
Modelling of stress and deformation distribution around endosteal implants

S. Konvičková¹, A. Kácovský¹, Z. Štrnad²

Quintessenz, Vol. 8, No. 12, 1999

The study uses a finite elements analysis to determine of the stress and deformation distribution in bone a around dental implant. Two types of implant shape were considered in the calculation – screw and cylinder, 12mm in length and 3,6mm in diameter. For each implant type three models of bone-implant interface were considered – 1. osseointegrated bioactive implant; 2. direct contact of a bioinert material with bone without bonding; 3. nonosseointegrated bioinert implant without a direct contact to bone (fibrous tissue encapsulation). For modelling we used a jaw segment corresponding to the position of a second molar. The implant was loaded by systematic forces. By calculation, equivalent stress and

deformation values in the bone around the cervical section of the implant were determined. The model of osseointegrated bioactive implant (1) exhibited significantly lower maxima compared to that in the other two model types. Regarding the shape of the implant, a screw-like design proved to be more suitable than a cylindrical one.

¹ Faculty of Mechanical Engineering, Department of Mechanics, CVUT, Prague, Czech Republic

² Lasak Ltd., Papirenská 25, Prague, Czech Republic