A framework for Soft-tissue Deformation

Umut Koçak* Karljohan Lundin Palmerius† Anders Ynnerman‡

Norrköping Visualization and Interaction Studio, Department of Science and Technology,
Linkoping University, SWEDEN

Abstract

For the simulation of soft tissue deformation Finite Element Method (FEM) has been the technique which achieves most physically realistic behaviour. However to provide stable force feedback, high refresh rates are needed which makes the use of FEM non-trivial.

To solve the challenge of the compromise between speed and realism, *asynchronous regions* which solves different parts of the model with different frequencies and different resolutions are used. The local neighbourhood of the contact is solved with higher frequency and resolution while the more remote regions are solved with lower frequency and resolutions. Different solution methods, implicit and explicit, can be used to solve different regions and the size of the regions can be adapted depending on the strain to maximize the efficiency.

Keywords: Haptics, FEM, soft-tissue deformation, surgery simulations

1 Introduction

The use of haptics in virtual environments has introduced the sense of touch in addition to the visual and audio outputs to increase the immersion into the environment. Haptics has been used for various aims like achieving guidance, force visualization or realism. Simulation of soft tissue deformation is one of the challenging fields demanding realism to be used in surgery simulations, especially since achieving a realistic deformation behaviour with sufficient refresh rates of force feedback has been an issue.

The challenge is to model the complex physical properties of soft tissue which results in time-consuming calculations together with stable and continuous force feedback. The refresh rate for the output force should be around 1 kHz to attain a continuous force sensing. By using today's fastest computing power, achieving the desired refresh rate is still not possible if the physical properties of soft tissue are to be simulated. Therefore a compromise between the realism and speed has been reached by sacrifising some of the material properties to achieve real-time simulations with force feedback.

Many different techniques exist to simulate deformation, for example mass-spring, chain-mail, finite element model . The mass-spring models are easy to implement and sufficient refresh rates can be reached, however the physical properties of materials cannot be taken into account and the model may respond un-realistically to larger deformations in these methods. For the chainmail methods, the inhomogeneous data and the un-realistic deformation behaviour have been problematic against the simplicity and speed of the method. Among these techniques, Finite element Method (FEM) is the one that results in most realistic deformations by allowing simulation of physical properties, however with a high computational burden drawback.

*e-mail: umut.kocak@itn.liu.se

†e-mail:karljohan.palmerius@itn.liu.se

‡e-mail:anders.ynnerman@itn.liu.se

Here we susggest the combined use of *asynchronous regions* [Koçak et al. 2009] and spatial level of detail for deformation to achieve a realistic force feedback with a refresh rate of 1 kHz. In the asynchronous regions approach the FEM equations are seperated into different regions, each of which can have different element sizes, be solved with different frequencies and even with different solution techniques(like implicit or explicit). By using this technique the local neighbourhood around the contact is modeled with higher resolution and the corresponding FEM equations are also calculated with higher frequency (1 kHz) than the more remote regions. Having remote regions modelled with lower resolutions and updated with a smaller frequency saves a computation power which is dedicated to attain both high resolution and the refresh rate within the local neighbourhood of the contact.

2 Related Work

There exists many different techniques to simulate deformation. The choice of the approppriate technique mainly depends on the aim of the application. Some methods are better to have a visually appealing deformation while for physically realistic deformation complicated techniques may be needed. In case of surgery simulations, it is crucial to have a physically realistic behaviour for the training and rehearsal purposes. In [Nealen et al. 2006], the reader can find an overwiew of pyhsically realistic deformation techniques in computer graphics field.

FEM has been one of the most common techniques used in surgery simulations because of the ease of modelling physical properties of organs realistically. Most of the soft tissues have a characteristic non-linear and visco-elastic behaviour, which comes with a computational burden. There exists numerous studies ignoring these physical properties while exploiting FEM for surgery simulations.

Although there exist several simulations of soft tissue deformation, none has accomplished to model all properties of soft tissue like visco-elasticity or non-linearity satisfying the requirement of a sufficient haptic refresh rate (1 kHz). Therefore the field is still active and waiting for more optimization techniques.

3 Asynchronous Regions

After loading the volumetric data, the finite element mesh which can have a different resolution than the data is created by using basic principles of FEM [Zienkiewicz et al. 2005]. The dynamic formula (1) is used to solve the system, where the stiffness matrix, displacements, forces, mass and damping are represented by $\mathbf{K}$, $\mathbf{u}$, $\mathbf{f}$, $\mathbf{M}$ and $\mathbf{C}$ respectively while $\dot{\mathbf{u}}$ and $\ddot{\mathbf{u}}$ refer to velocity and acceleration. The simulation is integrated into time steps to solve (1) by using implicit or explicit methods.

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{f} \quad (1)$$

During pre-processing nodes of each element are assigned to the data vertices spatially covered by the element. The assigned mesh nodes are used to interpolate the applied input and displacements between the model and the mesh. Having different resolutions for the data and the finite element mesh provides the ability to have

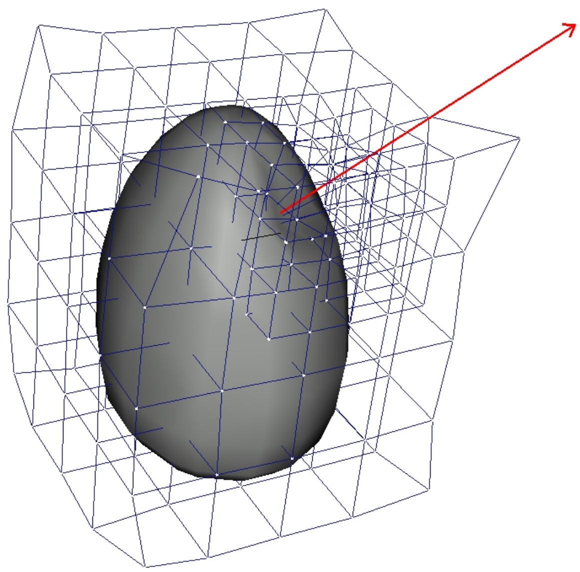

Figure 1: *The finite mesh and the data can have different resolutions in addition to the adaptive level of detail for finite mesh.*

lower finite mesh resolutions for the regions away from the contact while attaining the same visual resolution for the data. The concept is illustrated in figure 1.

The asynchronous regions are the seperate parts of the FEM model solved with different frequencies. The idea is exploited to solve the local neighbourhood of the contact (primary region) with 1kHz, while having lower frequencies for the remote regions (secondary regions). It is also possible to choose different types of solvers like implicit or explicit for the solution of different regions. There is no limitation for the number of regions also, a cubic data with 3 regions is illustrated in figure 2. During initialization, the number of regions and the default values for the sizes, frequencies and solver types of each region are chosen.

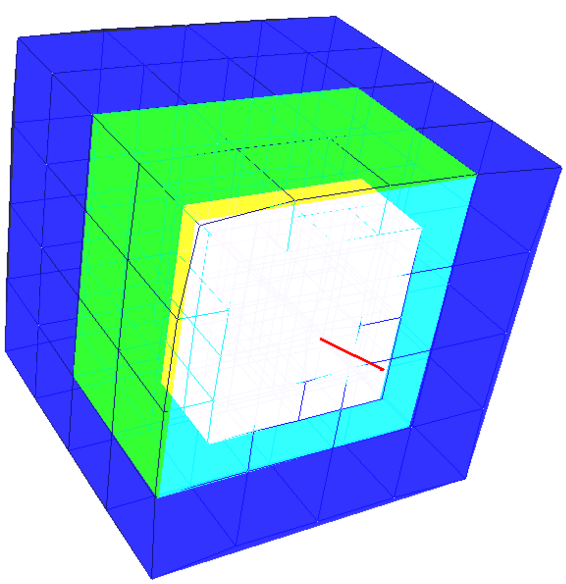

Figure 2: *Allocation of regions on the model.*

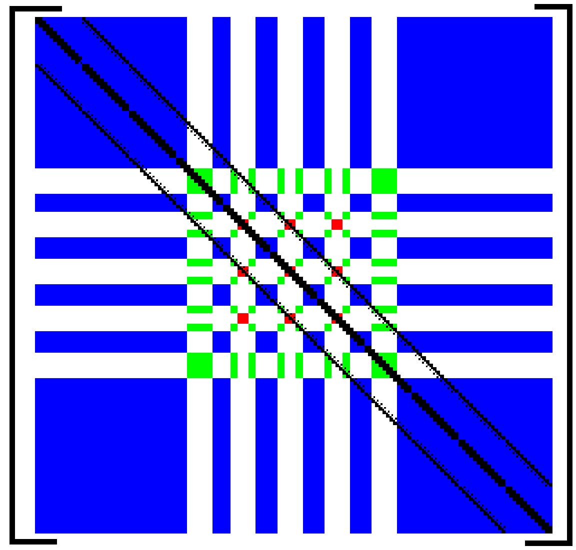

Figure 3: *The allocation of regions is illustrated by assigning different colors to the equations of each region.*

In case of contact with the model, the nodes of each region is updated in collision thread depending on the default sizes of regions. The size of the primary region is adaptable depending on the amount of the strain applied. The nodes of each region are solved aynchronously in different threads.

Seperating the model into different regions corresponds to seperating the stiffness matrix into different regions also. The allocation of the regions is illustrated in figure 3 by using different colors. Each thread solves the equations of its own region and updates a shared displacement data among the threads. For the nodes of the lower frequency regions, the interpolated displacements are used both in calculations and visual rendering.

4 Spatial Level of Detail

The level of detail in spatial domain is combined with the asynchronous regions to model the remote regions from contact with lower resolution also. The major possible drawback of the multi-resolution applications is the time consuming remeshing phase. Considering the possible frequent change of contact and the desired refresh rates, it is not feasible to apply remeshing in real-time. Therefore the finite mesh and independent stiffness matrices are evaluated for different levels during pre-processing. When the contact is changed, the node list of the regions are changed for each level on the fly without any need of remeshing. Each thread solves the nodes of its own region by using the stiffness matrix of its own level and updates the finest finite mesh.

5 Discussion

The use of asynchronous regions with multi-resolution mesh provides the ability of modelling the local neighbourhood of the contact with higher resolution by also attaining 1 kHz refresh rate for the primary region. Besides having different solution methods for different regions is exploited by solving the local region with explicit-solvers with a high frequency and having unconditionally stable implicit-solvers allowing larger time-steps for the more remote regions updated with a low frequency. The regions are updated on the fly when the contact node is changed in addition to the possibility of adapting their sizes depending on the magnitude of the strain.

As a future work, non-linearity and viscoelastcity will be integrated to the simulation in addition to importing the calculations to GPU in order to increase the performance.

Acknowledgements

This work has been funded by the Swedish Science Council through grant number 621-2005-3609 and by the Foundation for Strategic Research (SSF) under the Strategic Research Center MOVIII.

References

KOÇAK, U., PALMERIUS, K. L., AND COOPER, M. 2009. Dynamic deformation using adaptable, linked asynchronous fem regions. In *Proceedings of the Spring Conference on Computer Graphics*, 213–220.

NEALEN, A., MUELLER, M., KEISER, R., BOXERMAN, E., AND CARLSON, M. 2006. Physically based deformable models in computer graphics. *Computer Graphics Forum 25*, 4 (December), 809–836.

ZIENKIEWICZ, O. C., TAYLOR, R. L., AND ZHU, J. Z. 2005. *The Finite Element Method: Its Basis and Fundamentals*. Elsevier Butterworth Heinemann, Oxford.