Constitutive modeling of medical grade ultra-high molecular weight polyethylene

D.J.A. Senden

Master external assessment, MT08.07
# Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contents</td>
<td>i</td>
</tr>
<tr>
<td>Abstract</td>
<td>ii</td>
</tr>
<tr>
<td>1 Introduction</td>
<td>1</td>
</tr>
<tr>
<td>2 Modeling</td>
<td>3</td>
</tr>
<tr>
<td>2.1 Hertzian theory</td>
<td>3</td>
</tr>
<tr>
<td>2.2 Linear elasticity</td>
<td>4</td>
</tr>
<tr>
<td>2.3 Nonlinear constitutive model</td>
<td>5</td>
</tr>
<tr>
<td>3 Materials and Methods</td>
<td>8</td>
</tr>
<tr>
<td>3.1 Experimental setup</td>
<td>8</td>
</tr>
<tr>
<td>3.2 Numerical methods</td>
<td>10</td>
</tr>
<tr>
<td>4 Model Characterization</td>
<td>12</td>
</tr>
<tr>
<td>4.1 Results of uniaxial compression experiments</td>
<td>12</td>
</tr>
<tr>
<td>4.2 Material parameters</td>
<td>13</td>
</tr>
<tr>
<td>4.3 Relaxation spectrum</td>
<td>16</td>
</tr>
<tr>
<td>4.4 Comparison of material models in uniaxial compression</td>
<td>20</td>
</tr>
<tr>
<td>5 Flat Plate Indentation</td>
<td>21</td>
</tr>
<tr>
<td>5.1 Experimental verification</td>
<td>21</td>
</tr>
<tr>
<td>5.2 Investigating error sources</td>
<td>22</td>
</tr>
<tr>
<td>5.3 Influence of specimen thickness</td>
<td>23</td>
</tr>
<tr>
<td>5.4 Validity of Hertzian theory</td>
<td>24</td>
</tr>
<tr>
<td>6 Cup Indentation</td>
<td>26</td>
</tr>
<tr>
<td>6.1 Comparison of material models</td>
<td>26</td>
</tr>
<tr>
<td>6.2 Influence of minimum thickness</td>
<td>27</td>
</tr>
<tr>
<td>7 Conclusions</td>
<td>28</td>
</tr>
<tr>
<td>Acknowledgements</td>
<td>29</td>
</tr>
<tr>
<td>Bibliography</td>
<td>30</td>
</tr>
<tr>
<td>A Flat Plate Indentation Results</td>
<td>33</td>
</tr>
</tbody>
</table>
Abstract

Ultra-high molecular weight polyethylene is widely used in orthopaedic implants as a bearing material. An accurate prediction of the stress-strain behavior of these tibial- and hip inserts is of vital importance, since failure mechanisms, wear and creep, are strongly affected by the stress state of the material. In this finite element study, three constitutive models - linear elasticity, single- and multi mode nonlinear viscoelasticity - were compared for their respective abilities to predict the three-dimensional mechanical behavior of UHMWPE in ball head indentation, both on a flat plate and in a cup. Experimental verification was performed for the flat plate indentation. Uniaxial compression experiments were used to estimate the material parameters in each constitutive model. Due to its accurate description of the pre-yield regime, the multi mode model offers the most accurate stress predictions, especially in the medically relevant loading range.

This study was performed during an internship at the Swiss Federal Institute of Technology (ETH), Zürich, in the Polymer Technology group of prof.dr. P. Smith. It is a part of a master program in Mechanical Engineering at the Eindhoven University of Technology.
Chapter 1

Introduction

During the second part of the 20th century, total joint arthroplasty evolved into the widely accepted practice it is today; it successfully helps to restore the mobility of people with severely diseased joints. In contemporary hip prostheses, a metal or ceramic ball head articulating against a polymeric bearing surface is by far the most widely used bearing couple, although other options are available, such as: metal-on-metal and ceramic-on-ceramic bearings [1]. Ever since it was introduced to the orthopaedic society by a manufacturer of plastic gears in 1962, ultra-high molecular weight polyethylene (UHMWPE) has been the material of choice for the polymeric bearing surface [2]. Despite their widespread use, UHMWPE implants suffer from wear, limiting the longevity of the components and causing the generation of large amounts of submicron particles, ultimately leading to loosening of the implant [1]. Consequently, many researchers have investigated which mechanisms dictate the wear behavior of UHMWPE components [1, 3, 4]. An important focus in these studies is the influence of the stress state in these components. It has been found that wear is enhanced by high, localized stresses and strains at the articulating surface, the mechanism of which is mainly governed by the large-strain mechanical behavior of the material [5, 6, 7]. Plastic deformation, more specifically the accumulation of plastic strain, was also shown to have a pronounced influence on the generation of wear debris [7, 8]. More in general it was commented that the stress state is very complex and that different stress modes (e.g. compression, tension, shearing) contribute to different wear-, fatigue- and failure mechanisms [7, 9]. From these conclusions, the need for a method to accurately assess the stress state in UHMWPE components becomes evident. Since it is generally very cumbersome or simply impossible to measure these stresses experimentally, constitutive material modeling and computational techniques are used to predict them [6, 10, 11, 12]. Not only the material behavior has a clear influence on the stress state, it is also substantially affected by geometric characteristics, such as: component design, minimum thickness of the material and the level of conformity of the articulating surfaces [9, 13, 14], and loading conditions, e.g. rim loading.

Although more realistic material models are available, linear elasticity is still the most widely used in orthopaedic implant design practice to numerically predict the mechanical response of UHMWPE components, mainly because of its straightforward implementation in finite element analysis (FEA) and low computational costs. Also in the scientific community this material model has often been employed in numerical studies [13, 14, 15], occasionally extended to a bilinear or quadrilinear elastic model [9, 14, 16]. One of the few finite element studies involving an advanced viscoelastic material model was conducted by Bergström [17].

In this work, a nonlinear viscoelastic constitutive model, originally designed to describe the mechanical behavior of amorphous polymers, is used in combination with finite element analysis to describe and predict the mechanical response of UHMWPE components. This material model, introduced by Govaert [18] and Klompen [19], is slightly modified because of the absence of intrinsic strain softening in the behavior of UHMWPE; possible physical ageing effects are not taken
into account. Recently, the single mode model described by Klompen is extended to a multi mode model, as described by Tervoort [20], enabling a more accurate description of the nonlinear viscoelastic pre-yield regime in the mechanical response [21]. Both versions of the constitutive model are used and their respective abilities to describe the stress-strain behavior of UHMWPE are compared.

The objective of this study is to compare different computational approaches to predict the deformation behavior of UHMWPE components: analytical Hertzian theory, linear elastic FEA, single- and multi mode nonlinear viscoelastic FEA. The test geometry used is the indentation of a ball head in a flat plate of UHMWPE, employing a deformation cycle consisting of both loading and unloading. An experimental verification is performed in order to assess the predictive power of the various material models. Finally, a cup geometry is chosen to investigate the performance of the computational approaches in a more realistic setting.
Chapter 2

Modeling

2.1 Hertzian theory

Hertzian theory was applied to obtain a first-order approximation of the forces, displacements and contact stresses that can be expected in the UHMWPE specimens when they are loaded with a ceramic ball. This theory presents an analytical approach to calculate the force required to indent a specimen made of linear elastic material with a rigid spherical indenter for a given displacement. It also predicts the contact stress resulting from this indentation.

The following equations are taken from the account of Hertzian theory given by Fischer-Cripps [22, 23] for a force-controlled experiment. First, the relative curvature $R$ of the indenter and the specimen is calculated:

$$\frac{1}{R} = \frac{1}{R_{\text{ind}}} + \frac{1}{R_{\text{cup}}},$$  \hspace{1cm} (2.1)

where $R_{\text{ind}}$ and $R_{\text{cup}}$ are the radii of the indenter and the specimen, respectively. For a flat specimen, the latter is taken equal to infinity. For a concave specimen (e.g. a cup geometry) a negative value is taken for $R_{\text{cup}}$. The elastic mismatch factor $k$ is defined as:

$$k = \frac{9}{16} \left(1 - \nu^2\right) + \frac{E}{E_{\text{ind}}} \left(1 - \nu_{\text{ind}}^2\right),$$  \hspace{1cm} (2.2)

where the Young’s modulus $E$ and the Poisson’s ratio $\nu$ are specified for both the specimen and the indenter, the latter being indicated by the subscript ‘ind’. For a perfectly rigid indenter ($E_{\text{ind}} = \infty$), this equation reduces to:

$$k = \frac{9}{16} \left(1 - \nu^2\right).$$  \hspace{1cm} (2.3)

The radius of the circle of contact $a$ is defined by:

$$a^3 = \frac{4 kF R}{3 E},$$  \hspace{1cm} (2.4)

where $F$ denotes the indentation force applied to the indenter. The final step is the calculation of the maximum displacement $u$ of the indenter into the specimen:

$$u = \frac{9 k F R}{4 E},$$
\[ u = \frac{1 - \nu^2}{E} \frac{3F}{4a}. \]  

(2.5)

As mentioned before, it is in this case desired to calculate the force and contact stress for a given displacement. The equations must therefore be rewritten. Substitution and rearrangement of the previous equations, yields for the indentation force:

\[ F^2 = \frac{R}{\left(\frac{4k}{3E}\right)^2} u^3. \]  

(2.6)

The maximum contact stress is then given by:

\[ \sigma = \frac{3F}{2 \pi a^2}. \]  

(2.7)

### 2.2 Linear elasticity

Today, linear elastic material behavior, the simplest material model available, is still used by manufacturers of knee- and hip implants to calculate the stresses that occur in the tibial- or hip inserts in these devices. In general, finite element analysis is used, enabling an accurate description of the actual geometry and calculation of the full, three-dimensional stress state. Although strictly limited to small strains, this is a powerful method which has also been used by various researchers in this field [13, 14, 15], some extended the material model to a piecewise linear one (bilinear or quadrilinear) [9, 14, 16].

Hooke’s law defines linear elastic material behavior in one dimension as a linear relationship between stress and strain. For an isotropic material in three dimensions, this can be written as a tensorial relation between the Cauchy stress tensor \( \sigma \) and the true (logarithmic) strain tensor \( \varepsilon = \ln(U) \), where \( U \) represents the stretch tensor:

\[ \sigma = \kappa \text{tr}(\varepsilon) + 2G\varepsilon^d. \]  

(2.8)

The elastic material constants \( \kappa \) and \( G \) represent the bulk modulus and the shear modulus, respectively. These may be expressed in terms of two other elastic material constants, the Young’s modulus \( E \) and Poisson’s ratio \( \nu \):

\[ \kappa = \frac{E}{3(1 - 2\nu)} \]  

(2.9)

\[ G = \frac{E}{2(1 + \nu)}. \]  

(2.10)

Linear elastic isotropic material behavior is uniquely defined by two elastic constants, for instance \( E \) and \( \nu \).
2.3 Nonlinear constitutive model

In this section, both the single mode and the multi mode versions of the nonlinear constitutive model are discussed together, since their descriptions are very similar. The one-dimensional mechanical analogs for both models are depicted in Figure 2.1. The models consist of two parallel parts, one describing the contribution of strain hardening, the other describing the yield behavior. The former is modeled with a neo-Hookean spring (blue part), the latter with one (single mode) or a number of parallel (multi mode) Maxwell spring-dashpot elements (red part).

Figure 2.1: Mechanical analog (1D) of the nonlinear viscoelastic constitutive model for the single mode (left) and the multi mode (right) version.

Kinematics

Using the stress-free intermediate state, postulated by Leonov [24], the total deformation gradient tensor $\mathbf{F}$ is multiplicatively decomposed into an elastic and a plastic part, denoted by the subscripts ‘e’ and ‘p’, respectively:

$$\mathbf{F} = \mathbf{F}_e \cdot \mathbf{F}_p.$$  \hfill (2.11)

This result can be substituted in the calculation of the velocity gradient tensor $\mathbf{L}$, yielding an additive decomposition of this tensor into an elastic and a plastic part:

$$\mathbf{L} = \mathbf{\dot{F}} \cdot \mathbf{F}^{-1} = \mathbf{\dot{F}}_e \cdot \mathbf{F}_p^{-1} \cdot \mathbf{F}_e^{-1} + \mathbf{\dot{F}}_p \cdot \mathbf{F}_p^{-1} \cdot \mathbf{F}_e^{-1} = \mathbf{L}_e + \mathbf{L}_p.$$ \hfill (2.12)

Both the elastic and plastic velocity gradient tensors can be split into a symmetric and a skew-symmetric part, yielding the deformation rate tensor $\mathbf{D}$ and spin tensor $\mathbf{\Omega}$, respectively. Boyce [25] showed that taking the plastic spin tensor $\mathbf{\Omega}_p$ equal to the null tensor is a good assumption and that it makes the decomposition of Equation 2.11 unique.

$$\mathbf{L}_e = \mathbf{\dot{F}}_e \cdot \mathbf{F}_e^{-1} = \mathbf{D}_e + \mathbf{\Omega}_e$$ \hfill (2.13)

$$\mathbf{L}_p = \mathbf{F}_e \cdot \mathbf{\dot{F}}_p \cdot \mathbf{F}_p^{-1} \cdot \mathbf{F}_e^{-1} = \mathbf{D}_p + \mathbf{\Omega}_p = \mathbf{D}_p$$ \hfill (2.14)
Furthermore, incompressible plastic deformation is assumed, yielding a volume change ratio $J$ that can be expressed in only the elastic part of the deformation gradient:

$$J = \det(F) = \det(F_e).$$  \hfill (2.15)

An isochoric equivalent of both $F$ and $F_e$ can now be defined:

$$\tilde{F} = J^{-1/3}F; \quad \tilde{F}_e = J^{-1/3}F_e.$$  \hfill (2.16)

The (elastic) isochoric left Cauchy Green tensor is used as a strain measure in the constitutive model and is calculated according to:

$$\tilde{B} = \tilde{F} \cdot \tilde{F}^c; \quad \tilde{B}_e = \tilde{F}_e \cdot \tilde{F}_e^c.$$  \hfill (2.17)

**Stresses**

The total Cauchy stress tensor is additively decomposed into a driving stress $\sigma_s$ and a hardening stress $\sigma_r$, corresponding with the red and the blue part of the model (see Figure 2.1), respectively:

$$\sigma = \sigma_s + \sigma_r.$$  \hfill (2.18)

The contribution of the hardening stress is modeled with a neo-Hookean relation [26], governed by the hardening modulus $G_r$:

$$\sigma_r = G_r \tilde{B}^d.$$  \hfill (2.19)

The driving stress is modeled with a single or a number of parallel Maxwell spring-dashpot elements, depending on whether a single mode or a multi mode approach is used. Here, the general case will be treated in which the model consists of $n_m$ modes. In this case, the total driving stress is calculated by summing all modal driving stress contributions $\sigma_{s,m}$. Furthermore, these modal contributions are split into a hydrostatic and a deviatoric stress tensor:

$$\sigma_s = \sum_{m=1}^{n_m} \sigma_{s,m} = \sum_{m=1}^{n_m} (\sigma_{s,m}^h + \sigma_{s,m}^d).$$  \hfill (2.20)

Following Baaijens [27], the modal hydrostatic and deviatoric stress contributions are defined as:

$$\sigma_{s,m}^h = \kappa (J - 1) I$$  \hfill (2.21)

$$\sigma_{s,m}^d = G_m \tilde{B}_{s,m}^d.$$  \hfill (2.22)
in which $G_m$ denote the modal shear moduli. Because of its viscoelastic nature, the constitutive model is both history- and time-dependent. Therefore, changes in volumetric and elastic strains are governed by the following evolution equations (see [21] for derivations of these relations):

\[ \dot{J} = J_{\text{tr}} (D) \quad (2.23) \]

\[ \dot{B}_{\text{e}m} = \left( \dot{L} - D_{\text{p}m} \right) \cdot B_{\text{e}m} + B_{\text{e}m} \cdot \left( \dot{L}^c - D_{\text{p}m} \right). \quad (2.24) \]

For each mode, the modal plastic deformation rate tensor $D_{\text{p}m}$ is related to $\sigma_{\text{sm}}^d$ by a non-Newtonian flow rule which is governed by modal viscosities $\eta_m$ that depend on (equivalent) stress $\bar{\tau}$, pressure $p$ and temperature $T$:

\[ D_{\text{p}m} = \frac{\sigma_{\text{sm}}^d}{2\eta_m (\bar{\tau}, p, T)} = \frac{G_m}{2\eta_m} \tilde{B}^d_{\text{e}m}. \quad (2.25) \]

The modal viscosities $\eta_m$ are modeled as Eyring viscosities, modified to incorporate pressure dependence, as proposed by Govaert [18]. In this case, the mechanical behavior of UHMWPE is modeled only at a temperature of $37^\circ$C. The temperature dependence of the viscosities is therefore not taken into account and the contribution of the temperature term in the viscosity definition is simply incorporated in the initial viscosities:

\[ \eta_m = \eta_{0m} \exp \left[ \frac{\Delta H}{RT} + \frac{\mu p}{\tau_0} \frac{\bar{\tau}}{\sinh \left( \frac{\bar{\tau}}{\tau_0} \right)} \right] = \eta_{0m} \exp \left[ \frac{\mu p}{\tau_0} \frac{\bar{\tau}}{\sinh \left( \frac{\bar{\tau}}{\tau_0} \right)} \right]. \quad (2.26) \]

The pressure dependence parameter $\mu$ and the characteristic stress $\tau_0$ are model parameters, the initial modal viscosities are denoted by $\eta_{0m}$. The universal gas constant $R$ and the activation energy $\Delta H$ only play a role when the temperature dependence would be taken into account.

The equivalent stress $\bar{\tau}$ and pressure $p$ are defined as:

\[ \bar{\tau} = \sqrt{\frac{1}{2} \sigma_s^d : \sigma_s^d} \quad (2.27) \]

\[ p = -\frac{1}{3} \text{tr} (\sigma). \quad (2.28) \]
Chapter 3
Materials and Methods

3.1 Experimental setup

Uniaxial compression

In order to determine the parameters of the constitutive model for UHMWPE and thereby characterizing the nonlinear viscoelastic behavior of the material, a series of uniaxial compression tests were performed. These experiments were executed on a servo-hydraulic MTS Elastomer 810 Testing System that was equipped with a temperature control chamber to maintain a constant ambient temperature of 37°C. The setup consisted of two parallel metal plates between which the specimens were compressed. Friction was reduced by using highly polished metal plates and applying both PTFE tape and PTFE spray between the contact surfaces. The (finite) stiffness $k$ of the machine was measured and corrected for by calculating the true displacement $u$ as follows:

$$u = u' - \frac{F}{k},$$

(3.1)

where $u'$ is the recorded displacement and $F$ the corresponding indentation force measured by the load cell.

Two series of specimens were used, both made from medical grade GUR 1020 UHMWPE. One grade, however, consisted of conventional GUR 1020, whereas the other was highly crosslinked by using 7 Mrad e-beam radiation and thermal posttreatment. The latter material grade is commercially known as RexPol®. All specimens were cylindrically shaped with both the height and the diameter equal to 6 mm. The test specimens were machined from pucks, cut from compression molded sheets (Meditech, Vreden, Germany); the axis of the cylinders was always perpendicular to the compression molding direction.

A series of experiments was performed at constant true strain rates ranging between $10^{-1}$ and $10^{-5}$ s$^{-1}$; three samples were measured at each strain rate. The specimens were preconditioned at 37°C for 15 min and a maximum of 70% true strain was applied. From the force and (true) displacement signals measured by the machine, true strain and true stress were calculated, the latter with an incompressibility assumption:

$$\varepsilon_t = \ln (\lambda) = \ln \left( \frac{L}{L_0} \right) = \ln \left( \frac{L_0 - u}{L_0} \right),$$

(3.2)

$$\sigma_t = \frac{F}{A} = \frac{FL}{V_0} = \frac{FL}{A_0L_0} = \frac{F\lambda}{A_0}.$$  

(3.3)
Herein represent $\varepsilon_t$, $\sigma_t$ and $\lambda$ the true strain, true stress and draw ratio, respectively. Specimen length, cross-sectional area and volume are denoted by $L$, $A$ and $V$, respectively, the subscript '0' indicating that it is the initial value of the parameter.

**Ball head indentation**

To be able to assess the predictive power of the different computational approaches, a more complex loading geometry was chosen to experimentally verify the numerical predictions: ball head indentation on a flat plate.

An axisymmetric test setup was chosen to allow for two- instead of three-dimensional FEA simulations, hereby saving dramatically on computational costs. Furthermore, the experiments consisted only of short-term, non-cyclic, indentation-type loading along the axis of rotational symmetry. The indenter was chosen to be a ceramic ball head of the type that is also used in actual hip prostheses (Plus Orthopedics BIOLOX® forte Al$_2$O$_3$), with a radius of 14 mm. The specimens were cylindrically shaped, having a radius of 30 mm and a thickness of either 3, 6 or 9 mm. In these experiments, only specimens made from the conventional grade of GUR 1020 UHMWPE were used. Friction between the specimen and the indenter was reduced by spraying a thin graphite layer on the specimen and applying additional lubrication in the form of glycerol between the contact surfaces.

The test setup that was used is shown in Figure 3.1. A Zwick Z020 tensile/compression tester was used in combination with a temperature control chamber to maintain an ambient temperature of 37°C.

![Figure 3.1: Experimental setup for the ball head indentation tests with a spherical ceramic indenter and a flat specimen.](image)

After preconditioning at 37°C for 2 hours, the sample was placed and aligned in the tensile tester. Subsequently, the position of the surface was determined manually by slowly increasing the displacement until a compressive force of 2 N was reached. The indenter was then retracted and subsequently given a constant indentation speed of 0.5 mm/s, driving the ball head into the material. The maximum indentation depth (measured from the manually determined position of
the surface) was 0.6 mm in all cases. When the maximum depth was reached, retraction of the indenter was done with a speed of 0.05 mm/s. During both the indentation and the retraction phase, the test time, displacement and indentation force were recorded. The procedure used to correct the displacement data for the finite machine stiffness was entirely analogous to the one described for the uniaxial compression experiments.

3.2 Numerical methods

Mesh definition

The first step in setting up a finite element simulation is the construction of a mesh, in this case a two-dimensional one because of the axisymmetric nature of the simulated experiment. A vital requirement is that the mesh itself does not influence the results of the simulations. Verification of this was done by stepwise refining the mesh (i.e. decreasing the element size) until the results converged to a stable solution. In accordance with Bartel [16], convergence was defined to have been reached when a change of less than 2% was observed between two subsequent meshes. During this procedure it can clearly be observed that global variables (e.g. the indenter force) already exhibit convergence with a much coarser mesh than local variables (e.g. local stress components in a particular node). This is due to the fact that global variables experience a form of averaging over (a part of) the mesh.

For the flat plate indentation simulations, three different meshes were made, corresponding with the three thicknesses that were adopted in the tests: 3, 6 and 9 mm. To reach convergence of local variables in this type of simulation is a cumbersome task because of the extremely localized stresses and strains (the initial contact area is in principle infinitely small), a conclusion which can also be drawn from the work by van Melick [28]. For the experimental verification, however, only the global indentation force and -displacement are of interest and the local stresses and strains are actually not that relevant in our case. The meshes used in these simulations are therefore refined to such an extent that convergence was reached only for these (global) variables. An example of the mesh is shown in Figure 3.2, for a thickness of 3 mm. The indenter (also shown in the figure) is modeled as a rigid body, which is justified because of the large difference in modulus between the ceramic ball head and the UHMWPE specimen.

![Figure 3.2: Finite element meshes used in the actual simulations with the indenter modeled as a rigid body. a. The indenter radius is 14 mm for the flat geometry (left). b. The indenter radius is 18 mm for the cup geometry (right). The x-axis (horizontal) is the axis of rotational symmetry.](image-url)
In this numerical investigation, simulating the previously described ball head indentation test on a flat plate in order to perform a numerical-experimental verification was not the only objective. An axisymmetric ball-cup geometry, depicted in Figure 3.2b, was also studied, the dimensions of which were selected such that it is reminiscent to hip cups used in contemporary hip prostheses. The geometry consisted of a cylinder with an outer radius of 30 mm, where a cup with the shape of a half-sphere ($R_{cup} = 18.1$ mm) was machined into one of the flat sides. The minimum thickness of the geometry, i.e. the thickness along the axis of rotational symmetry, was either 3, 6 or 9 mm. Indentation of this geometry was done with a ball head having a radius different from the one used with the flat plates: 18 mm. In the practice of hip prosthesis design, it is very important to know the (maximum) contact stresses that occur between the articulating parts of the device. Adding more material between the cup and the opposing face will favorably lower these stresses, but result in larger prostheses, leading to higher bone loss upon implantation. In practice, 6 mm is used as a compromise. Since contact stresses are difficult to measure accurately, designers usually rely on predictions based on (linear elastic) finite element simulations. To be able to compare the contact stress predictions of the different material models described in Chapter 2, a mesh was needed that showed convergence of the contact stresses. The conformity between the indenter and the cup ensured a much larger contact area than with the flat plate indentation, resulting in less localized stresses. In Figure 3.2, the mesh with a minimum thickness of 3 mm is shown.

Simulations

The finite element software package MSC.Marc Mentat (version 2005r3) was used to perform the simulations; the linear elastic material model is standardly incorporated herein. For the nonlinear viscoelastic simulations, an implementation of the constitutive model into the Marc Mentat HYPELA2 subroutine (van Breemen [21, 29]) was used. The parameters of the constitutive model that actually define the mechanical behavior were determined by fitting the model on experimental uniaxial compression data. This is discussed in the next chapter, where also attention is given to the parameters that are used in both the single mode and the linear elastic models.

The boundary conditions that are required to define the rotational symmetry of the simulation, were automatically applied by the software. Furthermore, the displacement of every node on the bottom surface of the specimen was constrained in axial direction. The contact between the indenter and the specimen was modeled without friction. The meshes all consisted of linear quadrilateral elements.

In accordance with the actual ball head indentation experiments, the simulations are characterized by a displacement-controlled deformation. To ensure that the applied displacement (as a function of time) in the simulations is exactly the same as in the experiments, the measured $u(t)$ curve (corrected for the machine stiffness) was used as input for the simulations.
Chapter 4

Model Characterization

In this chapter, all model input parameters for the multi mode nonlinear constitutive model are determined from the results of the uniaxial compression experiments. The parameters for the single mode model are exactly the same, with the exception that the shear modulus $G$ and the initial viscosity $\eta'_0$ were equal to the sum of all modal shear moduli $G_m$ and initial viscosities $\eta'_{0,m}$ respectively. For the linear elastic material model, the Young’s modulus $E$ was selected equal to the initial Young’s modulus $E_0$ determined for the multi mode model. The Poisson’s ratio $\nu$ was assumed to be equal to 0.46, following Kurtz [7]. First, the results from the uniaxial compression experiments are discussed.

4.1 Results of uniaxial compression experiments

The results of the uniaxial compression experiments, described in Chapter 3, are presented as true stress - true strain curves in Figure 4.1. Note that measurements at a true strain rate of $10^{-5}$ s$^{-1}$ were only performed for the conventional UHMWPE. Three measurements were performed at every strain rate, the results showing excellent reproducibility at small to moderate true strains (up to around 50%). Furthermore, a significant influence of the strain rate on the mechanical response is apparent; the stress increases with increasing deformation rate. Of the two yield points that are present in the material response, only the first will be taken into account in the modeling of the behavior, since the strains that actually occur in knee- and hip prostheses are not expected to exceed the first yield point.

In Figure 4.2, the mean true stress - true strain curves are depicted in one graph to enable an easy comparison of the two materials. There is a significant discrepancy between the mechanical responses of the two materials and the differences between them appear to be inconsistent across the strain rate domain. The most obvious difference that can be observed, is the lower initial modulus of the highly crosslinked material. This lower stiffness is important for practical applications in the elastic regime at small deformations. However, in the rest of this work only the mechanical behavior of conventional UHMWPE will be discussed.
4.2 Material parameters

The material parameters that enter the model and that, therefore, need to be determined experimentally, are: the bulk modulus $\kappa$, the hardening modulus $G_r$, the (equivalent) characteristic stress $\tau_0$ and the pressure dependency parameter $\mu$.

The bulk modulus was assumed constant for all modes in the constitutive model and was calculated according to:
\[ \kappa = \kappa_0 = \frac{E_0}{3(1-2\nu_0)}. \] (4.1)

The initial Young’s modulus \(E_0\) was determined from the uniaxial compression measurements as the initial value of the first derivative of the true stress with respect to the true strain, measured at the highest strain rate. The initial, elastic Poisson’s ratio \(\nu_0\) was selected as 0.46, after Kurtz [7].

The hardening modulus that is employed in the constitutive model was determined in the same way as it was originally done by Klompen [19]. The method consists of plotting the true stress as a function of the strain measure \(|\lambda^2 - \lambda^{-1}|\), where \(\lambda\) is the draw ratio, and then fitting a linear function on the large strain regime of the curve. The slope of that curve is the hardening modulus \(G_r\). In Figure 4.3, where the procedure is visualized, it should be noted that in this case the hardening modulus was fitted on the part of the curve after the first yield point. It was already mentioned that the second yield point is not taken into account in the modeling of the material behavior.

![Figure 4.3: Visualization of the procedure to determine the hardening modulus \(G_r\).](image)

The method used to determine the characteristic stress \(\tau_0\) is also the same as presented by Klompen [30]. This parameter is a measure of the strain rate dependency of the yield stress, and calculation of its value thus requires the calculation of the yield stress, which is not trivial for UHMWPE. Kurtz [7, 31] determines the yield stress with the 0.2%-strain offset method and argues that this value bears a good correspondence with microscopic plastic deformation phenomena. In this implementation of the constitutive model, however, the yield stress is macroscopically regarded as the point where the strain hardening contribution to the material response becomes dominant over the viscoelastic contribution. For this reason, the yield stress is defined as the intersection of two linear fits: one in the initial viscoelastic regime, the other in the strain hardening regime. This method is visualized in Figure 4.4, where also the yield stress according to the 0.2%-strain offset method is depicted.
Figure 4.4: The yield stress of UHMWPE is defined as the intersection of two linear fits (dotted lines) and is indicated by the red circle. As a comparison, the red cross indicates the yield stress according to the 0.2%-strain offset method.

In Figure 4.5, the true yield stresses are plotted against the logarithm of the true strain rate. A linear fit (with respect to the logarithm of the true strain rate) was made through these data points, the slope of which is the characteristic stress $s_0$. The material parameter $\tau_0$ is an equivalent stress measure, which is why $s_0$ needs to be converted. In addition to that, a strain rate correction was required [30]. Combining both operations, $\tau_0$ was calculated from $s_0$ according to:

$$\tau_0 = \frac{\sqrt{3} - \mu}{3\ln(10)} s_0.$$  \hfill (4.2)

Figure 4.5: The yield stress (indicated with red triangles) linearly depends on the logarithm of the strain rate. The slope $s_0$ of the linear fit (dashed line) is a measure for the (equivalent) characteristic stress $\tau_0$. 
The pressure dependency of the material cannot be assessed using only uniaxial compression measurements; it follows, for instance, from a comparison of the material behavior in different loading geometries. For this reason, the pressure dependency will be neglected, i.e. \( \mu = 0 \). The influence of this assumption on the final results will be checked in Chapter 5.

Concluding this section, the values for the different material parameters are listed in Table 4.1.

<table>
<thead>
<tr>
<th>Material</th>
<th>( E_0 ) [MPa]</th>
<th>( \nu ) [-]</th>
<th>( \kappa ) [MPa]</th>
<th>( G_r ) [MPa]</th>
<th>( \tau_0 ) [MPa]</th>
<th>( \mu ) [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>UHMWPE</td>
<td>820</td>
<td>0.46</td>
<td>3417</td>
<td>12</td>
<td>0.78</td>
<td>0</td>
</tr>
</tbody>
</table>

### 4.3 Relaxation spectrum

As described in Chapter 2, the contribution of the driving stress is modeled by using a series of parallel Maxwell elements, a multi mode approach. The constitutive behavior of these elements is characterized by a discrete relaxation spectrum which needs to be determined, i.e. the shear modulus \( G_m \) and initial viscosity \( \eta_{0_m} \) must be defined for each mode. In the following, the strategy that was used to obtain this discrete relaxation spectrum is described.

Point of departure is the experimentally obtained true stress - true strain relation for uniaxial compression (i.e. \( \sigma(t) \) and \( \varepsilon(t) \) are known), from which first the strain hardening contribution is removed, such that only the driving stress remains \([19]\):

\[
\sigma_s = \sigma - G_r \left( \lambda^2 - \lambda^{-1} \right).
\] (4.3)

The nonlinear viscoelastic behavior of a single Maxwell element can be described by a Boltzmann integral to calculate the uniaxial stress at a certain time \( t \) \([32]\):

\[
\sigma(t) = \int_{\xi = -\infty}^{t} E \left( \psi - \psi' \right) \dot{\varepsilon}(\xi) d\xi,
\] (4.4)

where the nonlinearity is accounted for by the nonlinear timescale \( \psi \). The time variable \( \xi \) spans the entire history of deformation. The principle of time-stress superposition is used to calculate \( \psi \) \([32]\):

\[
\psi = \int_{\zeta = -\infty}^{\xi} \frac{d\zeta}{a_{\sigma} \{ \sigma(\zeta) \}} ; \quad \psi' = \int_{\zeta = -\infty}^{\xi} \frac{d\zeta}{a_{\sigma} \{ \sigma(\zeta) \}},
\] (4.5)

where \( a_{\sigma} \) represents the time-stress shift factor, calculated according to:

\[
a_{\sigma} = \frac{\sigma(\zeta) / \sigma_0}{\sinh (\sigma(\zeta) / \sigma_0)}.
\] (4.6)
The uniaxial true stress as a function of running time, $\sigma(\zeta)$, determines the magnitude of the shift factor, yielding a larger time-shift at higher stress levels. The characteristic stress $\sigma_0$ can be calculated from the equivalent characteristic stress $\tau_0$, determined in the previous section. This is done by using Equation 2.27 and taking into account the pressure dependence:

$$\sigma_0 = \frac{3}{\sqrt{3 - \mu}} \tau_0.$$  (4.7)

In a one-dimensional multi mode Maxwell approach, the relaxation modulus $E(t)$ is entirely defined by the discrete relaxation spectrum consisting of $n_m$ modal moduli $E_m$ and relaxation times $\tau_m$ [32]:

$$E(t) = \sum_{m=1}^{n_m} E_m \exp \left( -\frac{t}{\tau_m} \right).$$  (4.8)

Combining and rearranging Equations 4.4 and 4.8, yields:

$$\sigma(t) = \sum_{m=1}^{n_m} \left[ E_m \int_{-\infty}^{t} \exp \left( -\frac{\psi - \psi'}{\tau_m} \right) d\xi \right].$$  (4.9)

When a discrete spectrum of relaxation times $\tau_m$ is now chosen, the only remaining unknowns in Equation 4.9 are the corresponding modal moduli $E_m$, all other variables can be obtained from the true stress - true strain results of the uniaxial compression experiment. To calculate these moduli, Equation 4.9 is transformed into the following system of equations:

$$\sigma = ME.$$  (4.10)

The column $\sigma$ contains at every experimental time point the measured value of the true stress and thus has a length equal to the number of measurement values $n_t$. The column with unknown moduli $E$ has a length equal to the number of modes $n_m$. Therefore, the matrix $M$, containing the contributions from the relaxation times per mode for each measurement point, has a size $(n_t \times n_m)$. In the multiplication (right-hand side of Equation 4.10) the contributions of all modes are summed for each experimental time point. This system of equations can be solved by using a non-negative least squares method, automatically imposing a non-negativity constraint on the moduli $E_m$, which is required for obvious physical reasons.

The next step is to calculate the corresponding spectrum of shear moduli $G_m$ and initial viscosities $\eta'_0$, which the constitutive model requires as input parameters. This calculation involves the conversion of the relaxation modulus $E(t)$ into the shear relaxation modulus $G(t)$ for which the procedure to interconvert viscoelastic response functions, presented by Tervoort [26], is used. Point of departure for this procedure is the elastic conversion formula:

$$G = \frac{3\kappa E}{9\kappa - E}.$$  (4.11)

The correspondence principle states that the appropriate Laplace transform (in this case the Carson transform or $s$-multiplied Laplace transform) of an elastic response function is interchangeable
with the Laplace transform of the corresponding viscoelastic response function. In the constitutive model, it is assumed that the volumetric response remains elastic, i.e. the bulk modulus is a constant, which is not affected by the Carson transformation. In Equation 4.11, the Young’s modulus and shear modulus are therefore replaced with their respective Carson transforms:

\[ sG(s) = \frac{3\kappa_0 s\bar{E}(s)}{9\kappa_0 - s\bar{E}(s)} \]  

(4.12)

An overbar denotes the Laplace transform and \( s \) is the transform variable. Now, the Laplace transforms \( \bar{E}(s) \) and \( \bar{G}(s) \) can be replaced with the Laplace transforms of the corresponding viscoelastic response functions:

\[ E(t) = \sum_{m=1}^{n_m} E_m \exp \left( -\frac{t}{\tau_m} \right) \quad ; \quad G(t) = \sum_{m=1}^{n_m} G_m \exp \left( -\frac{t}{\tau_m} \right) \]  

(4.13)

\[ \bar{E}(s) = \sum_{m=1}^{n_m} E_m \tau_m \frac{1}{1 + \tau_m s} \quad ; \quad \bar{G}(s) = \sum_{m=1}^{n_m} G_m \tau_m \frac{1}{1 + \tau_m s} \]  

(4.14)

By combining Equations 4.12 and 4.14, and taking the relaxations times \( \tau_m \) equal in both spectra, a system of equations is obtained where the column of modal shear moduli \( \tilde{G} \) is the only unknown:

\[ \bar{b} = \mathbf{A}\tilde{G}. \]  

(4.15)

The column \( \bar{b} \) is defined as:

\[ \bar{b} = \frac{3\kappa_0 \bar{\delta}}{9\kappa_0 - \bar{\delta}} \sum_{m=1}^{n_m} E_m \frac{\tau_m}{1 + \tau_m \bar{\delta}} \]  

(4.16)

The matrix \( \mathbf{A} \) is of size \((n_s \times n_m)\) and is defined as:

\[ \mathbf{A} = \frac{\tau_m \bar{\delta}}{1 + \tau_m \bar{\delta}}. \]  

(4.17)

Herein, \( n_s \) is the number of points in which the domain of the transform variable \( s \) is discretized, i.e. the length of the column \( \bar{\delta} \). This system of equations (Equation 4.15) can be solved for the modal shear moduli \( G_m \) by a non-negative least squares method.

Finally, the modal initial viscosities are calculated according to:

\[ \eta'_0 \bar{m} = \tau_m G_m. \]  

(4.18)

This method was applied to the true stress - true strain curve obtained from an uniaxial compression test at a true strain rate of \( 10^{-1} \text{ s}^{-1} \). The resulting relaxation spectrum was subsequently used to predict the mechanical response of the material in exactly the same experiments, for a
number of different constant strain rates. This was done by using single element FEA simulations. In Figure 4.6, the results from both the uniaxial compression experiments and the simulations are compared, showing an excellent prediction for the measurement on which the relaxation spectrum was fitted (strain rate $10^{-1}$ s$^{-1}$). The strain rate dependency is captured only across limited range, between 1 and 2 decades in strain rate. For these predictions a relaxation spectrum consisting of 16 modes was used. The values of the modal shear moduli and initial viscosities in the discrete spectrum, bearing no direct physical significance, are listed in Table 4.2. Summing the modular values of the spectral parameters yields for the single mode model: $G = 281$ MPa and $\eta_0 = 1.1989 \times 10^7$ MPa s.

Figure 4.6: Comparison of the experimental results (circles) and FEA predictions (solid lines) of uniaxial compression tests. The corresponding true strain rates are (from the top downwards): $10^{-1}, 10^{-2}, 10^{-3}$ and $10^{-4}$ s$^{-1}$.

### Table 4.2: Discrete relaxation spectrum for UHMWPE.

<table>
<thead>
<tr>
<th>$m$</th>
<th>$G_m$ [MPa]</th>
<th>$\eta_{0_m}$ [MPa s]</th>
<th>$m$</th>
<th>$G_m$ [MPa]</th>
<th>$\eta_{0_m}$ [MPa s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>46.73</td>
<td>9.3178e+6</td>
<td>9</td>
<td>15.27</td>
<td>1.3605e+3</td>
</tr>
<tr>
<td>2</td>
<td>27.07</td>
<td>2.2804e+6</td>
<td>10</td>
<td>15.38</td>
<td>5.1487e+2</td>
</tr>
<tr>
<td>3</td>
<td>1.661</td>
<td>5.2578e+4</td>
<td>11</td>
<td>14.52</td>
<td>1.8267e+2</td>
</tr>
<tr>
<td>4</td>
<td>20.13</td>
<td>2.3950e+5</td>
<td>12</td>
<td>16.98</td>
<td>8.0238e+1</td>
</tr>
<tr>
<td>5</td>
<td>13.36</td>
<td>5.9700e+4</td>
<td>13</td>
<td>12.02</td>
<td>2.1350e+1</td>
</tr>
<tr>
<td>6</td>
<td>14.06</td>
<td>2.3620e+4</td>
<td>14</td>
<td>20.36</td>
<td>1.3582e+1</td>
</tr>
<tr>
<td>7</td>
<td>15.12</td>
<td>9.5423e+3</td>
<td>15</td>
<td>10.18</td>
<td>2.5523e+0</td>
</tr>
<tr>
<td>8</td>
<td>14.29</td>
<td>3.3874e+3</td>
<td>16</td>
<td>23.64</td>
<td>2.3550e-3</td>
</tr>
</tbody>
</table>
4.4 Comparison of material models in uniaxial compression

Obviously, the material models presented in Chapter 2 all describe the mechanical behavior of UHMWPE differently. The similarities and dissimilarities between the models are visualized in Figure 4.7 by plotting the prediction of the mechanical response in an uniaxial compression test for each model. The simulations were performed with the aforementioned single element FEA. At small deformations (strains below 0.01), all models converge to a linear elastic response. When the deformation is increased, the linear elastic model overestimates the stiffness of the material and the prediction of the stresses is therefore dramatically high at finite strains. The single mode model is not able to describe the yielding behavior of the material, but gives an accurate prediction of the stresses at higher strains. Indisputably the most accurate prediction of the stress - strain behavior across the depicted strain domain is obtained with the multi mode model.

![Figure 4.7: Predictions of the mechanical response during an uniaxial compression test at a true strain rate of 10^{-1} s^{-1}, using FEA. A comparison is shown between the linear elastic (dash-dotted line), single mode (dashed line) and multi mode (solid line) simulations on one hand, and the experimental results (circles) on the other.](image-url)
Chapter 5

Flat Plate Indentation

5.1 Experimental verification

The flat plate indentation experiments described in Chapter 3 were, as mentioned, performed to enable a numerical-experimental verification. In Figure 5.1, the experimentally obtained force-displacement curve (for a 3 mm thick plate) is shown together with the material response as it is predicted with FEA, using the three constitutive theories described in Chapter 2. To eliminate initial contact effects, all curves were cut off at 20 N and displacement-shifted to that data point. The linear elastic theory predicts, as expected, stresses that are much too high due to an overestimation of the stiffness. The nonlinearity in this response is a geometric nonlinearity induced by the spherical indenter. Obviously, no hysteresis is observed in the linear elastic response, this is inherent to the applied constitutive model. The single mode prediction appears to be the most accurate of the three, at least in the loading part of the indentation cycle. This is, however, a coincidental correspondence that will be addressed in the next section. The multi mode model predicts the material response well at the beginning of the indentation, but underestimates the stress response at larger displacements. Hysteresis is incorporated in both the single- and the multi mode model, but the unloading part of the indentation curve is accurately predicted by neither of them. It appears as if the initial unloading is modeled too stiff.

![Figure 5.1: Predictions of the mechanical response during ball head indentation on a 3 mm thick flat plate, using FEA. A comparison is shown between the linear elastic (dash-dotted line), single mode (dashed line) and multi mode (solid line) simulations on one hand, and the experimental results (circles) on the other.](image)
5.2 Investigating error sources

The correspondence between the FEA predictions and the experimental results, discussed in the previous section, is rather unsatisfactory, given the fact that the multi mode model accurately described the uniaxial compression experiments (Figure 4.6). One reason for the discrepancy could be that the pressure dependency of the material was neglected in these predictions, i.e. $\mu = 0$. To assess the influence of this simplification, a simulation was also performed with $\mu = 0.1$, a value that resembles a strong pressure dependence. In Figure 5.2, the result of these two multi mode simulations are shown together with the experimental results. It can be concluded that the influence of the pressure dependence is only marginal compared to the error between the FEA predictions and the experiments. Although an accurate assessment of the pressure dependency parameter $\mu$ enhances the validity of the multi mode simulations, it fails to explain the large error observed in the numerical-experimental verification.

![Figure 5.2](image)

Figure 5.2: For the multi mode model, the influence of not taking the pressure dependence into account is visualized by plotting the FEA predictions with $\mu = 0$ (dashed line) and $\mu = 0.1$ (solid line). The experimental results from the ball head indentation tests on a 3 mm thick flat plate are also depicted (circles).

In the finite element simulations, a frictionless contact is assumed between the bottom of the specimen and the supporting metal plate, i.e. the surface nodes that form the bottom of the specimen are only constrained (zero displacement) in axial direction (see Figure 3.2). This is obviously not the case in the actual experiment, but this assumption was not expected to have a significant influence because of the small radial displacements that actually occur along this surface. As it turns out, it does affect the response predicted with FEA significantly. Since the actual amount of friction is unknown, the two extremes of the situation were compared to assess the influence of this phenomenon. In Figure 5.3, the original simulation (frictionless) is compared with a simulation in which the bottom surface is completely constrained (zero displacement) in both axial- and radial direction. There clearly is an enormous influence of the friction on the bottom surface and it is, therefore, of imperative importance to accurately account for the amount of friction when comparing simulations and the experiments. The most straightforward method to do this is to ensure complete constraint of the bottom surface in the experiment, for instance by gluing the specimen to the supporting metal plate.
Figure 5.3: For the multi mode model, the influence of friction on the bottom side of the flat plate is visualized by plotting the FEA predictions with no friction (dashed line) and infinite stiction (solid line). The experimental results from the ball head indentation tests on a 3 mm thick flat plate are also depicted (circles).

For reasons of clarity, the numerical and experimental results of the ball head indentation tests on flat plates of 6 and 9 mm thickness were omitted here. The corresponding graphs are given in Appendix A.

5.3 Influence of specimen thickness

Although the previous section only discusses the results for the 3 mm thick plates, it is interesting to look at the influence of specimen thickness. In Figure 5.4, this is visualized by plotting the loading part of the indentation curve for the three thicknesses, both from the experiments and the multi mode simulations. A thicker plate is, in indentation with a rigid supporting plate, more compliant than a thinner one, a trend that is also visible in the graphs. However, the trend does not look the same for the experimental curves and the numerical predictions. When a volume of material is indented, a finite amount of material is involved in the actual deformation. When a specimen is thicker than this penetration depth of the deformation into the material, a further increase of the thickness will have no influence on the recorded response. For this reason, it seems logical that the increase in thickness from 3 to 6 mm will have a larger influence on the response than the increase from 6 to 9 mm, which is also predicted by the numerical simulations. In the experiments, however, the 9 mm curve does not follow this trend. Moreover, this curve exhibits a sudden change in stiffness at a displacement of around 0.2 mm and the stiffness in the regime prior to this change is much lower than would be expected on the basis of the other measurements and the simulations. The reason for this is still unclear, but could be related to the uncontrolled friction that occurs between the sample and the bottom plate.
5.4 Validity of Hertzian theory

There are three reasons why the Hertzian theory is essentially unsuitable to predict the material response of a specimen with a finite thickness, made from a strongly nonlinear viscoelastic material like UHMWPE: the theory only accounts for linear elastic material behavior, it treats the specimen as a semi-infinite body and it requires the radius of the contact area to be small with respect to the radii of the two contact bodies. However, a major advantage of this theory is that the solution can be analytically obtained without much effort and at low computational costs. With linear elastic FEA, and even more with single- or multi mode nonlinear viscoelastic FEA, quite the contrary is the case: it can be very time consuming, depending on the complexity of both the mesh and the material model. Hertzian theory could therefore be a very convenient option for orthopaedic implant engineers to get a rough indication of the mechanical response of a hip cup with a 'back-of-the-envelope' analysis. This raises the question of how reliable the predictions of the theory actually are, compared to experimental data as well as more advanced FEA predictions.

On the left side of Figure 5.5, the numerical-experimental verification (see Figure 5.1) is shown once more, now including the prediction of the Hertzian theory. This prediction is remarkably good, considering the aforementioned limitations of the theory. On one hand it dramatically overestimates the stiffness of the material due to the assumption of linear elastic material behavior. On the other hand, a contradicting effect occurs by assuming an infinite specimen thickness. When the specimen thickness increases, the structural stiffness of the component actually decreases, which is why Hertzian theory strongly underestimates the structural stiffness of the specimen. This effect is visualized on the right side of Figure 5.5, where the linear elastic FEA solutions for increasing specimen thickness appear to 'converge' to the Hertzian solution.
It can be concluded that counteracting effects in the Hertzian solution in this case yield a reasonable estimate of the mechanical response of an indented flat UHMWPE plate. However, it cannot be overemphasized that this is utterly coincidental. With other geometries (e.g. different indenter radius or specimen shape) this conclusion might not hold anymore; the same could be the case with other loading conditions (e.g. different indentation speed or maximum indentation).
Chapter 6

Cup Indentation

Although flat plate indentation, described in the previous chapter, is a convenient geometry to study material behavior and to assess the value of different computational techniques used to predict this behavior, it does not really have direct significance to the practice of orthopaedic implant design. Therefore, this chapter discusses FEA predictions of the mechanical response of a cup geometry (described in Chapter 3) subject to indentation with a spherical indenter. The radii of the indenter and the cup were 18 and 18.1 mm, respectively, a combination that is currently used in commercially available hip implants. Of the three minimum thicknesses that were employed in this study (3, 6 and 9 mm), 6 mm is the one with the most practical relevance; this value is regarded as the minimum allowable thickness of UHMWPE tibial- or hip inserts, when no metal backing is used [33].

6.1 Comparison of material models

The axial forces that actually occur in implanted hip- and knee prostheses during, for instance, a gait cycle, will generally not exceed 5 kN [9]. Therefore, the results in this chapter are presented with a special focus on this medically relevant range. Finite element simulation results, obtained using the different constitutive theories presented in Chapter 2, are shown in Figure 6.1 depicting both the force - displacement and the contact stress - displacement curves. When the indentation forces in this experiment are compared to what was found for the flat plate indentation, it is clear that they are much higher. Naturally, this is due to the fact that the articulating surfaces in the cup experiment have a much higher conformity, resulting in a significantly larger contact area and an increased contact stiffness. When the predictions of the various constitutive models are compared, there appears to be little difference between the response predicted by the linear elastic and the single mode model. The reason for this is that, due to the high contact stiffness, the displacements remain small within this force range. Consequently, the stress - strain response is restricted largely to the pre-yield regime. In Figure 4.7, it was already shown that in this regime the linear elastic and single mode models predict comparable mechanical behavior. Obviously, the multi mode model predicts a lower stiffness, indentation force and contact stress, since this model takes into account the relaxation behavior that the material exhibits at stresses below the yield stress.
Figure 6.1: Influence of constitutive material modeling on the FEA predictions of indentation force (left) and contact stress (right) in cup indentation (minimum thickness 6 mm). A comparison is made between the linear elastic model (dash-dotted line), the single- (dashed line) and multi mode (solid line) nonlinear viscoelastic model.

6.2 Influence of minimum thickness

The influence of the minimum thickness in these cup experiments is entirely analogous to that in the flat plate experiments; an increase in the thickness of the component yields a decrease in contact stress and indentation force due to a decrease in structural stiffness of the component. The multi mode predictions of the mechanical response in the cup indentation experiments are shown in Figure 6.2 for the three thicknesses. A direct link to non-metal-backed hip inserts is hard to establish because the cup geometry is a substantially simplified representation of such components. Nevertheless, a conclusion that can be drawn from the data presented here is that a minimum thickness of 6 mm for these components is indeed reasonable, since the corresponding contact stress remains below the yield stress throughout the range.

Figure 6.2: Influence of minimum component thickness on the indentation force (left) and contact stress (right) in cup indentation. Multi mode FEA predictions are plotted for a minimum thickness of 3 (black), 6 (red) and 9 mm (blue).
Chapter 7

Conclusions

The basis of this study consists of the mechanical characterization of the material behavior of UHMWPE, using results from uniaxial compression experiments at different strain rates. This characterization includes the determination of a number of material parameters and the discrete relaxation spectrum that shape the mechanical response predicted by the three constitutive theories that were compared in this study: linear elasticity, single- and multi mode nonlinear viscoelasticity. The correspondence that was obtained between the results from the uniaxial compression experiments and -simulations, was satisfactory, but with a limited applicability in terms of strain rate. It is expected that an improvement of the fit routine used to determine the discrete relaxation spectrum, will especially enhance the prediction of the unloading part of the indentation curve. The overestimated stiffness in the initial unloading regime is believed to stem from the poor characterization of the material behavior at low strain rates.

The numerical-experimental verification performed for flat plate indentation gave rise to a number of issues that need to be addressed in further research. The influence of the pressure dependency of the material was shown to be small, but it should nevertheless be quantified to complete the mechanical characterization. Furthermore, an unexpectedly large influence of the friction between the bottom of the sample and the supporting metal plate was found by simulating the two extreme cases. The level of friction in the performed experiments is unknown and will therefore contribute to a significant error, compared to the simulations.

In the practice of orthopaedic implant design, Hertzian theory is a convenient option to roughly estimate the mechanical response in a ball head indentation test. Despite the limitations of this theory, it can be concluded that for the experimental conditions used in this study, the predictions are remarkably good. In fact, the prediction of indentation forces and contact stresses are even more realistic than for the more elaborate linear elastic FEA. Since it unknown if this correspondence will also hold when the experimental conditions are different, it is dangerous to draw definite conclusions based only on Hertzian calculations.

In the final chapter of this work, the incentive has been given to an assessment of factors that are known to influence the stress response and performance of UHMWPE hip inserts. The discussion of the influence of the minimum thickness of these components could be extended to the influence of indenter radius, cup radius, material, etc.

Finally, an advantage of the proposed multi mode nonlinear viscoelastic material model is that, in principle, it should be able to accurately describe cyclic loading (fatigue) and creep phenomena. It was already pointed out by Estok [34] that quantification of creep in UHMWPE would enable a distinction between creep- and wear phenomena in orthopaedic implants, thereby helping to understand the contributions of the respective processes. In general, it is evident that, due to the long-term nature of the loading of actual joint replacements, fatigue- and creep effects may play a vital role in the performance of UHMWPE tibial- and hip inserts.
Acknowledgements

First of all I would like to express my gratitude to my supervisors, dr.ir. T.A. Tervoort and dr.ir. L.E. Govaert, without whom this report would not have existed. Furthermore, I am indebted to dr. Y. Dirix from Smith & Nephew for stimulating discussions, vital experimental support and critical reading of this report. I also gladly acknowledge ir. L.C.A. van Breemen for sharing his expertise on finite element simulations and the implementation of the viscoelastic model.
Bibliography


Appendix A

Flat Plate Indentation Results

In Chapter 5, the numerical and experimental results from the ball head indentation tests on a 3 mm thick flat plate were discussed. The same results are presented in this appendix, but then for specimen thicknesses of 6 and 9 mm, see Figures A.1 through A.3.

For the 6 mm specimen, basically the same discussion can be held as was done for the 3 mm thick one. The most apparent difference between these is that the friction between the bottom surface of the specimen and the supporting metal plate has a smaller influence than with the 3 mm specimen. The result of this is that, in the 6 mm case, the experimental values do not lie in between the simulated extremes. It nevertheless remains important to, as mentioned, level the amount of friction that occurs in the simulation and the experiment.

The numerical-experimental comparison for the 9 mm plate appears to be worse than in the other cases. The experimental results are, as mentioned, very different from what was expected, regarding the other experimental results and the simulations. It is impossible to draw conclusions from the numerical-experimental comparison as long as this problem remains unsolved.

![Figure A.1: Predictions of the mechanical response during ball head indentation on a 6 (left) and a 9 mm (right) thick flat plate, using FEA. A comparison is shown between the linear elastic (dash-dotted line), single mode (dashed line) and multi mode (solid line) simulations on one hand, and the experimental results (circles) on the other.](image-url)
Figure A.2: For the multi mode model, the influence of not taking the pressure dependence into account is visualized by plotting the FEA predictions with $\mu = 0$ (dashed line) and $\mu = 0.1$ (solid line). The experimental results from the ball head indentation tests on a flat plate are also depicted (circles). Results are shown for a 6 (left) and a 9 mm (right) thick plate.

Figure A.3: For the multi mode model, the influence of friction on the bottom side of the flat plate is visualized by plotting the FEA predictions with no friction (dashed line) and infinite stiction (solid line). The experimental results from the ball head indentation tests on a flat plate are also depicted (circles). Results are shown for a 6 (left) and a 9 mm (right) thick plate.