AnyBody – ANSYS Interface: CAE Technology for the Human Body

The CAE field is about to cross another frontier: analysis of forces in the human body. This article explains why this is important and how it is done using the AnyBody Modeling System and ANSYS.

Introduction
CAE technology has completely conquered most fields of engineering. There are few physical phenomena that cannot be analyzed by FEM, CDF, BEM, MBS and whatever the various technologies are called. However, when a human body is involved in the physical system, things get much more complicated and it is usually impossible to apply standard methods with good accuracy. Consider the problem of dimensioning a joint prosthesis, for instance a hip replacement. It is obviously catastrophic for the patient if the implant fails under load so there are good reasons to analyze the design thoroughly for failure against the loads applied to it. But what are the loads? This is actually a very difficult question to answer, and sophisticated finite element analysis of the bone/implant structure will do little good if we do not know the input forces.

The trouble with such biomechanical systems is that most of the forces are generated by muscles, and it is in practice impossible to measure muscle forces. This leaves us with no other option than to try to compute the muscle forces, and this is precisely what the AnyBody Modeling System does.

AnyBody Modelling System
AnyBody is basically a multibody dynamics system in which the user can build and analyze models of mechanisms. As such, AnyBody can be used to analyze all sorts of mechanical systems. However, the system has the ability to work with the additional complexities of the human body of which the most important is an ability to predict how a redundant set of muscles are activated to perform a given movement. The redundancy is due to the fact that the body has many more muscles than degrees of freedom, and the muscles can therefore produce the same movement by infinitely many different combinations of forces.

Another challenge of human modelling is the complexity of mechanical elements in the human body. AnyBody is therefore affiliated with a public domain repository of human models that its users can build on. The models can be used on the level of individual body parts, as smaller collections of body parts, for instance the pelvis and two legs, or as an entire human body depending on the application. Einstein’s rule of making models as simple as possible but not more simple than that certainly applies here. The models are completely open, so the user can modify them to suit specific purposes: implant a joint prosthesis, paralyze a muscle or shorten a bone. The user can also develop environments that the body can be hooked up to such as a workplace or a bicycle as shown in Figure 1.

The big question is naturally: How well does it work? Model validation is an important issue and a difficult one because the simulated quantities are difficult or impossible to measure in the living body. However, two groups (Brand et al, 1994, and Bergmann et al, 2001) have succeeded to implant instrumented hip joint replacement into patients, and this has allowed them to measure the joint reaction forces during gait and other activities.

The authors of this article have collected sets of gait movements and ground reaction forces from a textbook (Vaughan et al, 1992) and newly recorded from the University of Miami and have imposed them on the computational of Figure 2.
The computed hip reaction forces are then compared to the measured joint reactions of the Brand and Bergmann groups. The result is shown in Figure 3.

![Figure 3: Comparison of vertical hip joint reactions.](image)

The results show that as in all biological systems, the variation is considerable. However, the variations between simulations and experimental results is of the same magnitude as the variations between experimental data and variations between the right and left hand sides. This computational fidelity is very good for a biological system and indicates that the muscle forces that have been computed by the simulation, and which cause the joint reactions we see here, are also credible.

**Interface AnyBody to ANSYS**

The next logical step is to transfer the forces from the multi-body simulation to a finite element model. In one case, Ozen Engineering, Inc. analyzed the performance of a spinal implant (Divringi et al, 2008) by running AnyBody model of a lateral bending as shown in Figure 4.

![Figure 4: Lateral bending – standing lift motion.](image)

Research suggests that the moment on spinal implants are more pronounced during ventral flexion (bending over), lateral bending(bending to the side). Since the extreme stresses will be ideal for testing the limits of the implant, an ideal human body movement would combine each of these elements into a motion. AnyBody provides a repository of projects with a variety of motions for ready use or modification. One of the included studies is the standing lift motion, which fulfils the above criteria nicely. In this study the subject is holding a 10 kg box while bending from side to side. The posture for this movement is bent over and the primary movement is lateral bending, which should stress the spinal implant a good amount.

Since the movement of the spine vertebrae are restrained by the implant, they must be similarly restrained in the human body model. Due to the assumption that there is perfect immobilization in the absence of true coupling between the softwares, the segments are set stationary to each other. AnyBody uses a spinal rhythm model for determining individual vertebra movements by approximating the segments as a bending beam with variable stiffness. In this way the vertebra positions are determined by constraint equations that have the thorax/hip angle as its main input and the spine keeps its natural shape when bending. To effect the immobilization from the implant, the stiffness coefficient between the two vertebra was set to zero. This ensures that after these vertebra assume their initial position they will not move relative to each other.

**A Generalized Tool**

To aid in this coupling a generalized tool was developed in house at Ozen Engineering, Inc. The tool, tentatively titled Any2Ans, has several capabilities which aid the process of
bridging AnyBody and ANSYS. Written in Java, the tool is portable across all platforms. After starting Any2Ans, it is then pointed to the main file of the standing lift study. This file is the start point for the entire study and includes references to several files, which can refer to still other files. Any2Ans reads through this master file and every file it refers to and builds an internal representation of the entire AnyBody project. This representation is programmatically searched for the physical elements used in the study and extracts them. The element geometries, including the bones, are converted from the internal format that AnyBody uses into the very common STL file format for future use. Any2Ans then runs AnyBody in batch mode and sets it to output the movement data of the geometries extracted from the study. The results are read and the model is displayed by the program (Figure 5).

After the AnyBody result has loaded, bone geometries may be selected or excluded as per the requirements of the simulation and exported into a single STL file. The entirety of the movement can be replayed within the program window and the exported geometries will be in the arrangement pictured. The motion is converted to the appropriate coordinate system and output into an XML file readable as a load history in ANSYS. Any motion exported will be relative to the arrangement of the resultant STL file and may be shifted and looped as needed.

Creating the ANSYS Simulation

The exported bone geometries are surfaced meshed and converted into volume bodies in ICEM CFD before being imported to ANSYS DesignModeler. This is necessary because the AnyBody bone geometries are primarily for visualization purposes and can have surface discontinuities or abnormalities that would cause problems with finite element analysis. In cases where the bones themselves are directly undergoing FE analysis, more detailed bone geometries would be desirable. The implant is recreated from the detailed drawings in the patent document and affixed to the vertebrae. The design is created in a procedural manner so the design parameters of the device can be altered and refined with little extra work. The model is advanced from the design state to the simulation stage where specific types of analysis can be set up. The goal is to apply boundary conditions to the vertebral bodies and observe their effect on the implant itself (Figure 6).

The contacts are set up for the implant to best reflect how it would mechanically function, the details of which are gone into in this study. The contact between the bone and implant is set to be perfect bounding. Affixation of the implant to the bone, though certainly significant and the subject of a large body of research, is beyond the scope of this study. The material selected for the implant is 316L Stainless Steel, for which there is data on alternating stress cycle response and strain life information, allowing long term reliability evaluation of the implant based on this simulation. The static simulation is now set up and run for both implant variations.

Creating the ANSYS Simulation

The exported bone geometries are surfaced meshed and converted into volume bodies in ICEM CFD before being imported to ANSYS DesignModeler. This is necessary because the AnyBody bone geometries are primarily for visualization purposes and can have surface discontinuities or abnormalities that would cause problems with finite element analysis. In cases where the bones themselves are directly undergoing FE analysis, more detailed bone geometries would be desirable. The implant is recreated from the detailed drawings in the patent document and affixed to the vertebrae. The design is created in a procedural manner so the design parameters of the device can be altered and refined with little extra work. The model is advanced from the design state to the simulation stage where specific types of analysis can be set up. The goal is to apply boundary conditions to the vertebral bodies and observe their effect on the implant itself (Figure 6).

The contacts are set up for the implant to best reflect how it would mechanically function, the details of which are gone into in this study. The contact between the bone and implant is set to be perfect bounding. Affixation of the implant to the bone, though certainly significant and the subject of a large body of research, is beyond the scope of this study. The material selected for the implant is 316L Stainless Steel, for which there is data on alternating stress cycle response and strain life information, allowing long term reliability evaluation of the implant based on this simulation. The static simulation is now set up and run for both implant variations.
Results
The comparison of joint reaction forces between the AnyBody generated values of the L1/L2 joint and the ANSYS generated reaction forces of the L2/L3 joint shows that the implant provides significant support in its intended area. This result also supports the validity of the simulation, with the implant having the desired effect on the body.

The stresses on the implant seem to be extremely high at first. Upon further inspection, the high stresses are heavily localized around a small number of elements near contact surfaces and can be ignored. However, the implant still shows a significant amount of yielding in several places, hovering at or below a safety factor of 1 in several areas. In this simulation the 316L stainless steel is insufficient for the needs of this implant (Figure 8).

Figure 8: Safety factor contour plots on the implant

Conclusion
By improving the coupling of AnyBody and ANSYS, new ways of analyzing medical devices become apparent. This spinal implant was tested not by a best guess max moment, force or stress but by a specific motion of the human body. The results achieved in this case study are satisfactory but there are improvements that could be made, both to this case study and future studies using this general procedure.

References

Authors
John Rasmussen, Aalborg University
Metin Ozen, Ozen Engineering, Inc.