Textbox 1

A. Mechanical Properties

The term **stress** refers to a deforming force (e.g. pressure, stretch, shear, torque, etc.) applied to an object, expressed in terms of the unit area over which it is applied. The deformation of the object that results from the stress is referred to as **strain,** which is a normalized measure of deformation of an object compared to its original shape (e.g. $\Delta L/L$ where ΔL is the change in length of an object which started out with a length, L)

The resultant deformation depends, among other things, on the material properties of the tissue involved. The simplest relationship between stress and strain is known as **linear elastic** behaviour in which strain is directly proportional to the stress applied. The relation between these two varies from one material to another and is known as **Young's modulus**, E, where:

$$E = \text{stress}/\text{strain}.$$

(The units of both stress and Young's modulus are N/m^2. Strain is a ratio and is therefore dimensionless.) Stress and strain have a directional component – there may be completely different stresses acting on, say, the front and the side of an object. Direction is therefore an important component to the accurate description of both stress and strain which are best described as vectors. Vector geometry is potentially complicated and so a simplified, directionless, value of "average" stress (**von Mises stress**) or strain (**von Mises strain**) [1] is often used for the purposes of modelling.

Three-dimensional structures have the capacity for compression in one axis to be partially compensated for by expansion in a perpendicular axis. How much this can happen is a property of individual materials and is described as the **Poisson ratio** which represents the ratio between transverse and axial strains. For the purposes of modelling, any material is usually described in terms of its Young's modulus and its Poisson ratio. What these values actually are for each material may, or may not, be easy to determine. Non-biological materials lend themselves to mechanical testing and so these parameters are usually well-described. Biological materials (tissues), on the other hand, are not so easily tested and their values may not actually be known. In this case it is necessary to make assumptions based on the mechanical properties of similar tissues which have been tested.

In reality, the behaviour of most materials, particularly biological tissues, in response to strain is much more complex and involves **hyperelastic** and/or **viscoelastic** behaviour. In addition, there is often a difference between the mechanical properties in different planes (**anisotropy**). Description of these is beyond the scope of this article, but the interested reader is referred to [2]. Suffice it to say that the mathematical representation of both these behaviours is much more complicated than that of linear elastic behaviour. For that reason, most models start out by assuming linear elastic properties (as here), accepting the fact that future development will almost certainly require much more sophisticated mathematics to generate a better simulation of reality.

B. Finite Element Modelling

In finite element modelling (FEM), a complex structure is divided up ("discretised") into a large number of small **elements** (**nodes**), the whole being referred to as a **mesh**. Each element is linked to a number of adjacent elements, potentially transmitting stresses onwards to each of them. The resultant deformation for any one element will be determined by the sum of all displacements of the nodes connecting the element to the surrounding elements. Appropriate material properties (Young's modulus and Poisson ratio) must be used to model the structural response of the elements, in this case a range of different tissues. In addition, there is the issue of how different components (and therefore elements) interface with each other when they come into contact. For example, is movement between elements possible and, if so, is friction involved or not?). Modelling the interaction between elements representing different components involves what are known as **boundary** assumptions.

Thus, generating a model requires many different considerations. For example, how many elements should be modelled? What is a sensible number of element-to-element interactions to model? What are the material properties of each element? What interactions are possible between adjacent elements, both within tissues and between tissues (boundary conditions)? And so on. As can be imagined, there is an enormous amount of computation involved in solving the millions of simultaneous equations required to determine the behaviour of each element when a particular stress is applied to the model as a whole.

Finite element models must be **verified** and **validated**. The former involves ensuring that the computer code correctly represents what is being modelled. The latter involves assessing the predictions of the model against reality. Validation is often particularly difficult when dealing with biological models because appropriate information may not be available.

All finite element models simulate a condition and yield results, but validation is required to know both how well the results from the model mimic the simulated condition and to determine if the model has predictive value and can therefore be relied upon to provide insight into the modelled system. While unvalidated models can be used to provide some insight in limited circumstances when experimental data is not available, the validity of this insight will only be as good as the assumptions and approximations used.

REFERENCE

1. Ueno K, Melvin JW, Li L, Lighthall JW. Development of tissue level brain injury criteria by finite element analysis. J Neurotraum. 1995;12:695–706.
2. Fung YC. Biomechanics: Mechanical Properties of Living Tissues, 2nd edition. New York, NY: Springer-Verlag, 1993.