DEVELOPMENT OF A PLUG ASSISTED THERMOFROMING SIMULATION.

R.McCool¹, P.J.Martin² and Eileen Harkin Jones³

¹rmccool01@qub.ac.uk; ² p.j.martin@qub.ac.uk; ³ e.harkinjones@qub.ac.uk
Queen’s University of Belfast, School of Mechanical Engineering, Ashby Building, Stranmillis Road, Belfast, Northern Ireland, BT9 5AH

Abstract - Thermoforming is a very important process in the manufacture of thin gauge food packaging products. Industry methods currently rely on trial and error, however there is a growing need to develop robust finite element simulations of the process. This paper describes the development of a simulation of the plug assisted thermoforming of high impact polystyrene (HIPS) products, using the commercially available finite element software ABAQUS. The simulation takes into account the key elements associated with the process which include the deformation response of the material, and the friction and heat transfer between contacting surfaces. The evolution of the simulation from an isothermal condition with friction between contacting surfaces, to a non-isothermal simulation with friction and heat transfer effects is presented. The output of the simulation is detailed in the paper and its performance is correlated with experimental observations.

Introduction
Thermoforming is widely employed in the manufacture of thin gauge packaging for the food industry. Essentially in the thermoforming process a pre-extruded thermoplastic sheet is clamped, heated to a desired softening temperature and shaped into a cool mould by a combination of mechanical movement and air pressure. The desired output from thermoforming is to have a controlled product wall thickness distribution, as in many products excessive material thickness in certain regions is common, and in addition companies rely on trial and error methods in achieving an acceptable product. However the possibilities to simulate the process with the aid of computer software have been investigated in recent years, and the most common approach is to use the finite element method, which is also the chosen method in this work. A major stumbling block in the development of such simulations is the need for researchers to understand the key elements of the thermoforming process which are the high strain/strain rate material deformation and the tool sheet contact. And a successful process simulation must include these aspects through an accurate material deformation model, a temperature dependant friction model and a model which is capable of predicting the heat transfer effects in the process.

Plug assisted thermoforming and key process elements
In the plug assisted thermoforming process there are two distinct stages in which the movement of a mechanical plug is followed by the application of positive air pressure. A process diagram is shown in Figure 1. Initially the heated sheet is clamped followed by the advancement of the plug into the mould thus pre-stretching the sheet to form a preform. Air is applied to the top surface of the sheet at the point when the plug is fully advanced, the sheet is then forced to take up the shape of the mould and the forming is complete.

![Figure 1 – Plug Assisted Thermoforming Process.](image)

The interaction between the plug, sheet and mould material strongly affects the final part wall thickness distribution. These are known as the three key elements, the first key element is the mechanical stress/strain response of the material and its ability to stretch through different modes of biaxial stretching. During the mechanical pre-stretching of the sheet by the movement of the plug, the slip along the surface of the plug is related to the coefficients of friction between
the contacting surfaces. However sheet slip is only achievable through extension of the sheet, therefore this effect is highly dependant on the ability of the sheet to stretch. This ability for the sheet to slip is the second key element. Finally the ability for heat to flow from the sheet affects its local temperature and consequently its ability to stretch by either free or sliding extension, therefore the third key element is the heat transfer effects. To develop a simulation of the plug assisted thermofoming process all the key elements associated with the process must be included, also accurate replication of the geometry and process conditions.

**Experimental**

Experimental thermoforming tests were carried out on a purpose built laboratory thermoforming machine using tooling and forming parameters that were designed to mimic industrial conditions. The product shape was a typical conical pot, as shown in Figure 2. The tooling consisted of an aluminium alloy mould and a closely matched flat bottomed plug made from syntactic foam. All experimental work was carried out on 1.31 mm thick extruded HIPS (Nova 643N) sheet. In this study the commercial Finite Element package ABAQUS was used to create the thermoforming model. ABAQUS has been widely employed in other polymer process simulations and it has been successful in modeling stretch blow moulding.

Initially an isothermal simulation of the process was developed and the capabilities of the model were increased by systematically adding the effects of heat transfer through conduction and convection.

**Modeling of the Geometry**

The single shot experimental thermoformer at Queen’s University of Belfast was used to produce test pots to allow comparison with simulations. The tooling used has the major dimensions as outlined in Figure 2, and produces a pot of major dimensions 75mm base diameter, 100mm lip diameter and 90mm in height from an initial sheet thickness of 1.31mm. Taking advantage of the tooling symmetry resulted in the ability to model the tooling axi-symmetrically within ABAQUS. Subsequent wall thickness measurements were recorded from a point on the centre of the pot base (0mm) to the outer point on the pot lip (133mm).

**Figure 2 – Thermoforming Tooling.**

**Modeling of the Material**

Accurate material models are a fundamental part of any FE simulation and construction of such models requires data obtained from material tests carried out under conditions comparable to actual processing conditions. A purpose built biaxial test rig has been developed by Queen’s University Belfast (QUB) and work has focused on accurate measurement of the deformation behaviour of polymer materials. Research carried out by Tshai, into the biaxial stretching of HIPS has led to the development of a HIPS deformation model. It is based on a Van der Waals model combined with a Prony series relaxation function, and uses the Williams, Landel and Ferry, WLF ref, equation to introduce temperature dependency. Killian and Vilgis developed the Van der Waals strain energy function. For further details of the material model readers are referred to reference. The models performance in simulating the biaxial deformation response of the HIPS material used in this study was in very good agreement over a range of strain rates.

**Modeling of Contact Friction**

Research at QUB has employed a sled test mechanism to identify friction coefficients. In these experiments samples of plug material were pulled across the surface of sheet materials at varying contact conditions. Frictional shear stresses may develop at a contact interface if the two contacting bodies have rough surfaces. If the frictional shear
stress reaches a critical value, the bodies will slide. For the purpose of this work a temperature dependant friction model measured at QUB\(^3\) is used for the combination of plug and sheet material used in this study. Within ABAQUS isotropic Coulomb friction using the penalty method was selected in this work. In the initial isothermal simulation single values of friction were tested, as the simulation incorporated heat transfer the temperature dependant friction model was implemented and its performance was evaluated.

**Heat Transfer**

To accurately model the thermal effects in the plug assisted thermoforming process, all components in the system, i.e. the plug, the sheet and the mould are assumed to be in a non-isothermal condition. Heat transfer by conduction depends on the conductivity of the material within an interface as well as the temperatures of the respective surfaces, and is modelled in ABAQUS by *Gap Conductance*. Investigations into the effects of heat transfer were conducted using thermal imaging equipment and the experimental thermoformer which was setup to allow plug only tests and subsequent measurement of sheet temperature during plugging, similar tests were carried out at QUB\(^3\). Further details of the approach used in modeling of heat transfer and friction, are detailed in paper P02-006: *MODELING OF CONTACT IN THERMOFORMING*.

**Results and Discussion**

Figure 3 shows the experimental wall thickness distribution as measured from a cup produced at the base settings (Sheet Temperature 120ºC, Plug Temperature 100ºC). Also shown in figure 3 is the output from an isothermal simulation showing how the wall thickness distribution varies by changing the coefficient of friction between the plug and the sheet and for simplicity the contact between the sheet and the mould was assumed “rough”. When a high value of coefficient of friction was chosen between the plug and the sheet more material resulted in the base of the cup (0 to 35mm) where as the material in the lip region (120 to 133mm) was reduced. A major area of discontinuity is in the side wall of the cup from 50 to 90mm where the simulation under predicts the thickness, this may be due to the contact between the sheet and the plug, or material behaviour, and further investigations are being carried out to establish the cause of this.

![Graph showing wall thickness distribution](thick_dist.png)

**Figure 3** – Effect of Plug Sheet Friction on Wall Thickness under Isothermal Conditions.

Figure 4 shows the results of increasing the sheet temperature up to 140ºC on the wall thickness distribution. In this case the fully coupled temperature dependant friction model is incorporated and heat flow is permitted between contacting surfaces by using a fixed arbitrary gap conductance value of 500. The results show that as the sheet temperature is increased the amount of material in the base region increases, this maybe due to the fact that at higher temperatures the friction coefficients also increase.
Conclusion
A finite element simulation of the plug assisted thermoforming process was constructed using ABAQUS, the model was initially constructed isothermally and was subsequently improved to incorporate the effects of heat transfer. It was shown that even under isothermal conditions the model was capable of predicting relatively accurately the profile of the final wall thickness distribution as achieved experimentally by systematically changing the level of friction between the sheet and the plug. By adding a temperature dependant friction model and the effects of heat transfer the capabilities of the simulation were improved. It is important to stress however that even though the simulation incorporates all the key elements associated with the process, it does not automatically result in accurate simulation. A balance has to be struck between all key elements so that they perform in an inter-related manner, for example, if the sheet temperature cools during forming this has a bearing on the contact friction and the ability of heat to flow from the sheet, and it is these complex interactions that must be accurately modeled.

Acknowledgements
The authors wish to thank Wilsanco Plastics Ltd for their support of this work.

References
6. NewMet www.newmet.co.uk/.