Day, A.J., Allport, John, Fischer, W.P., Coates, P. and Mimaroglu, A.

Finite element modelling of polymer deformation processes

Finite Element modelling of polymer deformation processes

AJ Day, JM Allport, WP Fischer, PD Coates, A Mimaroglu,

University of Bradford,
Interdisciplinary Research Centre in Polymer Science and Technology,
Bradford, West Yorkshire, UK.

Abstract

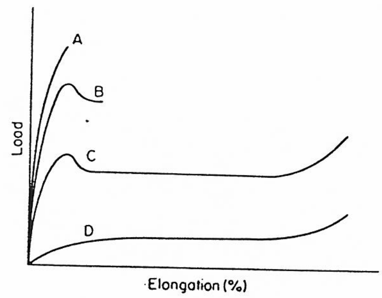
Three examples of the use of ABAQUS for the finite element analysis of polymeric or elastomeric components and products, drawn from a programme of research at the University of Bradford into polymer material modelling and characterisation for Computer Aided Engineering, are presented. The analysis of a die drawing process applied to Polyethylene pipe is described; this includes a moving mesh, pipe wall/die surface friction, and non-linear material properties. The modelling of filled elastomer material for use in driveline couplings and vibration control using Hyper elastic material models is then described: the Mooney-Rivlin and the Ogden models have been used with limited success, but statistical mechanics models have produced the best correlation with experimental test data. The high strain behaviour of Polypropylene in free-boundary drawing of sheet specimens is finally briefly described using a stepwise sequential analysis procedure to include non-linear material properties derived using novel image analysis procedures. It is concluded that ABAQUS provides an excellent research tool for analysing polymer deformation processes.

1. Introduction

The successful use of Engineering polymers relies upon the ability to understand and predict the load/deformation behaviour as part of the design process. The geometric form of polymer products and components is not always complicated, but the complex material behaviour, often associated with large scale deformation and time-varying contact behaviour, requires sophisticated mechanical and mathematical modelling. This is part of a programme of research into the solid phase deformation of polymers at the University of Bradford, within the Interdisciplinary Research Centre (IRC) in Polymer Science and Technology at Leeds, Bradford and Durham, UK.

The mechanical behaviour of polymers depends upon the type of polymer, processing history, as well as the conditions under which they are being used. Modulus, for example, is usually strain, strain-rate, pressure and temperature dependent in polymers, and this complicated behaviour arises because of the viscoelastic nature of polymer materials under normal operating conditions at room temperature and above (Ward 1990). The relationship between stress and strain can also be anisotropic and highly non linear as effects such as orientation of the polymer chains occur. Polymer deformation is often described as being "recoverable" (rubber-like), or permanent (un-recoverable), or a combination of the two. Dynamic properties of polymers are usually substantially different from static properties; the visco-elastic behaviour dissipates energy providing damping and alters the output phase from the input. At the same time heat is generated which causes sufficient temperature rise to alter material properties. Typical polymer behaviour under test conditions of uniaxial loading is illustrated in Figure 1 (Ward 1990).

Figure 1, Load-elongation curves for a polymer (Ward 1990)



To take all these features into account in the engineering analysis of polymer components and products is a mammoth task, and therefore a number of approximations or models have been generated which incorporate one or more of the important characteristics of the material being studied. For example, in small strain elasticity, polymer non-linear material behaviour can be modelled in ABAQUS using a defined stress-strain characteristic with incremental loading. For large strain problems however, it is necessary to define a constitutive relationship between stress and strain which correctly represents the behaviour of rubber-like materials under loading conditions; these are known as hyper-elastic material models. Alternatively a stepwise approach can be adopted in which the deformation parameters such as stress, strain, strain rate and temperature can be calculated and the corresponding material properties can be selected and applied in each separate step of the analysis. In this paper examples of all three of these methods for modelling polymer material behaviour are presented and discussed.

2. The solid phase processing of Polyethylene pipes

2.1 Process description

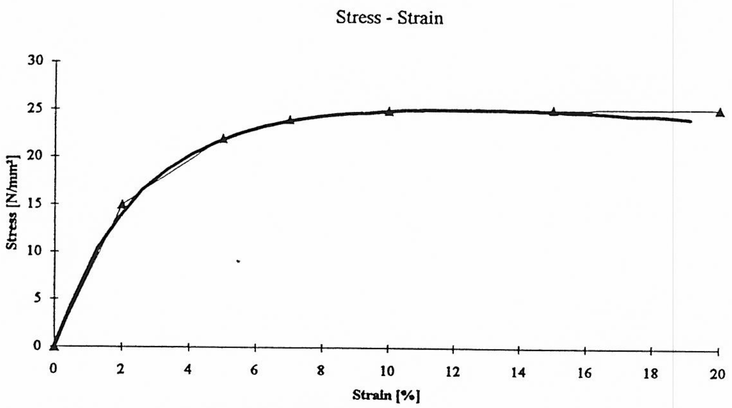
Polyethylene pipes are extensively used in the gas and water distribution industries both for new installations and for the renewal of existing distribution systems, some of which can be 100 years old or more. One way of renewing existing pipelines which have become too old for safe or reliable operation is to re-line the existing pipe, which is usually cast iron or steel, with a polymer pipe liner. Methods have been developed to minimise the resulting reduction in cross sectional area, one is to utilise the viscoelastic properties of the polymer pipe material to recover to as near to its original diameter (prior to insertion) after relining is completed. During the process a relatively thin-walled polyethylene pipe is drawn through a metal die, which reduces its outside diameter or cross-sectional area sufficiently to permit easy passage through an existing cast iron pipe. On release of the towing load the polymer pipe recovers towards its original diameter and against the inside diameter of the original pipe.

ABAQUS version 5.2 is being used to model this process, with the objective of being able to predict parameters such as installation drawing load, optimum pipe diameters, die profile design, stress and strain conditions in the pipe wall, residual stresses, and the viscoelastic recovery for different materials. The process essentially consists of forcing a continuous tube through an appropriately shaped die by a draw force or displacement applied to the leading end of the tube. The two major kinematic parameters of the process are the nominal reduction ratio R, which is the diameter at the exit of the die (r) over the original tube size (r_0) and the die angle α, which is the half angle of the conical portion of the die.

2.2 Material behaviour

Experimental tensile test data have been fitted mathematically over the appropriate strain range to give a variable stress/strain property relationship for inclusion in the ABAQUS input file as several straight lines with an accuracy better than ±1% of the stress. A linear elastic region from zero to 2% strain modelled the material to reach a "knee point". Plastic behaviour was then modelled to a maximum strain of 20%. A stress/strain curve in tension and the modelled approximation for ABAQUS can be seen in Figure 2.

Figure 2, **Polyethylene uniaxial tensile characteristic.**



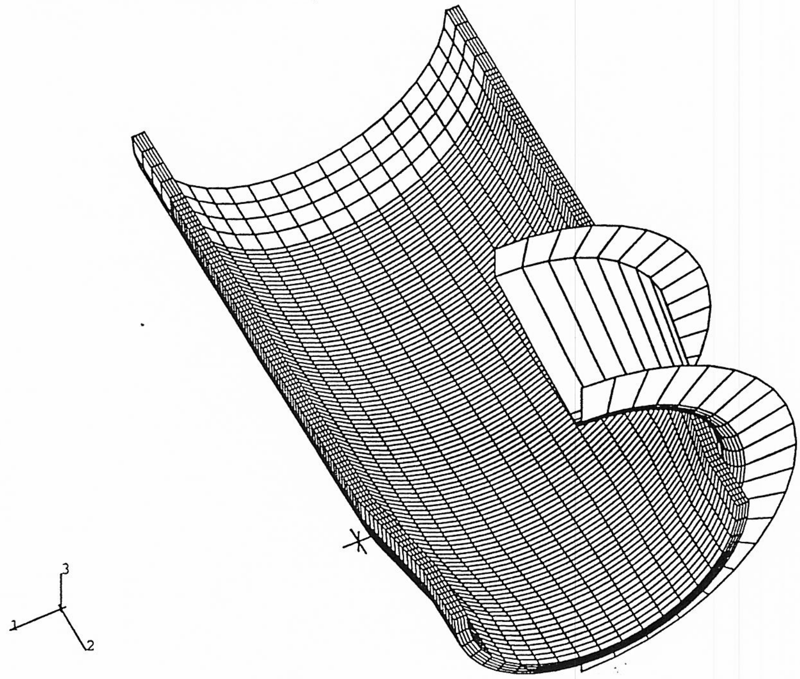
2.3 Analysis procedures

The deformation process has been modelled in a predeformed axisymmetric finite element model, using ABAQUS element type CAX4R with a very long length of material before the die because only this part of the material undergoes the full deformation during the process. A long mesh with approx. 4000 elements was first created to investigate model stability, but subsequently this detailed mesh was reduced to about 2500 elements. This size of mesh was required to reach a steady state condition in the deforming zone as the elements go from an unstrained state through the entire deformation process in the die: by an initial elastic analysis it was found that the displacement required was 2.5 times the length of the die for a typical die angle of 12.5 degrees. The undeformed mesh shown as a three dimensional body can be seen in Figure 3.

The die surface, the relative motion of the pipe to the die surface, and the friction at the die/pipe interface was defined using the *INTERFACE option. The rigid surface was modelled employing the *RIGID SURFACE option, with the reference node being fixed for all four possible degrees of freedom. Friction was employed to calculate the shear forces between the surfaces by applying interface elements (ABAQUS interface element type IRS21A) between the two bodies. The rigid surface was smoothed with a radius of 1 mm to ease convergence.

The FE mesh was displaced incrementally through the die by applying a prescribed displacement in the axial direction on the front surface or trailing end of the pipe by applying a prescribed ramped *AMPLITUDE. During the analyses the asymmetric matrix solver was utilised as well as the NLGEOM switch for large displacements. The default elastic slip of the interface elements was not changed, as element sizes were small and displacements were relatively large.

Figure 3, Pipe finite element mesh.



2.4 Results

A plot of the radial stresses in the pipe wall after a displacement of 130 mm is shown in Figure 5. This clearly indicates two-point pipe/die contact which is also observed in practice. A displacement of 130 mm was found to be sufficient to reach a stable solution at a die angle of 12.5 degrees; the load stabilised after approximately 120 mm. These results also demonstrated that the linear four-noded reduced integration axisymmetric elements were the element type providing the best accuracy and efficiency of solution compared with other higher order elements or not-reduced integration elements. Some summarised results are illustrated in Figures 6 and 7.

Figure 5, **Radial stresses in the pipe wall.**

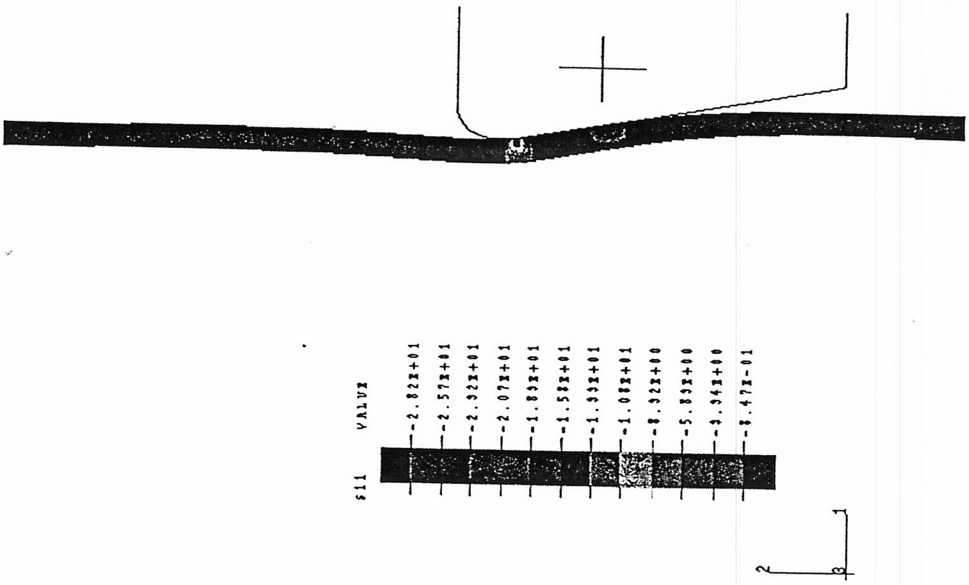
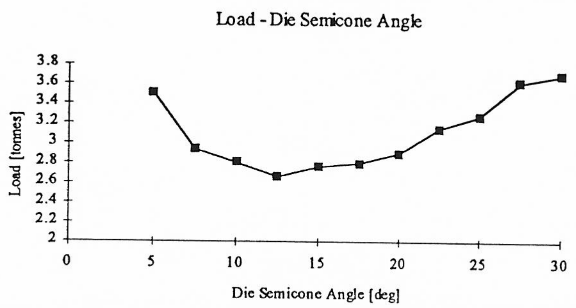
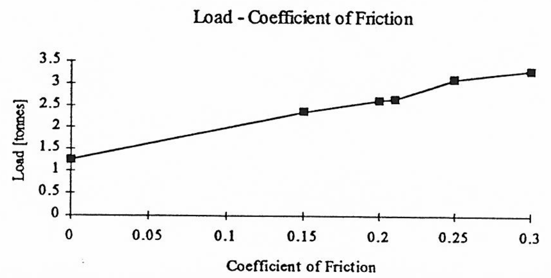


Figure 6. **Relationship between the die semi-angle and the drawing load.**



These results are in good agreement with experimental test data. Figure 7 shows the predicted relationship between the friction factor and the drawing load.

Figure 7, Predicted drawing force:coefficient of friction.



The predicted stress contours (not shown here) indicate the generation of high strains at the inside of the pipe during the deformation process which could affect the subsequent material behaviour of the pipe. Further investigations into the die profile and the heat generated by friction are currently being made.

3. Finite element analysis of elastomers

3.1 Background

Elastomeric components are incorporated in a wide range of products such as driveline couplings for vibration damping and isolation. The design of these components is critical to the satisfactory performance of the product, yet empirical formulae and "rules of thumb" are still utilised for design purposes, and product optimisation has traditionally been carried out by building and testing prototypes. Finite Element Analysis of elastomers is not widely used because of the problems encountered in creating accurate material models. Elastomers, unlike metals and most common engineering materials, do not behave in a linear, isotropic fashion and so require more complex constitutive equations.

3.2 Analysis Models

Mathematical models describing the stress-strain relationship for elastomeric materials follow one of two approaches. They can either use a phenomenological invariant based model (such as the Mooney-Rivlin and Ogden models implemented in ABAQUS) or a statistical mechanics approach. Phenomenological models consider the bulk properties of the material whereas statistical mechanics models look at the molecular level and attempt to model the internal behaviour of the elastomer. The usual form of expression used is an elastic strain energy relationship in terms of the principal stretches. The Mooney-Rivlin model for example has the relationship:

$$W=C_1 (\lambda_1^2 + \lambda_2^2 + \lambda_3^2 - 3) + C_2 (1/\lambda_1^2 + 1/\lambda_2^2 + 1/\lambda_3^2 - 3) \qquad \text{(eqn. 1)}$$

but most models cannot cover different states of deformation fully. For example, a model which captures the behaviour of a material in tension usually requires modification before it can be used for compressive analysis. For mixed forms of loading, such a model can at best be only a compromise.

The work briefly described here is mainly concerned with developing constitutive equations for specific elastomers used in driveline couplings. Because of this, relatively simple geometries are used with the emphasis on the material behaviour. The material properties are input to ABAQUS via the *UHYPER subroutine; the models can be adapted and changed easily to "fine tune" their characteristics. The parameters necessary to define the material model are derived from tensile testing, and good correlation between simulations and actual tests in a variety of loading configurations using the tensile input data have been obtained.

3.3 Modelling and experimental correlation

Finite element models of two standard ASTM test specimens have been prepared for material model development and correlation purposes. Tensile test comparison was made using the gauge length of an ASTM "C" dumb-bell test piece using a 3-dimensional finite element. model consisting of 450 C3D8H elements. The large number of elements was to allow the high extensions to be modelled without causing excessive distortion in the elements. Compression test comparison was made using an ASTM standard compression button, because of the much simpler geometry, this could be modelled by as few as 4 CAX8H elements without loss of accuracy. Loading in both cases was uniaxial, modelled by fixing one end of the specimen in the load direction and applying a load or displacement to the other end. All the nodes at each end were constrained to give equal displacement.

A more detailed model of an actual rubber component, a coupling block, was also generated and used. This involved compressive loading between two curved contact planes which was found to cause major instability problems. To overcome these problems, exact initial contact between the rubber block and each contact plane had to be defined: the enhanced facilities available in ABAQUS V5.2 should obviate this problem.

3.4 Results

The results obtained for a typical material, a highly filled rubber, under tension and compression are shown in Figures 8 and 9. These indicate very good agreement between predicted and measured values for exactly the same material model in tension and compression. The analysis is being continued to validate the material model in actual component design.

Figure 8, **Comparison of predicted and measured tensile stress/strain behaviour.**

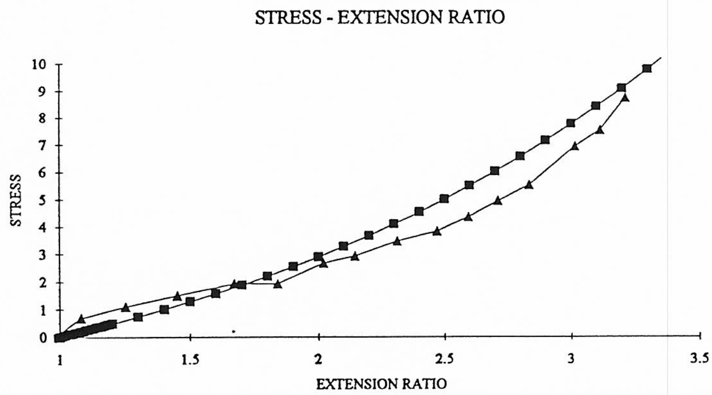
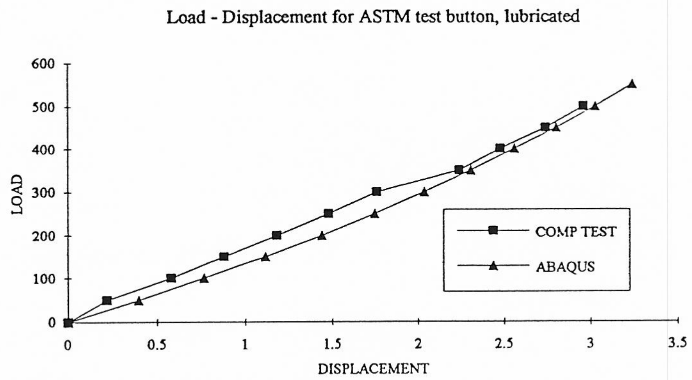


Figure 9, **Comparison of predicted and measured compressive load/displacement behaviour.**



4. High strain behaviour of Polypropylene

4.1 Process description

The solid phase deformation of Polyolefins is of considerable scientific and practical interest because significant enhancement of the strength properties of oriented polymers can be usefully achieved. For example, Polypropylene sheet is used in the manufacture of high strength grids for ground reinforcement and other purposes by the bi-axial stretching of polypropylene sheet into which a matrix of holes have been introduced. As the sheet is stretched, the polymer material around the holes stretches and orientates to generate a high strength grid. This process has been studied extensively (Coates 1987) and the strength and processing conditions are known to be affected by the initial geometric form of the individual holes in the matrix. The process has therefore been modelled using finite element analysis to allow these effects to be quantified and the design of the original sheet to be optimised.

4.2 Material behaviour

When subjected to uniaxial tensile force, Polyolefins respond with a small recoverable (elastic) deformation which is quickly overtaken by an unrecoverable (plastic) deformation as the polymer chains start to orientate or straighten out, as illustrated in curve C of Figure 1 (Ward 1990). As this orientation occurs there is usually a boundary between the region of high orientation and the adjoining unoriented region which is called the "neck". (This differs from the neck in metal deformation which occurs as an immediate precursor to fracture.) Modelling this material behaviour directly is not possible because the neck is essentially a localised discontinuity, and a stepwise approach has been adopted for finite element analysis using local material properties generated from extensive experimental studies of test specimens using image analysis procedures to capture the true local stress/strain behaviour. This analysis essentially therefore used the tangent modulus, assumed linear over the strain increment, based on detailed stress/strain experimental data.

4.3 Analysis procedures

A 2-D finite element model of a uniaxial tensile test specimen was prepared as shown in the undeformed mesh of Figure 10. This was a relatively simple plane stress model with 68 elements sufficient to provide geometric accuracy over the full extension process without requiring re-meshing. Appropriate boundary conditions including XSYMM and YSYMM were applied and a uniaxial tensile load was applied as incremental displacement steps. At the end of each step the stress, strain, strain rate and temperature rise in the polymer were used to determine the correct properties for the next increment.

4.4 Results

Examples of predicted deformed shapes are shown in Figure 10. These illustrate the formation of a neck, or localised deformation of the cross section, which is in general agreement with actual observations in the experimental testing of polymers as illustrated in Figure 11. Further finite element analyses are currently being conducted in which temperature and strain rate effects are included, thus allowing the neck propagation to be modelled. Predicted deformations are being correlated against observed test results using image analysis, and heat generation and temperature rise effects are being correlated against Infra-Red temperature measurements.

Figure 10, FE mesh and deformed shapes of sheet test specimen.

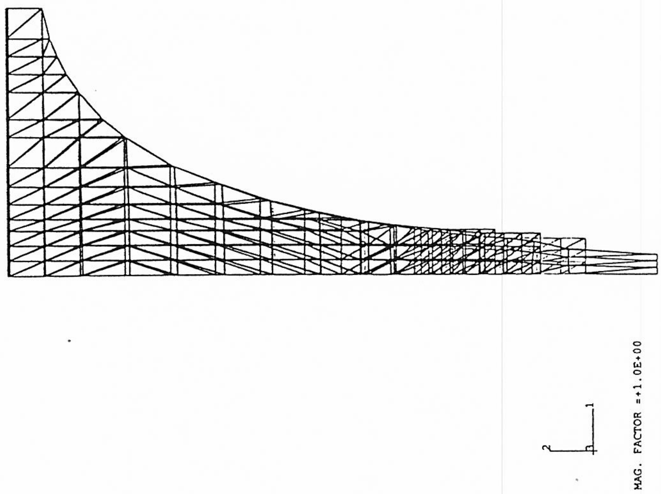
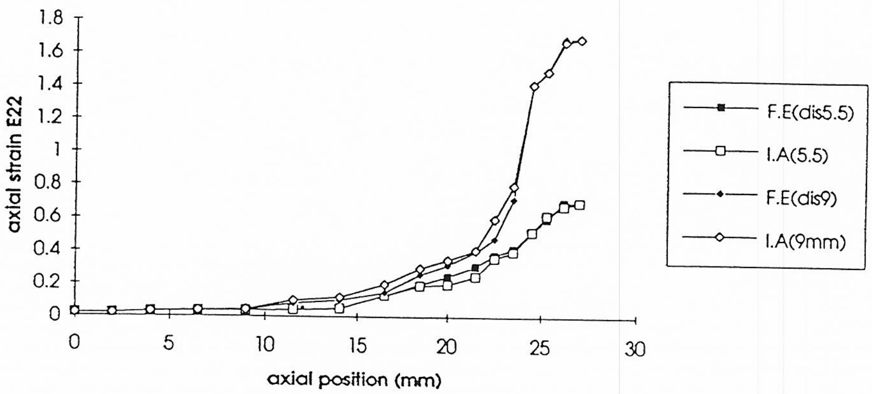


Figure 11, Comparison of predicted and measured strains for sheet test specimen.



5 Conclusions

5.1 Polymer material modelling

Non-linear material behaviour of polymers can be modelled for the purposes of finite element analysis using ABAQUS in three distinct ways:

(a) The loading can be applied incrementally as a stepwise linear model in which the tangent modulus is used to define the relationship between stress and strain. Considerable experimental data are required to define the material properties to include the effects of time, temperature, strain rate, etc., and tensile testing with image analysis has provided such data for finite element. analysis. Quantitative agreement of analytical with experimental results has been achieved for local deformation characteristics.

(b) The stress/strain characteristic can be approximated as an elastic/plastic material which models visco-elastic behaviour as permanent deformation. In this case it is necessary to establish a material stress/strain characteristic as close to the conditions of the deformation process as possible.

(c) The relationship between stress and strain can be expressed mathematically as a "Hyper-elastic" material, in which large strain behaviour is correctly computed from an appropriate material model.

All three of these approaches have been used successfully, as outlined in this paper.

5.2 The ABAQUS program

The ABAQUS program is an extremely useful research tool; in particular the facility for user-defined constitutive relationships is invaluable.

5.3 Elastomer material modelling

For filled elastomer compounds, the statistical mechanics approach to material modelling is better than either the Mooney-Rivlin or the Ogden model.

5.4 Modelling complexity

The combination of non-linear material behaviour, and complex boundary conditions such as interfaces, gaps and friction, makes substantial demands on computer power and availability.

Acknowledgements

The work presented in this paper is part of the Polymer Engineering research activity at the University of Bradford and the Interdisciplinary Research Centre for Polymer Science and Technology at the Universities of Leeds, Bradford and Durham, UK. The Authors are grateful to the Science and Engineering Research Council and the Royal Commission for the Exhibition of 1851 for their support, and to the Industrial Companies who are also supporting the research. Thanks are also due to all colleagues, particularly Mr. N Thirugnanasothy of the University of Bradford Computer Centre, for their invaluable help and discussions.

References

Arruda, EM, Boyce, MC, "A three-dimensional constitutive model for the large stretch behaviour of rubber elastic materials", pre-print of paper accepted for publication in Journal of the Mechanics and Physics of solids, 1992.

Coates, PD, Ellis, DI, Pourmahnaei, SM, "Drawing thick section polypropylene at high strain rates" Plastic and rubber processing and applications, Vol. 8, pp 165 - 172, 1987.

Ward, IM, "Mechanical properties of solid polymers", 2nd Edition, John Wiley & Sons Ltd., 1990.