Evaluation and modelling of knee kinematics based on medical imaging and cadaver test data

Dries Lagrou

Supervisors: Prof. dr. ir. Benedict Verhegghe, Prof. Jan Victor
Counsellors: Dr. ir. Matthieu De Beule, Ir. Wouter Devriendt

Master's dissertation submitted in order to obtain the academic degree of Master of Science in Electromechanical Engineering

Department of Electronics and Information Systems
Chairman: Prof. dr. ir. Jan Van Campenhout

GE03
Chairman: Prof. Jan Victor

Faculty of Engineering and Architecture
Academic year 2013-2014
I’d like to take the opportunity to thank some people that were directly involved with this master dissertation:

- Matthieu De Beule for the enthusiastic drive, overview and recommendations.
- Nic Debusschere: for taking over Wouter Devriendt his role in this scription. They were always present with accurate, smart tips throughout the whole thesis. Thank you for your support.
- Prof. Van Hoof for the anatomical knowledge, CT and MRI-scans and dissections your department did.
- Benedict Verhegghe and Francesco Iannaccone for my questions in pyFormex/Linux and clustering.
- Diogo Moreira Campos Ferreira de Almeida, Matthias Verstraete, Dr. Nico Van Der Hauwaert, Dr. Annick Verhaeghe and Maarten Vanneste.
- The reader of this for at least getting to line 10. You already read 0.8%.

Beside the people that are directly involved, I’d like to thank my family and friends for their support during my education and this work.

Special thanks for my mother.
Thank you father.

“The author gives permission to make this master dissertation available for consultation and to copy parts of this master dissertation for personal use. In the case of any other use, the limitations of the copyright have to be respected, in particular with regard to the obligation to state expressly the source when quoting results from this master dissertation.”

Dries Lagrou, 2nd of June, 2014
Purpose and overview

The main purpose of this master dissertation creating a foundation for the development of a finite element knee simulator. The simulator aims to develop a graphical user interface (GUI) for non-engineers to use when knees are tested. In future surgeons could be assisted by the GUI to provide some anatomical or kinematics information so patients would be treated more accurately. Examples are knee replacements, reconstruction ACL/PCL, ...

Three big steps are:

1. **MRI/CT-scans**
   First of all, MRI and CT scans are provided. CT-scans are better for bone registration (femur, tibia and fibula) and MRI give better resolution to the soft tissues in the knee (ligaments, menisci and cartilage). All parts are segmented in Mimics to reconstruct a 3D knee model. That knee model is processed further for finite element simulations (FEA).

2. **Kinematic tests**
   Markers are placed on a cadavers femur and tibia. With eight cameras the position of the markers is traced. This data is handled in pyFormex to render the 3D kinematics and automatic generate an input-file for FEA.

3. **Modelling/Simulations**
   Some parameters and assumptions have to be changed/made in order to get realistic results and stable calculations.

Some future works has to be done. This is discussed in the conclusions of the master dissertation.

**Keywords:**

Biomechanics, Human knee kinematics, Image segmentation, Finite element simulations
Evaluation and modelling of knee kinematics based on medical imaging and cadaver test data

Dries Lagrou
Supervisors: Benedict Verhegge, Jan Victor M.D., Matthieu De Beule, Nic Debuisschere, Wouter De Vriendt

Abstract – This article studies the kinematics and kinetics of the knee joint. The process of developing a python based graphical user interface (GUI) for mimicking knee kinematics in finite element analysis (FEA) software is described. The correct anatomy is acquired from CT & MRI scans, kinematics from 3D stereo-photogrammetry of rigidly mounted markers. The main objective is creating a foundation for automatic FEA simulations as part of the development of a virtual total knee simulator.

Keywords – human knee kinematics, image segmentation, finite element simulation

I. INTRODUCTION

The knee joint is the largest and most intricate joint of the human body. Many attempts have been done to describe the relative movement of the femur to the tibia (=knee kinematics). Limitations of planar mechanisms have become clear and mathematical descriptions are too complex for clinical use. Hollister et al. captured the motions by LED’s on tibia and femur. She simplified the description of the motion with two angles: one for flexion – extension and the other one covering endo-exorotation and varus-valgus. Although several recent techniques capture motion accurately, no useful clinical description is developed such as the one Hollister have defined.

In this article, kinematics are captured with the methodology suggested by Jan Victor [1] and described according to Grood and Suntay [2].

II. METHODOLOGY

The methodology consists of four steps. Kinematic tests are performed to capture the kinematics. The tested knee is then imaged and via segmentation a 3D model is created. Kinematic data is applied to the 3D model resulting in the relative movement of the tibia to the femur. Via the developed GUI, four ligaments are added and an input file is automatically generated to be compatible with the finite element software Abaqus. The last two steps are done in pyFormex. This is open source, in-house developed, python based software for programming. (http://www.nongnu.org/pyformex). Abaqus v. 13-6 (Dassault Système) is used for FEA.

A. Kinematic Tests

Two holders containing three optical markers each are rigidly mounted onto the femur and tibia of a cadaver. The specimen is set in the focus of a tracking system and five open chain flexion – extension cycles are performed. The absolute position of the all markers is acquired at a frequency of 35 Hz.

Figure 1: kinematic test setup
B. Segmentation

The 3D knee model is constructed with CT and MRI scans of the tested knee. Bone tissue with markers are reconstructed using the CT scans because of the better delineation than using MRI. The MRI scan is used for soft tissue. Segmentation is done in Mimics v. 16.0 (Materialise).

C. Kinematics of knee

As pyFormex is a powerful program to transform large geometrical models with mathematical operations, it will be used to obtain the kinematics. A script in pyFormex imports the 3D knee model and the kinematic data. The coordinates of each marker in the 3D model is picked and the corresponding coordinates of the marker in the kinematic data are picked. By projecting the markers of the 3D model onto the coordinates of the kinematic data for every step, the movement of the both bones are traced. By following the recipe of Grood and Suntay [2], kinematics are analysed. Results of the rotation angles are shown in following graph and are discussed in the article of Vanneste [5].

![Kinematics of the knee. In red flexion angle, blue tibial rotation and green varus angle.](image1)

D. Abaqus input file generation

The time consuming computer aided engineering (CAE) part in Abaqus is bypassed with a pyFormex script showing a GUI that generates an input file for stable simulations in Abaqus/standard.

First, a model is created in the Abaqus/CAE for determining the optimal simulations strategy and setup. The surface models of the 3D knee model are meshed in 3-matic v. 8.0 (Materialise) and the optimal parameters for a stable outcome are determined (material models, calculation settings, contact properties, meshing setups,...).

A hyperelastic neo-hookean material is used to model the mechanical properties all the ligament [4]. Menisci and cartilage are excluded.

A pyFormex script is developed, reproducing the input file of the optimal model out of the segmented parts. The imported surface models are meshed automatically by a link with tetGen (www.tetgen.org).

III. RESULTS

The generated .inp-files are imported in Abaqus and surfaces are adjusted (dummy surfaces are generated).

In the GUI different setups of mesh densities could be chosen. These are used for the mesh sensitivity test. Convergence, vulnerability to kinematic data faults and calculation time are considered for the best meshing setup.

From this simulation following was obtained. For every step, the mean maximal principal stress have been calculated and plotted as a function of time in figure 3.

![mean maximal principal stresses in function of time](image2)
Region I and II describe a flexion – extension cycle (see figure 2). The excessive values in region IV are due to faults in kinematic data. Although the same simple material models are used for every ligament, the high load on the ACL and MCL is observed due to the mobility of the medial condyle. The low values for the LCL are because isotropic material model. This gives a high, unrealistic curvature to the LCL. The compression and expansion in curvatures results in low average stresses.

MCL mechanical tests executed by Vanneste learned that the maximal stress is 2 MPa. This value is obtained by applying the neo-hookean constants suggested by Peña [4]. Theoretical linearity of the stresses to the neo hooke material constant is seen in following graph. The red graph was obtained with linearity, the green by overdoing simulations.

![Graph showing stress over time](image)

Figure 4. Simulations with C = 0.7324 MPa (blue) and C = 1.44 MPa (green, red).

A high flexion reaction moment is observed in the reaction moments around the femoral reference point. In full extension the knee lock mechanism could be seen as the magnitude of the resulting moment is primary determined by the tibial rotation moment. A negative abduction moment conforms the varus seen in the kinematic tests.

In this project several steps have been taken towards the development of a finite element knee simulator. A script is developed that reads the segmented bone and ligament tissues and generates a .inp-file with stable abaqus simulations. With few adaptions, menisci and cartilages are included. For full automatic .inp-file generation, surface determination has to be developed further.

With simple material models some known kinematics are found back on the graphs. But no accurate data is available for validation. Other MRI-scans can be used for further validation of the soft tissue displacement. A great difference is seen between the presented material models found in literature. The linear material model was submitted to simulations and values 20 times higher are observed. Simulations couldn’t be finished. Some kinematic faults are still left due to a correction that has to be done for fitting soft tissue with the bone tissue movement from the kinematic data.

REFERENCES


# Table of contents

**PART I: INTRODUCTION**

- **H1. THE KNEE: ANATOMY AND KINEMATICS**
  - 1.1. Anatomy of the knee joint
  - 1.2. Kinematics of the knee joint

- **H2. METHOD & UTILITY OF THE MASTER DISSERTATION**
  - 2.1. General method / process
  - 2.2. Utility of the master dissertation

**PART II: IMAGING AND IMAGE PROCESSING**

- **H3. IMAGING**
  - 3.1. CT-scans
  - 3.2. MRI scans

- **H4. SEGMENTATION**
  - 4.1. A first segmentation
  - 4.2. Bone/Marker/Sphere segmentation in CT-images
  - 4.3. Bone registration in MRI-images
  - 4.4. Segmentation in MRI-images: generals
  - 4.5. The knee 3D model

- **H5. MESHING**
  - 5.1. General meshing of soft tissue
  - 5.2. General meshing of hard tissue
  - 5.3. Volumetric meshing in TetGen
  - 5.4. Meshes imported into pyFormex: stl_read.py

**PART III: KINEMATICS**

- **H6. KINEMATIC TESTS**
  - 6.1. Preparation
  - 6.2. Tests

- **H7. KINEMATIC ANALYSIS**
  - 7.1. Generals: From .csv-files to kinematics and strain analysis
  - 7.2. Transformatie.py
  - 7.3. Kinematics: from .csv file to moving object
  - 7.4. Kinematics analysis strategy
  - 7.5. Kinematic analysis implemented in pyFormex
  - 7.6. Elongation analysis

**PART IV: MODELLING**

- **H8. A FIRST MODEL WITH ABAQUS**
PART V. APPENDIX MATERIAL MODELS

1. LINEAR ANISOTROPIC MATERIAL MODEL .......................................................... 71
2. NEO HOOKE - HYPERELASTICITY ........................................................................ 72
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D</td>
<td>Two dimensional</td>
</tr>
<tr>
<td>3D</td>
<td>Three dimensional</td>
</tr>
<tr>
<td>AA</td>
<td>Abduction - abduction</td>
</tr>
<tr>
<td>ACL</td>
<td>Anterior cruciate ligament</td>
</tr>
<tr>
<td>avg</td>
<td>Average</td>
</tr>
<tr>
<td>C</td>
<td>Compression modulus</td>
</tr>
<tr>
<td>CAD</td>
<td>Computer aided design</td>
</tr>
<tr>
<td>CAE</td>
<td>Computer aided engineering</td>
</tr>
<tr>
<td>CC</td>
<td>Contact constraint</td>
</tr>
<tr>
<td>CF</td>
<td>Femoral cartilage</td>
</tr>
<tr>
<td>COR</td>
<td>Centre of rotation</td>
</tr>
<tr>
<td>CP</td>
<td>Patellar cartilage</td>
</tr>
<tr>
<td>csv</td>
<td>Comma separated values</td>
</tr>
<tr>
<td>CT</td>
<td>Computed tomography</td>
</tr>
<tr>
<td>CTL</td>
<td>Tibial Cartilage lateral</td>
</tr>
<tr>
<td>CTM</td>
<td>Tibial Cartilage medial</td>
</tr>
<tr>
<td>D</td>
<td>Bulk modulus</td>
</tr>
<tr>
<td>DEPVAR</td>
<td>Dependent variables</td>
</tr>
<tr>
<td>DOF</td>
<td>Degree of freedom</td>
</tr>
<tr>
<td>E</td>
<td>Young's modulus</td>
</tr>
<tr>
<td>EE</td>
<td>Endo-exorotation</td>
</tr>
<tr>
<td>EF</td>
<td>Extension facet</td>
</tr>
<tr>
<td>EFC</td>
<td>Extension facet centre</td>
</tr>
<tr>
<td>FE</td>
<td>Flexion-extension</td>
</tr>
<tr>
<td>FEA</td>
<td>Finite element analysis</td>
</tr>
<tr>
<td>FF</td>
<td>Flexion facet</td>
</tr>
<tr>
<td>FFC</td>
<td>Flexion facet centre</td>
</tr>
<tr>
<td>FOV</td>
<td>Field of view</td>
</tr>
<tr>
<td>FS</td>
<td>Finite sliding</td>
</tr>
<tr>
<td>G</td>
<td>Gliding modulus</td>
</tr>
<tr>
<td>GUI</td>
<td>Graphical user interface</td>
</tr>
<tr>
<td>inp</td>
<td>Abaqus input</td>
</tr>
<tr>
<td>LCL</td>
<td>Lateral collateral ligament</td>
</tr>
<tr>
<td>LML</td>
<td>Lateral meniscal ligament</td>
</tr>
<tr>
<td>MCL</td>
<td>Medial collateral ligament</td>
</tr>
<tr>
<td>MML</td>
<td>Medial meniscal ligament</td>
</tr>
<tr>
<td>MP</td>
<td>Mega pixels</td>
</tr>
<tr>
<td>MRI</td>
<td>Magnetic resonance imaging</td>
</tr>
<tr>
<td>NTS</td>
<td>Node to surface</td>
</tr>
<tr>
<td>odb</td>
<td>Output database</td>
</tr>
<tr>
<td>pgf</td>
<td>pyFormex geometry file</td>
</tr>
<tr>
<td>RF</td>
<td>Reference point</td>
</tr>
<tr>
<td>RM</td>
<td>Reaction moment</td>
</tr>
<tr>
<td>s</td>
<td>Second</td>
</tr>
<tr>
<td>sec</td>
<td>Second</td>
</tr>
<tr>
<td>SS</td>
<td>Small sliding</td>
</tr>
<tr>
<td>STL</td>
<td>StereoLithographies</td>
</tr>
<tr>
<td>STS</td>
<td>Surface to surface</td>
</tr>
<tr>
<td>TC</td>
<td>Tie constraint</td>
</tr>
<tr>
<td>Tet</td>
<td>Tetraedric</td>
</tr>
<tr>
<td>TKR</td>
<td>Total knee replacement</td>
</tr>
<tr>
<td>UZ</td>
<td>Universitair ziekenhuis</td>
</tr>
<tr>
<td>VV</td>
<td>Varus-valgus</td>
</tr>
<tr>
<td>v</td>
<td>Poisson ratio</td>
</tr>
</tbody>
</table>
PART I: Introduction
The knee is the largest human joint, connecting the femur to the tibia and fibula. It allows locomotion with minimal energy and high stability in all ranges of motion. Another function is transmitting, absorbing and redistributing forces caused in daily activities.

1.1. Anatomy of the knee joint

The knee joint mainly consists of four sorts tissues.
- Four bones: Femur, Tibia, Patella and Fibula.
- Four ligaments: ACL, PCL, LCL and MCL.
- Two menisci: Lateral and Medial (LML, MML).
- Four cartilages. CF, CTM, CTL, CP

The four bones are cushioned with cartilage (figure 1.1). Onto the bottom of the femur, c-shaped femoral cartilage surrounds the knobs (= condyles femoral) of the femur. The femoral cartilage is connected to the cartilage of the patella and both medial and lateral tibial cartilage, attached to the two tibial top surfaces (=condyles tibial). The menisci are placed in between the femoral cartilage and tibial cartilage. At the medial side the meniscus is c-shaped, the lateral meniscus is more o-shaped. They thicken from the centre to the outer. Cartilage and menisci assure near frictionless movement and absorb impact. Figure 1.2

Figure 1-1: cartilages covering the bones
Three ligaments connect the femur to the tibia: ACL, PCL and MCL. The LCL connects femur and fibula. The collateral ligaments (LCL, MCL) are aside of the knee, limiting the lateral-medial motion of the knee. The cruciate ligaments connect the femur and tibia in between the condyles. The ACL from anterior tibia to posterior femur, the PCL is connected contrarily. These two ligaments limit the back (ACL) and front (PCL) movement. Anatomical terms are introduced further.

The patella protects the front of the knee (=kneecap) and is connected by ligaments to the muscles for forces transmission to the tibia. Thanks to a grove in the femoral cartilage, the patella helps stabilizing the knee joint.

Synovium can be considered as a bag around the knee bones filled with fluid. It nutrients and lubricates cartilage. The fluid provide anti-inflammatory elements in case of injury.
Some anatomical terms are included in this thesis. Three planes in human body are used: the letters in between the brackets indicate the numbers in figure 1.4.

- Sagittal plane (c): the vertical plane dividing the body into a right and left part. The midsaggital plane is located in the middle.
- Frontal/Coronal plane (a): the vertical plane dividing back and front parts.
- Transverse/Axial plane (b): the horizontal plane dividing upper and lower parts.

To point out relative positions, following anatomical terms are used:

- Anterior/Ventral: describing the structures in front or towards the front of the body.
- Posterior/Dorsal: describing the structures behind or towards the back of the body.
- Superior/Cranial: describing the structures above or toward the head/upper part of the body.
- Inferior/Caudal: describing the structures below or away from the head/upper part of the body.
- Proximal: describing the structures away from the origin of the body part a limb to the body trunk.
- Distal: describing the structures towards the origin of the body part a limb to the body trunk.
- Lateral: describing the structures away from the sagittal midplane.
- Medial: describing the structures closer to the sagittal midplane.

The bones are subdivided in three main regions (figure 1.6: original uploader Chriudel derivative work: Osado)

- Epiphysis: the rounded end of the bone. In the femur, the distal epiphysis is close to the knee joint and the proximal epiphysis is involved with the hip joint. In the tibia, the distal epiphysis is involved with the ankle joint and the proximal epiphysis with the knee joint.
- Diaphysis: the midsection of the femur.
- Metaphysis: the transition in between epi- and diaphysis.
1.2. Kinematics of the knee joint

The definition of kinematics is: ‘the study of the relative bone movement without the influence of forces’. The forces and moments are studied in kinetics. Within the knee joint there are three moveable articulations (figure 1.7): lateral femorotibial (2), medial femorotibial (1) and femoropattellar (3). From these three, only the bicondilar femorotibial connection is relevant for this kinematic analysis. It’s a mobile trocho-ginglymus (pivot hinge joint): the hinge structure allows translation when rotating, the pivot limits the translation.

![Figure 1-7: Articulations in the knee](image)

1.2.1. Generals

Some anatomical terms are introduced first. Within knee movements, there are three rotations that can be related to planes in the knee. These planes are shown in figure 1.8. The flexion-extension angle is expressed with theta. When the patients are standing, theta is zero. The range of motion (ROM) of theta is -5° to 160°. The frontal axis shows the varus-
valgus/abduction-adduction rotation. This angle barely changes with (natural) movements. It can be determined in standing position. In the transverse axis, internal-external rotations make the tibias position change. When the knee is loaded, this value can be up to 25°. Unloaded this value is slightly less: 14°.

By combining all those rotations and translations, some relative movements of femur to tibia can be described. They are pictured in figure 1.9. The picture shows how rolling, sliding and rotating purely (spinning) works. These three movements are applied to the lateral and medial condyles of the tibia in function of the femur flexion to key out the movement of the knee joint.

---

**Figure 1-8: Knee rotations**

**Figure 1-9: Knee motions of the condyles**
The movement of the knee differs when the joint is loaded or unloaded. For this scription, an unloaded and open chain setup is used. An open chain knee movement means that the foot is moving free in space.

The description of the knee kinematics is started with the finding that the distal edge of a femoral condyle can be described with two circle segments: flexion facet FF and extension facet EF. This is shown in figure 1.10. The segments are expressed by a radius from a central point named extension facet centre EFC and flexion facet centre FFC. The FF, EF, FFC and EFC are fixed to the femur. On the tibia there are corresponding surfaces to the EF & FF (figure 1.11). The EFC expresses the movement of the knee in the first 20°, the FFC for the higher flexion angles. By expressing the anterior-posterior position of the EFC/FFC within their range, the movement of the knee can be demonstrated. This is explained below.

The dotted lines on figure 1.12 expresses the connection between the EFC’s on both condyles, the black ones the connection between FFC’s. When the flexion angle ranges from 0° to 20°, the medial and lateral EFC move posterior indicating the movement is a rotation around their EFC’s with a little progressive movement of the femur backward on the tibia (rollback / backward sliding). In the range from 20° to 90° the FFC is used. The points are moved anterior because of the relative position of the FFC to the EFC. The points on the lateral condyle are going faster posterior, causing the tibia to exorotate. As the points are closer together in the medial part, the backward sliding is greater at the medial. Above 90°, the lines are more parallel. This indicates that the flexion does not induce anymore exorotation of the tibia. Above 120°, the knee is slightly sliding forward / spinning.

During the last 20 degrees of the knee extension, the screw-home mechanism locks the femur in the tibial plates by a medial anterior tibial glide. This is shown in figure 1.13.

The kinematics of the knee are mainly function of the cruciate ligament (ACL and PCL) performances. They both elongate when flexion is rising up to 120°. The ACL generates higher anterior forces to guide the bones, the PCL makes the condyles pivots and roll. The menisci mainly function as shock absorbers and force distributors for the cartilage. From kinematic point of view, it has been observed that the medial meniscus is less mobile than the lateral one.

As summary: the knee joint kinematics depends on the loading. When unloaded and open chain, the movement of the tibia to the femur can be described in following sequence:

<table>
<thead>
<tr>
<th>flexion femur</th>
<th>Femoral motion</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt; 20°</td>
<td>Backward slipping / screw-home</td>
</tr>
<tr>
<td>20° - 90°</td>
<td>Rotation &amp; small Backward slipping</td>
</tr>
<tr>
<td>90° - 120°</td>
<td>Small backward slipping</td>
</tr>
<tr>
<td>120° &lt;</td>
<td>Small forward slipping / rolling</td>
</tr>
</tbody>
</table>

Table 1-1: summary of the femur motions relative to the tibia
Figure 1-10: The EF and FF with the circles describing the EFC and FFC (Iwaki)

Figure 1-11: EF and FF on tibia and femur (Iwaki)

Figure 1-12: Projection of the EFC (striped) and FFC (bold)
1.2.2. How is the movement captured

Kinematics of the knee are complex. Different models are made in order to understand and reconstruct these movements: from over mathematical approaches (Hollister et al.) to CT/MRI-analysis (Iwaki et al.) and newer technologies (Victor et al.). In the past many extreme complex mathematical models were produced to mimic the knee. These models became too complex for clinical use, however some models did describe the knee motion properly. In 1995 Hollister managed to accurately simplify the descriptive models. She defines the motion of the knee with two axes: one for the flexion-extension and the other one covers valgus-varus and exo-endorotation. Relative movement is captured by LED’s on the tibia and femur. The flexion-extension axis was found by CT analysis. Surgeons were able to use this model as the axes have intuitive connotation for clinicians.

A kinematic model was produced by CT/MRI-analysis and analysing the FFC and EFC (see above). Kinematics of different knees in different flexion angles could be analysed in a sagittal view and reconstructed in 3D. The motion of the EFC’s and FFC’s gave insight in the 3D-motion of the knee.

Other techniques are: analysis with CT fluoroscopy. A CT-scans of the same knee is segmented first. In vivo measurements of real time movements are made with CT fluoroscopy video analysis. The in vivo projection in the video is used for fitting the segmented bones’ 3D place. By doing this for every flexion angle, the movement can be reconstructed.

The kinematic model presented by Jan Victor will be used for this master dissertation. The procedure and modelling will be explained in next head chapter.

1.2.3. Deficiencies

Knee kinematics is largely influenced by the cruciate ligaments. A deficiency in ACL or PCL makes the 3D kinematics change causing instability and damage of other tissues in the knee. Cruciate ligaments are mainly damaged in sports with a lot of sharp, intensive turns. This
makes them elongate largely and intensively. When damaged, a grown laxity and rotational instability is noticed. Cartilage and menisci are differently and higher loaded as the kinematics change. Studies have shown that this can be defeated with (patient specific) ACL/PCL-reconstructions. Hereby the ligament is completely removed and replaced by a new one. The reconstruction can be performed anatomically/non-anatomically according to the surgeon. Afterwards, kinematics are changed and some problems arise due to stability and damage. The development of the total knee simulator in UZ Gent, the surgeon will helped to perform maximally.

If a knee is severely damaged, a total knee replacement (TKR) can be applied (figure 1.14). Hereby, cartilage on the femur and tibia are accurately cut, shaping the femoral component and tibia component to make the new knee fit. The cruciate ligaments are removed but collateral ligaments left in place. Movements are influenced by the shape of both components and the plastic spacers that act as menisci. Kinematics can be analysed and compared to natural kinematics. Surgery parameters can be analysed and improved. By calculating back from the normal kinematics, TKR components can be improved.

Other deficiencies influencing the kinematics of the knee, are: meniscal tears, collateral ligament damage (most common is the MCL) and dislocated or fractured patella.
2.1. General method / process

A right knee of a cadaver, free from articular disorders was the specimen we worked on. The specimen was conserved and prepared by the department of anatomy at UGent. The cadaver was conserved so that mobility is maintained. In this way differences between in vivo movements and the cadaver movements are reduced as much as possible.

The knee went through following steps:

Kinematic tests are performed and images are taken.

- Two markers were rigidly mounted into the femur and tibia by a medical doctor. Three spheres are attached to the extremities of each marker.
- Tests were performed within the optitracker: flexion – extension (to 90° flexion), deep flexion - extension (to 150 ° flexion) and endorotation - exorotation in three flexion angles. The position was written to excelfiles (csv-files).
- CT-scans were performed. The whole lower right limb with markers, individual markers and spheres were imaged. That scans were used to segment bones and markers.
- MRI scans were prepared by professor Van Hoof. The knee was amputated so that it fitted the scanning machine. The metal markers had to be removed.
  Four MRI scans were taken in flexion angles 0°, 20°, 60° and 90°. The image of flexion angle 20° was used for segmentation of soft tissue, the other ones can be used for validation of the obtained model.

The images need to be processed.

- In Mimics 16.0, CT-scans are used to segment bone tissue (femur, fibula and tibia) and markers. Markers measures are tested for accuracy.
- In Mimics 16.0, MRI-scans are used to segment the soft tissue (four ligaments, three cartilages and two menisci). Segmented parts from CT and MRI are anatomically correct registered on to each other, fitting the bone tissue to the soft tissue. The outcome of this step is a 3D knee model, containing the stl-files of every tissue in correct relative positions.

A first model is created for the influence of Abaqus parameters.

- Out of the stl’s, Volumetric meshes are generated (3 matic v. 8.0) for the soft tissue and surface meshes for the rigid bones. An Abaqus / CAE - model is created and the influence of
H2. Method & Utility of the master dissertation

different settings is analysed. This knowledge is used for developing Abaqus input files in pyFormex. This procedure continues until a stable simulation strategy is obtained.

Stl files of the image processed tissues and excel files of the kinematics are joined with a pyFormex script.

- The spheres of both markers from initial stl’s are projected onto the markers’ position from the optitracker for every time step, resulting in the movement of the segmented bone tissues.
- Soft tissues are inserted to the pyFormex script and combined with the movement. Meshes, material models, contact properties, boundary conditions and kinematics are loaded and an .inp-file is created to run simulations in Abaqus.

At last, simulations are performed in Abaqus and postprocessing is done. Parameters are adjusted in order to improve simulations. This feedbacks the pyFormex script and meshing. This procedure continues until a stable simulation strategy is obtained.

2.2. Utility of the master dissertation

This master dissertation is framed within a greater project at UZ Gent, aiming to develop a virtual knee simulator tool mimicking the experimental knee tests. From CT/MRI-scans, a GUI guides non-engineers through the procedure of segmenting, meshing, evaluating kinematics and run simulations within finite element analyses (FEA). Supplementary information can be provided. Because some hiatuses on knee kinematics are still present, the tool can provide information about how the knee works. This also might be useful to assist surgery and test techniques of surgeries.

Within this scription, we managed to get an imaged knee segmented (stl’s). A pyformex script automatically meshed the stl’s and combine them with the kinematic files from the optitracker, resulting in (semi-)automatic stable simulation in Abaqus.
PART II: Imaging and Image processing
Development of the a 3D knee model for FEA is done in three steps. Images showing the exact anatomy are taken first (=imaging). Based on the images, rough 3D models of all the individual tissues are obtained (= segmentation). The rough geometrical models are processed in order to get realistic 3D surfaces. The last step is meshing: 3D models are triangulated properly for FEA.

For the first Abaqus model, the resulting STL’s are processed to volumetric meshes that are necessary for the simulations that will be done in Abaqus. Segmentation is done in mimics v16.0, meshing in 3-matic v8.0.

When Abaqus input files will be created in pyFormex, only processed geometrical models of the tissues are used as input. Volumetric meshing is automatized by a link to tetGen.

For this project, two sorts of medical images are used: CT and MRI images.

### 3.1. CT-scans

CT-scans rely on the attenuation of X-rays within tissues. X-rays are shot through the scanned object and captured on a collector screen on the other side. When a tissue attenuates a lot of X-rays, barely no electrons are caught on the screen. This results in white regions on the collector screen (bone tissue). Small absorption leads to black regions (air and soft tissue). By rotating around the scanned object and performing a certain amount of shots, a 2D map of the attenuation coefficients can be reconstructed mathematically. This rotation is done in several axial positions resulting in an array of 2D maps of the axial positions and via interpolation a 3D map (=CT scan) is obtained.

Attenuation of the X-rays is mainly influenced by the density of the tissue. As the density of the bone and soft tissue differs significantly, bone is delineated clearly. This makes that CT-scans are used for segmentation of bone tissue. Because metal is prohibited in MRI scanners, spheres and markers are imaged with the CT as well.

### 3.2. MRI scans

The hydrogen concentration within tissues is the physical parameter captured in Magnetic Resonance Imaging (MRI). A homogeneous, strong magnetic field in the scanner forces the spin of all hydrogens nuclei align the direction of the field. A radiowave with the same frequency of the spinning hydrogen nucleus makes the hydrogens spin resonate so that it deviates from the alignment. When the radiowave is removed, the hydrogen nuclei realign the magnetic field and
photons are emitted. These photons are captured by a screen. By adapting the amplitude of the radiowave gradients and sequences, the concentration of hydrogen can be measured in every pixel of the field of view (FOV). This technique delineates the soft tissues better.

Geometrical constraints of the FOV in the MRI-scanner made it impossible to visualize the whole limb, so amputation of the proximal part of the femur and the ankle was needed. This was done by the Department of Anatomy.

For this study, four MRI-scans were made: at 0°, 20°, 60° and 90° flexion. Only one scan will be used for segmentation. The other ones can be used for validation of the kinematic simulations by comparing soft tissue positions.
As suggested above, a hybrid segmentation method. Hereby CT-scans are used for bone tissue segmentation and MRI-scans for soft tissue segmentation. This is followed by registration of the surface models of the bone tissues into the MRI images.

All this is done in Mimics v16.0 from Materialise NV (Leuven, Belgium). This is an acronym for Medical Image Segmentation for Engineering on Anatomy and is part of Mimics Innovation suite, a toolbox for performing a multitude of engineering operations starting from medical imaging data. Medical images can be processed in order to get highly accurate 3D models of patient’s anatomy. In this master dissertation, Mimics was also used for segmentation, boolean operations between tissues and registration of segmented bones within the MRI-images. The output of mimics are stl-files describing the physical surface of the studied tissues.

Generally, segmentation is started from thresholding or region growing followed by a lot of manual work. In following sections a clear overview is given of the segmentation strategy. Nowadays a lot of research is done for developing automatic segmentation tools. Some recent papers tend to have produced such a tool with MRI-scans for bone and cartilage tissue (automatic segmentation pdf, ). No commercial tools are available for lower limb segmentation.

4.1. A first segmentation

A first segmentation was done on a first knee. From that, a general for segmentation was developed and applied to this knee. Some knowledge of segmentation was gained too.

The first knee was fully extended. In that position the PCL is bended and not stressed and thus having a high cross-section. This causes that the PCL is segmented too thick and material models from the real ligaments do not fit. The PCL is more aligned and stressed with higher flexion. Another plus for segmentation in higher flexion angle is the fact that validation will be done with the MRI-scans in different angles. If segmentation is done in full extension, there will be a higher error induced in the 90° flexion than if segmentation was done with some flexion.

For those reasons, segmentation of MRI was done in 30° flexion. No angle was specified for CT-scans as the rigid tibia and femur will be positioned onto the MRI-scans.
4.2. Bone/Marker/Sphere segmentation in CT-images

Four parts have to be segmented: femur and tibia, both connected and disconnected to their markers and spheres. The connected version is needed for fitting position of the spheres of the positions from the kinematic tests. The disconnected version is needed for principal axes later in pyFormex script.

Markers are made out of metal with a very high attenuation coefficient. On the histogram of the CT-scans we notify the high frequencies at high values, showing the markers. The two markers are segmented by selecting only the high values in the histogram. First, the largest part is kept. A boolean operation “two markers minus largest marker” isolates the smallest marker. Some isolated pixels are removed with region growing within every marker.

Spheres have other material properties and are processed manually. In order to segment efficiently and have isomorphic spheres, only one is segmented slice by slice. Via translations, all other spheres are set in place.

![Figure 4-1: the markers each having three spheres](image)

To validate the CT-scan and segmentation, the real dimensions of the markers are measured and compared to the dimensions in Mimics. The distance between all spheres is measured virtually and physically (table 3.1).
The femur is segmented by selecting a range of [1430, 4037] units and labelled by maskFemur. Some pixels close to the markers are eradicated in maskFemur with the ‘Edit Mask in 3D’-function. ‘Dynamic region grow’ removes the isolated parts. Close to the CF, no smooth surface was obtained by automatic segmenting. A smoother surface is manually obtained with axial ‘multislice interpolation’. Off the caudal part of the distal epiphysis, interpolation was done coronal so that more slices could be used to determine the actual edge of the femur. The anatomy was checked and improved in 3D view and sagittal view until a realistic mask was achieved.

For segmenting the tibia (with fibula), a similar approach was used. These stl’s of the femur and tibia are the unconnected versions, shown in figure 3.5.

Spheres, markers and the conforming bone tissue have to be merged the connected versions. The translated spheres have to be merged to the extremities of the marker, resulting in a holder. The latter has to be merged to the femur or tibia. This can be done with Meshlab, a tool developed with the support of the 3D-CoForm project. “MeshLab is an open source, portable, and extensible system for the processing and editing of unstructured 3D triangular meshes. The system is aimed to help the processing of the typical not-so-small unstructured models arising in 3D scanning, providing a set of tools for editing, cleaning, healing, inspecting, rendering and converting this type of meshes.” (http://meshlab.sourceforge.net/).

Merging stl’s can be done by importing parts that have to be merged. The option ‘filter’ -> ‘mesh

Table 4-1: A comparison between the physical and virtual measures

<table>
<thead>
<tr>
<th>Length on figure 3.3 or 3.4 [mm]</th>
<th>Measured length [mm]</th>
<th>Relative fault [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>110.75</td>
<td>111.78</td>
<td>-0.9</td>
</tr>
<tr>
<td>106.31</td>
<td>106.84</td>
<td>-0.5</td>
</tr>
<tr>
<td>101.10</td>
<td>101.41</td>
<td>+0.3</td>
</tr>
<tr>
<td>101.62</td>
<td>102.35</td>
<td>+0.7</td>
</tr>
<tr>
<td>109.42</td>
<td>109.21</td>
<td>-0.2</td>
</tr>
<tr>
<td>118.53</td>
<td>118.24</td>
<td>+0.2</td>
</tr>
</tbody>
</table>
layer’ -> ‘flatten visible layers’ joins all visible stl’s to one. Then, the layer can be exported as stl. By first importing the three spheres with corresponding marker, these can be joined to the whole holder. The resulting holder can be merged with the corresponding bone. The output is the connected model of both tibia and fibula (figure 3.6).

Bone tissues are processed in 3-matic as they are too raw for further processing. Some smoothing and wrapping is applied. This procedure is explained later on.

![Figure 4-3: The femoral unconnected version](image1)

![Figure 4-4: The femoral connected version](image2)

### 4.3. Bone registration in MRI-images.

When the stl’s of the bone tissues are imported in mimics of the MRI-scans, they do not match the bone tissue of the scans. In order to get the stl’s of the bone tissue in the right place, three steps have to be taken. A rough dummy mask of the femur/tibia has to be created by selecting pixels on the MRI scans, the stl has to be relocated close to the dummy mask and a global registration from the stl to the dummy mask has to be performed. This procedure is first done with the disconnected models. Registration is more accurate as physically the same tissues are used. The influence of the connection to the markers in registration can be seen as well.

For the tibia (& fibula), the threshold range was chosen between 910 and 1403 units. Only the greatest part was kept and every tissue superior to the tibia was removed by ‘Edit Mask 3D’. With the ‘multislice interpolation’ and ‘Edit Mask 3D’-function close pixels are removed. The obtained model of the tibia is rough (figure 3.7). For registration no further refining has to be done. Since the imported tibia is far from the rough model, it is moved close to the tibia (figure 3.8). This needs to be done because otherwise Mimics does not converge registration. In the options of global registration, a high number of iterations is chosen because the registration is not very demanding. (figure 3.9, 3.9b).
An analogue approach was used for the connected model. No difference in positioning of the tibia was observed figure.

The same procedure is done for the femur. No difference in positioning of the connected and disconnected femur was observed neither. We conclude that the connection of the marker has no influence on the registration of the stl’s to the MRI-scans.

Figure 4-5: The rough tibia from semi-automatic segmentation in MRI scans

Figure 4-6: The tibia is brought close to the rough mask

Figure 4-7: Registration of the tibia from CT scans to the rough tibia

Figure 4-8: Registration settings
H4. Segmentation
4.4. Segmentation in MRI-images: generals

The main strategy for segmenting tissues starts with a first guess of the main thresholding / region growing. Next, some labour-intensive tools (‘multislice interpolation’, ‘Edit Mask in 3D’, ...) are applied in order to (un)mark pixels that belong to the mask of the tissues. Interpolation was done within the view with the most slices containing the tissue. Other views are used for checking and improving segmentation. Raw segmented parts of every tissue are obtained.

Because tissues are physically connected (i.e. cartilage to bone, menisci to cartilage and ligaments to bone), some overlap is needed. A boolean operation was applied to remove the redundant pixels of the mask. Tissues are connected properly then.

The boolean operation was done within Mimics by exporting the raw segmented parts as stl’s. These stl-files are edited by smoothing/wrapping in 3-matic as they are too raw for processing. By importing the edited stl’s, a boolean operation can be performed and stl’s are re-exported to 3-matic for meshing.

The surfaces that do not overlap, need a high anatomical correctness and a high demand is set for segmentation. For example the CF: the bone needs a high anatomical correctness with the surface facing mensici and the cartilage must have overlap into the femur (figure 3.10). No high anatomical accuracy is set to surface facing the femur. The femur is said to be the master, whereas the CF is the slave. A table gives the master-slaves in the model. (figure 3.11)

<table>
<thead>
<tr>
<th>Surface in between</th>
<th>‘Master’</th>
<th>‘Slave’</th>
</tr>
</thead>
<tbody>
<tr>
<td>bone and cartilage</td>
<td>bone</td>
<td>cartilage</td>
</tr>
<tr>
<td>bone and ligament</td>
<td>bone</td>
<td>ligament</td>
</tr>
<tr>
<td>cartilage and meniscus</td>
<td>ligament</td>
<td>Meniscus</td>
</tr>
</tbody>
</table>

Table 4-2: master and slave surfaces in segmenting. In boolean operations the slavesurface is subtracted with the mastersurface. A higher demand / accuracy is set to the mastersurfaces.

Figure 4-9: Overlap of the CF (purple) in the registered femur (red contour)

Figure 4-10: Boolean operation: Red - Green
4.4.1. **Segmentation of the cartilage**

Three cartilages have to be segmented: medial tibial cartilage (CTM), lateral tibial cartilage (CTL) and femoral cartilage (CF). Because of the irregular white value at the surface of the cartilage, no semi-automatic function could be used. ‘multislice interpolation’ is used in sagittal and coronal view for both CTM and CTL. Because of the high curvature of the CF different views are used. The superior parts of the CF are interpolated within the axial view and for the inferior parts the sagittal and coronal views were used.

Surfaces facing the meniscus are segmented with high demand and accuracy. In 3-matic, the stl’s are processed by smoothing and wrapping. Because a small overlap between CF to both CTM and CTL was seen, a boolean operation was needed. Here, the CF was considered to be the master. Boolean operations are executed between tibia - CTM, Tibia - CTL and femur – CF.

4.4.2. **Segmentation of the ligaments**

Four ligaments need to be segmented: the anterior cruciate ligament (ACL), the posterior cruciate ligament (PCL), the lateral collateral ligament (LCL) and the medial collateral ligament (MCL). Although ligaments are clearly seen on the images, automatic segmentation is not possible. A lot of manual work is done with ‘Multislice interpolation’ and ‘Edit Mask in 3D’ according to the previously explained strategy. Ligaments are smoothed, wrapped and then cut by both femur and tibia (fibula for LCL) in a boolean operation.

4.4.3. **Segmentation of the menisci**

Two menisci have to be segmented: the medial meniscal ligament and the lateral meniscal ligament. First, ‘region growing’ is applied as the tissue is significantly darker. As the menisci overlap the cartilage tissue, only the inner and outer boundaries are checked. Then the meniscal overlap is cut by the CF and CTM/CTL.

4.5. **The knee 3D model**

All stl’s are joined and the knee model (figure 3.12) was positively evaluated by orthopaedist Dr. Nico Vanderhauwaert (AZ St. Augustinus Veurne) and Prof. Van Hoof of the UGent department of anatomy.
Figure 4-11: Overview knee

Figure 4-12: Back of the knee
Figure 4-13: Front of the knee
H5. Meshing

Meshes are assigned to the different tissues so that calculations can be done. It’s done in 3-matic v8.0. Like Mimics, 3-matic is part of the Mimics Innovation Suite of materialise (Leuven, Belgium). The stl-files from Mimics are imported and processed to high quality volumetric / surface meshes.

5.1. General meshing of soft tissue

3-matic is used twice for the soft tissues: for editing the parts before and after boolean operation in Mimics. (see generals of segmenting)

The rough surfaces from Mimics are smoothed, wrapped and locally proceeded. Auto-remesh was applied. The surface mesh was exported Mimics for the boolean operation.

After the boolean operation, triangles are damaged and some smoothing of the sharp edge is needed as well. Sharp edges cause peak tensions and highly deformed volumetric triangles, resulting in errors in Abaqus simulations. ‘Quality preserve triangles’ with a low geometrical error reduces .

The fix wizard controls the mesh errors during processing. Supplementary operations are done until no errors are given.

The inspection page evaluates the quality of the mesh. A histogram shows the frequencies of measured equi-angle skewness. When a large amount of orange-red bars is shown, meshing is improved or redone (figure 5.1).

Only if both fix wizard and inspection page are positive, a volumetric mesh with low demand is created and exported to Abaqus. Volumetric and surface mesh are controlled with ‘verify mesh’.

As a low demand volumetric meshing in 3-matic generates a good mesh in Abaqus (max. 2% of error), automatic volumetric meshing in pyFormex is not set a high demand (figure 5.2).

Figure 5-1: Inspection page. All the equi-angle skewnesses are in the green-yellow field
5.2. General meshing of hard tissue

No boolean operation nor volumetric meshing has done for the hard tissue. The raw segmented parts from Mimics are processed with smoothing/wrapping and local operations. A high quality surface mesh is made with a low geometrical error. Control was performed only in Abaqus as the mesh will not be transformed in pyFormex.

5.3. Volumetric meshing in TetGen

The surface meshes of the soft tissues are imported into the pyFormex script and volumetric meshes are generated by an external program: tetGen, made by Hang Si (Berlin). “Tetgen is a robust, fast, and easy-to-use software for generating tetrahedral meshes suitable in many applications”. It partitions the 3D surface stl’s into tetrahedrons by Delaunay triangulation. Delaunay triangulation aims to create a 3D space with tetrahedrons of which all circumspheres of each tetrahedrons does not contain a point of another tetrahedron. This is shown in following 2D-picture (figure 3.15): V1,V2 and V4 make the triangle T1 and the green circumsircle does not contain V3. This triangulations are said to be a Delaunay triangulated. This is not the case figure 3.16. Non-Delaunay triangulated meshes tend to have sharp edges, which results in bad meshes. Hang Si implemented his algorithm for Delaunay triangulation in tetGen.

How the meshing-tool was used will be explained in the program.
5.4. **Meshes imported into pyFormex: stl_read.py**

The meshes in stl-format are imported into pyFormex by the program stl_read.py. Meshes from both hard and soft tissue are read and converted to a common a .pgf-file. Only this file is read by the pyFormex script.
PART III: Kinematics
H6. Kinematic tests

6.1. Preparation

A fresh frozen cadaver was prepared by the Department of Anatomy with an agent preventing the cadaver from stiffening. Two markers with each three infrared reflecting spheres were rigidly mounted on both femur and tibia of the right leg. No recommendations were considered for precise positioning.

The optitrack system (natural point – optitrack) was installed. Four posts were placed in a square, each containing two cameras focussing on the central place in the square. The eight cameras (1.3 MP) were calibrated by a dummy object with known measurements. By evaluating the error of the geometry measurements produced by the optitrack, the whole system was tested. During this calibration the coefficients of the direct linear transformation were determined. This transformation relates the coordinates viewed on the screen (2D) to the physical coordinates (3D). When the error was too high, some artefact from the surrounding were looked for and removed. An accuracy of 0.2 mm was noticed.

![Kinematic test setup](image)

Figure 6-1: Kinematic test setup - This specimen is placed in the focus of the tracking system

6.2. Tests

The cadaver was placed in the central region of the focussing cameras and the six spheres were visualized on the Motive software. After stabilizing of the movement of the spheres, tracking was
started. The 3D positions relative to the defined coordinate system of every makers was written at a frequency of 35 Hz to .csv-files.

Three rotations of the knee were tested: flexion-extension (FE), endo-exorotation (EE) and varus-valgus (VV). Tests of EE and VV were performed at three FE-angles. For every test, five cycles were executed. Tests were performed in open chain and manually by a medical doctor.

Figure 6-2: Kinematic testing - FE
Kinematic tests are post-processed with pyFormex. It is a program for generating, transforming and manipulating large geometrical models of 3D structures by sequences of mathematical operations. Unlike traditional CAD systems, pyFormex uses a powerful python based scripting language, enabling to play with large range of parameters. pyFormex is often used to create 3D renderings, Abaqus models and simulations from medical scan images. “You are only limited by your own imagination” (www.nongnu.org/pyformex).

Within this chapter the kinematic tests will be combined with the results of the meshing chapter. The surface geometry models will be submitted to the movements of their markers from the optitrack .csv-files, resulting in the rendered 3D movements.

7.1. Generals: From .csv-files to kinematics and strain analysis

The optitrack system writes all the positions of every sphere to an .csv-file. In order to determine which sphere in the .csv-file belongs to which physical sphere, Transformatie.py is created. The spheres are positioned onto the coordinates of the csv-file. If this is done for every step, the movement is acquired. Kinematics are analysed according to the paper: “a joint coordinate system for the clinical description of three-dimensional motions: application to the knee” by E.S. Grood and W.J. Suntay (1983).

The segmented ligament should fit both femur and tibia precisely if simulations are performed in Abaqus. This is needed because of the attachments in Abaqus. Structures that are too far from each other, can’t get attached. This requirement is not fulfilled for any step of the kinematics above. This is solved by applying a correction to the relative position when the relative positions in kinematics are ‘sufficiently identical’ to position fitting the ligaments precisely (i.e. the anatomical correct position). After the correction, relative movements are calculated further.

7.2. Transformatie.py

The coordinates of the spheres in the segmented parts are different from the conforming coordinates in the csv-files since two totally different coordinate systems are used in segmenting and kinematic tests.

The coordinates of all spheres of the pgf-file are picked in pyFormex. The csv-file has 6 set variables for every step: one set of x,y,z-coordinates for all six spheres. The right sequence of the markers is not known before. The conforming points have to be searched first. This is done with transformatie.py by going through all the possibilities. Only when the tibia and femur are aligned, we can conclude that the right sequence has been found.
An example is given:

Left in figure 7.1 (pyFormex) the coordinates of points 1-6 are picked. The coordinates from 1’-6’ are given in the csv-files, resulting in a total different positioning of the tibia to the femur. In order to get a relation between the two coordinate systems, the conforming points have to be placed onto the right sphere. In this example, point 1 conforms the 3rd set in the csv-file, (1-> 3’, 2 -> 1’, 3 -> 2’, 4-> 4’, ... ).

An important note has to be made:

By querying the centre of spheres in pyFormex NOT THE EXACT coordinates are picked (finite size of spheres, accuracy of distances in CT-scans) and in the results of the optitrack, NOT THE EXACT coordinates of the centre of the spheres are written (an accuracy of 0.2 mm is measured, it’s not known on which part of the spheres coordinates are captured). Because makers are placed far from contact femur-tibia, a small fault on the coordinate of one marker so that a greater fault is induced on the femur/tibia contact. This is shown in figure 7.2. A correction to the position of the tibia will be performed to counter this problem. Soft tissues will fit precisely the anatomical correct positions.
7.3. Kinematics: from .csv file to moving object

After importing the stl-files, the relative movement of the tibia to the femur is obtained by three steps:

- The raw movement generated by the .csv-datafile is obtained.
- A correction is applied to the tibia.
- The relative movement of tibia to femur is calculated off the correction.

7.3.1. Raw movements

For the femur and tibia, the positions of each sphere from the segmented parts is written to another .csv-file. This file is named ‘origin .csv-file’. This coordinates have to be matched onto the coordinates of the correspondent sphere in the csv-file. This is done for every step so that a continuous movement is obtained.

In pyFormex a function is defined for doing that:

\[ \text{A.position}(X,Y) \]

The geometrical rigid object A contains a nodeset X with points X1, X2 and X3. The nodeset Y also contains three points Y1, Y2 and Y3 and is isomorph to X. By applying the function A.position(X,Y), a geometrical object is obtained with the form of A and X positioned aligned with Y. This procedure is shown with a pyFormex example.
Figure 7-3: Object A (red) has X1, X2, X3. A.position(X,Y) aligns the X and Y and object A is moved (blue). The figure left shows the circumstances before, right after.

This function is applied to both femur and tibia. For every step, the spheres on the stl’s of both femur and tibia are positioned onto the conforming positions of the spheres in the .csv-datafile (transformative.py). The positioned tibia and femur are displayed and the movement is seen. A low pass filter can be applied to soften noise and kinematic faults.

### 7.3.2. Correction for correct anatomy

An anatomical correct model was obtained with the segmentation. Calculations showed that the angle between the principal axes of segmented tibia and femur is 32°. By running the raw movement, the angle between the principal axes of tibia and femur are evaluated for every step. When that angle is within the range of 32° +/- 1°, the models are considered to be 'sufficiently identical'. The position of the tibia is corrected to the anatomical correct place. This is done by applying the femur displacement to both femur and tibia so that they stay in an anatomically correct relative positioning to each other. The same displacement is given to the soft tissues.

Figures 7.4 - 7.5 show the correction of the tibia. The blue tibia is generated by the kinematic data; the red tibia is its correction.
Figure 7-4: Correction of the tibia. Postanterior view (left), anteriorposterior view (right). The tibia is moved medial and superior.

Figure 7-5: Correction of the tibia: Medial lateral view (left), Lateral medial view (right)
7.3.3. **Relative movement**

Now relative movements of the tibia to femur are applied. If not, the correction of the tibia is nullified. Another plus is fixed boundary condition can be set for the femur. This enables more stable and faster simulations.

For relative movement following pseudo-code is used:

\[
\text{partnew} = \text{part}.\text{position}(\text{old position marker of part}, \text{new position marker of part}).\text{position}(\text{new position marker fixed part}, \text{old position marker fixed part})
\]

The movement of the part \text{part}.\text{position}(\text{old position marker of part}, \text{new position marker of part}) is cancelled with the negative movement of the fixed part (\text{position}(\text{new position marker fixed part}, \text{old position marker fixed part})).

7.4. **Kinematics analysis strategy**

Kinematics are analysed with the paper “a joint coordinate system for the clinical description of three-dimensional motions: application to the knee” by E.S.S Grood and W.J. Suntay (1983). The paper is concerned with the description of the knee joint motion for both biomechanicians and clinical physicians. It is considered to be the international standard for analysing knee motion and is highly cited in knee kinematic papers.

In order to describe the relative rotational movement (Flexion/extension (FE), endorotation/exorotation (EE) and abduction/adduction (AA)) of the two bones, three coordinate systems are used. Two fixed to the rigid bodies and a joint one to describe relative movement. In the fixed coordinate system of the femur, the axis around which FE occurs can be described. Two points are selected in an anatomical way explained in the paper so that this axis can be reconstructed. This is also done for the axis on the tibia describing EE. These two axes are called the body axes. By taking the cross product of the two body axes the floating vector F is defined. The body axes and F describe the relative coordinate system. The vector F can be projected onto the planes perpendicular to the two body axes. In those two planes, an anatomical vector was constructed (procedure by paper). The angle between this vector and the projected floating axis determines FE and EE. AA + 90° could be captured by the angle between the two body axes.

The relative translation displacements (lateral tibial (LTD), anterior tibial (ATD), joint distraction (JD)) are the movements of two –anatomically correct - selected points projected onto the body axes and floating axis.

The implementation in pyFormex is discussed in next section and can be reviewed in the .py-file.
Figure 7-6: The fixed coordinate systems. The body axes are X (femur) and z (tibia). The planes perpendicular to the body axes are coloured grey. (Grood and Suntay)

Figure 7-7: The H vector connects the origins and is projected on the body axes and floating axis F. (Grood and Suntay)

Figure 7-8: The definition of the rotations (Grood and Suntay)
7.5. **Kinematic analysis implemented in pyFormex**

All the anatomical points from the paper of Grood were selected manually with a 'query'. The coordinates of these points are evaluated in every step. In every step the body axes and the axes in the plane perpendicular to the body axes were calculated and all these vectors are joined in a dictionary. When all vectors from one step were known, the floating axis F (crossproduct of the body axes) and translations vector H (origin femur – origin tibia) could be determined. FE and EE were calculated with the body axes and plane axes, AA with the body axes. LTD, ATD and JD were projected (dotproduct) onto the body axes and floating axis. All the latter values (FE, EE, AA, LTD, ATD and JD) are saved for every step in the dictionary kinematicCoordDic. The nomenclature within the dictionary is conform with the paper of Grood & Suntay.

AA, FE, EE, LTD, ATD, JD were plotted with Gnuplot and the results are shown below. These results are discussed in the master dissertation of Maarten Vanneste.

![Figure 7-9: Rotations defined by Grood and Suntay](image)

![Figure 7-10: Displacements defined by Grood and Suntay](image)

7.6. **Elongation analysis**

In the .pgf-file the insertions can be determined with the query function in pyFormex. All point numbers of the ACL, PCL, LCL and MCL-insertions on both femur and tibia were remembered. In every step, the coordinates of the points are determined and stored in the dictionary strainElem. When the points of the insertions on both femur and tibia are known, the distance between the conforming points of a ligament are calculated and saved in a dictionary. Elongations are calculated by the formula:

\[
\text{Elongation} = \frac{l - l_{\min}}{l_{\min}}
\]
PART IV: Modelling
The FEA software used for simulations and modelling is Abaqus. The software is suited for powerful and complete solutions for sophisticated engineering problems in a large range of industries.

One of the products of Abaqus is the Abaqus / CAE (Computer Aided Engineering). This product has an intuitive GUI (graphical user interface) for creating simulations. All parts can be imported or created (CAD), material models can be assigned, assemblies brought together, steps are chosen, loads are created... This is all done in a visual way. After creating the whole simulation setup an .inp-file is created for the calculations in Abaqus/ Standard. The big disadvantage of the CAE is the fact that it is very time consuming and non-parametric.

The .inp-file can be generated with pyFormex scripts in an automatic way. So, the time consuming CAE is thus bypassed and parameters are included easily.

Before developing this pyFormex script, a first model was created with Abaqus/CAE to check the influence of different parameters (meshing/element types, contact properties, boundary conditions, material models,...) and the optimal parameters and strategies are sought. From that, problems with pre-processing are highlighted and automatic .inp-file generation in pyFormex could be done more efficiently.

All modules of the CAE are reviewed in next and optimal settings are highlighted.

### 8.1. Parts

This module is used for creating the geometries. These parts can be generated with CAD-tools or can be imported from other models or from other programs generating meshes.

For this project, all the parts are meshed in 3-matic v. 8.0 (see Image and Image Processing, First processing). In Abaqus, the meshed parts are joined in one common model with ‘Copy Objects...’ in the Model tabulation from the toolbox. As the parts are meshed in 3-matic, no changes can be applied to the ‘ophran mesh’. For the hard tissues a surface mesh was imported (rigid body); for the soft tissues a volumetric mesh.

For the ligaments some supplementary geometrical information has to be included since an orthotropic linear elastic material model is used. An orientation between the insertions (axis 3) has to be present. This is done by double-clicking onto the orientation tab in the part. For the set of all elements two points are selected close to the insertions.
8.2. Property

This module is used for assigning properties to the nodes and elements. Both geometrical and material information are ascribed to all the different parts: densities, thicknesses and material models,...

Generally, properties are assigned by first creating a material. That material is included in a section definition with some supplementary information. The sections are assigned to the conforming element sets of the tissue part.

Hard tissue is considered to be rigid. The rigid property is considered as boundary conditions. In Abaqus, the rigid boundary condition is accompanied by a shell material definition. A low thickness (0.001mm) is chosen for the shell. This material is included in the rigid section that’s assigned to the elements of the bone tissues.

For the soft tissues, a solid homogeneous material model is applied. For the cartilages and menisci, a linear elastic model was selected as suggested by Isabelle Waterplas (2012). Densities are included for Abaqus’ will to 1.0 $e^{-9}$ ton/mm³. The set ($E$ [MPa]; $\nu$ []) for resp. cartilage and meniscus are (10 ; 0.45) and (58 ; 0.48). Ligaments are suggested to be modelled with an anisotropic linear elastic material (Elastic -> Engineering Constants). These constants are shown in following table. The orientation connected to the elements is automatically selected.

Table 7.1 (axis 3 is longitudinal to the ligament)

<table>
<thead>
<tr>
<th>Material constant</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_1$</td>
<td>34.5 MPa</td>
</tr>
<tr>
<td>$E_2$</td>
<td>34.5 MPa</td>
</tr>
<tr>
<td>$E_3$</td>
<td>345 MPa</td>
</tr>
<tr>
<td>$\nu_{12}$</td>
<td>0.22</td>
</tr>
<tr>
<td>$\nu_{13}$</td>
<td>0.022</td>
</tr>
<tr>
<td>$\nu_{23}$</td>
<td>0.022</td>
</tr>
<tr>
<td>$G_{12}$</td>
<td>14 MPa</td>
</tr>
<tr>
<td>$G_{13}$</td>
<td>168 MPa</td>
</tr>
<tr>
<td>$G_{23}$</td>
<td>168 MPa</td>
</tr>
</tbody>
</table>

An isotropic hyperelastic material model for the ligaments was suggested by E. Peña. Neo Hooke material models for $C = 0.7324$ MPa and $D = 0$ MPa (=incompressible) are applied. Since the tension of a incompressible neo hooke is linear to the C-value. The influence of other C-values is analysed easily.

In the appendices, the material models are discussed more extensively.

For both ligament material models simulations are run.
8.3. Assembly

The independent meshed parts are joined within the assembly module. Referential points for femur, tibia and fibula are selected and a set is created, referring to each of these referential points. Surfaces are created for the contact definitions later (section 8.5).

8.4. Step

Because large deformations will take place, nlgeom has to be enabled in the step definition. The maximum amount of steps is raised to 1000 and minimal step to $1.0 \times 10^{-5}$ sec. When the simulations take steps lower than this, no good outcome is obtained. As the aim of the first model is checking influence of the main parameters / settings, the output of the simulations is accessory and defaults are kept. No dynamic influences are expected, so a standard analysis is advanced to explicit calculations.

8.5. Interaction

Two types of interactions are introduced: fixed interactions and moveable interactions. With the fixed interactions, two structures are rigidly tied to each other. This is the case for the interaction between with the bones for the ligaments and cartilages. The interaction is also given in between the menisci and cartilage of the tibia. A Tie Constraint is created for this interactions by double-clicking on the ‘Constraints’ in the model database. The tissue that is the least dense meshed is chosen as master surface, the slave is higher dense meshed. Because of the boolean operations after segmentation, the places of fixed interaction are delineated clearly. Surfaces are easily selected by hiding all the other tissues and selecting the surfaces ‘by angle’. For the bones, all elements are selected. In the options ‘Surface To Surface’ is preferred to ‘Node To Surface’ because the latter postulates total no overlap of any node in to the master surface. This causes peak tensions and highly deformed meshes. ‘Surface to Surfaces’ postulates no average overlap and is more appropriate for irregular surfaces as here. “S-to-S discretization reduces likelihood of snagging”. It is thus a safer approach for simulations. Adjustment of the slave is not enabled because it deforms the surface mesh, causing the quality of the volumetric mesh. Initial clearances are close enough for doing that.

For the moveable contacts, Contact Properties are chosen. First, the interaction property is created. For this, Surface to Surface is chosen with finite sliding. “Small-sliding formulation: Approximation intended to reduce solution cost; limited applicability” and “Finite-sliding formulation: General applicability” (http://marketing.intrinsys.co.uk). Interactions are created with previous defined interaction properties.
The rigid bodies are defined within the ‘constraints’ part. The elements are selected and for the RF the correspondent set is assigned.

### 8.6. Load / Boundary Conditions

For testing the parameters / setup, different sets of loads are applied to the reference point of the femur. The sets of the RP of the femur is submitted to different loads. The magnitudes of their amplitude gets the knee bended with a realistic movement. The reference points of both tibia and fibula are fixed.
8.7. Mesh

This module is neglected as meshing was done in 3-matic. Control on mesh quality in Abaqus was done before. C3D4H elements are opted for the Neo Hookean material model, C3D4 for the linear elastic. For the rigid tissues R3D3 elements are used.

8.8. Conclusions from the first model

From the first model some conclusions for the development of the pyFormex script can be made.

8.8.1. Material models

The orthotropic linear elastic material converges much slower and when the flexion rises, triangles close to the tie constraints are highly deformed. This causes peak tensions and errors because of too low time step increments. Figure 8.4 shows the differences in tension distribution. For the Neo Hooke material model a more uniform tension distribution is seen.

The isotropic hyperelastic material with neo-hooke description is preferred for automatic Abaqus .inp-file generation.

Figure 8-4: Stresses in early flexion with a linear elastic model (left) and a neo-hookean material model (right) with 80% of the peakstress. A more uniform distribution is seen with Neo Hooke.
8.8.2. Contact Constraints

Cartilages have a small thickness and by applying forces, these surfaces are deformed largely. When the forces on the cartilages are too high, the Tie Constraint is not fulfilled anymore and cartilages disappear in the tibia/femur. Triangles are largely deformed and simulations stock. Figure 8.6 shows the influence of too high contact constraint on the cartilage. The triangles right on the figure are highly deformed, causing unstable simulations and errors. As the simulations generated by the pyFormex-script will be kinematic driven, it is possible that the femur and tibia come too close due the inaccurate kinematic description in the contact region. No Contact Constraints will be included in the pyFormex generated .inp-file.

![Figure 8-5: A overview (left) and detailed view (right) showing the cartilage overlap in the tibia. The orange line shows the edge of the tibia. In the green cadre triangles are deformed exessively.](image)

8.8.3. Meshing

From the segmentation and meshing of the first model, it has been observed that high quality surface meshes are the basis of high quality volumetric meshes. Even with low demand for the volumetric meshing, no errors are seen in Abaqus.
8.8.4. **Minorities**

- No high demand should be set for the surface determination. When elements are too far for connecting, they are automatically ignored in the analysis.

- As fibula and tibia are taken fixed, they can be considered to be one tissue that rigidly connected to each other. Only one referential point is thus needed.

- The generated nomenclature of the .inp-file has to be analysed for adapting the fe_abq.py-file in pyFormex for the non-standard options in Abaqus. This will be explained later on.
The part handles the development of the pyFormex script for generating an Abaqus input file for finite element modelling and simulations.

Following steps are taken:

- An .pgf-file is automatically meshed and element types are assigned
- Relative movement of the tibia to the femur from the kinematics are exported to the .inp-file. A boundary condition is set to the fixed femur.
- Material models (neo hooke, linear elastic orthotropic, DEPVAR 12) are assigned to the different tissues
- Contact properties (only Tie Constraints) are applied

The represented script is based on a pyFormex-script of Wouter Devriendt. He managed to reproduce the kinematic tests in an Abaqus simulation. Adaptations of the existing script are done: soft tissues are included and parameters are changed for speeding up and stabilizing the simulations. As discussed in the conclusions of the first model, only the ligaments with the kinematic data of hard tissue are be included in the .inp-file. The simulations can be broadened easily with the contact constraint of menisci and cartilages by enabling the lines of comment and including the tissues. This is indicated in the script.

9.1. General .inp-file generation with pyFormex

Generally, The script has four steps. An assembly is created first: all geometrical parts are imported and if they are not meshed, meshing is applied. The meshed parts are joined in an assembly. Properties are saved in a property database and assigned to all the needed elements and nodes: material models, element types, reference nodes,… A third step consists of applying the boundary conditions and loads to the conforming elements and nodes. Finally all outputs and calculation parameters are defined.

The setup defined in previous four steps are used for writing the .inp-file. This is mainly done with fe_abq.py. This pyFormex script provides a standard part of the Abaqus feature range. Because the neo hooke and orthotropic linear elastic material model are not included in the standard, some adaptations have to be done to this script. This is discussed in section 9.6.

An overview of the steps is given:
9.2. Assembly of the imported tissues – Mesh generation with tetGen

The pgf-file, generated in read_stl.py is read and all geometry models are loaded. Because the assembly needs all meshed parts, meshing is done. The geometry models of the hard tissues are used as surface mesh. Automatic volumetric meshing of the soft tissue geometry models is obtained with a link to tetGen. Following code is used:

```python
tetgen(stlpart, quality=meshquality, volume=None, filename=None, format='.stl').setProp(propertynumber)
```

The maximum size of the triangles is set with ‘meshquality’. If False, no requirement is set. If True, 1mm is set. A float for meshquality sets the maximum size directly.
In the GUI, one can choose the meshing setup. The maximum edge lengths and number of elements for every setup are shown in following tables.

<table>
<thead>
<tr>
<th>Meshing setup (GUI)</th>
<th>Coarse</th>
<th>Middle</th>
<th>Fine</th>
<th>Very Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum Edge Length [mm] / meshquality</td>
<td>False</td>
<td>6</td>
<td>1.8</td>
<td>1.4</td>
</tr>
</tbody>
</table>

Table 9-1: The used meshquality / maximum edge length for the meshing setups

<table>
<thead>
<tr>
<th>Meshing setup (GUI)</th>
<th>Coarse</th>
<th>Middle</th>
<th>Fine</th>
<th>Very Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACL</td>
<td>4 275</td>
<td>12 539</td>
<td>20 748</td>
<td>29 749</td>
</tr>
<tr>
<td>PCL</td>
<td>3 539</td>
<td>10 458</td>
<td>16 740</td>
<td>23 587</td>
</tr>
<tr>
<td>MCL</td>
<td>2 704</td>
<td>9 171</td>
<td>13 494</td>
<td>19 460</td>
</tr>
<tr>
<td>LCL</td>
<td>2 381</td>
<td>7 885</td>
<td>10 918</td>
<td>14 920</td>
</tr>
</tbody>
</table>

Table 9-2: Amount of meshes for every setup

In the tetMesh()-definition on line 257 of the script, the option ‘q’ within tetGen is chosen. This enables the program to modify and refine the surface mesh in order to get higher quality volumetric meshes.

Because the meshes are used in Abaqus, the meshes are controlled with the ‘Verify Mesh’-function to control meshing errors. No errors are generated for any setup or tissue. Only 1.83 % of elements are warnings.
An assembly of all meshed, numbered parts is created by mergedModel() form fe.py. This assembly will be used directly for the Abaqus data export.

### 9.3. Property database

#### 9.3.1. Assigning material model/ reference points / sections

The properties are assigned to the different element sets. A property database is created for enclosing all assignments.

A rigid section is created with the generateRigidSection() -function. That rigid section is applied to the R3D3 element sets of the bone tissue.

For the soft tissue properties, material and section characteristics of all tissues are loaded with the function inreadmaterial(). The material and section definitions depend on the selected material model (GUI).

#### 9.3.2. Making surfaces

There’ll be interaction between the different tissues: ligaments and cartilages are fixed to bones and cartilage has contact with menisci. For those interactions, surfaces need to be defined first.

This can be done manually in Abaqus/CAE or automatically in pyFormex.

For automatic surface generation, a sequence of operations is done on the tissuesurface so that for any 3D model of the tissue, the same surface is selected.
When the surfaces are defined manually, a dummy surface is created in pyFormex and adjusted in Abaqus/CAE. Because the latter could be done very fast and accurately, no automatic surface generator is developed.

For an interaction two surfaces are involved. A general name is given to the surfaces: “S ‘name of tissue - name of interacted tissue’”. Two types of interaction are involved in the knee: connected or gliding. A connected interaction is modelled with a tie constraint TC, a gliding interaction with contact constraint CC. Optimal settings of these are taken over from the first model.

As discussed in the conclusions before, no CC are be used. The automatic creation of CC is still included in the script, but disabled with hashtags. Table 9.4 summarizes the surfaces per tissue, table 9.5 gives an overview of the interactions. The parts in grey are not involved in this project.

<table>
<thead>
<tr>
<th>Tissue</th>
<th>Surface</th>
<th>Tissue</th>
<th>Surface</th>
</tr>
</thead>
<tbody>
<tr>
<td>Femur</td>
<td>SFemur</td>
<td>CF</td>
<td>SCf-Fem</td>
</tr>
<tr>
<td>Tibia</td>
<td>STibia</td>
<td>SCf-Mml</td>
<td></td>
</tr>
<tr>
<td>ACL</td>
<td>SAcl-Fem SCf-LML</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PCL</td>
<td>SPcl-Fem SCtcm-Tib</td>
<td></td>
<td></td>
</tr>
<tr>
<td>LCL</td>
<td>SLcl-Fem SCtcm-Tib</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MCL</td>
<td>SMcl-Fem</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 9-3: Used surfaces in the knee. The grey tissues are not involved in this master dissertation but can be included.

<table>
<thead>
<tr>
<th>Interaction</th>
<th>Interaction</th>
<th>Surfaces (master/slave)</th>
<th>Interaction</th>
<th>Interaction</th>
<th>Surfaces (master/slave)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Femur - ACL</td>
<td>TC</td>
<td>SFemur</td>
<td>Femur - CF</td>
<td>TC</td>
<td>SFemur</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SAcl-Fem</td>
<td></td>
<td></td>
<td>SCf-Fem</td>
</tr>
<tr>
<td>Femur - PCL</td>
<td>TC</td>
<td>SFemur</td>
<td>CF - LML</td>
<td>CC</td>
<td>SCf-Lml</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SPcl-Fem</td>
<td></td>
<td></td>
<td>SLml-CF</td>
</tr>
<tr>
<td>Femur - LCL</td>
<td>TC</td>
<td>SFemur</td>
<td>CF - MML</td>
<td>CC</td>
<td>SCf-Mml</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SLcl-Fem</td>
<td></td>
<td></td>
<td>SMml-CF</td>
</tr>
<tr>
<td>Femur - MCL</td>
<td>TC</td>
<td>SFemur</td>
<td>Tibia - CTM</td>
<td>TC</td>
<td>STibia</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SMcl-Fem</td>
<td></td>
<td></td>
<td>SCtcm-Tib</td>
</tr>
<tr>
<td>Tibia - ACL</td>
<td>TC</td>
<td>SFemur</td>
<td>Tibia - CTL</td>
<td>TC</td>
<td>STibia</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SAcl-Fem</td>
<td></td>
<td></td>
<td>SCtcl-Tib</td>
</tr>
<tr>
<td>Tibia - PCL</td>
<td>TC</td>
<td>SFemur</td>
<td>CTM - MML</td>
<td>TC</td>
<td>SCtcm-Mml</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SPcl-Fem</td>
<td></td>
<td></td>
<td>SMml-Ctm</td>
</tr>
<tr>
<td>Tibia - LCL</td>
<td>TC</td>
<td>SFemur</td>
<td>CTL - LML</td>
<td>TC</td>
<td>SCtcl-Lml</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SLcl-Fem</td>
<td></td>
<td></td>
<td>SLml-Ctl</td>
</tr>
<tr>
<td>Tibia - MCL</td>
<td>TC</td>
<td>SFemur</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>SMcl-Fem</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 9-4: Interactions in the knee. The interactions in grey are not used in this project.
A row with all surfacenames and a row of TC-interactions ([master surface name, slave surface name]) is created. This enables automatic property assignment. The code generating the surfaces and Tie constraints is given below. No adjustments of the slave and no rotations are enabled, these are derived from the optimal first model.

```python
P.Prop(name = surfacenames[i], set = dummy set, surftype = 'element', label = 'S2')
P.Prop(constraint= '1', name = "TieConst" + str(i), adjust = 'no', master = TCinteractions[i][1], slave = TCinteractions[i][0], norotation = True)
```

### 9.4. Boundary conditions / load

The boundary conditions are set by the kinematic data. The movement of the tibia and fibula is described by the results of the previous kinematic analysis. As the bones are rigid, the movement of the whole body will be ascribed to the movement of the reference point. The movement of the bones is get by selecting three points on the femur and tibia. Onto the bones, three random points are chosen (point nr.1, 500 and 1500) and are evaluated before (coordsbefore) and after the movement of the bones (coordsafter). From coordsbefore and coordsafter, the rotation matrix, translation matrix and axis-angle representation around the ref. point. can be determined by the function computeDisp(coordsbefore, coordsafter, ref. point. of the conforming bone). The axis-angle representation and translation vector are used for displacements in Abaqus and are saved for every step in a dictionary.

The following code is used in a for-loop going over all timestep.
(part is the bone in the previous time step)

```python
coordsprev = array([part.coords[1],part.coords[500],part.coords[1500]])
if femur:
    newfixed = new
    oldfixed = old
    firsts = False
partnew = part.position(old, new).position(newfixed,oldfixed) #fixed femur
coordsnext= array([partnew.coords[1],partnew.coords[500],partnew.coords[1500]])
R, T, phi = computeDisp(coordsprev, coordsnext, RP,  axisAngle = True)
disp[nameRP].append(np.concatenate((T, phi))
```

In the dictionary disp the translation and axis-representing rotations are saved under the name of the RP.

Values for the rotation and translation of the femur are low (magnitude of $10^{-6}$ mm). A fix
boundary condition is set to the femur to improve the speed of calculations.
In pseudocode:

Femur:
P.nodeProp(set= set femur, bound=[fixed], ampl= (disp[‘RPfemur’]), tag='step1')

Tibia:
P.nodeProp(set= set tibia, bound=[1], ampl= (disp[‘RPtibia’]), tag='step1')

9.5. Calculation / output settings

In the GUI the minimal time increment and time step could be chosen. Nonlinear geometrical
effects are enabled in the step definition as high deformations occur and the maximal increments
are augmented to 1000. Default history and field output were chosen.

An .inp-file finally is created with:

all = AbqData(mergedModel([all meshes]), prop= the property database, steps=[step1], bound=['init'])

9.6. Job in Abaqus

For simulations in Abaqus surfaces need to be assigned because some dummy surfaces were
created in pyFormex. This is discussed further.

9.7. Adapting the fe_abq.py

First, a directory ‘mypyplugins’ is created in the current work directory and the fe_abq.py is
copied into this map. From the .inp-file from the first model, the Abaqus nomenclature for the
material model is derived. This nomenclature should be generated by the new fe_abq.py file.
In the first model inp-file, a neo hooke model is declared as:

*Material, name=ACLMATERIAL
*Density
1.2e-06,
*Hyperelastic, neo hooke
0.732, 0.
In the definition fmtMaterial(), the material property is created to match the required nomenclature:

```python
elif mat.elasticity.lower() == 'hyperelastic':
    out += '*HYPERELASTIC, %s % mat.type.upper()
    
    if mat.type is not 'neo hooke'
        out += ', N=%i\n %order
        out += fmtData(mat.constants)
```

This is also done for the linear orthotropic elastic material.

In the main program a referral to the new fe_abq.py is made by adding a new path that has to be checked for plugins.

```python
import sys
sys.path.append("/home/dries/Desktop/Dropbox/KNEE TEST CASE/mypyfplugins")  #directory of the map with the new fe_abq.py
from fe_abq import *
```
Pre-processing within pyFormex

Part V: Simulations & Conclusion
H10. Mesh Sensitivity Analysis

Before explaining how .inp-files are handled further in Abaqus, the optimal meshing setup is sought.

Meshing settings are important for the simulations. The element types and amount of meshes influence the simulations. By executing the simulations with different element types and amounts of elements, the sensitivity of the model to meshing can be evaluated. Another aspect is the time needed for calculations. Time for calculations rises exponentially with the amount of elements but. In the mesh sensitivity analysis the amount of meshes for convergence is determined. From that, the optimal meshing setup is determined.

As element types C3D4H are specially created for hyperelastic materials, no influence of the element types is involved. Only the amount of elements are modified. Because no explicit calculations are done, no influence of the step time is taken into account.

In the pyFormex script, four settings for meshing densities could be chosen. Following table gives the amount of elements for each ligament in every setting. The name of the conforming Abaqus file is given as well.

<table>
<thead>
<tr>
<th>Setting (name file)</th>
<th>Coarse (kneeNHCoarse.cae)</th>
<th>Middle (kneeNHMiddle.cae)</th>
<th>Fine (kneeNHFine.cae)</th>
<th>Very Fine (kneeNHVeryFine.cae)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACL</td>
<td>4 275</td>
<td>12 539</td>
<td>20 748</td>
<td>29 749</td>
</tr>
<tr>
<td>PCL</td>
<td>3 539</td>
<td>10 458</td>
<td>16 740</td>
<td>23 587</td>
</tr>
<tr>
<td>MCL</td>
<td>2 704</td>
<td>9 171</td>
<td>13 494</td>
<td>19 460</td>
</tr>
<tr>
<td>LCL</td>
<td>2 381</td>
<td>7 885</td>
<td>10 918</td>
<td>14 920</td>
</tr>
</tbody>
</table>

Table 10-1: Number of meshes for the meshing settings

All the models are submitted to the same kinematic data from the tests. Because this project aims to find a stable strategy for every movement, the test with highest displacements is chosen. Although some discontinuities are found in the kinematic data, no problems with convergence for any model is seen for the hyperelastic Neo Hooke material.

The sensitivity test is done in three steps. The progress of the average maximum principals in the ACL are analysed during one and a half flexion-extension cycles. Afterwards a sensitivity test is performed in some selected steps. In the last step the time for calculations is evaluated.
10.1. General progress

After calculations are finished, the .odb-file is opened. In the XY-data manager all the maximum principals from every point in the ACL are loaded with ‘create XY-data’ - ‘operate on field output’. For every step the average of all maximum principals are calculated with ‘create XY-data’ - ‘operate on XY-data’. All the plots of the maximum principals are given in the ‘avg (( ))’-option. This is done for the four meshing setups.

![Figure 10-1: Progress of the mean maximal principal ACL stresses in function of time for the different meshing settings](image)

The ACL NH Coarse graph is differing significantly from the others. A smoother progress is seen and stresses are higher. This can be seen in the blue cadre. From the other ones, the Fine setup is different from the other two. Lower values are seen, but the progress is the same. The difference is due to the manual ACL-Fem surface determination. This surface has to be determined by the user and thus differs from one simulations to another. Off the middle setup, convergence is observed for the general progress.

The red cadre shows that the more accurately ACL’s are meshed, the more vulnerable they are for discontinuities in the kinematic data. This peak tension is less observed in coarser meshed ACL’s. Discontinuities are shown in H11.

Some conclusion can be made:

- The coarse model is not accurately enough.
- A fine / very fine setup is too accurately meshed and is vulnerable for data discontinuities.

The points indicated by the orange line are discussed more into detail.
10.2. Sensitivity test for some selected points

In the selected points, the influence of the amount of meshes is visualised. The first point \((t = 1s)\) shows the influence of the ACL-fem surface definition, this is not the case for the third point \((t = 2.4 \ s)\). The second orange line \((t = 2.04s)\) stresses the influence of the amount of meshes for data faults.

Following graph shows the mean maximal principals in function of the amount of meshes.

![Graph showing mean maximal principals in function of the amount of meshes.

Figure 10-2: mean maximal principals of the ACL stresses in function of the amount of elements in different times

From the green graph, convergence is considered at the middle model and no influence is seen from manual input. From this graph, convergence is judged from the middle meshing setup. The red graph shows convergence for data faults when more than 20 000 elements are chosen. No (non-)convergence can be deducted from the blue graph. The oscillating progress suggests that no convergence is obtained.

10.3. Time for calculations

Another important part of the mesh sensitivity test is the time of calculations. These are found back in the .dat-file. Following graph shows the total CPU time for all the tissues and the weighted time for the ACL. The latter is the value for the time calculations on the ACL relative to its amount of meshes to the total amount of meshes.
Mesh Sensitivity Analysis

Figure 10.3: Calculation time in function of the amount of ACL meshes

An exponential behaviour is seen in function of the number of meshes. This favours the least amount of meshes.

10.4. Conclusion of the mesh sensitivity test

The middle model is the best option and will be used for further analysis. For quick solutions a coarse model is suggested, for accurate solutions a fine model.

<table>
<thead>
<tr>
<th>Model</th>
<th>Coarse</th>
<th>Middle</th>
<th>Fine</th>
<th>Very Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>General progress</td>
<td>--</td>
<td>+</td>
<td>+</td>
<td>++</td>
</tr>
<tr>
<td>Convergence</td>
<td>-</td>
<td>0</td>
<td>++</td>
<td>++</td>
</tr>
<tr>
<td>Error vulnerability</td>
<td>++</td>
<td>+</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Calculation time</td>
<td>++</td>
<td>+</td>
<td>-</td>
<td>--</td>
</tr>
<tr>
<td><strong>TOTAL</strong></td>
<td>+</td>
<td>+++</td>
<td>+</td>
<td>+</td>
</tr>
</tbody>
</table>

Table 10-2: Summary of the sensitivity test
11.1. Preparation

The .inp-file generated by the pyFormex script is loaded in Abaqus. All geometries, materials, section, reference points + movements,…etc. are present in the Abaqus / CAE. For the surfaces a dummy was created in pyFormex. These have to be adapted manually by double clicking the surface in the model tree: assembly - -> ‘surfaces’. Following table summarizes how this is done.

<table>
<thead>
<tr>
<th>Surface</th>
<th>Selection procedure</th>
</tr>
</thead>
<tbody>
<tr>
<td>SFemur</td>
<td>Select by angle (20°)</td>
</tr>
<tr>
<td>STibia</td>
<td>Select by angle (20°)</td>
</tr>
<tr>
<td>Acl – Fem*</td>
<td>Individual Selection</td>
</tr>
<tr>
<td>Acl – Tib*</td>
<td>Select by angle (10°)</td>
</tr>
<tr>
<td>Pcl – Fem*</td>
<td>Select by angle (10°)</td>
</tr>
<tr>
<td>Pcl – Tib*</td>
<td>Select by angle (20°)</td>
</tr>
<tr>
<td>Lcl – Fem*</td>
<td>Select by angle (10°)</td>
</tr>
<tr>
<td>Lcl – Tib*</td>
<td>Select by angle (20°)</td>
</tr>
<tr>
<td>Mcl – Fem*</td>
<td>Individual Selection</td>
</tr>
<tr>
<td>Mcl – Tib*</td>
<td>Individual Selection</td>
</tr>
</tbody>
</table>

* indicates that femur and tibia have to be hidden in the assembly for doing the procedure.

Because of the boolean operation in segmenting, surfaces are delineated and most of them could be selected by angle. This is not the case for the MCL. The ACL-Fem has to be selected individually because the surface determination with ‘select by angle’ is too high, results in excessive stresses later.

The influence of the ACL-fem surface can be seen in figures from H10. The fine meshing setup is considered to be converged but because a smaller surface was chosen, lower tensions are seen.

A job is created and calculations are started.
11.2. Simulation results

From the simulations, the average maximal principal stresses of every ligament is plotted and the resulting moments in the femur reference point are evaluated. The kinematics of the bones is reviewed and a comparison with the linear elastic model is made at last.

11.2.1. General progress

11.2.1.1 Mean maximal principal stress

The progress is evaluated with the mean maximal principal stresses for every tissue. It is explained previously how this is done. A middle meshing setup is used.

Four regions are shown on the graph (I, II, III, IV). The knee starts in \( t = 0 \) sec from its segmentation position. In region I, the knee is fully flexed and returns to the initial position. The knee is then extended to full extension and returned \( (t = 2.75 \text{ s}) \). Region I and II describe a full gait cycle. Region III does the same as region I and in IV a kinematic fault is shown.

From experimental knowledge and the introduction, we know that the medial side of the knee is the most mobile one and is generally higher loaded. This is found back in the graph. In all regions the ACL is hardest loaded and also the MCL is highly loaded in region I. The low MCL stresses in region II are explained by the spinning of the femur in initial flexion. The medial condyle center of rotation (COR) is close to the insertion of the MCL. This makes that the MCL is not loaded, but the ACL is.
The PCL has modest tensions in region I, but very low values in region II. This follows the observation of the curved, unloaded PCL’s in low flexion.

In full extension the LCL has its longest measures. A pretension should be included. The isotropic material model forces the LCL to bend unrealistically. In bended structures, no high average stress is present because of the compression and expansion. A anisotropic material model should be used for the LCL to get more realistic results.

Figure 11-2: Progress of the mean maximal principal stresses for every tissue

Figure 11-3: A high curvature is seen for the LCL. No isotropic material model can be used
The general values of the stresses are low. From the material test of the master dissertation of Maarten Venneste, maximal stresses of 2 MPa are measured for the MCL. By reviewing the paper of pena, neo hooke material constants twice higher are suggested for the MCL. Because the stresses are linear to the material constants with incompressible neo hooke materials, following graph red is obtained. This linearity is tested by running the calculations with $C = 1.44$ MPa (green graph). Almost no difference is seen. We conclude that the theoretical linearity is a very good approximation for the calculations with other $C$-values. The maximal values fit those of the experimental measurements.

![Graph](image)

**Figure 11-4:** Progress of the mean maximal principals of the ACL with $C = 0.7324$ MPa and $C = 1.44$ MPa. The latter is obtained via calculations (green) and linearity of the Neo Hooke model (red)

### 11.2.1.2 Resulting moments

The resulting moments around the reference points in the femur are plotted. Moments on femur are opted because of the fixed position. In the XY-data manager, the reaction moments are chosen for all time steps.

In region I, the magnitude of the reaction moment is mainly determined by the RM3 femur. The 3-axis is almost perpendicular to the sagittal plane so that we conclude that the RM3 is the resulting moment that has to be generated by the muscles for FE. The magnitude of the reaction moment is almost fully determined by the FE until the orange line ($t = 2.1s$). From that moment, the RM1 is overtaking. As the 1-axis is near perpendicular to the transversal plane, causing EE. This overtaking can be relied to the lock mechanism in full extension (see introduction). Off the blue line, the RM1 is overtaking. As the 2 axis is perpendicular to the coronal plane, negative RM2 values in the full extension region suggests a varus patient.
For the exact progress of the moments describing EE and AA, knowledge of the kinetics is needed. This could be done in future.

![Figure 11-5: Progress of the reaction moments on the RF of the femur](image)

### 11.2.2. Kinematics

Because of the non-fit between the kinematic data from the tests and the segmentation, a correction was done. That correction influences the knee kinematics in Abaqus. In the kinematic data some discontinuities are present as well. This is shown first.

During the kinematic tests it occurs that some markers are blocked for the cameras (examinator, camera faults, ...) or software faults are present. Dummy values are generated in the .cvs files. In region IV in previous graphs, such a discontinuity is shown. Excessive stresses and strains are measured.

The correction for difference results in a more posterior position of the tibia relative to the femur. Relative displacements of the tibia before the correction are used further and no 100% correct movement of the tibia relative to the femur is made. The correction is needed for fitting the ligaments with the femur and tibia. Otherwise no constraints could be created. Following pictures show the faults due the correction as well.
11.2.3. The linear elastic model

For the linear model used by Isabelle Waterplas, an option is included in the pyFormex script. A .inp-file is generated and submitted to Abaqus. Surfaces are created and simulations are started.

For every setup (amount of meshes, element type,...) no convergence is obtained because some parts of the PCL are loaded too hard. Elements are deformed extensively and no step higher than the minimal (1e-5 s) can be made.

For comparing the two material models, a simulations with only the ACL and MCL is done. For the ACL and MCL calculations are quitted at around resp. 90° and 110° flexion because of heavy distorted elements. The mean maximal principal stresses are plotted until convergence. The obtained values are around 20 times higher than the Neo Hooke material.
H11. Simulations

Figure 11-8: Very high peak tensions are observed in early calculations (12 MPa, $t = 0.12$ s)

Figure 11-9: Maximum angle achieved with the linear material model for the MCL (left) and ACL (right)

Figure 11-10: Progress of the mean maximal stresses in the ACL, using the linear elastic material model vs neo-hookean material model.
The last part of the master dissertation is the conclusion. The project aimed to create a stable Abaqus simulation strategy for knee kinematic data in an automatic way. A short review of the different parts is given with some recommendations for future work.

- **Segmentation & Meshing**

  From the CT and MRI-scans bone, markers and soft tissues are segmented manually. These are merged and a knee model was obtained. A high quality surface mesh was applied to all the tissues so that volumetric meshing was done more easily. This relieves the demand on automatic mesh generation.

- **Kinematic tests**

  These markers are captured during movement of the knee and are written out to an excel file. The three movements are tested (FE, AA, EE). FE is focused in this master dissertation. A suggestion for future tests: a lot of post-processing time could be gained by fixing the femur during the tests. No relative movements have to be calculated and less faults will be induced. Another advantage is the fact that the error on the marker position of the femur can be evaluated. Mounting the markers close to the place of interest reduces the errors.

- **First Model**

  A first model was created to gain knowledge of simulation strategies. An optimal strategy was obtained and used for further analyses.

Some important conclusions are taken that have to be reviewed in future:

- A linear elastic model causes divergence in high flexion angles.

- Contact constraints are very demanding for the simulations. Because of the inaccurate data of the kinematic tests, the bone tissues are likely to overlap.
Cartilage triangles will be highly deformed, resulting in errors and unrealistic simulations.

- **pyFormex modelling**

  The First Model was automatized with a pyFormex script. This was done by mimicking the optimal strategy and parameters derived from the first model. Automatic volumetric meshing was included as well.

  Some options can be reviewed in future
  
  - Automatic surface generation. This could be done by mean of geometrical functions. No difference in simulations results will be present due ACL-Fem surface determination.
  
  - In the pyFormex script, a user defined material model DEPVAR12 is included for testing material models achieved by the mechanical tests of Maarten Vanneste.

- **Simulations**

  The generated .inp-files from pyFormex are imported in Abaqus and surfaces are adjusted. Calculations are started and results obtained. From the mesh sensitivity test, the optimal linear meshing setup was derived.

  Future work:
  
  - A review of the correction of the kinematic data to the anatomy should be done for disabling bone overlap and thus including cartilages/menisci with their constraints.
  
  - The influence of hexahedral and quadratic tetrahedral meshing should be analysed.
  
  - By comparing the two material models a big difference in magnitude is seen. This shows the diversity of values seen in literature.
  
  - Validation of the simulations can be done by reviewing the MRI-scans with other knee angles.
PART V. Appendix material models
1. Linear anisotropic material model

In the master dissertation of Isabelle Waterplas, a linear anisotropic elastic material model was used. It’s a modified material model that was suggested in literature (Soni, Chawla, Mukherjee & Malhotra, 2001) and adjusted by Waterplas.

The material models of the tissues are elastic and transversal so that materials have different properties in the longitudinal direction compared to the cross direction. Stress – strain relation is given by following matrices:

\[
\begin{bmatrix}
\frac{1}{E_p} & -\frac{\nu_{pz}}{E_p} & -\frac{\nu_{pz}}{E_z} & 0 & 0 & 0 \\
-\frac{\nu_{pz}}{E_p} & \frac{1}{E_p} & -\frac{1}{E_z} & 0 & 0 & 0 \\
-\frac{\nu_{pz}}{E_p} & -\frac{1}{E_z} & \frac{2G_{pz}}{E_z} & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{1}{2G_{pz}} & 0 & 0 \\
0 & 0 & 0 & 0 & \frac{1}{2G_{pz}} & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{1 + \nu_{pz}}{E_p}
\end{bmatrix}
\]

The second matrix expresses the hookes law (fits equations of equilibrium). The first is the inverse matrix to calculate the displacements. Z is the longitudinal direction, p is in-plane with the longitudinal direction. E is the youngs’ modulus, G the shear modulus. Nominal strains and stresses are used.

Transverse Young’s modulus was chosen 10 time higher than in plane Young’s modulus. This makes that the material has a low bending resistance, which is observed in real ligaments.
Some materials (rubber) return to their initial position even with a nonlinear stress-strain curvature. These materials are often incompressible so that the linear elastic equations become impractical. A energy approach rather than solving equations of equilibrium are used.

For the hyperelastic material model the total energy could be written in function of only the Green-Lagrange finite strain tensor. The Piola-Kirchoff stress tensor is calculated by the derivatives (equilibrium equations).

\[ U = A(\varepsilon_{ij}^G) \]

\[ S_{ij} = \frac{\delta A}{\delta \varepsilon_{ij}^G} \]

The elastic energy should be independent from the coordinate system and from that could the elastic energy function be written as:

\[ A = A(J_1, J_2, J_3) \]

With \( J_1, J_2, J_3 \) the independent variables. As ligaments are considered to be incompressible, \( J_3 = 1 \).

For the Neo Hooke material model the function is posed as:

\[ A = \frac{\mu_1}{2} * (J_1 - 3) + \frac{K_1}{2} * (J_3 - 1)^2 \]

thus

\[ A = \frac{\mu_1}{2} (J_1 - 3) \]

After a lot of calculations, following theoretical expression could be obtained:

\[ \sigma_{ij} = \mu_1(\varepsilon_{ij}^G - \frac{1}{3} \varepsilon_{ij}^G \delta_{ij}) \]

\[ \delta_{ij} \text{ – kronicker delta function} \]

\[ \varepsilon_{ij}^G \text{ – Cauchy Green tensor} \]


List of Figures

Figure 1-1: cartilages covering the bones .......................................................... 2
Figure 1-2: General Anatomy of the human knee (Medivisuals.com) .................. 3
Figure 1-3: The synovium ................................................................................. 4
Figure 1-4: The body planes .............................................................................. 5
Figure 1-5: Relative positions .......................................................................... 5
Figure 1-6: Bone regions .................................................................................. 5
Figure 1-7: Articulations in the knee ................................................................ 6
Figure 1-8: Knee rotations ............................................................................... 7
Figure 1-9: Knee motions of the condyles ........................................................ 7
Figure 1-10: The EF and FF with the circles describing the EFC and FFC (Iwaki) ................................................................. 9
Figure 1-11: EF and FF on tibia and femur (Iwaki) ........................................... 9
Figure 1-12: Projection of the EFC (striped) and FFC (bold) ......................... 9
Figure 1-13: screw-home mechanism ............................................................. 10
Figure 1-14: Total knee replacement .............................................................. 11

Figure 4-1: the markers each having three spheres ........................................... 18
Figure 4-2: Distances between the of the spheres tibial (left) and femoral (right) component ......................................................... 19
Figure 4-3: The femoral unconnected version .................................................. 20
Figure 4-4: The femoral connected version ...................................................... 20
Figure 4-5: The rough tibia from semi-automatic segmentation in MRI scans ................................................................. 21
Figure 4-6: The tibia is brought close to the rough mask .................................. 21
Figure 4-7: Registration of the tibia from CT scans to the rough tibia ............. 21
Figure 4-8: Registration settings ..................................................................... 21
Figure 4-9: Overlap of the CF (purple) in the registered femur (red contour) .......................................................... 23
Figure 4-10: Boolean operation: Red - Green ................................................ 23
Figure 4-11: Overview knee ............................................................................. 25
Figure 4-12: Back of the knee ......................................................................... 25
Figure 4-13: Front of the knee ......................................................................... 26

Figure 5-1: Inspection page. All the equi-angle skewnesses are in the green-yellow field .............. 27
Figure 5-2: Control in Abaqus. Only 0.6% of the elements are warnings (red cadre) ............ 28
Figure 5-3: Delaunay triangulated .................................................................. 29
Figure 5-4: Non Delaunay triangulated ................................................................ 29
Figure 6-1: Kinematic test setup - This specimen is placed in the focus of the tracking system ........................................ 31
Figure 6-2: Kinematic testing - FE .............................................................................................................................................. 32

Figure 7-1: the position in the pyFormex pgf- file (left) and the first coordinates in the csv-file (right) ........................................... 34
Figure 7-2: This figure shows that a fault in the spheres(left, green) position is doubled in the part of interest(left, red). The measures are shown right. ........................................................................................................... 35
Figure 7-3: Object A (red) has X1, X2, X3, A.position(X,Y) aligns the X and Y and object A is moved (blue). The figure left shows the circumstances before, right after ................................................................. 36
Figure 7-4: Correction of the tibia. Postanterior view (left), anteriorposterior view (right). The tibia is moved medial and superior. ................................................................................................................................................. 37
Figure 7-5: Correction of the tibia: Medial lateral view (left), Lateral medial view (right) ............................................................... 37
Figure 7-6: The fixed coordinate systems. The body axes are X (femur) and z (tibia). The planes perpendicular to the body axes are coloured grey. (Grood and Suntay)................................................................. 39
Figure 7-7: The H vector connects the origins and is projected on the body axes and floating axis F. (Grood and Suntay) ................................................................................................................................................... 39
Figure 7-8: The definition of the rotations (Grood and Suntay) .............................................................................................. 39
Figure 7-9: Rotations defined by Grood and Suntay ................................................................................................................... 40
Figure 7-10: Displacements defined by Grood and Suntay ......................................................................................................... 40

Figure 8-1: Relative sliding between bodies (3ds.com) .............................................................................................................................. 45
Figure 8-2: Contact discretization (3ds.com) ................................................................................................................................. 45
Figure 8-3: Summary of the general contact properties ........................................................................................................... 45
Figure 8-4: Stresses in early flexion with a linear elastic model (left) and a neo-hookean material model (right) with 80% of the peakstress. A more uniform distribution is seen with Neo Hooke. ...... 46
Figure 8-5: A overview (left) and detailed view (right) showing the cartilage overlap in the tibia. The orange line shows the edge of the tibia. In the green cadre triangles are deformed exessively. .......... 47

Figure 9-1: Control of the automatic generated meshes. No errors are shown and only 1.83% are warned ........................................................................................................................................................................ 51

Figure 10-1: Progress of the mean maximal principal ACL stresses in function of time for the different meshing settings ................................................................................................................................................................. 58
Figure 10-2: mean maximal principals of the ACL stresses in function of the amount of elements in different times .................................................................................................................................................. 59
Figure 10-3: Calculattion time in function of the amount of ACL meshes .......................................................................................... 60

Figure 11-1: Reccomandations for ACL-fem surface determination .......................................................................................... 62
Figure 11-2: Progress of the mean maximal principal stresses for every tissue ............................................................................ 63
Figure 11-3: A high curvature is seen for the LCL. No isotropic material model can be used .............................................. 63
Figure 11-4: Progress of the mean maximal principals of the ACL with C = 0.7432 MPa and C = 1.44 MPa. The latter is obtained via calculations (green) and linearity of the Neo Hooke model (red) ...... 64
Figure 11-5: Progress of the reaction moments on the RF of the femur .......................................................................................... 65
Figure 11-6: Kinematic data faults. The figure lefts is at t = 4.94, the figure right at t = 5.02 ........................................... 66
Figure 11-7: Kinematic faults due to the correction. Tibia is overlapping the femur ........................................... 66
Figure 11-8: Very high peaktensions are observed in early calculations (12 MPa, t = 0.12 s)............ 67
Figure 11-9: Maximum angle achieved with the linear material model for the MCL (left) and ACL (right)................................................................................................................................................. 67
Figure 11-10: Progress of the mean maximal stresses in the ACL, using the linear elastic material model vs neo-hookean material model ............................................................................................................................................ 67
0. List of Tables

List of Tables

Table 1-1: summary of the femur motions relative to the tibia ......................................................... 8

Table 4-1: A comparison between the physical and virtual measures ................................................. 19
Table 4-2: master and slave surfaces in segmenting. In boolean operations the slave surface is
subtracted with the master surface. A higher demand / accuracy is set to the master surfaces. ........ 23

Table 9-1: The used mesh quality / maximum edge length for the meshing setups ......................... 50
Table 9-2: Amount of meshes for every setup ......................................................................................... 50
Table 9-3: Used surfaces in the knee. The grey tissues are not involved in this master dissertation but
can be included. .................................................................................................................................... 52
Table 9-4: Interactions in the knee. The interactions in grey are not used in this project ............ 52

Table 10-1: Number of meshes for the meshing settings ........................................................................ 57
Table 10-2: Summary of the sensitivity test .......................................................................................... 60

Table 11-1: Selection procedure for the surfaces used. * indicates that femur and tibia have to be
hidden in the assembly for doing the procedure ................................................................................... 61