Simulation of the Molten Glass Sheets Flow

Paul Baias[1], Eva Pelster [1], Dr. David Wenger[1]
[1] Wenger Engineering GmbH, Einsteinstr. 55, 89077 Ulm, Germany, mail@wenger-engineering.com

Abstract: Thin glass sheets have nowadays a lot of commercial applications. The optimization of the production process is of extreme importance to ensure a high quality product. The optional functionality of such product is often linked to a surface homogenous in thickness. This uniform distribution can oftentimes be controlled by the forming process and depends on the flow of the molten glass as well as the geometry of the production device.
To gain more knowledge of the influence of differ the forming geometry a parametric geometry was built in Comsol Multiphysics to simulate the molten glass flow for different production setups.

Keywords: flow, fluid dynamics, glass, thin sheets, two phase flow

1. Introduction

Thin glass sheets play an important role in everyday life, for example television screens or mobile phones. To gain further knowledge on the production process of thin glass sheets a numerical model to simulate the flow of molten glass is generated in this study. The modeled part of the process simulates the end of the production process where the molten glass flows down the walls of a trough. After the trough reservoir is filled, the molten glass overflows and the cascading sheets meet at the underside of the trough forming a single sheet. The sheet continues to fall under gravity or an external applied force and form the final glass sheet, as seen in Fig. 1.

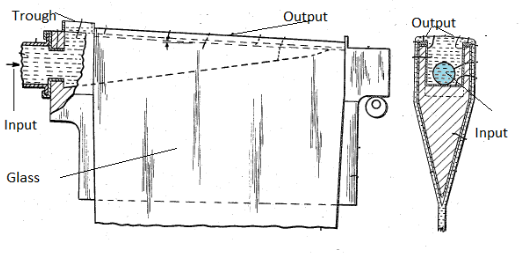

Figure 1. Geometry of the glass forming device after Stuart M. Dockerty& Corning [1]

This glass production process is called the fusion process and was developed by Corning; this process has the advantage that the outer surfaces of the molten glass do not come in contact with other materials other than the surrounding air while it is being formed [1]. This procedure ensures a glass surface optimal in purity.

As we can see the whole production process relies on the system design generating the correct fluid dynamics and this is well defined by Comsol Multiphysics. Modeling with Comsol Multiphysics offers the advantage that studies can be carried out before first prototypes are available, reducing the costs of testing. The goal is to model a design that provides us the best control of the production process. The study will help us to understand and achieve a desired outcome consisting in a sheet of desired thickness and uniform cross-section, free of impurities.

2. COMSOL simulation

First a parameterized geometry was built. Aim was to simulate a glass flow through the inlet with a defined mass flow rate. The overflow rate is studied at the output surface. For a uniform glass sheet we expect at the output a uniform specific mass flux over the whole length of the output surface.

The flow is simulated as a free surface flow; we proceed by using the laminar two-phase-flow (glass/air). Comsol Multiphysics has two methods, the Level-Set-Method or the Phase-Field-Method.

In this study the Level-Set-Method was used. At the input we defined a mass-flow, the walls condition we are choosing for the simulation are the no slip walls condition and finally the output boundary we defined by pressure 0 Pa.

The material properties are also defined, for air the data from the Comsol Multiphysics material library was used. For this study isothermal conditions were assumed, so the relevant material data for glass are defined by T = 1273.15 K.

A time dependent solver was used to simulate the glass flow.

3. Calculation

3.1 The modelled geometry

The modelled geometry consists of the fluid domains only, in this case the molten glass as well as the surrounding air. The air region was expanded, in order to ensure, that the modelled domain was big enough, see Fig. 2. To be able to change the geometry quickly, the entire setup was parameterized. This way the model can be of further use, i.e. when trying to find an optimal shape for desired output behaviour. In our study the variable is the inclination angle of the trough reservoir in order to use the effects of the gravity in optimizing the output flow rate.

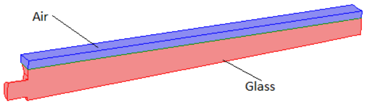

Figure 2. Geometry of the modeled fluids.

The chosen parameterized dimension may have the highest impact on the flow rate at the output boundary. We will select for the inclinations of 0° and 7° (see Fig.3) and simulate the results in order to study the influence on the output flow rate.

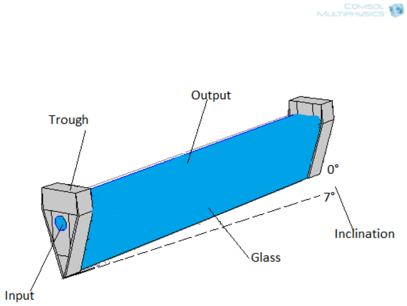

Figure 3. Simplified Geometry for calculation with inclination

3.2 The modelled equations

For the simulation of the fluid dynamics the Laminar Two Phase Flow- Level Set mode was chosen.

The Level Set interface uses the incompressible formulation of the Navier-Stokes equations:

$$\rho\frac{\partial \mathbf{u}}{\partial t}+\rho(\mathbf{u}\cdot\nabla)\mathbf{u} = \nabla\cdot[-p\mathbf{I}+\mu(\nabla\mathbf{u}+\nabla\mathbf{u}^T)]+\mathbf{F}_g+\mathbf{F}_{st}+\mathbf{F}_{ext}+\mathbf{F} \quad 3.2.1$$

$$\nabla\cdot\mathbf{u} = 0 \quad 3.2.2$$

If the level set method is used to track the interface, it adds the following equation:

$$\frac{\partial\phi}{\partial t}+\mathbf{u}\cdot\nabla\phi = \gamma\nabla\cdot\left(\varepsilon\nabla\phi-\phi(1-\phi)\frac{\nabla\phi}{|\nabla\phi|}\right) \quad 3.2.3$$

The density is a function of the level set function according to

$$\rho = \rho_1+(\rho_2-\rho_1)\phi \quad 3.2.4$$

and the dynamic viscosity is

$$\mu = \mu_1+(\mu_2-\mu_1)\phi \quad 3.2.5$$

where ρ_1 and ρ_2 are the densities of Fluid 1 and Fluid 2, respectively, and μ_1 or η_1 and μ_2 or η_2 are the dynamic viscosities of Fluid 1 and Fluid 2. In our calculation Fluid 1 is air and Fluid 2 is glass see Fig.3.

At the input we define a mass flow rate from 0.05 kg/s see Fig.4.

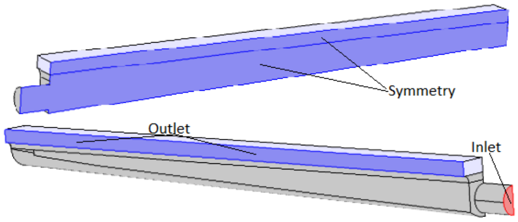

Figure 4. Input and output of molten glass.

The dynamic viscosity for air and glass and the density both dependent from the temperature were taken for both materials at T=1273.15 K.

The wall boundary condition which is suitable for walls in contact with the fluid-fluid interface is the wetted wall condition. If this boundary condition is used, the fluid-fluid interface can move along the wall. This boundary condition enforces the slip condition.

For the calculations we have set $\beta = h/100$, where h is the mesh element size and β the slip length. This boundary condition adds a weak boundary term which is a result of the partial integration of the surface tension force in the Navier-Stokes equations. Additionally the contact angle θ_w has to be defined as well. This represents the angle between the fluid interface and the wall. For this calculation $\theta_w = \pi/8$ was chosen.

On the y direction the gravitational force is present, to incorporate the effects of gravity a volume force was introduces. This adds a term on the right-hand side of the incompressible flow equation Eq. 3.2.1.

3.3 The COMSOL model- solver

The time dependent solver was used; this gives us the possibility to study the process at each time step. So we have the advantage to follow the flow behaviour step by step and to have the best control over the process.

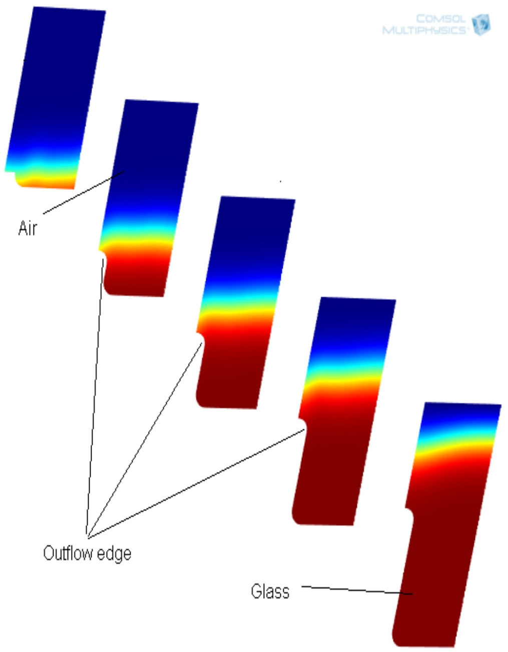

Figure 6. Outflow edge and boundary at 1800 s

3.4 The simulated results

In Fig. 7 the results for a 0° inclination are represented as 3 D plots of the mass flow rate over the outflow boundary at different time steps. The colour range distribution of the plot represents the uniformity of the glass distribution.

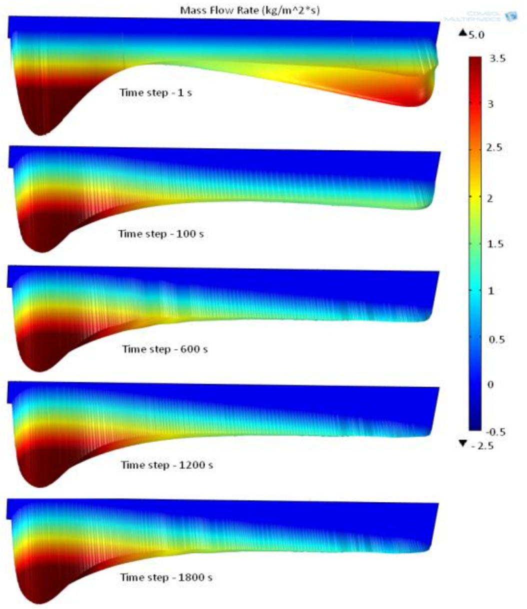

Figure 7. Flow Profile at the outflow boundary for 0°

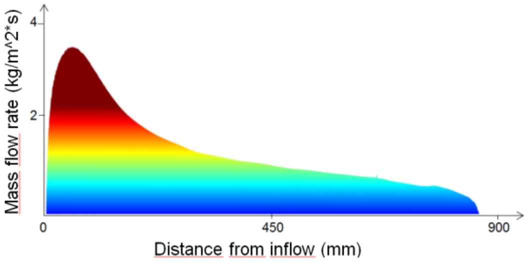

Figure 8. Mass flow rate at the outflow boundary for 0° after 1800s.

Because the interface boundary between the two fluids at the distal end from the inlet was set over the outflow edge there is a higher mass flow at the beginning until the normal level is reached. We can observe that the distribution of the mass flow is not uniform. The flow profile remains relatively constant after 600 s.

Fig. 6 shows the glass profile at t=1800s for cross sectional areas along the length of the trough.

Fig. 9 shows a comparison of the resulting contours of the flow at t=1800s. Here the influence of the 7° inclination is clearly visible in the shape of the contour. It changes towards a more homogenous contour which is commonly better for the production process.

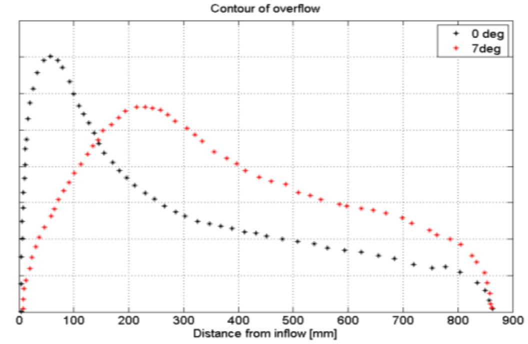

Figure 9. Contour of outflow shape at outlet for t = 1800s.

4. Conclusions

In this study a method for calculating the flow of molten glass with Comsol Multiphysics was presented. Therefore a parameterized geometry was built in order to model part of a glass sheet forming process. It was shown that the model is able to simulate the overflow and vary the input to be able to optimize the geometry and therefore the process. Additional tests should be carried with the model to gather more knowledge of the influence of different parameters.

5. References

1. S.M. Dockerty. *Sheet Forming Apparatus*. U.S. Patent 3,338,696, 1967.