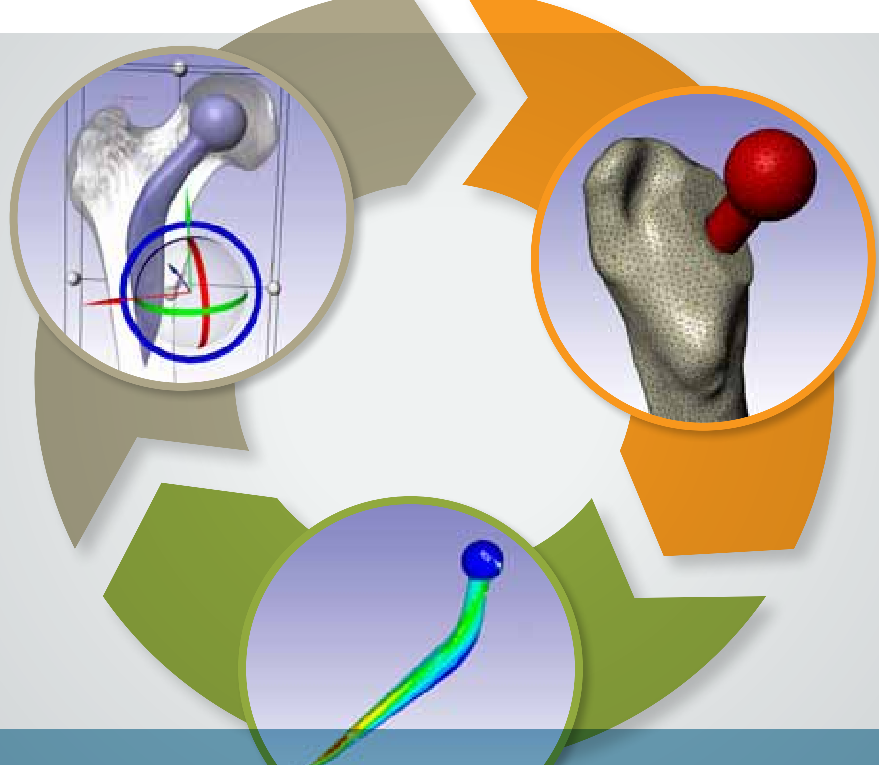


IMPLANT POSITIONING IN TOTAL HIP REPLACEMENTS

Case Study

Developing an Automated Positioning Tool in Simpleware and ANSYS

Implant positioning represents a significant challenge for total hip replacements. Implants need to be well-fitted and positioned within the medullary canal, and should ideally maximise the femoral bone-implant contact area. However, experimental testing of implant positions is limited by cost. An alternative is to use computational modelling to comprehensively analyse implant position at an early stage of product development. While this approach is not intended to replace experiments, it can help surgeons to better understand the effect of implant position on primary or secondary stability.

Simpleware software and ANSYS were combined to create an automated workflow to integrate a CAD-designed implant into a CT scan of a femur, generating Finite Element (FE) models for micromotion analysis. Simulation results were used to generate response surfaces, demonstrating the effect a change in position can have on micromotion.

Characteristics

- Integration of CAD implant and CT scan of a femur in Simpleware software
- Multiple implant positions/orientations automated through scripting
- Generation of FE meshes in Simpleware software for export to ANSYS
- Simulation of implant micromotion in ANSYS
- Post-processing of results to generate response surface
- Prediction of best and worst case scenarios for implant position

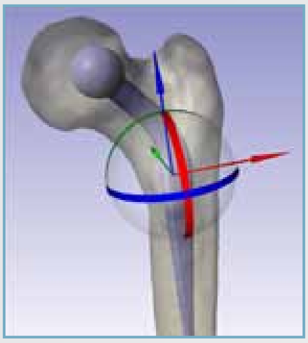
Thanks to:

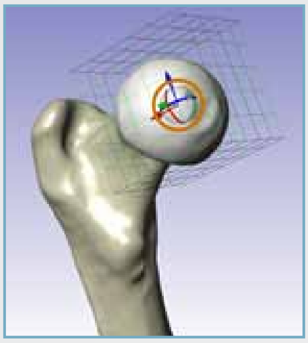
ANSYS

University of Southampton School of Engineering Sciences

IMAGE PROCESSING AND CAD INTEGRATION

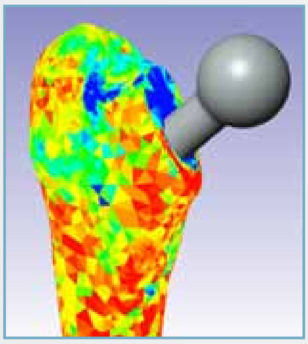
Simpleware software modules ScanIP and +CAD were used to combine a segmented femur model taken from a CT scan with a CAD-designed implant. An FE mesh was then generated in the Simpleware +FE module and exported to ANSYS Workbench for micromotion simulation. The Simpleware API, employing Python scripting, was used to automate the process of generating multiple implant positions without the need for time-intensive manual adjustments.





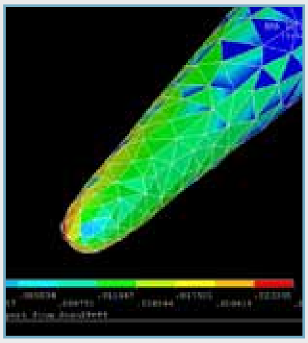
FE MESH GENERATION

FE meshes were generated for each implant position with approximately 10,000 nodes and 38,000 elements for the femur, and approximately 2,000 nodes and 6,000 elements for the implant, which was modelled as titanium. Standard material properties based on the original scan's Hounsfield units were assigned to the femur using Simpleware software's automatic mapping algorithm. Node sets were added to simulate constraints and loading conditions for the implant and femur. The mesh was also refined at the implant-bone interface for realistic simulation.

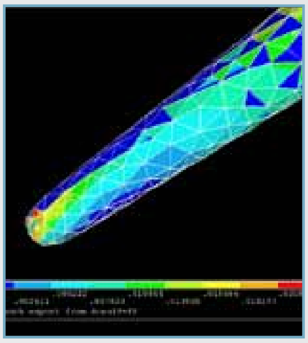


SIMULATION OF MICROMOTION

The initial FE models were exported to ANSYS Workbench for micromotion simulation. A Response Surface Model (RSM) was generated by interpolating 425 successful simulation points with the Kriging regression method. ANSYS Workbench then used the RSM to determine the implant position that leads to the highest and lowest possible values of micromotion. These results can be interpreted by a surgeon to predict what the best and worst positions are for an implant.



Best Position



Worst Position

CONCLUSION

An automated tool was successfully developed using Simpleware and ANSYS software for combining image data and FEA, with the goal of analysing interactions between an implant and a femur. The use of the response surface methodology enabled insights into the sensitivity of micromotion to positioning. With feasibility for the tool established, further work might focus on examining multiple implant designs and carrying out analyses on patient populations. The tool therefore has significant applications for future biomedical research.