COMPUTER AIDED OUTPUT IMPROVEMENT OF A HIGH CAPACITY BLOWN FILM EXTRUSION LINE

Benedikt Neubert and Johannes Wortberg, University of Duisburg-Essen, Institute of Product Engineering, Engineering Design and Plastics Machinery, Germany

Abstract

For improving the output of high capacity blown film extrusion lines usually, the limiting factor, namely the air-cooling ring, is substituted or modified. Therefore, the production process has to be interrupted which is time and cost intensive. Primarily the major disadvantage of this experimental strategy is the uncertainty about the outcome. In detail, not all the thermodynamic and fluidic phenomena caused by the changing cooling configuration, and their impact on the formation of the bubble, are predictable in advance.

To overcome these problems and to understand all the effects, which take place inside the bubble formation zone a numerical procedure has been developed and validated in previous works [1, 2, 3]. The so-called Process Model is capable of simulating the formation of the bubble with regard to changing cooling configurations and rheological behavior. According to industrial concerns, the modeling procedure was adapted to fulfill the requirements for simulating a high capacity blown film process [4]. In this paper, the first results for the numerical optimization of an industrial high capacity blown film process, using the adapted Process model, will be presented. Furthermore, a developed evaluation strategy for the CFD-results will be used to point out the positive effects of the modified cooling configuration.

Based on the simulation results, the experimental validation will prove the applicability of the computer-assisted designing and optimizing strategy. For this purpose, the best virtual outcome will be manufactured and transferred to the current high capacity blown film line. It will be shown that output improvements of approximately 10 % are achievable without neglecting the quality of the final film product.

Introduction

In the production environment, the cooling of the produced film inside the bubble formation zone is typically done by applying air supplied cooling rings. Therefore, conventional single or dual lip geometries are being used to reduce the temperature of the film. One or more air-jets emerge from the external air-ring and flow tangentially on top of the outer film surface. Additionally, within the industrial environment, an internal cooling device supports the outer cooling of the extrudate. This state-of-the-art technology doubles the effective surface for the heat transfer and simultaneous stabilizes the bubble due to the higher internal pressure load. As a result, the external air supply can be increased [5, 6]. In terms of a production rate improvement the internal bubble cooling can have a major impact. Depending on the system configuration (die size and film width), the improvement can vary between 20 % for small dies with diameter between 200 and 250 mm and 100 % for dies larger than 900 mm [5].

Usually conventional cooling systems are being developed, optimized and tested using the experimental trial and error method. Especially for production lines, this is a time and cost intensive procedure. To reduce these costs, a numerical approach can help to accelerate the development and optimization process of such cooling devices.

For this Bussmann et al. [1, 7] developed the first coupled integrative model, called Process Model, to simulate the blown film process with respect to changing process conditions. Deviating from previous simulation models it is possible to compute the film behavior not only for a stationary but also for changing cooling configurations. For this purpose, the Process Model consists out of two submodules. The first submodule calculates the film contour based on the framework from Pearson and Petrie [8, 9]. To obtain a realistic rheological film contour prediction, a modified rheological material description by Phan-Thien and Tanner [10, 11] was implemented. Inside the second submodule (2D-CFD-Simulation [2]), the temperature profile of the film, the pressure profiles on the inner and outer bubble surface, and the streaming velocities of the cooling air are computed. Additionally Bussmann et al. stated that the capability in estimating the exact shape of the bubble is strongly depending on the temperature dependence of the material modeling. In order to find a final quasi-stationary film contour, an iterative loop between the above-mentioned submodules is integrated [3, 5, 6]. Figure 1 shows the structure of the Process Model. Besides the basic simulation, respectively calculation of the blown film behavior, the Process Model can be used for the optimization of existing flow-situations and for the development of novel cooling systems [3, 12, 13, 14].
In this paper, the successfully for laboratory and high capacity blown film extrusion lines verified and validated Process Modell [1, 3, 4], will be applied for improving the output of an industrial blown film line. A comparison between the flow situations of a reference process and the virtually optimized process will demonstrate the positive effect on the heat transfer caused by the geometrical changes of the cooling system.

**Simulation of an Industrial High Capacity Blown Film Process**

For simulating a high capacity blown film extrusion line, it is important to take a closer look at the industrial process conditions and the adaptations of the simulation domain. The following table gives an overview of the proportions that were used for the numerical and afterwards for the experimental investigations.

**Reference Process Conditions**

<table>
<thead>
<tr>
<th>Blow up ratio (BUR)</th>
<th>Mass Flow Rate [kg/h]</th>
<th>Cooling Air Volume Flow [m³/h]</th>
<th>Haul-Off Speed [mm/min]</th>
</tr>
</thead>
</table>
|                     |                       | a) outer 4,000 – 6,000  
b) inner 1,000 – 2,000 | > 35                   |

The considered blow up ratios differ between 1.5 and 3.5. Compared to previous investigations [1, 3] the mass flow rates and the haul-off-speed increase by the factors 10 and 4. Taking the used spiral mandrel system (> 500 mm) into account, the output per die circumference is in the range between 1.0 and 1.5 kg/mm. Furthermore, an adaption of the simulation domain (CFD-Simulation) has to be considered (Figure 2). As a result, the supplementing of an inner domain (Domain 2; red parts) ensures the correct calculation of the differential pressure due to the determination of the inner and outer height-dependent pressure load [11]. By the use of the membrane theory, it is now possible to determine the dynamic formation of the contour inside the bubble formation zone. Changing cooling configurations, like the use of internal bubble cooling devices (IBC) or additional implemented air guiding elements, are being examined.

As the final product, a coextruded 3-layer film is being produced. The thickness distribution is assumed to be homogeneous. It consists out of different types of polyethylene and a masterbatch.

**Required Input Data**

For generating a high accuracy of the modeling and simulation procedure, it requires various input data. The biaxial stretching of the film and the resulting rheological stresses have to be represented in a physically correct form. That implies the measurement of the material-specific, height-dependent film velocity profile inside the bubble formation zone. Figure 3 shows the recorded average velocity data (Laser-Surface-Velocimeter; InteliSENS SL mini 3600 Co. Proton Products; sampling rate: 1 Hz; 3 minutes) including the minimum and maximum values for each measuring point. The corresponding height is normalized to the resulting frost line height and the velocity to the haul-off-speed. An internal quality monitoring system allows a direct valuation. Thereby, the quality value (QV) should not be lower than 60 percent to guarantee good quality measurements.
To ensure a numerical stable contour calculation the measured velocity data has to be approximated. A recording in the normalized height between zero and 0.39 can not be performed due to geometrical limitations. The velocity of the film at the die exit can generally be calculated by the consideration of the ratio between the output and the geometrically fixed die gap area with regard to the temperature dependent density of the polymer. The fluctuations of the velocity can be explained by the measurement regulations and the movement of the bubble surface.

Further, the Process Model requires material-characteristic caloric input data. Beside the temperature-dependent thermal conductivity, the density and the heat capacity were measured. Compared to monolayer films, the layer composition, the film structure, the thickness distribution and synergetic effects between the layers have a major influence on the final film properties. Nevertheless, a consideration of these effects on a modeling basis is rather difficult to implement. Hence, the components were mixed up and measured with respect to their over-all percental weightage inside the coextruded film.

Additionally, the material-characteristic activation energy $E_0$ was determined by performing rheological examinations. Thereby, $E_0$ is an input parameter for the calculation of the temperature-shift factor $\alpha_T$, which is for its part an input parameter for the Carreau model [15]. As a result, the temperature-dependent behavior of the viscosity and the relaxation time can be modeled in a good approximation.

**Virtual Optimization**

For the virtual optimization, the reference process is defined as the initial state. In terms of an optimization, the simulated normalized reference frost line height of 1 has to be lowered, without changing the reference process parameters. Excluding the installed conventional single-lip cooling ring, the flow situation will be improved by the adjustment of the passively acting cooling configuration. In a first step, the heat exchange between the cooling air and the film should be intensified. As the final evaluation criterion, the optimization should result in an output improvement while keeping the reference frostline height constant. Additionally, the improvement must not influence the stability and the key quality factors formalized for the final film product. This includes, but is not limited to, the layflat-width and the measured inline and offline thickness distribution of the film.

**Influencing factors**

The focus will be on stalling the interactions between the fast streaming cooling air and the ambient air. Predöhl [16] was able to prove that the so-called entrainment-effect causes a reduction of the maximum and the average cooling air streaming velocity, which in turn has a negative impact on the heat removal out of the film. According to this, a modification of the cooling configuration should result in an increase of the streaming velocity in the machine direction. Further, a stabilizing effect of the bubble might be achieved by generating the Venturi-effect, which sucks the bubble towards the installed geometries.

**Evaluation of CFD-Results**

The primary goal of the developed evaluation tool (Figure 4) is the quantitative comparison of two simulated and quasi-stationary CFD-results out of the Process model. A graphical user interface allows the interpretation of the fluidic phenomena that occur due to changing cooling configurations. Furthermore, the tool provides the opportunity for identifying even further potentials for improving the used cooling configuration.

![Image of flowchart](image-url)

Figure 4. Structure of the evaluation strategy.

Therefore, the tool is partitioned into two modules. The first module contains the processing of the calculated film contours $r(x)$ and the thickness-profiles $h(x)$ inside MATLAB. It is now possible either to get all the necessary thermodynamic or fluidic information in radial or axial direction inside the bubble formation zone. All the required data is provided by the CFD-analysis (case & data files). For every iteration a polyline can be generated which is equal to a transformation of the selected contour in radial direction. On these polylines all the important data is accessible.

In addition to the height-dependent profiles of the temperature, the external and internal pressure load and the heat transfer coefficients, the streaming velocity can be computed in specific distances to the bubble surface. As mentioned above, the goal lies in modifying the oncoming flow conditions so that the average and...
maximum streaming velocity increases. For this purpose, one has to take a closer look at the boundary interface between the laminar and the turbulent flow and the resulting velocity profiles. As a function of the radius of the curvature, the streaming velocity profiles can be visualized. Figure 5 shows the outcome of the evaluation strategy for the reference process state. It can be seen that the profiles change in dependence on the height above the melt outlet. At a normalized height of 0.2, the velocity reaches a value of about 36.6 m/s and the profile is relatively sharp. Continuing upwards the interactions between the cooling and the ambient air increase. This results in a widening of the velocity profiles in the normalized distance to the bubble surface. At a normalized height of 1.2, the maximum velocity decreased to 9.8 m/s due to the entrainment effect. Inside the red crosshatched area, an air-guiding element is installed.

As stated above, the tool provides the opportunity for identifying further potentials for the enhancement of the cooling configuration. A specific observation of the suction effect of the surrounding ambient air, caused by the installed passively acting air-guiding elements, is now possible. For that, inside the graphical user interface, the operator can select specific geometrical control points of the cooling configuration. Between these points, which are a part of the real geometrical setting, the volume flow of the aspirated ambient air can be computed. Assuming that the system is rotationally symmetric and all the fluidic phenomena take place over the entire circumference, the aspirated volume flow for the reference process reaches a value of approximately 0.6 m³/s. This volume flow provides great potentials in intensifying the flow and as a result the cooling situation. Due to the suction effect, the maximum and average streaming velocities of the cooling wall jet increase.

As a further development, the utilization of this effect induces the geometrical modification of the cooling configuration. The basic idea involves the precooling of the aspirated air. Therefore, a targeted design and conception phase requires a fully understood flow situation. The mixing of the precooled ambient and the regular cooling air reduces the average temperature of the wall jet. This leads to a higher differential temperature between the cooling air and the film. As an outcome, the heat transfer is intensified.

Starting from this point, various geometrical changes were simulated by the use of the Process Model. To achieve a better flow situation, different versions of an additional air-guiding element were tested virtually, while keeping the position of the first element constant. Concerning this, the positioning, the dimensions and the stalling angle of the second element have been modified. As a goal, the height dependent maximal velocities should increase. The comparison between the height-dependent resulting maximal velocities of the reference and the most promising approach is provided in Table 3.

![Figure 5. Height dependent velocity profiles as an outcome of the evaluation strategy.](image)

**Table 3. Comparison of the height-dependent velocities.**

<table>
<thead>
<tr>
<th>Normalized height</th>
<th>( v_{\text{max}} ) [m/s] reference state</th>
<th>( v_{\text{max}} ) [m/s] optimized state</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2</td>
<td>36.6</td>
<td>37.6</td>
</tr>
<tr>
<td>0.4</td>
<td>23.3</td>
<td>24.7</td>
</tr>
<tr>
<td>0.6</td>
<td>17.3</td>
<td>18.4</td>
</tr>
<tr>
<td>0.8</td>
<td>13.5</td>
<td>14.5</td>
</tr>
<tr>
<td>1.0</td>
<td>10.9</td>
<td>11.5</td>
</tr>
<tr>
<td>1.2</td>
<td>9.8</td>
<td>10.4</td>
</tr>
</tbody>
</table>

It becomes clear that the maximum streaming velocities at each height coordinate have risen. The growth percentage vary between approximately 2.7 and 6.4%. Furthermore, the aspirated ambient air between the upper lip of the cooling ring and the lower edge of the air-guiding element can be intensified to a volume flow of almost 0.7 m³/s.

**Experimental Validation and Optimization**

These effects indicate that the implementation of an additional air-guiding element has a positive impact on the flow situation. In the next step, the final evaluation criterion for the optimization, namely the output-improvement, will be analyzed. Therefore, in a first step
the outcome of the virtual optimization has to be validated experimentally.

Figure 6 shows the outcome of the experimental validation procedure. The blue line represents the experimental recorded and the red line to the simulated contour for the optimized cooling configuration. The comparison indicates that the modified Process model is capable of simulating and optimizing an industrial high capacity blown film process. Especially up to a normalized height of approximately 0.5, a very good accordance has been achieved. Starting from this point, the diameter of the simulated contour gets larger than the experimental contour, what can be explained by the experimental recording procedure. The maximal deviation of the normalized radius, illustrated by the grey bars, reaches 0.046. Moreover, the experimental frost line height (dashed blue line) could be predicted in a very good accordance. The normalized frost line height of the simulated contour (dashed red line) lies approximately two percent higher than the experimental one.

As an indicator for the optimization of the cooling situation, the normalized frost line height of the optimized state should be lower, compared to the reference state (dashed black line). Due to the improved flow situation, a considerable reduction of the normalized frost line height of almost 0.04 has been reached. From the point of view of the industrial partner, the final evaluation criterion lies in the improvement of the output without neglecting the key quality factors for the final film product. The experimental investigation demonstrates that the final output could be increased by five percent until reaching the reference frost line height. Accepting a little higher frost line height, the output could even further be increased by ten percent compared to the reference state. For both processes neither the demanded thickness distribution nor layflat-width were influenced in a negative way.

Conclusions

In this paper, the applicability of the computer-assisted designing and optimizing strategy for a high capacity blown film line ( > 750 kg/h ; 1.0-1.5 kg/mm) was demonstrated. It could be shown that the combination out of the modified Process Model and the developed evaluation strategy for the CFD-results can be used for the improvement of the cooling configuration. Furthermore, the fluidic phenomena can be visualized and interpreted. Due to the implementation and the proper positioning of an additional passively acting air guiding element the frost line height could be decreased without changing the process parameters. As the final evaluation criterion, an improvement of the mass flow rate up to 10% is the result.

Further work will deal with the investigation of potentials in increasing the mass flow rate of high capacity blown film lines. Therefore, additional air guiding elements will be tested on a virtual and experimental basis. Moreover, the basic idea of precooler the aspirated air is of special interest. Conceptual studies will demonstrate the potential of this idea.

References


**Acknowledgment**

This project is supported by the German National Science Foundation (Deutsche Forschungsgemeinschaft, DFG), project-nr. WO 302/57-1. We owe a special thanks to our project partner Mondi Gronau GmbH for the support.