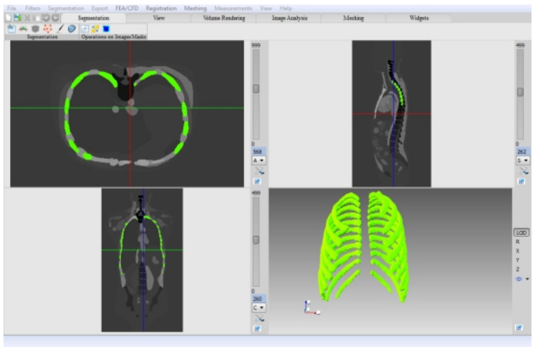
Liver Injury under Impacts on Abdomen

A finite element human torso model was developed from high resolution CT data. ImageSim platform (VOLMO LTD (UK)) was used to create a detailed model consisting of Liver , Rib cage and Lungs. Model assembly and alignment was done in TSV Pre (TechnoStar Japan) . The Analysis was performed using LS-DYNA971

CT data conversion into STL Model.



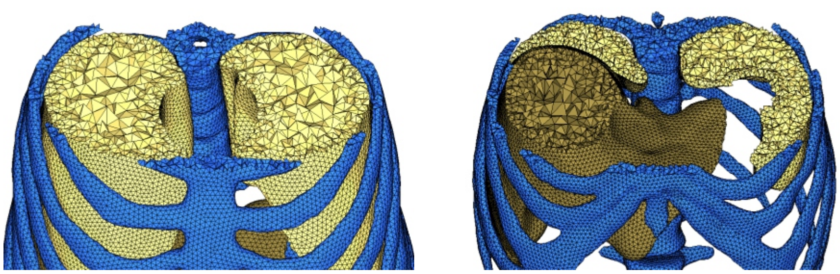
CT data of healthy adult male was imported into ImageSim & the geometry of the components of whole model were reconstructed from CT data using ImageSim. The model consists of geometrically detailed liver, rib cage and lungs.

Steps to generate a clean STL model

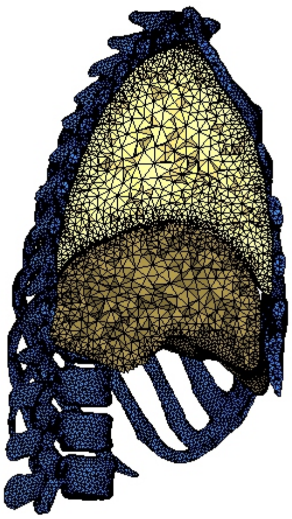
- Filtering and Contrast Enhancement
- Image Segmentation
- STL generation and export

Developing FE Model.

The FE Model used in this study was developed in TSV Pre using STL models created in ImageSim. TSV Pre is a robust CAE environment and can be invoked from ImageSim. Automatic volume mesher was used to generate high quality tetrahedral meshing.



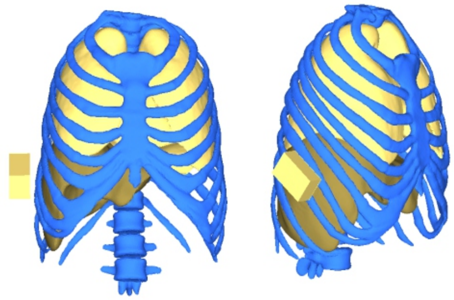
Top Cross section View in different Section



Side Cross section View

Impact Scenario Setting.

Torso Model was hit by rigid body with an effective mass was 489.393 g from frontal, lateral and rear directions with impact velocities 6 m/s, During the impact process only the lower part of the spine was fixed. Termination time was set to 6 ms and time interval between outputs was set to 0.1 ms.



Results

The deformation , displacement , stress and strain distribution of model parts were calculated and used to analyse the biomechanism underlying liver injury. Some of the results are shown below

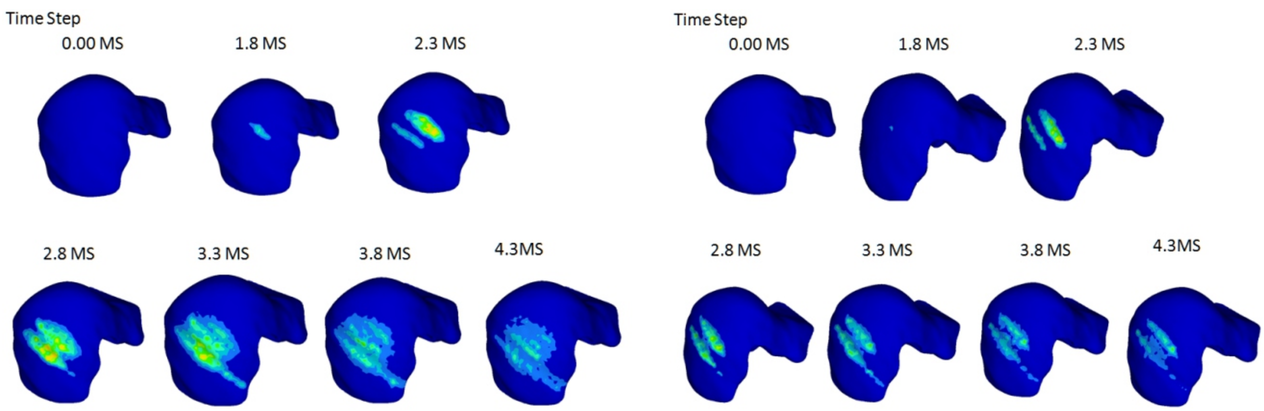
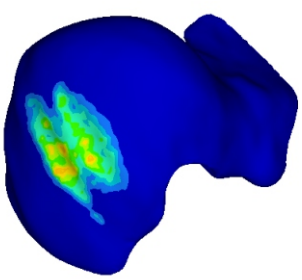


Figure :- Stress(A) & Strain(B) Distribution on liver during Impact

Conclusions

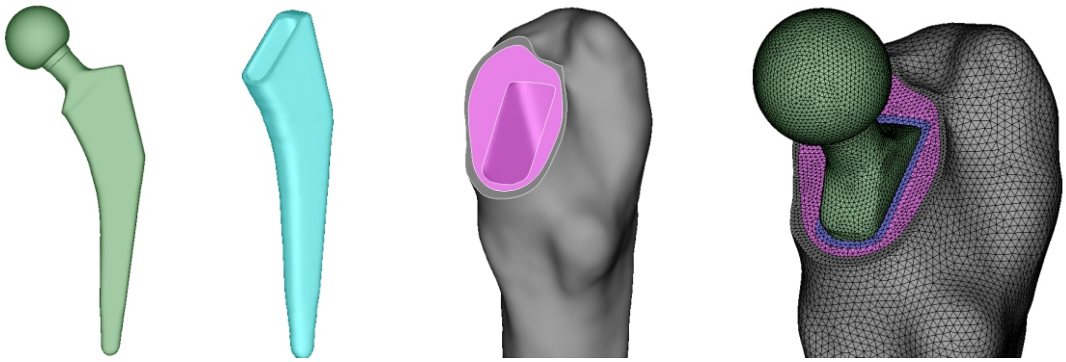
We have described liver injury under blunt impact based on FE analysis, specifically with respect to impact to the abdomen. Development of a three dimensional human torso FE model, which consists of geometrically- detailed liver, lungs and rib cage was used to simulate impact on the abdominal different impact velocities and three impact directions. The output was compared with the experimental results of the liver rupture of the victim[1]. The region of greatest strain (figure below) matched with location of injury



[1] Blunt Liver Injury with Intact Ribs under Impacts on the Abdomen: A Biomechanical Investigation Yu Shao1,2., Donghua Zou1., Zhengdong Li1, Lei Wan1, Zhiqiang Qin1, Ningguo Liu1, Jianhua Zhang1, Liangwei Zhong3, Ping Huang1*, Yijiu Chen1* 1 Department of Forensic Pathology, Institute of Forensic Sciences, Ministry of Justice, P. R. China, Shanghai, China, 2 Department of Forensic Science, Shanghai Medical College, Fudan University, Shanghai, China, 3 College of Mechanical Engineering, University of Shanghai for Science and Technology, Shanghai, China

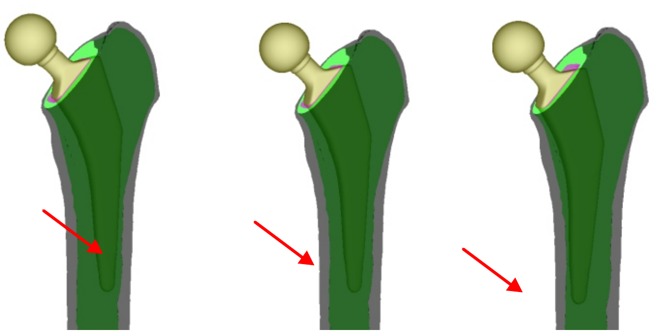
Optimisation of implant position in cemented femur in Total Hip Replacement

Total hip replacement (THR) is one of the most common and effective surgical procedures performed worldwide with the purpose of improving the quality of life of patients suffering from hip disorders. There are two major types of artificial hip replacements : cemented and uncemented. This paper presents finite element approach to measure micromotion along the stem /cement and cement/ bone interface. A response surface model (RSM) was created and used to determine the best implant position for lowest value of micromotion.
In this study finite element model of femur was created from 3D CT scan data using ImageSim software from VOLMO LTD (UK). Surface model of femur in STL format was exported into CAE environment (TSV Pre) where femur resection , implant positioning and alignment was done. The final assembled model of bone, cement and implant was used as the base model for creating new models. The loading and boundary condition have been applied for walking condition.



Model Generation using Script
Java scripting functionality within the tool provided a robust environment for automatically rotating the implant to a new angle, exporting STL, Volume meshing, applying load boundary condition and finally exporting volume mesh. This technique provided an efficient approach for generating new models with different implant positions in the femur bone. Around 25 models were generated in less than six hours.

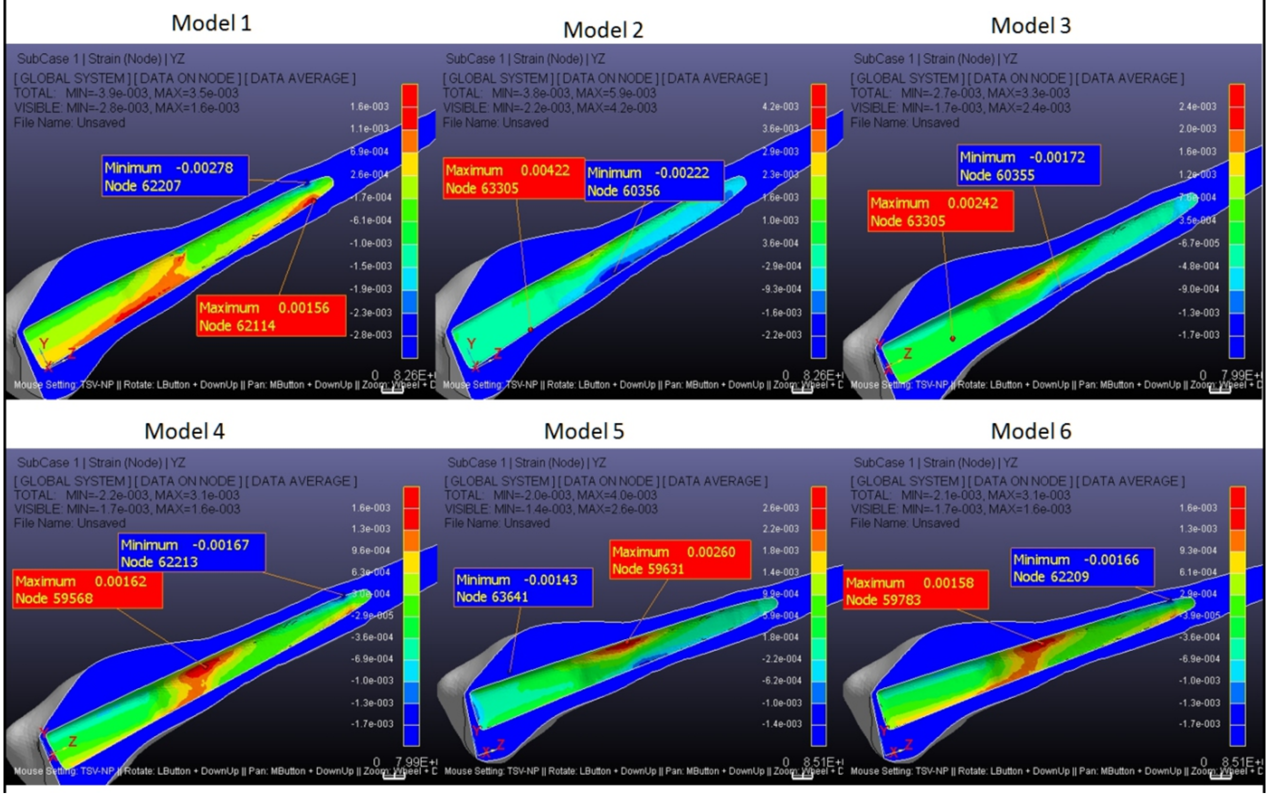
Boundary and Loading Condition
Loading and boundary condition applied are for walking, at five different locations forces have been applied including reaction and muscle force. These models were then simulated in DYNAMIS solver available internally in the environment.



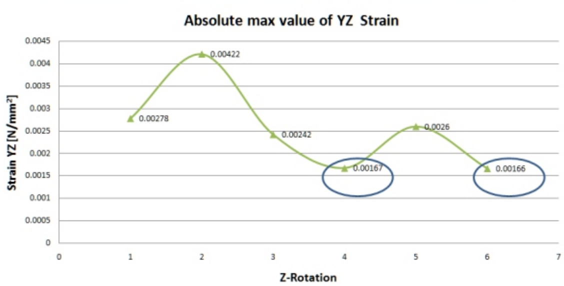
Example of Models with different
Implant angle with respecte to Femur bone

Result : Initial results obtained are shown below.

Inner Bone Structure Strain YZ result– (Rotation in Z models)



Rotation -Z	Strain Value along YZ (N/mm²)		
	maximum negative Strain	maximum positive Strain	absolute max value
1	-0.00278	0.00156	0.00278
2	-0.00222	0.00422	0.00422
3	-0.00172	0.00242	0.00242
4	-0.00167	0.00162	0.00167
5	-0.00143	0.0026	0.0026
6	-0.00166	0.00158	0.00166



rotation -Z	Estimated Micromotion in millimeters using		
	maximum negative Strain YZ	maximum positive Strain YZ	absolute max value of Strain YZ
1	-0.005004	0.002808	0.005004
2	-0.003996	0.007596	0.007596
3	-0.003096	0.004356	0.004356
4	-0.003006	0.002916	0.003006
5	-0.002574	0.00468	0.00468
6	-0.002988	0.002844	0.002988

Average Mesh length : 1.8 mm