Real-time Finite Element Analysis with Virtual Hands - An Introduction

Disi Lian
School of Engineering and the Built Environment
SC040, Telford Campus
University of Wolverhampton
TF2 9NT, Telford, UK
E-mail: d.lian@wlv.ac.uk

Ilias A. Oraifige
School of Engineering and the Built Environment
SC121, Telford Campus,
University of Wolverhampton
TF2 9NT, Telford, UK
E-mail: i.a.oraifige@wlv.ac.uk

F. Richard Hall
School of Engineering and the Built Environment
MA109, University of Wolverhampton, WV1 1SB,
Wolverhampton UK
E-mail: f.r.hall@wlv.ac.uk

ABSTRACT

Computer aided Finite Element Analysis (FEA) tools have been used since the late 1950's. The asynchronous plot of simulations and the corresponding results consumes a large amount of engineering cycle time; additionally, applying FEA parameters on a 3D CAD model with 2D mouse/keyboard has always been a complex process. This paper presents the prototype of a software toolkit that performs real-time FEA simulations, the software is associated with Virtual Reality (VR) technologies which gives users a better sense of spatial presence. Interactive product design and real-time FEA is discussed in this paper.

Keywords

Interactive, Real-Time Simulation, Virtual Reality, Collision Detection, Finite Element Analysis, Virtual Hands.

1 INTRODUCITON

Putting VR technologies into practical usage in the engineering field has always been an interesting goal within researcher's community. The performance limit of computing hardware and the efficiency of software algorithms are critical obstacles in this field. This paper presents a real-time FEA toolkit which reduces the time taken to implement linear-elastic FEA simulations; the human-machine interface of this system employs VR technologies—namely tactile gloves and virtual hands—to increase users' sense of special presence.

CAE tools have been used since the late 1950's, the importance of such tools to modern engineering can never be over stated, aerospace and automotive industries save billions of dollars each year by using such tools. FEA simulation software is one of such kinds; engineers can perform analysis of complex structures with little or no physical prototypes [Zor03] with the help of sophisticated FEA tools. However, performing FEA simulations on complex structures is usually a very time consuming procedure, this is partly because graphical simulation results are not displayed synchronously to background calculations in most FEA systems. A real-time FEA system will enable engineers to observe simulations as it is calculated, moreover, engineers can stop the simulation at any point that deters the success or failure of it. Such a system is more cost effective in terms of money and time.

Additionally, current FEA systems usually have 3-dimensional (3D) User Interface (UI); however, the complexity of the UI countervails the benefit of using a 3D human-machine interface. On the other hand, VR technologies have been around since the late 1980s' [Bry97], such technologies are particularly good at providing comprehensive UI. A virtual hand integrated UI may offer a better sense of spatial presence to the user than UIs using 2D mouse, this is especially true when the observation of conceptual models is such a great deal at the early stage of design.

The performance of computer hardware has been improved rapidly in recent years that performing real-time simulations is not only possible but has been well demonstrated by the computer game industry. As far as the application in engineering field is concerned, many researchers have also made lots of efforts. Michael Ryken and Judy Vance [Ryk00] have developed a CAVE like system that intends to integrate VR technology with FEA, although the practicality of a system at such scale is arguable, the possibility and productivity of implementing such a system are obvious. Jeffrey

WSCG POSTERS proceedings
WSCG'2004, February 2-6, 2004, Plzen, Czech Republic.

Berkley's virtual suturing application [Ber02] also involves real-time Finite Element Modelling (FEM) methods, several useful algorithms such as matrix equation inversion are introduced in their research; Ling Lu and Odd Tullberg have discussed the possibility of achieving real-time graphical feedback of FEA results at the cost of accuracy in their work [Lu01]; Yeh incorporated finite element data into a VE that the designer can interactively modify the transverse force using 3D mouse and observe the resulting deformation that each deformation level is represented by different colour [Yeh98], the FEA module of this system is powered by MSC/NASTRAN.

2 APPROACH

2.1 System workflow

This paper introduces a real-time FEA system that deals with linear-elastic analysis. The numerical calculation functionality is based on a solver that integrates key-frame method with the PCG solver of ANSYS 6.1. A pair of virtual hands is generated for the system; users conduct the virtual hands through a pair of customised Fakespace pinch gloves. The entire system is based on a PC with a 2.2GHz Intel P4 processor, 512MB RAM, and 64MB graphics memory. Fig. 1 shows the system workflow of the toolkit developed in this research. A 3D interactive interface, a background numerical calculation module and a Data Interpreter between them form up the infrastructure of the system.

The front-end GUI provides users with a pair of virtual hands to manipulate IGES formatted CAD models. At the back-end, the batch processing module collects information sent by the UI, and then performs necessary calculations to find the solution for given FEA simulations.

The toolkit has two working modes: Manipulation Mode and Interactive FEA Mode. At the pre-processing stage, the system works under the Manipulation Mode, users are able to navigate the CAD model and apply material information on it. At the backend, a very important pre-process—stiffness

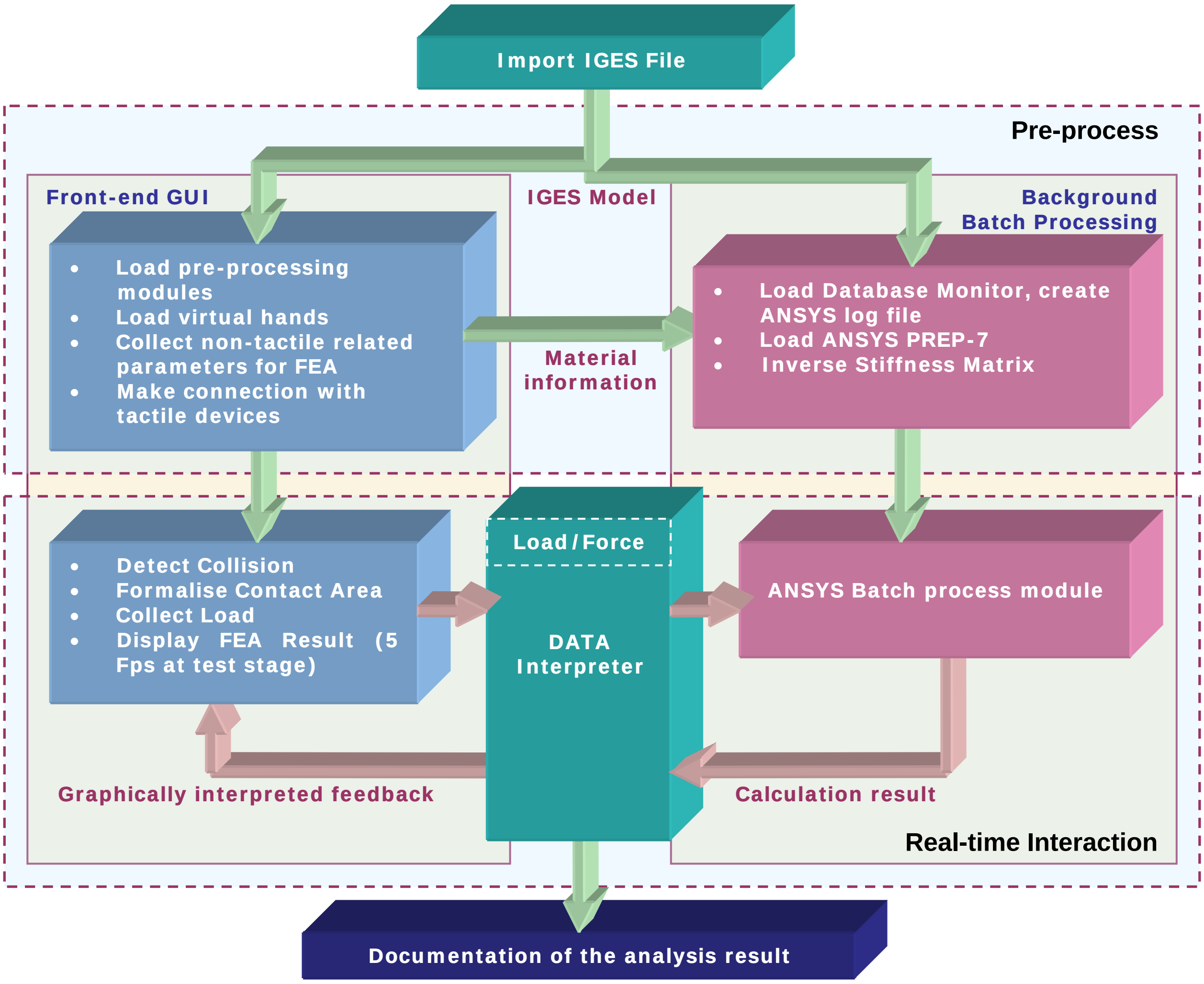

Figure 1 – System workflow of the real-time FEA toolkit

matrix inversion—is performed, the pre-processor collects the material information and inverses the stiffness matrix ([K] in Eq.1) in the finite element equation for static linear-elastic problems.

$$[K]\,[d] = [f] \qquad \text{Eq.1}$$

This inversed matrix will be used later at the real-time FEA stage. Several different sets of inversed matrix can be saved each reflects a unique simulation environment; however, only one set can be used at a time. The major purpose of inversing the stiffness matrix at this stage is that it is solely related to the material of the structure to be analysed, and therefore it does not change according to force or pressure applied in a linear-elastic analysis, a pre-processed matrix [K] can well serve the whole calculation without consuming computing resources at the real-time FEA stage.

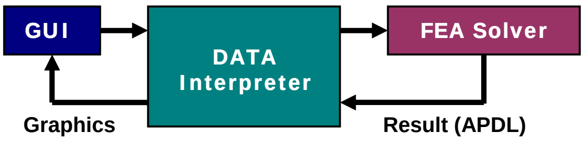

Figure 2 – Interactive FEA loop

On the completion of stiffness matrix inversion, the CAD model is meshed, and then the toolkit is switched to the Interactive FEA mode, under which the toolkit works in a complete loop for each frame (Fig. 2). Raw force data collected from the tactile gloves is converted to APDL language command line by the Data Interpreter and then sent to the backend FEA solver; the optimised force feedback is sent back to the gloves to trigger the vibrator and generate the sense of touch. The FEA solution is sent back to the front-end GUI through the Data Interpreter. Each critical step is saved in the analysis log file by the post-processing unit. The processing speed of this FEA loop is set at 5 fps.

2.2 Collision detection and virtual hands

IGES 5.3 file is one of the industry standards for FEA. It can be taken as polygonal meshes, which is ideal for a system that deals with collision detection and FEA simultaneously.

The collision detection algorithm this toolkit employs is a mixture of the concept of OBB [Kri98] and Sphere Box [O'su99]. Virtual objects are boiled down to a hierarchical tree in which the first three levels are oriented bounding boxes because OBB tree is effective in dealing with slim objects such as the fingers on virtual hands. For further levels beyond the first three, sphere volumes are used to perform the exact contact.

The collision detection module is also responsible for the correct functioning of the pair of virtual hands associated with the system, and the gesture control of

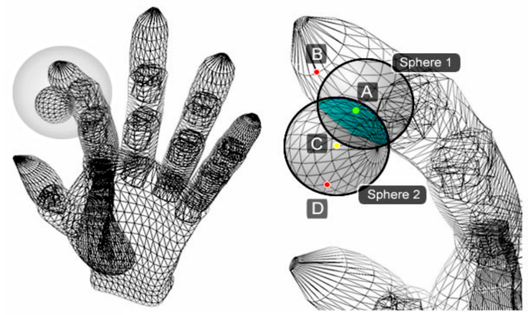

Figure 3 – Determine contact area

the virtual hands is greatly integrated with the tactile gloves. There are three gesture sensors at the joints of the phalanxes on each finger. These three sensors measure the degree of relative rotation of each part of the virtual hand. The sensors have their counter parts in the pinch gloves which updates the gesture of the virtual hands by sending signals collected from the tactile gloves. Apart from the rotation sensors, two position tracking sensors is equipped on the palm of both virtual hands and pinch gloves, these two sensors controls the translation and orientation of the whole hand base for all the fingers of the whole hand.

Figure 4 – Interaction using pinch gloves

2.3 Interactive FEA

Currently, this FEA toolkit solves linear structural problems. The FEA module uses PCG solver of ANSYS V6.1 as a black-box processing unit, the PCG solver is able to deal with 50,000 to 1,000,000+ DOFs per second and is very fast for structural solid elements and shell elements. In addition to the default FEA solver, the following algorithms are applied to the process to achieve better performance.

2.3.1 Stiffness matrix inversion

For linear elastic analysis, the Finite Element equation can be described as

$$[K]\,[d] = [f] \qquad \text{Eq.1}$$

Where K is an n x n stiffness matrix, d and f are n x 1 vectors that represent nodal displacements and the force applied in the analysis. The ultimate goal of a FEA simulation is to find the corresponding d from the known value of f, and determine the corresponding stress at distorted area. The key point here is that in order to find a FEA solution, Equation 1 has to be inversed to calculate the displacement value according to f, obviously, it is not necessary and computationally expensive to process matrix inversion every time the applied force is changed for every frame. Because [K] is only related to the material, a pre-calculated inversed stiffness matrix can be applied through the whole analysis procedure as far as the same analysis environment is used

The inverse of the stiffness matrix can be determined in approximately αn3 computations, where n is three times the number of nodes and α refers to efficiency gains that can be achieved through sparse LU [1] decomposition techniques, thus, equation 1 can be rearranged as the following equation:

$$\begin{Bmatrix} d_{noContact} \\ d_{Contact} \end{Bmatrix} = \begin{bmatrix} Ki_{aa} & Ki^{T}{}_{ab} \\ Ki_{ab} & Ki_{bb} \end{bmatrix} \begin{Bmatrix} f_{noContact} \\ f_{Contact} \end{Bmatrix} \quad \text{Eq.2}$$

$d_{Contact}$ and $f_{noContact}$ are know factors, Ki is the inverse of [K]. From Equation 2, the final equations can be derived to obtain the unknown variables at run-time.

$$d_{noContact} = Ki_{aa} f_{noContact} + Ki^{T}{}_{ab} f_{Contact} \quad \text{Eq.3}$$

2.3.2 Key - Frame method

In addition to the stiffness matrix inversion technique, the performance of real-time analysis can be increased by introducing a key-frame mechanism in combination with deformation predicting module. Key-Frame method is usually used in video editing software for image compressing purpose. The key point of this concept is to concentrate only on the key frames of the analysis.

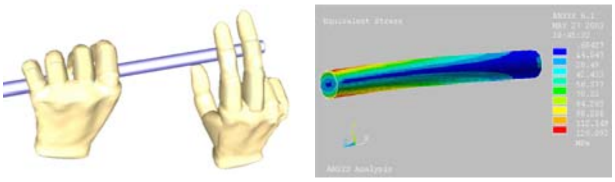

Figure 5 - Illustration of FEA Simulation

During operation, the load sent from the Data Interpreter is divided into finite time steps with a frame rate at 5Fps, and there is one key-frame for every five normal frames. The ANSYS PCG solver only calculates the solutions for key-frames, and a simpler and less accurate deformation predictor using Spectral Lanczos Decomposition Method (SLDM) [Dru94] smoothes up the rest frames. For every frame after a key frame, the initial position information of the analysed model in the database for SLDM solver will be updated with the displacement solution found from the PCG solver of ANSYS.

3 CONCLUSIONS

Introducing VR technology to interactive FEA is an important research area in Virtual Prototyping. For years, efforts have been made in developing virtual design tools, many other applied certain FEM techniques on generating simulations, the research presented in this paper has made an attempt to provide engineers with a time-saving interactive FEA toolkit, which not only implements FEA simulations in a comprehensive and interactive way, but also demonstrates the possibility of real-time numerical calculation on a PC. It is hoped that the introduction of this research itself will draw more attention in the application domain and therefore lead to more effective algorithms and applications in this field, further development will be made according to the analysis at the end of the description of each module.

4 REFERENCES

[Ber02] Berkley, J. Haptic Rendering of Deformable Bodies using Real-Time Finite Element Analysis: An Application to Suturing Simulation. Doctoral Thesis, University of Washington.

[Bry97] Bryson, S. Virtual Reality. The Computer Science and Engineering Handbook 1997. pp.1354-1374. 1997

[Dru94] Druskin, V. and Knizhnerman, L. Application of the Lanczos method to solution of some partial differential equations, Computer. Application. Math. 1994, v. 50, pp. 255-262.

[Kri98] Krishnan, S., *et al.,* Rapid and Accurate Contact Determination between Spline Models using ShellTrees, Computer Graphics Forum, 1998.

[Lia03] Lian,D., Oraifige, I.A., Hall, F.R. Usage of virtual hands in computer-aided finite element analysis. Num3D-3D Solid Digitising & Modelling 2003, Paris, France. pp.195-202, 24th April, 2003.

[Lu01] Lu, L., Connell, M. and Tullberg, O. The use of Virtual Reality in Interactive Finite Element Analysis by Java3D API. Applied Virtual Reality in Engineering and Construction, proceedings of the conference AVR II and CONVR 2001, pp 60-68.

[O'su99] O'lSullivan, c. and Dingliana, J., Real-Time Collision Detection and Response Using Sphere-Trees, 15th Spring Conference on Computer Graphics. pp 83--92. Budmerice, Slovakia, April 1999.

[Ryk00] Ryken,M.J. and Vance, J.M. Applying Virtual Reality Techniques to the Interactive Stress Analysis of a Tractor Lift Arm. *Finite Elements in Analysis and Design* 35, pp. 141-155, 2000.

[Yeh98] Yeh, T.P. and Vance, J.M. Applying Virtual Reality Techniques to Sensitivity-Based Structural Shape Design. ASME Journal of Mechanical Design December 1998, Vol. 120, No. 4, pp. 612-619.

[Zor03] Zorriassatine, F. *et al*., A Survey of virtual prototyping techniques for mechanical product development. Proc. Instn Mech. Engrs Vol. 217 Part B: J. Engineering Manufacture. IMechE 2003.

[1] Please refer to the book Numerical Recipes in C by Press, W.H., et al.