

SIMPLEWARE, ANYBODY & ANSYS

Case Study

From 3D Medical Image to Musculoskeletal Simulation

This case study highlights the ease of integrating subject specific models with a musculoskeletal modelling system to provide a movement specific loading case for a finite element (FE) analysis. Simpleware was used initially to reconstruct the humerus of a 26 year old male from MRI data. The 3D model of the humerus was exported to the AnyBody Modeling System™ to carry out a dumbbell lifting task. The calculated muscle forces from this system were combined with the volume mesh from Simpleware for FE analysis in ANSYS®. This example shows how subject specific geometries can influence the stresses recorded from muscle forces for a particular activity.

Characteristics

- » Based on in vivo patient data
- » Segmentation tools to aid generation of accurate bone model
- » Details of structure captured aiding landmark identification
- » Intuitive, interactive landmark placement aided by a range of views
- » Surface and volume mesh rapidly generated for direct export to FE
- » Landmarks and geometry used by AnyBody system for subject specific dynamic physiological load case calculation
- » Analysis performed using AnyBody loads and Simpleware mesh in ANSYS®

In collaboration with:

ANYBODY
TECHNOLOGY

ANSYS

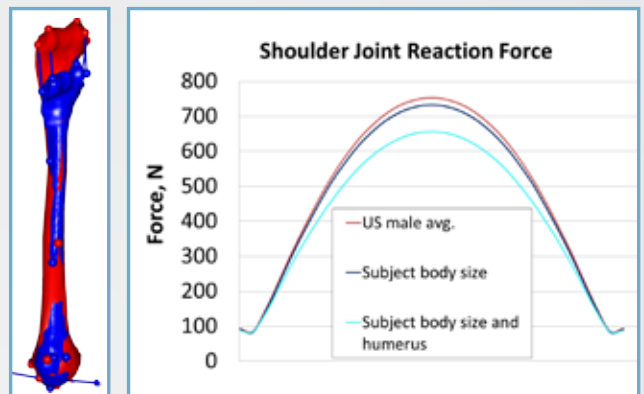
IMAGE PROCESSING AND LANDMARKING

A humerus model was generated from MRI data of a 26yr old healthy male, using the semi-automated segmentation tools available in Simpleware ScanIP. Interactive measurement tools allowed for the generation and rapid interactive placement of landmark points, using both 2D and 3D views. Each point could be named and exported with its coordinate values. Finally, high quality surface and volume meshes were generated to allow direct use of load cases computed by AnyBody in subsequent FE analysis.



MUSCULOSKELETAL MODELLING IN ANYBODY

The bone landmarks generated within Simpleware were then used for generating a subject-specific AnyBody analysis, computing a dynamic physiological load case for the humerus for dumbbell lifting. Output was dynamic individual muscle and joint forces. Input was an average US male body model, a model scaled to match the height and weight of the subject, and the same model also including the subject-specific humerus shape derived from Simpleware. Notice how inclusion of subject-specific data into the model affects the results.



STRESS ANALYSIS IN ANSYS®

The volume mesh generated in Simpleware was exported directly to ANSYS®, along with the subject specific load cases determined by the AnyBody Modeling System™. AnyBody exports the load steps in a script file so that they can be run over multiple time steps to simulate, in this case, the arm lifting the dumbbell. The results show the changing strain fields through the bone as the arm raises. The image shown is of the highest strain observed, occurring mid way through the lift.

