

Parameterised contact model of pelvic bone and cartilage: model development

1.0 RATIONALE

Contact stresses in the human hip play an important role in the development and progression of joint disease and cartilage degeneration. Finite element (FE) modelling is an effective tool to investigate the contact mechanics of human hip joint. FE models can now be generated in a highly subject-specific manner by segmenting geometries from three-dimensional scans and the parameterised models have the potential to allow the effect of individual geometrical features to be tested in isolation, through the rapid generation of models with controlled geometric variations.

This operating protocol is applied to the development of finite element models for the natural hip and the parameterised models for the acetabulum and cartilage. This document details the development of finite element models of the natural hip within the study entitled “Geometric parameterisation of pelvic bone and cartilage in contact analysis of the natural hip: An initial study”. The methods include the measurement of the geometry of the acetabulum and cartilage and the development of parameterised modelling based on the FE models.

2.0 MATERIALS

The specific images used in this study were:

- Images of the left pelvic bone and femur of a 38 year-old male, who was healthy at the time of death. The solid model was downloaded from BEL repository.
- Images of a cadaveric right hip, from a 55 year-old male, who died due to the alcoholic cirrhosis of the liver. The pelvic and femur bones were carefully dissected and all soft tissues were removed, which were then scanned using micro-CT scanner at a cubic voxel size of 73.6 μm and energy of 70 kVp, 114 μA .

3.0 SOFTWARE AND FILE FORMATS

Micro-CT scanner (*.dcm)

Simpleware ScanIP (*.sip, *.STL, *.igs, *.inp)

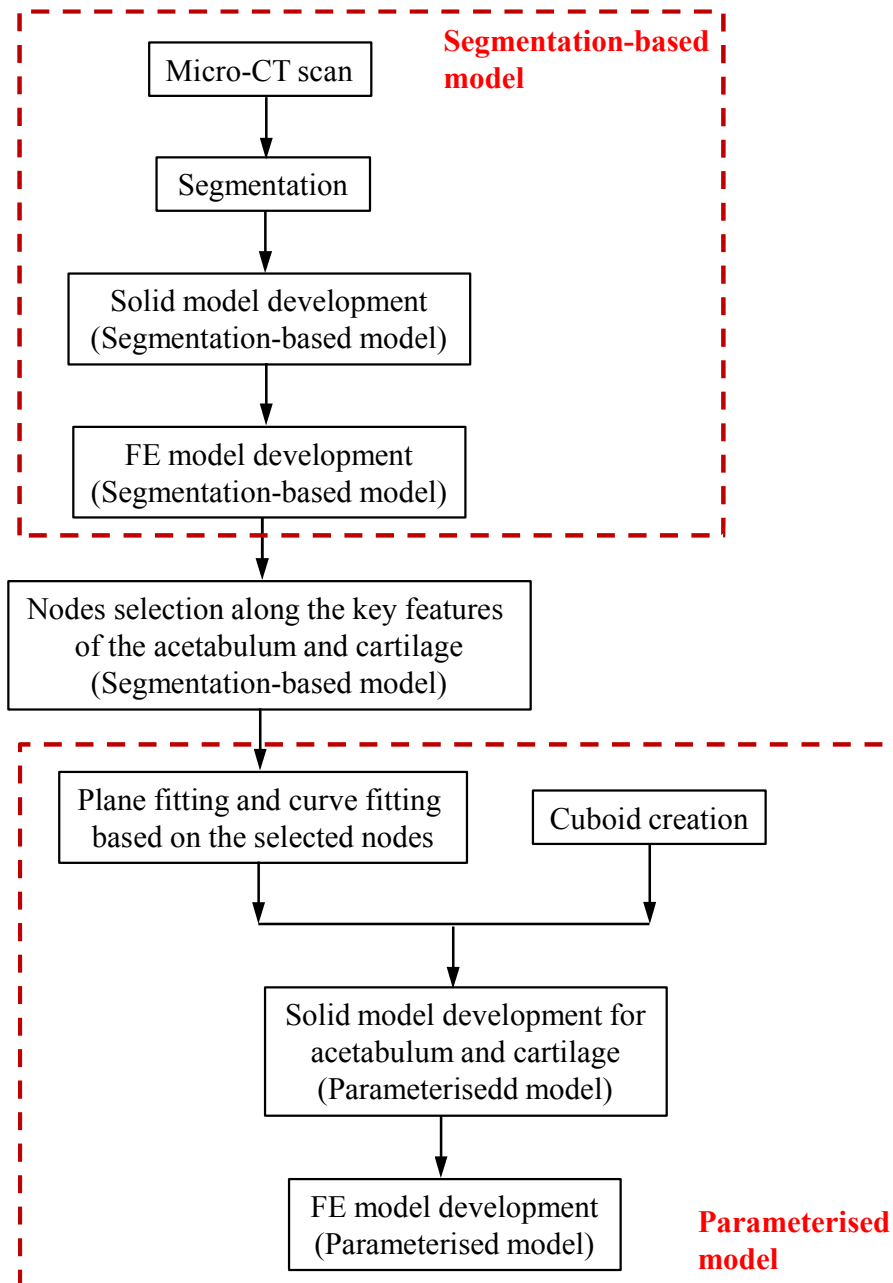
Matlab software (*.m);

I-deas software (*.igs, *.inp);

Abaqus software (*.STL, *.inp).

4.0 PROCEDURE

The flowchart of the methodology was as follow:



Part 1: Solid model development for segmentation-based model

(1) Scan the acetabulum and femur using micro-CT scanner;

- (2) Import the micro-CT images (*.dcm) into an image processing software package (Simpleware ScanIP) for segmentation;
- (3) The surface of the acetabulum and femur were meshed using three-noded triangular elements and exported in *.STL format;
- (4) The meshed component in *.STL format was imported into a surface-generation software package (Geomagic Studio) to develop the solid model which was then exported in *.igs format.

Part 2: Mesh generation for segmentation-based model

Acetabulum and femur

- (1) Import the solid models of acetabulum and femur in *.igs format into I-deas;
- (2) Move the centre of acetabulum cavity and femoral head into the origin of the coordinate system in I-deas and rotate the acetabulum and femur to the desired angles (e.g. 63° inclination angle and 15° anteversion angle in present study);
- (3) Make the acetabulum cavity spherical by cutting a thin layer on the cavity surface using a spherical surface;
- (4) Partition and delete the femoral head from the femur and create a new sphere to represent the femoral head;
- (5) Mesh the outer surface of the pelvis and femur using shell triangular elements with maximum size of 3 mm after a mesh sensitivity analysis;
- (6) Mesh the volume of the pelvis and femur using tetrahedral element with maximum size of 3 mm.

Acetabular and femoral cartilage

- (7) Generate the acetabular cartilage by extruding the surface of acetabulum cavity with thickness of 2 mm in I-deas;
- (8) Create the femoral cartilage by creating a 2 mm thickness hemispherical shell in I-deas;
- (9) Partition the cartilage solid models into several volumes (e.g. 11 volumes for acetabulum cartilage and 4 volumes for femoral cartilage in the present study, as shown in Figure 1);

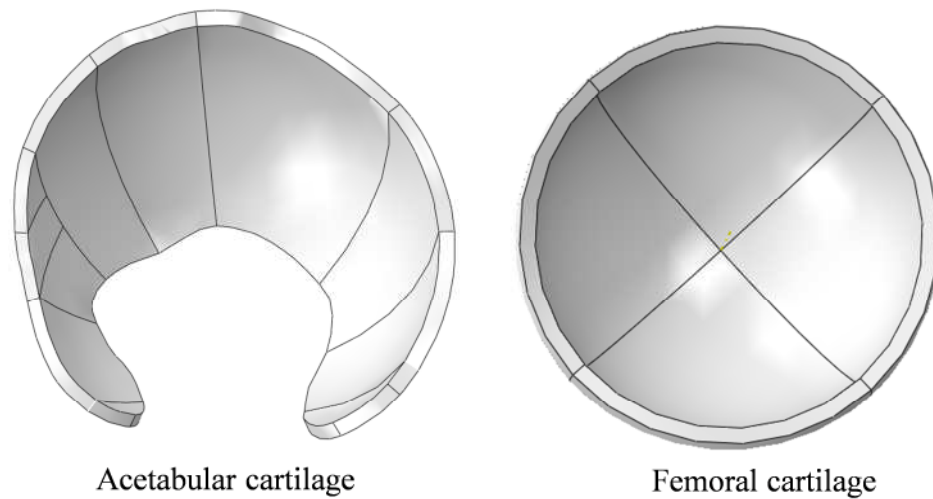


Fig 1. The partition for the acetabular cartilage and femoral cartilage

- (10) Mesh each volume of the cartilage using map mesh technique and sweep mesh technique (in the present study, the number of the elements were 32 in short direction, 140 in long direction and 3 in the thickness direction for acetabular cartilage; 64 in circumferential direction and 2 in thickness direction for femoral head cartilage, as shown in Figure 2);

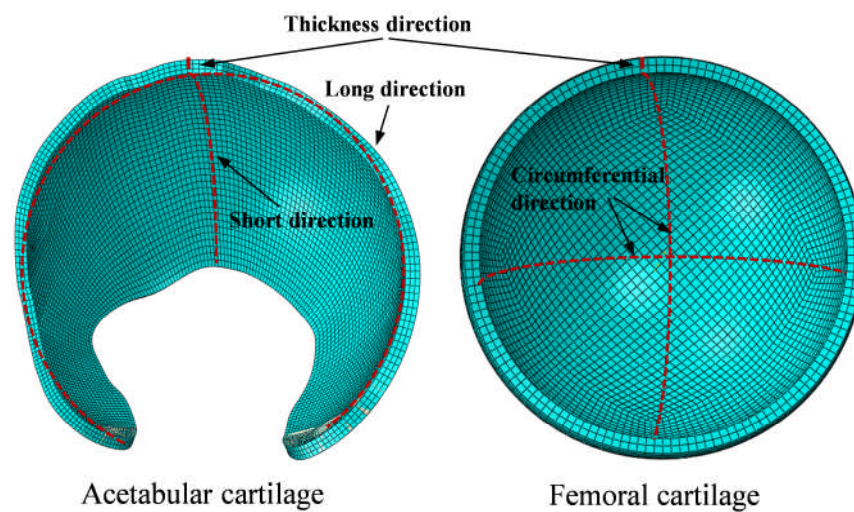


Fig 2. The mesh of the components of acetabular and femoral cartilage

- (11) Output the meshed components of acetabulum, acetabular cartilage, femoral cartilage and femur as “*.inp” format file in I-deas.

Part 3: Geometry measurement of acetabulum and cartilage

- (1) Import the *.inp file of acetabulum and cartilage into Abaqus;
- (2) Pick several nodes on the posterior edge of the acetabulum, as shown in Figure 3;

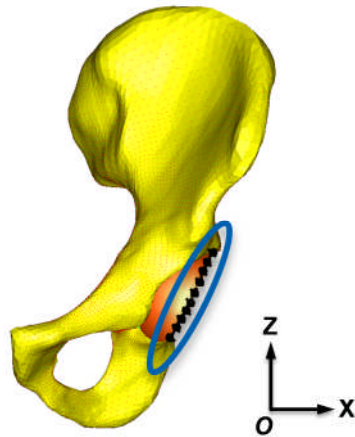


Fig 3. The nodes selected along the posterior edge of acetabulum

(3) The picked nodes were fitted to a plane using least squares regression in Matlab (the cup depth d was then defined as the distance between the plane and the centre of the cavity);

(4) Pick several nodes on the anterior edge of the acetabulum, as shown in Figure 4;

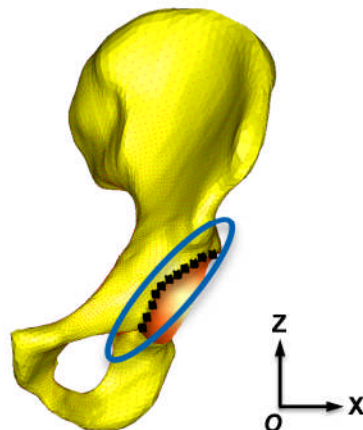


Fig 4. The nodes selected along the anterior edge of acetabulum

(5) Project the nodes on the coronal plane (i.e. the xoz plane in Abaqus coordinate system as shown in Figure 4) which were then fitted to a circle using least squares regression (the centre o_1 with radius r_1) in Matlab;

(6) Pick several nodes on the fossa of the cartilage, as shown in Figure 5;

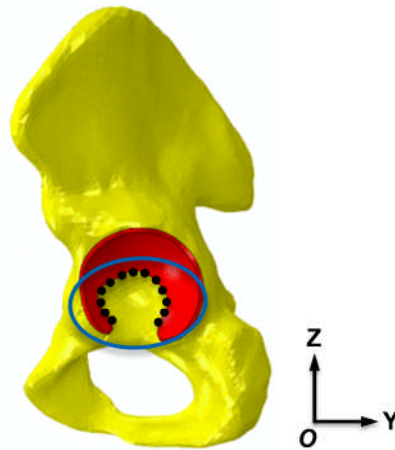


Fig 5. The nodes selected on the fossa of the cartilage

- (7) Project the nodes on the sagittal plane (i.e. the yoz plane in Abaqus coordinate system as shown in Figure 5) which were then fitted to a circle using least squares regression (the centre o_2 with radius r_2) in Matlab.

Part 4: Solid model development of parameterised model

- (1) Build a cuboid with the sizes of 70 mm by 80 mm by 85 mm along the x-axis, y-axis and z-axis respectively in I-deas to represent the pelvis, the geometric center of the cuboid was placed on the position with coordinate value of (0,0,20);
- (2) Cut the cuboid pelvis by a plane with desired inclination angles (e.g. 63° in the present study) to generate the posterior edge of the acetabulum. The plane was obtained in Part 3 step (3).
- (3) Cut the cuboid pelvis using a spherical surface to create the acetabular cavity. The centre (o) and radius of the spherical surface were the same as the ones in the corresponding segmentation-based model;
- (4) Cut the anterior edge of the cuboid pelvis using a cylinder by extruding the circle generated in Part 3 Step (5);
- (5) Generate the cuboid acetabular cartilage by extruding the surface of acetabulum cavity with thickness of 2 mm;
- (6) Mesh the cuboid pelvis using shell triangular elements and tetrahedral element (reference to part 2 step2 5 and 6);
- (7) Partition and mesh the acetabular cartilage and femoral cartilage using map mesh technique and sweep mesh technique (reference to part 2 steps 9 and 10);

- (8) Output the meshed components of cuboid pelvis, acetabular cartilage, femoral cartilage and femur as “* inp” format file.

Part 5 Finite element (FE) model development for segmentation-based model and parameterised model

- (1) Input the “*inp” file of components into Abaqus;
- (2) Develop the FE model by assembling all the components of pelvis, acetabular cartilage, femoral cartilage and femur in default position and orientation;
- (3) Set the material properties for the components of the FE models
 - Cortical bone (shell element): 17 GPa for elastic modulus and 0.3 for Poisson’s ratio;
 - Cancellous bone (tetrahedral element): 0.8 GPa for elastic modulus and 0.2 for Poisson’s ratio;
 - Acetabular and femoral cartilage: 12 MPa for elastic modulus and 0.45 for Poisson’s ratio;
- (4) Set the interaction between components:
 - Use a “tie” constraint to prevent movement at the interface between the pelvis and acetabular cartilage, and the interface between the femur and femoral cartilage;
 - Set a surface-to-surface contact formulation for the interface between the acetabular and the femoral cartilage, with a frictionless property;
- (5) Set boundary conditions for the FE models:
 - In the segmentation-based model S1a, the nodes situated at the sacroiliac joint and about the pubic symphysis were fully constrained. In models S1b and S2, the nodes located at the truncated area were fully constrained, as shown in Figure 6;

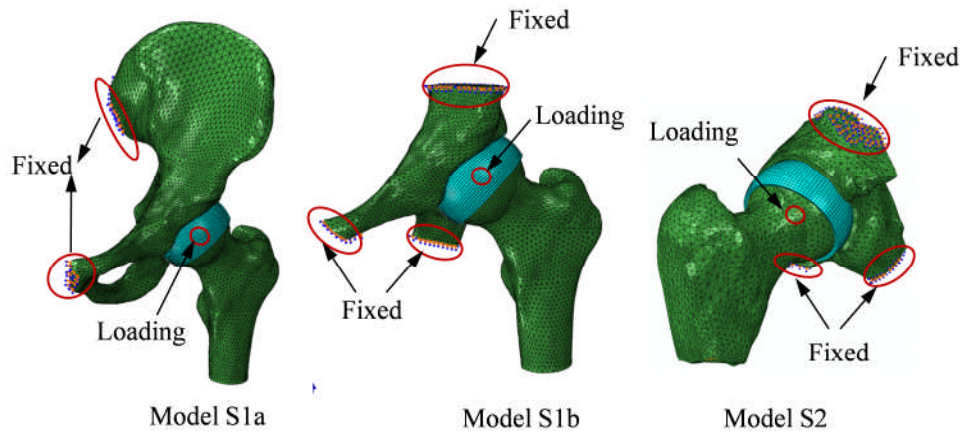


Fig 6. The boundary conditions for the model S1a, model S1b and model S2

- In the parameterised models P1 and P2, the nodes at the upper and lower planes of the cuboid pelvis were fully constrained;

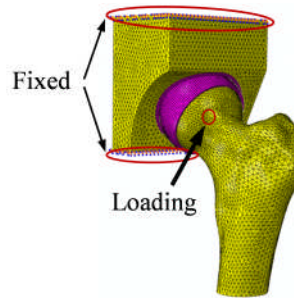


Fig 7. The boundary conditions for the model P1 and P2

- A region of nodes (with radius of approximately 4 mm) at the centre of the femoral component was constrained as a rigid body to a reference point (for load application);
- Constrained the rotation of the femoral head;

(6) Apply loads to the FE models:

- Apply a small displacement to the reference point which was slightly larger than the radial clearance between the head and the cavity (e.g. 0.8 mm in the present study) and along the direction of contact force at 0% instance of the gait in order to make initial contact between acetabular cartilage and femoral cartilage;
- Apply the physiological loading of seven different human activities (normal walking, ascending stairs, descending stairs, standing up, sitting down, knee bending and standing) to the FE models. The physiological loading was obtained from the Hip 98 database. In order to consider the specific direction

and orientation of the forces, the resultant hip joint forces were resolved to three components and converted to the FE model coordinate system. The resultant hip joint forces were then discretized into 13 steps and apply to the reference point at the centre of the femoral head;

(7) Create job and run the job;

(8) Record the maximum contact pressures and contact areas for each instance of the gait for each FE models;