pressure angle, using the asymmetric teeth, altering the geometry of root fillet curve and so on. Some research work is also done using the stress redistribution techniques by introducing the stress relieving features in the stressed zone to the advantage of reduction of root fillet stress in spur gear.

II. FINITE ELEMENT METHOD

In FEA modeling element quality greatly effects the accuracy of analysis results. many modern finite element analyzer solvers have routines to compensate for some measure of poor quality element but it is not a good practice to rely on these compensations. the FEA modeler must take into consideration element quality and thereby judge whether the analysis results are meaningful. The ideal four node (quad) plate element is a planner square two types of error can result from translating a single node. if one of the nodes is translated in the plane remaining nodes, interior angles change & edge lengths vary between sides introducing skew and aspect ratio into the element if one of the nodes is translated out of the planes of others result is war page. with first order tria elements war page is not possible but aspect ratio & skew remain valid it measures element quality. The element checks in Hyper mesh test their properties and provide feedback as to quality of element.



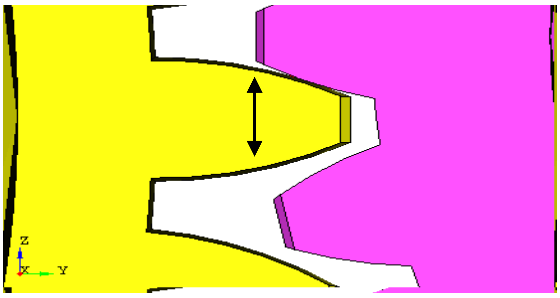
"Figure 2. Flowchart"

III.STRESS CONTOUR OF GEAR & PINION

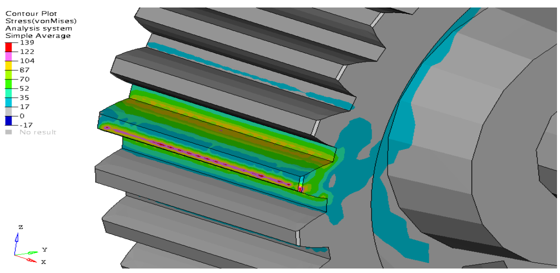
A. Before Optimization

Case-I

Tooth thickness at location 1 = 3mm



"Figure 3.Contact point at location 1"

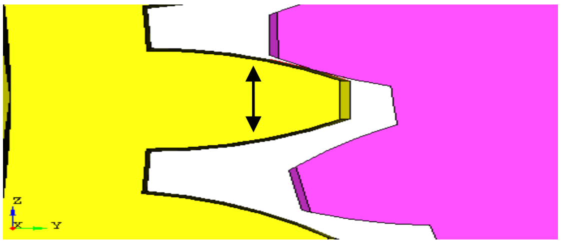


"Figure 4.Contact stress at location 1"

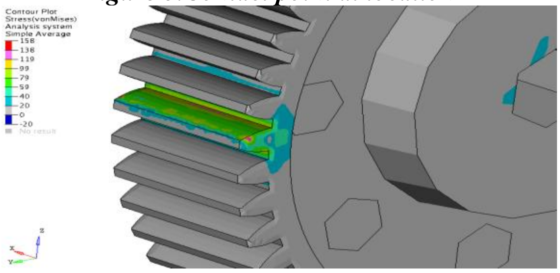
Maximum Stress is 132.47 N/mm^2

Case-II

Tooth thickness at location 2 = 2.6mm



"Figure 5.Contact point at location 2"

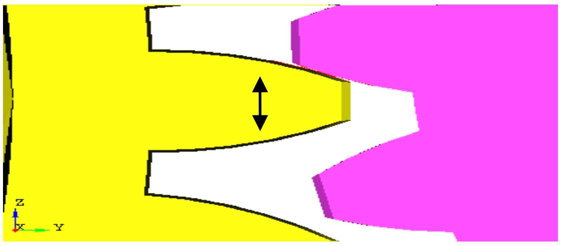


"Figure 6.Contact stress at location 2"

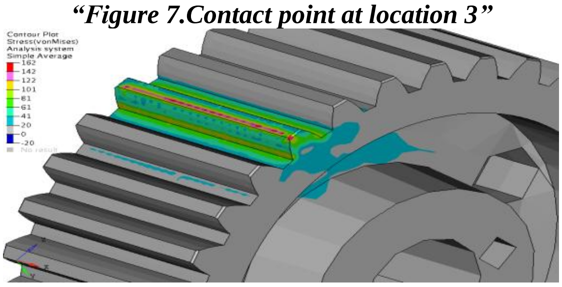
Maximum Stress is 143.67 N/mm^2

Case-III

Tooth thickness at location 3 = 2.33mm



"Figure 7.Contact point at location 3"

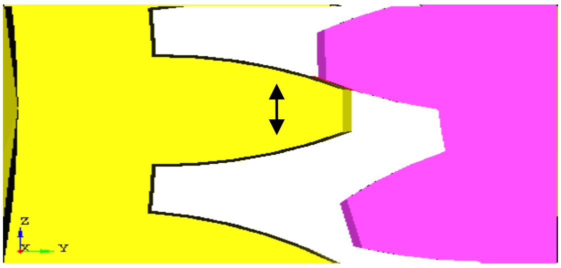


"Figure 8.Contact stress at location 3"

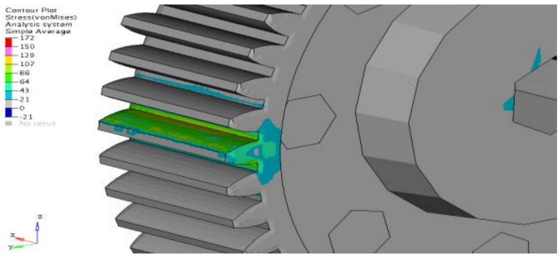
Maximum Stress is 153.65 N/mm^2

Case-IV

Tooth thickness at location 4 = 1.917mm



"Figure 9.Contact point at location 4"



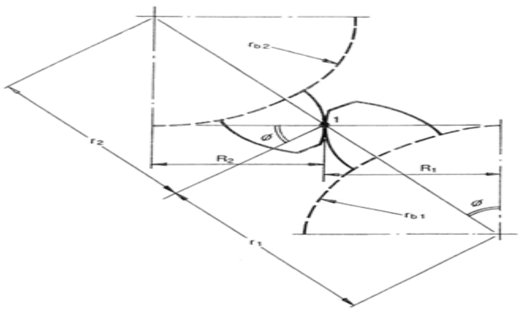
"Figure 10.Contact stress at location 4"

Maximum Stress is 160.75 N/mm^2

At first position of contact area the tooth thickness is 3 mm, in second contact location it is 2.6 mm, in 3rd position it is 2.33 mm while in last contact location it is 1.917mm. The reason of changing thickness is as pinion rotates gear because of tooth geometry the contact area is changing that's why thickness also.

IV. HERTZ CONTACT STRESS (CONTACT STRESS ANALYSIS)

The transfer of power between gears takes place at the contact between the acting teeth. The stresses at the contact point are computed by means of the theory of Hertz. The theory provides mathematical expressions of stresses and deformations of curved bodies in contact.[2] Fig. 10 shows a model applied to the gear-two parallel cylinders in contact. Where E1 and E2 are the Modulus of Elasticity of pinion and gear respectively, υ1 and υ2 are the Poisson's ratios of pinion and gear respectively and F is the face width of pinion. Same equation can be apply for teeth, assuming for R1 and R2 the respective radii of the in volute curve at the contact point, Let us assume that the contact stress Analysis in Pinion. Where rp1 and rp2 are the pitch radii of the pinion and gear and φ is the pressure angle. The stress correlations derived heretofore and Eq. are based on a number of simplifying assumptions, such as pure bending of short beam and elliptic distribution of stresses at tooth contact. A question therefore arises concerning their accuracy. Contact takes place at point 1, and then the respective radii are equal to R1= rp1 sin Ǿ, R2=rp2 sin Ǿ



"Figure 11.Parralel cylinders in contact"

Problems, design needs and safety requirements make far in depth and complicated study of this contact. The current project aims to arriving at these very solutions. The elastic compression of two-dimensional bodies in contact cannot be calculated solely from the contact stresses given by the Hertz theory. Some account must be taken for the shape and size of the bodies themselves and the way in which they are supported. In most practical circumstances such calculations are difficult to perform, which have resulted in a variety of approximate formulae for calculating the elastic compression of bodies in line contact such as gear teeth and roller bearings in line contact . The pitting problems, design needs and safety requirements make far in depth and complicated study of this contact. The current project aims to arriving at these very solutions. Hertzian Contact Stress is given by,

$$P\rho = Y_m\, Y_p \sqrt{\frac{Ft}{bd1} * \frac{u+1}{u}}$$

Where,

$P\rho$= Hertz contact stress in Mp_a

Y_m= Material Coefficient

Y_p= Pitch Point Coefficient

F_t= Tangential Load in N

b= Face Width in mm

d1= Dia. of Pinion in mm

u=d2/d1 (unity)

Allowable maximum stress is given by

$$P\rho = P_p/FOS$$

Y_m is material coefficient is given by

$$Y_m = \sqrt{0.35 * \frac{2E1 * E2}{E_1 + E2}}$$

$$= \sqrt{0.35 * \frac{2 * 2.1 * 10^\wedge 5 * 2.1 * 10^\wedge 5}{2.1 * 10^\wedge 5 + 2.1 * 10^\wedge 5}}$$

$$= 210.32$$

Where, Y_p is Pitch point coefficient is given by,

$$Y_p = \sqrt{\frac{1}{\cos^2 a * \tan a}}$$

$$Y_p = \sqrt{\frac{1}{\cos^2 20 * \tan 20}}$$

$$Y_p = 1.76$$

$D_{1=}$ module (m) * Number of teeth on Pinion

D_1= 2*31

D_1= 62 mm

$D_{2=}$ module (m) * Number of teeth on Gear

D_2= 2*52

D_2= 104 mm

u= D2/D1

u= 104/62

u=1.67

=325.7014/FOS

=325.7014/2

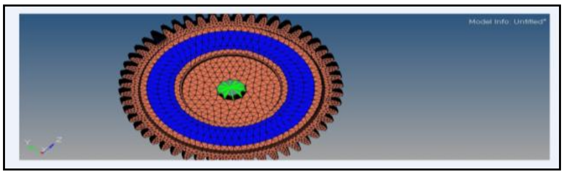
$= 162.85\ Mp_a$

Hertzian Contact Stress is 162.85 N/mm^2

V. DESIGN SPACE FOR TOPOLOGY OPTIMIZATION

A. Design Variable: we remove the material from Gear of tooth because to create design space. Optimization will be performing over the design variable. We define the separately property & material.

B. Constraint: Stress limit should not go beyond the Endurance limit while performing topology optimization then force is provided tip of the tooth which will generate equal and stresses in contact analysis is 160.75 N/mm^2

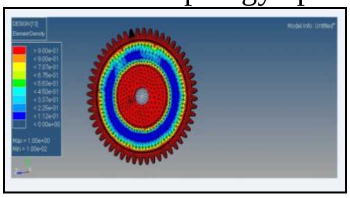


"Figure 12.Design space for topology optimization"

1. To minimize the volume fraction of gear
2. Constraint: - stress should not be more than of yield stress i.e. 355 N/mm^2
3. Endurance limit 0.4 times ultimate tensile strength = Ultimate tensile strength of material is 700 N/mm^2

$$= 0.4*700$$
$$= 280\ N/mm^2$$

Blue color shows where topology optimization is perform

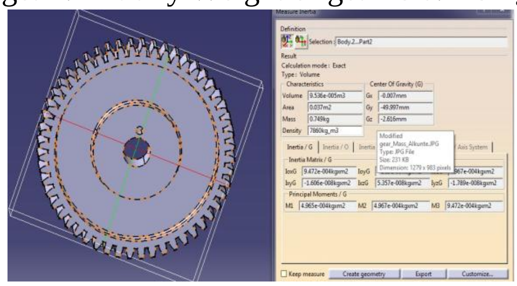


"Figure 13.Element density plot of gear"

C. Design Modification

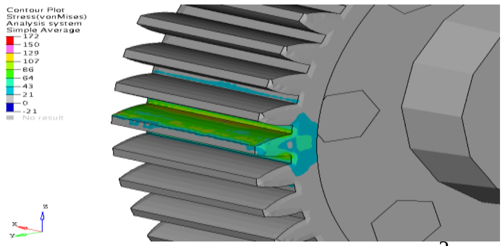
As the original design shows the maximum deformation of 0.02 mm and maximum contact stress as 280.75 Mpa. The modification is suggested towards minimization of stress as well as mass. Even a slight reduction in the stress and mass results in great increase in the fatigue life of a gear. Hence a design modification with 2 holes, 4 holes,6 holes, and 8holes are used to predict the mass and stress for a given loading condition

1. FE Model of Existing gear : Initially Weight of gear is 0.749Kg



"Figure 14.FE Model of existing gear"

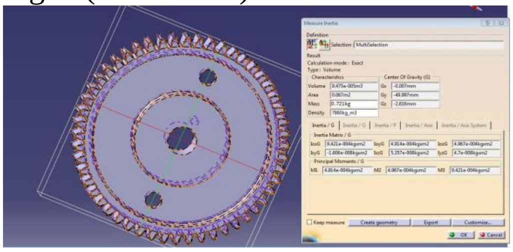
Analysis Result



Contact stress is 160.75N/mm^2

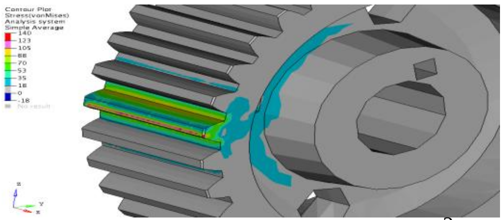
Case-I

FE Model of modified design 1(For 2 holes)



"Figure 15.FE Model of 2 holes"

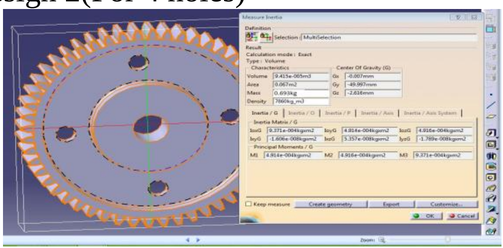
Analysis Result



Contact stress is 158.62 N/mm^2

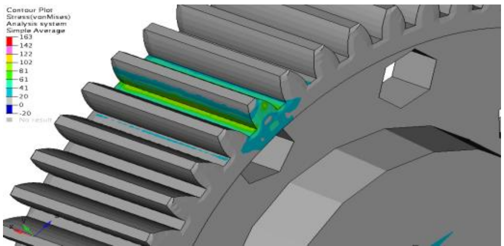
Case-II

FE Model of modified design 2(For 4 holes)



"Figure 16.FE Model of 4 holes"

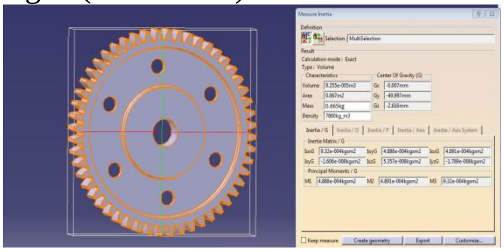
Analysis Result



Contact stress is 151.65N/mm^2

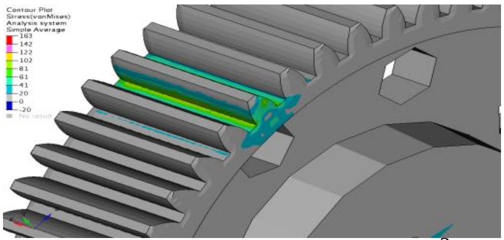
Case-III

FE Model of modified design 3(For 6 holes)



"Figure 17.FE Model of 6 holes"

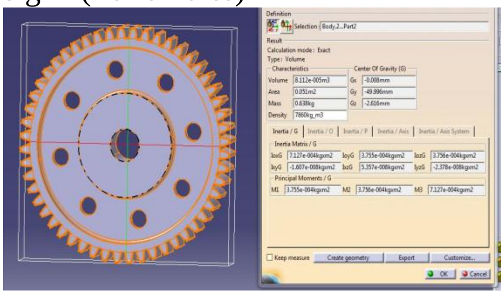
Analysis Result



Contact stress is 141.61N/mm^2

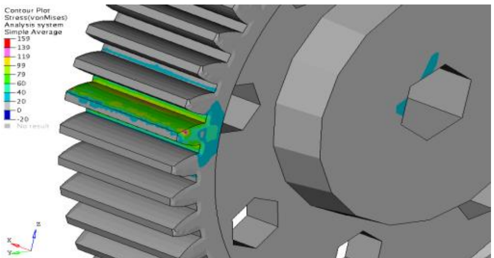
Case-IV

FE Model of modified design 4(For 8 holes)



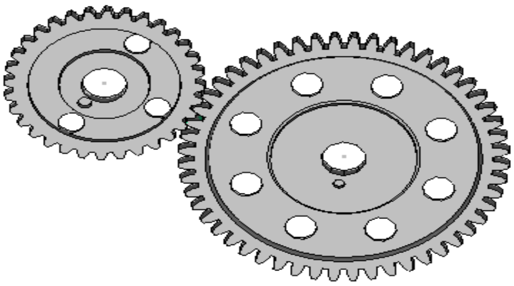
"Figure 18.FE Model of 8 holes"

Analysis Result



Contact stress is 130.28 N/mm^2

Geometry of gear and pinion after topology optimization



"Figure 19.Geometry of gear and pinion after topology"

We have created eight holes of 5mm radius uniformly along radial circumference of 32 mm from the center of the gear.

RESULT AND SUMARRY

Sr. No.	Analytical Result	Analysis Result	After Optimization
1	162.85 N/mm^2	160.75 N/mm^2	130.28 N/mm^2

"Table1.Contact stress"

From above result we can conclude that Maximum contact stress is 160.75N/mm^2 and after optimization stress is 130.28N/mm^2 somewhat similar. Before topology weight of Gear is 749gm and After Topology weight of gear is 638gm so there is 111gm reduction. It means that Ease of incorporating the new feature for weight reduction over the existing process of manufacturing and the magnitude of volume of mass (or weight) reduced.

CONCLUSION

1 The contact stress level in the optimized spur gear tooth is reduced about 22.5%
2. Contact stress value is within permissible Limits
3. The total weight reduction of spur gear is about17.38%
4. The design deformation is well within the permissible limit of maximum 0.02 mm.
5. Stress relieving feature having a shape of circular hole is used in the path of stress flow. When compared to elliptical hole it will be easy for manufacturing.

REFERENCES

[1] Dhavale, A.,S., Abhay, Utpat, 2013"Study of Stress Relief Features at Root of Teeth of Spur Gear," International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622, Vol. 3, Issue 3, May-Jun 2013, pp.895-899
[2] Mr., Bharat, Gupta, Mr., Abhishek, choubey, 2012'' Contact Stress Analysis Of Spur Gear, " , International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 1 Issue 4, June.
[3] Ruben, D.,Chacon, Luis J., Andueza, " Analysis Of Stress Due To Contact Between Spur Gears, " Advances in Computational Intelligence, Man-Machine Systems and Cybernetics, ISBN: 978-960-474-257-8,
[4] Ali, Raad, Hassan, 2009"Contact Stress Analysis of Spur Gear Teeth Pair," World Academy of Science, Engineering and Technology 58
[5] Vivek, Karaveer,,Ashish, Mogrekar, 2013" Modeling and Finite Element Analysis of Spur Gear, " International Journal of Current Engineering and Technology, ISSN 2277 - 4106, Vol.3, No.5
[6] Nidal, H,. Abu-Hamdeh, Mohammad, A. Alharty, " A Study on the Influence of using Stress Relieving Feature on Reducing the Root Fillet Stress in Spur Gear," Proceedings of the 2014 International Conference on Mathematical Methods, Mathematical Models and Simulation in Science and Engineering,ISBN:978-1-61804-219-4
[7] Seok-Chul, Hwanga, Jin-Hwan, Lee, b, Dong-Hyung, Lee, c, Seung-Ho, Hana, Kwon-Hee, Lee, a, 2013 " Mathematical and Computer Modelling " Mathematical and Computer Modelling 57 (2013) 40–49

