G Model
CIRP-1849; No. of Pages 4

ARTICLE IN PRESS

CIRP Annals - Manufacturing Technology xxx (2018) xxx–xxx

Contents lists available at ScienceDirect

CIRP Annals - Manufacturing Technology

journal homepage: http://ees.elsevier.com/cirp/default.asp

Smart pressure distribution estimation in biological joints for mechanical bio-inspired design

Elia Picault [a], Emmanuel Mermoz (2)[a,b], Thomas Thouveny [a], Jean-Marc Linares (1)[a,*]

[a] Aix Marseille Univ, CNRS, ISM, Marseille, France
[b] Airbus Helicopters, Aéroport de Marseille Provence, Marignane, 13700, France

ARTICLE INFO

Keywords:
Biologically inspired design
Finite element method
Mechanical connection

ABSTRACT

Designing new mechanical links using bio-inspiration requires the knowledge of operating contact pressure in biological joints. However, the contact pressure magnitude and distribution are difficult to measure experimentally without disrupting the functioning of the articulation. In this paper, a new methodology to estimate the pressure distribution in biological joints is presented. A robust finite element model was developed based on in-vitro precise measurements of shapes, relative positions and loads in order to get accurate results. Furthermore, the envelope of the contact area was obtained through thermal imaging for comparison with the numerical results and qualitative validation of the FE model.

1. Introduction

1.1. Mechanical bio-inspired design

Current mechanical connections technology based on generic links and simple geometries have reached their limits within the framework of severe loads despite the application of surface treatments to improve mechanical properties. Fig. 1a shows a highly stressed pivot axis on which the contact area is severely damaged. A new path in research opened these last years, based on bio-inspiration and biomimicry [1,2]. Bio-inspired design could be a way to improve mechanical connections lifespan.

1.2. Numerical modelling of biological joints

To understand the functioning of biological joints, either for medical applications or in the context of mechanical bio-inspired design, numerical models are widely used. The finite element (FE) method was first applied to biological joints in the 80s with the study of the human knee [3–5]. Nowadays, the standard in medical biomechanics is anatomically based subject-specific 3D model. In those models, fluids and most soft tissues other than ligaments are almost always neglected. Secondary bones, ligaments, articular cartilage and disks are often neglected too. Bone is routinely assumed to be linear elastic isotropic and homogeneous although sometimes modelled as rigid or anisotropic. The same is true for articular cartilage which in practice is found to be nonlinear viscoelastic and heterogeneous. Some rare studies that focus on the role of articular cartilage consider part of this complex behaviour [5]. Finally, ligaments were often modelled as 1D nonlinear springs but tend to be represented by transversely isotropic hyper-elastic elements [6]. Geometries of bones and cartilage are usually acquired with computed tomography (CT) or magnetic resonance imaging (MRI), occasionally with digitizing systems [7]. Mechanical properties of bone are mostly derived from CT or MRI scans, whereas those of soft tissues are obtained from literature or by fitting the model to experimental data. In medical biomechanics, the main issues with FE models of biological joints are a poor knowledge of in-vivo loading and boundary conditions [4], as well as uncertainty of mechanical properties of biological materials (high variability among specimens and difficulties to obtain significant samples of fresh biological material) [5]. Biological geometry generation and meshing is also a sensitive aspect which requires a certain expertise due to the surfaces complexity, the low thickness of some components (e.g. cartilage) and the compromise between accuracy and computation time that must be achieved (especially for computer-assisted surgery). In the larger field of mechanical bio-inspired design, other issues arise due to the difficult access to biological material and to medical imaging equipment, as well as a lack of data on non-human biological links in the literature.

1.3. Scope of the study

In this context of mechanical bio-inspired design, our aim is to study the contact pressure magnitude and distribution in biological joints with no access to medical equipment. The present paper has a twofold purpose. First, it proposes a new methodology to develop a robust FE model of an articulation and presents a qualitative comparison between results obtained numerically and experimentally. Second, it provides a first investigation of the tested biological joint behaviour from a mechanical point of view.

* Corresponding author.
E-mail address: jean-marc.linares@univ-amu.fr (J.-M.).

https://doi.org/10.1016/j.cirp.2018.04.116
0007-8506/© 2018 Published by Elsevier Ltd on behalf of CIRP.

This methodology has been used to model a lamb elbow joint (Fig. 1b). This choice results from a compromise between the type of link aimed for the bio-inspired design study (comparable to a pivot link) and the accessibility of biological material. The FE model accounts for humerus and radius-ulna bones (Fig. 1c) with articular cartilage, whereas soft tissues have been neglected. In numerical study of contact, the results quality strongly depends on good representation of the bodies' shape and relative positions, as well as on the applied load. Therefore, an optical 3D scanner and a multi-component dynamometer are employed to obtain accurate geometrical and load data. Knowledge of in-vivo loading conditions is difficult to achieve however. Thus, this work is limited to a preliminary study of contact distribution, without seeking for the applied loads to be perfectly representative of those seen in-vivo. Doing so it is assumed that the contact area in an articulation is dictated first by the geometry and second by the load. Contact pressure magnitude and distribution are also difficult to measure experimentally without disrupting the joint behaviour. In literature, the main approach used to that end is the insertion of pressure sensitive films in the joint. However, this approach is intrusive, ill-suited for curved surfaces and gaps around 10–20% between real and measured contact area and magnitude can be obtained [8]. Here a thermal camera is used to estimate the contact area envelope based on local heating of the surface.

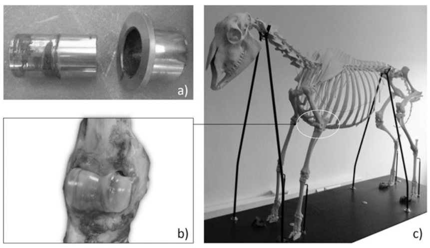

Fig. 1. (a) Damaged pivot axis. (b) Sheep elbow joint with radius-ulna active surface inside humerus. (c) Sheep skeleton with elbow joint.

In the next section, the methodology is described in detail. First, the experimental set up and measurement process are presented. Then, the numerical process leading to the FE model is explained. Finally, the obtained results are compared and analysed.

2. Principle of experimental setup and finite element model

2.1. Methodology overview

The methodology detailed hereafter is summarized in Fig. 2. An experimental test bench is set up to run several tests (static and cyclic) on the articulation. Various measures are made before, during and after the tests to collect the data necessary for the FE model construction and validation. Next, a number of preliminary stages in the model construction are carried out, beginning with bones and cartilages reconstruction, their relative positioning and the computation of the load seen by the articulation in agreement with the previous measurements. Finally, the FE model is built with the meshing of the reconstructed geometries and the definition of materials, loads and boundary conditions.

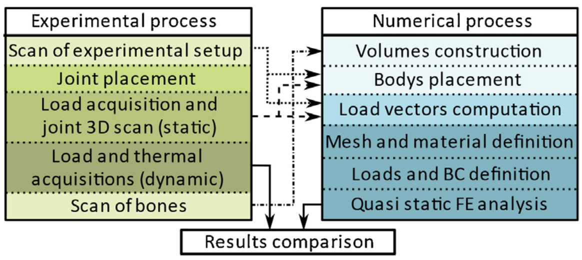

Fig. 2. Methodology overview.

2.2. Experimental process

The experimental setup is shown in Fig. 3. The humerus of the biological joint (1) is sealed in a flange (2) mounted on a KISTLER dynamometric platform (3), itself fixed on a support (4). A system of pulleys and wires (5) links the radius-ulna (referred to as radius hereafter) to weights (6) on one side and to an operating handle on the other side (not visible on Fig. 3). The handle can be fixed at several positions on the wire to adjust the radius lead angle for static loading (Table 1) or it can be actuated by a tester to simulate passive flexion-extension at a standardized rate similar to that of gait. For both loading scenarios, the first step (before placing the joint to limit its drying) is to scan several markers placed on the dynamometric platform and its support using a GOM ATOS 3D optical scanner (7). Three markers on the lateral side (3a) of the platform, one on its frontal side (3b) and one on the support (4) will be used afterwards to define the numerical model global reference frame. The articulation is then fixed on the platform and the wires are attached to the radius (without weights). Static tests are performed first. The handle is adjusted in position and the dynamometric platform is started in order to record the load seen by the joint with a resolution of 0.05 N. A first part of the acquisition (35 s at 300 Hz) is realized unloaded to detect the drift of the platform. Then an 8 kg weight (approximately one quarter of a lamb weight) is set up and acquisition continues during a time sufficient for the system to stabilize (120–180 s at 300 Hz). Simultaneously, several components of the test bench are scanned to obtain the radius position and the load directions relative to the global reference frame. To that end, additional elements allowing an optimal placement of markers detected by the optical scanner are introduced on the wires (8) and the radius (9). The same process is repeated for each inclination of the radius. For cyclic testing, the operating handle is actuated to operate around one hundred cycles (5 cycles per 10 s during 240 s) and a FLIR thermal camera (10) is used to record the temperature on the humerus surface with a sensitivity of 0.05 °C. To compare afterwards the 2D thermal image and the 3D FE model with the proper orientation, the thermal camera is partially digitalized. However, to know its position in the global reference frame it is necessary to scan also a physical path between the camera and the test bench. Though a metal straight (11) with markers is added to the experimental setup. After the tests, the articulation is dismantled and active surfaces are digitalized. The humerus is scanned sealed in the flange (2) itself fixed on the dynamometric platform (3), whereas the radius-ulna is scanned with the plastic cylinder (9) attached.

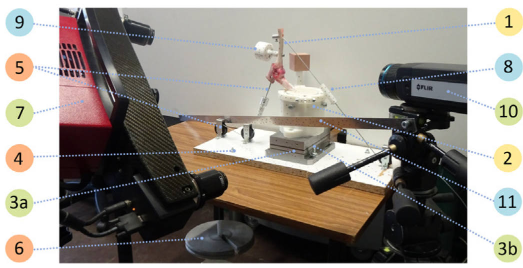

Fig. 3. Experimental setup. (1) Biological joint. (2) Flange. (3a) Lateral side and (3b) frontal side of dynamometric platform. (4) Support. (5) Wires and pulleys system. (6) Weight. (7) 3D optical scanner. (8) Plastic cylinder with markers (load direction). (9) Plastic cylinder with markers (radius local reference frame). (10) Thermal camera. (11) Metal straight with markers (physical path between test bench and thermal camera).

2.3. Numerical process

Using the data measured from the experimental set up, the finite element model is then built in several steps. First, CATIA V5 is used to generate part files containing the global and local reference frames, the humerus and radius bones and cartilages plus the load axes. The bone and cartilage parts (Fig. 5a) are created using the

Please cite this article in press as: Picault E, et al. Smart pressure distribution estimation in biological joints for mechanical bio-inspired design. CIRP Annals - Manufacturing Technology (2018), https://doi.org/10.1016/j.cirp.2018.04.116