P-46

고응력 외상에의한 고관절용 세라믹/세라믹 쌍의 비선형 유한요소법 분석

Non Linear Finite Element Analyses of Ceramic/Ceramic Pairs of Total Hip Replacements Using High Trauma-Like Loads

다니엘 카리요[1*] , 이수완[1]
[1]선문대학교 공과대학 전자재료공학과

1. Introduction

The ceramic components of a hip prosthesis have been proven to be reliable materials when manufactured according to ISO regulations (ISO 6474). However, disadvantage of ceramic joint prostheses is some clinical history of sudden fracture, both of the femoral head and the acetabular component [1-3] and this type of failure has been directly related to trauma loading conditions [3]. Many factors have been implicated in ceramic head fracture, including trauma, demanding physical activity, obesity, component mismatch, small ball diameter, defective component manufacture, and implantation error [4]. Delayed fracture after trauma has been shown to occur in ceramic heads in vitro, owing to the development of hoop stresses within theceramic after impaction on the trunnion [4, 5]. Moreover high peak trauma-like loads are often overlooked in the design procedures of the ceramics ball heads (i.e. proof testing procedures). The aim of this investigation is to make a computer simulated model of the implanted hip joint, which replicates the physiological impact loading conditions of the ball head. Stress concentration regions and contact pressure distributions are to be determined. This data may be helpful to visualize the incidence of geometric parameters in the loading of the ball head caused by sudden impact, and to improve design approach as well as proof testing procedures of ceramic ball heads.

2. Experimental Procedure

The stem model simulated was the Euro-cone (12/14) which is usedin more than 70% of all stem-ceramic couplings and is standardized by ISO-7206. Models for the ball head were constructed for 22, 28, 32 and 36 mm diameters. Two different sizes for the ceramic bore were considered for ball head diameters higher than 28mm(including 28mm ball heads), being the bore depth the varying parameter. Given the fact that manufacturing tolerances are a major issue in the reliability of the taper fit, influencing the magnitude of the ball head stresses as well as the concentration regions [6, 7]; they were taken in to consideration when modeling the 12/14 taper. A non-linear 3D finite element model was generated in the ANSYS software to calculate stress distribution and contact pressure in the stem/ball joint and stress distribution in the ceramic cup. The materials considered for the stem were Ti6Al4V alloy and alumina for the ceramic materials (ball and cup). The coefficient of friction considered was 0.35, this value has been proven to be adequate in this kind of analysis [8].

3. Results and Discussion

Table 1shows the peak values of stress related parameters calculated for the different stem ball configurations as a function of the size of the cone, the material and the diameter of the ball head. It can be seen that for the size1 ball heads, the Von Mises stress increase dramatically with decreasing the ball diameter. In the size 2 case this tendency was not found. This behavior leads to the idea that the Von Mises stress magnitude is directly related with the size of the cone in the ball head, instead of the ball diameter. High stress configurations were found for the cases of 22 and 28 mm. Moreover, the stress exceeds the bending strength in the case of size 1 alumina ball heads of 22 mm in diameter, which is unacceptable. Whenthe transmitted stresses to the ball head are high, concentrators due to manufacturing imperfections or stem impingement may give raise to fracture of the ceramic component. This effect is slightly relieved by the use of zirconia due to its better mechanical properties in a structural point of view. The contact pressure and frictional stresses also are lesser for the size 2 than the size 1 case, which corroborates that a purely geometric approach in the design of the ball head by using a deeper cone may be suitable to decrease risk of failure. The deeper cone also promotes a more even stress distribution around the contact region.

Table 1. Peak values calculated for the Von Mises stress, contact pressure and friction stresses for the different geometries simulated..

Diameter [mm]	Size 1						Size 2					
	$\blacklozenge_{VM}$ [MPa]		Contact Pressure [MPa]		Frictional Stress [MPa]		$\blacklozenge_{VM}$ [MPa]		Contact Pressure [MPa]		Frictional Stress [MPa]	
	Al_2O_3	ZrO_2	Al_2O_3	ZrO_2	Al_2O_3	ZrO_2	Al_2O_3	ZrO_2	Al_2O_3	ZrO_2	Al_2O_3	ZrO_2
22	652	571	80	82	113	101	-	-	-	-	-	-
28	448	372	85	94	120	101	216	215	66	65	106	103
32	235	221	97	101	120	107	215	214	73	74	103	95
36	234	221	97	100	118	105	212	216	88	93	110	104

The von mises stress distribution as well as the contact pressures on the stem/ball joint were calculated. When the trauma-like loading is simulated in the system, a peak stress is found near the bottom of the ball. In both cases size 1 and size 2 the ceramic material can resist with this stress magnitude, however, due to the sensibility of ceramic materials to stress concentration, a geometrical stress reliever could be useful. Notice that the values obtained for the size 2 geometry are lesser than the ones for the size 1, probably caused for the greater moments that may arise from the size 1 configuration when a large part of the stem head lies outside the ballhead. An irregular contour of contact pressure was found in opposition with the regular stress-related contour seen for the size 2 case. Also the pressure for the size 1 is much lower than the size 2 case, low contact pressure in a given interface may produce microsliding, in this type of system microsliding could lead to fretting corrosion of the materials. Some limitations of these models should be pointed out. The simulation was performed in a quartersection of the volume to save CPU time in the runs, so a symmetric load has to be applied. This is not the case for real physiologic loading. Moreover, impact loading could have magnitudes and directions highly dissimilar than those of routine activities.The constraints applied to the system were somewhat simple for the same reasons. The taper lock should be the provider of the stems constrain in the vertical direction when loaded, however, a zero displacement in this direction was applied to the stem to help the model converge. The same case with the boundary condition considered for the ball perimeter (zero in all directions).

4. Summary

Non linear finite element analyses were performed in various configurations of stem-ball head. High stresses were found for the cases when the stem tended to penetrate less into the ball head. An upgraded design of the cone may improve the loading conditions of the ball head to resist trauma-like loading more effectively than manipulating the ball diameter. When the surgeon needs to use small ball heads (i.e. 22 mm), the use of zirconia seems to be appropriate also. After simulating a trauma like loading of the materials, it was found that the deepness of the cone to locate the stem is of major importance for the performance of the device. Further work, considering more sizes for the cone design should be performed in order to determine an optimal depth for the cone in relation to the diameter of the ball head. Also the simulation of contacts pairs including polyethylene and CoCr is important for further research.

References

[1] S.A. Maher, J.D. Lipman, L.J. Curley, M. Gilchrist, T.M. Wright: J. Arthroplasty Vol 18 (2003), p. 936

[2] O. Mahoney and J. Dimon: J. Bone Joint Surg. [Am] Vol 72 (1990), p. 663

[3] I. Pulliam and R. Trousdale: J. Bone Joint Surg. [Am] Vol 79 (1997), p. 118

[4] C. McLean, H. Dabis and D. Mok: J. Arthroplasty Vol 17 (2002), p. 503

[5] R. Michaud and S. Rashad: J Arthroplasty Vol 10 (1995), p. 863

[6] N. Shareef and D. Levine: Biomat. Vol 17 (1996), p. 623

[7] B. Cales and Y. Stefani: J. Biomed. Mat. Res. Vol 43 (1998), p. 62

[8] B.Weisse, M. Zahner, W. Weber and W. Riegerb: J. of Biomech. Vol 36 (2003), p. 1633