J.Hard Tissue Biology.14(2)Proceeding,2005

Examination of Properties of Bioabsorbable Osteosynthetic Material Using Finite Element Method

Tomoaki Kawamoto and Toshio Sugahara

Department of Oral and Maxillofacial Reconstructive Surgery, Okayama University Graduate School of Medicine and Dentistry.
2-5-1 Shikata, Okayama, Okayama 700-8525, JAPAN

Abstract:To overcome the disadvantages of metallic osteosynthetic devices that should remove after the operation, bioabsorbable, ultra high strength poly-L-lactid osteosynthetic devices were developed, and have been applied to the treatment of bone fractures[1.5.6]. On the other hand, it is not known how much stress is actually generated on these devices when an external force was added and minor segment was displaced. In order to know the stress distribution generated in the devices, using 3-dimensional-CAD software, we created the solid model of these devices and the blocks, and assembled these parts just as the two blocks were fixed with these devices. We simulated of fixed one block in the two blocks, and moved a constant amount in four directions to the other block, and analyzed the stress generated in these devices using 3-dimensional stress analysis software. In these simulations, the biggest stress was generated when the plate was tensiled.

Key Words : poly-L-lactid, osteosynthetic device, finite element method

Introduction

Osteosynthetic devices, such as miniplates and screws, are presently used to repair fractured human skeletal bone, and stainless steel and titanium are generally used to these metallic devices. However there are disadvantages in the use of these metallic devices, for example, corrosion, carcinogenicity for long-term implantation or osteoporosis beneath the miniplates due to stress shielding[4]. In order to overcome these disadvantages, bioabsorbable, ultra high strength poly-L-lactid osteosynthetic devices were developed, and have been applied to the treatment of bone fractures[1,2,3)]. We have also used these devices in the treatment of maxillofacial trauma, and successful fracture healing was obtained. On the other hand, it is not known how much stress is actually generated on these devices, when an external stress was added and minor segment was displaced. In order to know the stress distribution generated in the devices, using 3-dimensional-CAD software, we created the solid model of these devices and the blocks, and assembled these parts just as the two blocks were fixed with these devices. We simulated of fixed one block in the two blocks, and moved a constant amount in four directions to the other block, and analyzed the stress generated in these devices using 3-dimensional stress analysis software. We report these methods and the results of these simulations.

Materials and Methods

The simulations were performed on the computer. For the model creation software, 3-dimensional-CAD soft SolidWorks2000 (Developing agency : SolidWorks : U.S., domestic selling agency : Itochu Techno-Science Corporation) and as the analysis software, COSMOSWorks6.0 (Developing agency : Structural Reseach & Analysis Corporation : U.S., domestic selling agency : Yokogawa Techno-Information Service Inc.) were used. The plate as the device was made referring to the blueprint in the manufacturing origin. These parts were assembled just as the two blocks were fixed with these devices. Automatic division generation of each element was performed for this analysis using a tetrahedron element (Fig.1). The contact part of the plate and the screw was set not to unite. Full restraint of the forefront side of the aluminum block was set. The displacement of four directions was set to other aluminum block as follows Fig.2). Case 1 is in the opposite direction. Case 2 is in the lateral direction 1mm. Case 3 is in the downward 1mm. Case 4 is to set the 1mm upward displacement to one side of the aluminum block and the 1mm downward displacement to another side of the block so that the plate was twisted. For the analysis, the manufacturer's data of Young's module and Poisson's ratio data on the PLLA was used (Table 1). The static analysis was performed.

Results

In four simulations, when the block was stretched in the opposite directionn (Case1), the biggest stress was generated in the plate. The second big stress was generated when horizontally moving it (CCase2), and the third big stress was generated when moving it to the vertical direction (Case3). When twisting (Case4), the generation of the stress was the smallest(Fig.3)

Discussion

In order to examine the reliability of this simulation, we created a model of the same form as the tensile test on the computer, and then a simulation under the same conditions as the tensile test of PLLA plate was performed. We compared the results of the tensile test with the simulation, and investigated the fundamental relationship between displacement and load. The load-displacement curve obtained from the tensile and compression tests correlated well with the simulation data, and supported the validity of the simulation[4)].

Although a PLLA block is created by stretch and has initial crys-

---

Correspondence to Tomoaki Kawamoto:Department of Oral and Maxillofacial Reconstructive Surgery, Okayama University Graduate School of Medicine and Dentistry.2-5-1 Shikata, Okayama, Okayama 700-8525, JAPAN,E-mail: tom_kat@md.okayama-u.ac.jp

tallinity, it also has bend intensity. The device is manufactured by cutting this block, and is generally accepted that a stretched object has an anisotropic character. In order to solve this problem, a forging method was used. The anisotropic character of the PLLA block increased, and a high intensity and strong viscoelasticity block was manufactured.

In the case when the plate bends by translating the block, the generation of the stress was smaller than the stretched case. When the block was stretched, the biggest stress was generated in the manufactured plate. It is shown that it is difficult to stretch it though it is easy to transform this to bend the plate. However, the big stress that destroy the plate was not generated. This shows that this plate has enough strength as an osteosynthetic device.

It may be necessary to develop the material that doesn't cause the transformation in the plate when external stress that bend the plate is given to it.

Conclusion

In cases of four all, the stress as the plate and the screw were destroyed had not been generated.

It is thought that the PLLA plate and the PLLA screw have enough strength as an osteosynthetic device.

Reference

1) Herrmann JB, Kelly RJ and Higgins GA. Polyglycolic acid sutures. Laboratory and clinical evaluation of a new absorbable material, Arch. Surg., 100, 486-490, (1970).
2) Cutright DE and Hunsuck EE. "Fracture reduction using a biodegradable material, polylactic acid," J. Oral Surg., 29, 393-397, (1971).
3) Vert M, Christel P, Chabot F, and Leray J. Bioresorbable plastic materials for bone surgery. In: Hastings GW and Ducheyne PEd. Macromolecular Biomaterials. Boca Raton, Florida: CRC Press, 119-141, (1984)
4) Kawamoto T and Sugahara T. Properties of Bioabsorbable Ultra High Strength Poly-L-lactid(PLLA) Device. Proceedings of 7th Japan International SAMPE Symposium & Exhibition, 455-458, (2001)

Table 1 Material Properties

	Young's Module	Poisson's Ratio
PLLA(Compress)	1,500Mpa	0.35
PLLA(Tensile)	2,075Mpa	0.35
Stainless Steel	210,000Mpa	0.28
Aluminum	69,000Mpa	0.33

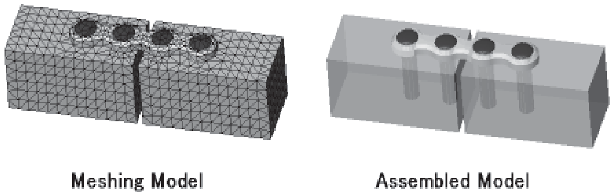

Fig.1 Assembled Model and Meshing Model of this Examination

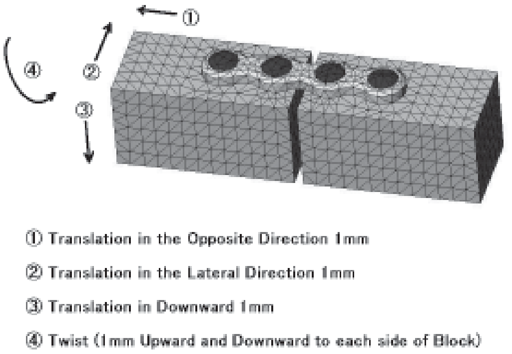

Fig.2 Displacement of Four Directions

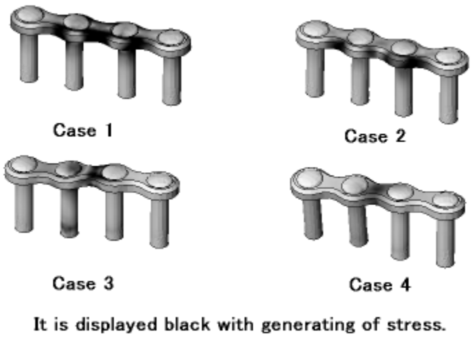

Fig.3 Distribution of vonMises Stress of the Devices