On determining the critical velocity in the high-pressure die casting machine’s shot sleeve using CFD

S. Kohlstädta,b,*, M. Vynnyckya, S. Goekec, A. Gebauer-Teichmannb

aDivision of Processes, Department of Materials Science and Engineering, KTH Royal Institute of Technology, Brinellvägen 23, 100 44 Stockholm, Sweden
bVolkswagen AG - Division of Components Manufacturing, Dr. Rudolf-Leiding-Platz 1, 34225 Baunatal, Germany
cKassel University - Institute of Mechanics, Mönchebergstr. 7, 34125 Kassel, Germany

Abstract

This paper investigates the critical plunger velocity in high-pressure die casting during the slow phase of the piston motion and how it can be determined with CFD methods in open source software. The melt-air system is modeled via an Eulerian volume-of-fluid approach treating the air as a compressible perfect gas. The turbulence is treated via a RANS approach using the Menter SST $k-\omega$ model. Two different strategies for mesh motion are presented and compared against each other. The solver is validated via analytical models and empirical data. A method is then presented to determine the optimal velocity using a 2D mesh. As a second step, it is then being discussed how those results are in line with the results obtained for real world 3D geometries and simulating also the ingate system of the die.

Keywords: compressible two-phase flow, fluid-structure interaction, high-pressure die casting, shot sleeve, critical velocity

1. Introduction

High-pressure die casting (HPDC) is an important process for manufacturing high volume and low cost automotive components, such as automatic transmission housings and gear box components [1–4]. Liquid metal, generally aluminium or magnesium, is poured into a shot sleeve chamber and further on injected through complex gate and runner systems and into the die at high speed, typically between 50 and 100 ms$^{-1}$, and under very high pressures up to 100 MPa. The normal high-pressure die casting process typically consists of three phases. These phases are shown in the following figure (figure 1 shows these phases. They are from left to right: pre-filling, die-filling (the shot), dwell-
pressure. The content of this paper will evolve around the processes during the first phase only.

![Figure 1: Three phases of die filling.](image)

(a) pre-filling
(b) die-filling
(c) dwell-pressure

One aspect of this process are the flow processes that take place in the shot-sleeve of the high-pressure die casting machine. For each combination of piston diameter, melt height in the chamber and chamber length there is exactly one critical velocity that can be determined analytically [5, 6] or measured experimentally [7]. Faura et al. also defined the optimal acceleration parameters for reaching that velocity [6].

In order to achieve a sound casting process the plunger speed and its acceleration profile have to be selected carefully. Figure 2 illustrates this claim. One observes here that three distinguished patterns can be seen. In the picture on the right, the plunger operates below the critical velocity of the system. The air-melt interface, i.e. the wave separates from the plunger and propagates freely inside the chamber. This is to be avoided as it may entrap air behind the ultimate melt front inside the casting.

The two extrema are further shown in figure 3. While figure 3(a) shows a process setup where the plunger speed is far lower than the speed that the accumulating wave in the chamber naturally wants to propagate with. Figure 3(b) shows the other extremum. Here, the plunger propagates too fast. The melt accumulates in front of the plunger much faster than the propagating wave can transport the material away from the melt-plunger interface. The result is a crashing of the wave as soon as the melt air interface hits the ceiling of the round chamber.

These figures indicate that there is only one good velocity for each melt height and shot-sleeve diameter combination that has to be found for every
2. Model equations

We model the two-phase flow of molten metal and air in high pressure die casting by using the VOF method [8], wherein a transport equation for the VOF function, $\gamma$, of each phase is solved simultaneously with a single set of continuity and Navier-Stokes equations for the whole flow field; note also that $\gamma$, which is advected by the fluids, can thus be interpreted as the liquid fraction. Considering the molten melt and the air as Newtonian [9], compressible and immiscible fluids, the governing equations can be written as [10–12]

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0,$$

$$\frac{\partial}{\partial t} (\rho \mathbf{U}) + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = -\nabla p + \nabla \cdot \{\left(\mu + \mu_{\text{tur}}\right) \left(\nabla \mathbf{U} + (\nabla \mathbf{U})^T\right)\} + \rho \mathbf{g} + \mathbf{F}_s,$$

$$\frac{\partial \gamma}{\partial t} + \nabla \cdot (\gamma \mathbf{U}) + \nabla \cdot (\gamma (1 - \gamma) \mathbf{U}_r) =$$

$$-\frac{\gamma}{\rho_g} \left(\frac{\partial \rho_g}{\partial t} + \mathbf{U} \cdot \nabla \rho_g\right),$$
where $t$ is the time, $\mathbf{U}$ the mean fluid velocity, $p$ the pressure, $\mathbf{g}$ the gravity vector, $\mathbf{F}_s$ the volumetric representation of the surface tension force and $^T$ denotes the transpose. In particular, $\mathbf{F}_s$ is modelled as a volumetric force by the Continuum Surface Force (CSF) method [13]. It is only active in the interfacial region and formulated as $\mathbf{F}_s = \sigma \kappa \nabla \gamma$, where $\sigma$ is the interfacial tension and $\kappa = \nabla \cdot (\nabla \gamma / |\nabla \gamma|)$ is the curvature of the interface. The term $\mathbf{U}_r$ is a supplementary velocity field for compressing the phase-interface introduced by the solving scheme for the $\gamma$-field (MULES) [10, 14]. The material properties $\rho$ and $\mu$ are the density and the dynamic viscosity, respectively, and are given by

\begin{align}
\rho &= \gamma \rho_l + (1 - \gamma) \rho_g, \\
\mu &= \gamma \mu_l + (1 - \gamma) \mu_g,
\end{align}

(4) (5)

where the subscripts $g$ and $l$ denote the gas and liquid phases, respectively. We take $\rho_l, \mu_g$ and $\mu_l$ to be constant, but use the approach to assume air as a compressible perfect gas, i.e. its density changes with pressure and temperature. Hence, the equation of state for our model reads

\[ \rho_g = \frac{p}{R_s T}, \]

where $R_s$ is the specific gas constant and $T$ is the temperature. Equation (6) introduces a new field into a set of unknowns. $T$ also requires a PDE to be solved as its value may change over space and time and its quantity is transported according to the equation below.

\[ \frac{\partial}{\partial t} (\rho T) + \nabla \cdot (\rho T \mathbf{U}) = \nabla \cdot (\alpha_{eff} \nabla T) \]

\[ - \left( \frac{\gamma}{c_{v_l}} + \frac{1 - \gamma}{c_{v_g}} \right) \left( \nabla \cdot (\rho \mathbf{U}) + \frac{\partial (\rho K)}{\partial t} + \nabla \cdot (\rho K \mathbf{U}) \right), \]

(7)

where $K = \frac{1}{2} \mathbf{U} \cdot \mathbf{U}$ is the kinetic energy, $c_{v_g}$ and $c_{v_l}$ denote the specific heat capacities at constant volume for the gas and liquid phases, respectively, and $\alpha_{eff}$ is given by

\[ \alpha_{eff} = \frac{\gamma k_l}{c_{v_l}} + \frac{(1 - \gamma) k_g}{c_{v_g}} + \frac{\mu_{tur}}{\sigma_{tur}}, \]

where $\sigma_{tur}$ is the turbulent Prandtl number, whose value is set to 0.9 [15].

This set of equations describes a system where air is being treated as compressible. Simulations were also done with treating air as incompressible and the entire system as isothermal. This simplifies the set of equations. All time derivatives of the density can be omitted and no equations of state and no temperature PDE is needed anymore.

Furthermore, $\mu_{tur}$ in equation (2) denotes the turbulent eddy viscosity, which will be calculated via the Menter $k$–$\omega$-SST model [16]. The implementation of
the Menter $k-\omega$-SST model inside the OpenFOAM framework has shown to be robust and also in excellent agreement with experimental data [17].

Equations (1)-(7) require boundary conditions. In the HPDC shot sleeve situation under consideration, there are essentially two types of boundaries: an outlet and a wall. At the former, the normal velocity, the temperature and the liquid fraction are treated with the \textit{zeroGradient} condition; thus,

\begin{equation}
\nabla U \cdot \mathbf{n}_O = 0, \quad (8)
\end{equation}

\begin{equation}
\nabla T \cdot \mathbf{n}_W = 0, \quad (9)
\end{equation}

\begin{equation}
\nabla \gamma = 0, \quad (10)
\end{equation}

where $\mathbf{n}_o$ denotes the outward unit normal vector at the outlet. Furthermore, it is necessary to set conditions for the turbulence quantities relevant to the turbulence model. This is done via the turbulence intensity and length scale for the largest eddies, which were set at 0.01\% and 2 mm, respectively. At walls, the tangential and normal components of velocity are zero. One exception is the moving wall that connects to the plunger. There the velocity of the fluid was prescribed to have the exact same velocity as the moving wall, i.e.

\begin{equation}
\mathbf{U} = \mathbf{U}_{\text{movingWall}} \quad (11)
\end{equation}

In addition, conditions are required for the temperature; for simplicity, we assume that all walls are thermally insulating, so that

\begin{equation}
\nabla T \cdot \mathbf{n}_W = 0, \quad (12)
\end{equation}

where $\mathbf{n}_W$ denotes the inward unit normal vector at a wall. Note that, whilst these are the basic wall boundary conditions, they are taken care of in different ways according to the turbulence model used. We omit the details here, which can instead be found in [16, 18].

Lastly, we also require initial conditions. We assume that the cavity initially contains melt to a certain height $h_0$ and above that only gas. Both phases are at rest and the air is at room temperature, $T_0$. The melt has the temperature $T_{\text{melt}}$ of liquid molten aluminium alloy 4600, i.e. 823 K. Thus, we set

\begin{equation}
\mathbf{U} = 0, \quad (13)
\end{equation}

\begin{equation}
T = \begin{cases} 
T_{\text{air}}, & \text{if } h > h_0 \\
T_{\text{melt}}, & \text{if } h \leq h_0
\end{cases}, \quad (14)
\end{equation}

\begin{equation}
\gamma = \begin{cases} 
0, & \text{if } h > h_0 \\
1, & \text{if } h \leq h_0
\end{cases}, \quad (15)
\end{equation}

\begin{equation}
\gamma = \quad (16)
\end{equation}

The model parameters are given in Table 1; note that it is usually $c_{p_g}$ and $c_{p_l}$, the specific heat capacities at constant pressure, that are tabulated, rather than $c_{v_g}$ and $c_{v_l}$. However, for air, the value of the isentropic expansion factor
or heat capacity ratio, $c_p/c_v$, can be assumed to be constant and equal to 1.4 for air in the given temperature range [19, 20] with acceptable accuracy. On the other hand, the aluminium melt was treated as incompressible in this model; thus, $c_v = c_p$.

One additional comment at this position on the material parameters. These (table 1) are the parameters used in the regularly used commercial code MAGMA for the particular alloy. The first step in this research work, however, was to validate the solver against existing studies in the field. This was done in one case based on a study by Korti and Aboudi [21] and the referenced studies in this paper (see figure 7 later in the paper). In the interest of optimal comparability, the physical model parameters of the previous publications were used for the benchmark study, while the reported results in the results section rely on table 1’s values. The differences on some occasions can probably be attributed to the previous authors using the values for pure liquid aluminium, while we are using the values for the particular alloy, which is majoritively in use at the Volkswagen Kassel foundry.

### 3. Mesh motion strategies

Two different mesh motion strategies have been applied in order to include the motion of the piston and the therefore shrinking fluid domain into the model. Both simulations start with the base mesh at $t = 0$. Figure 4 shows a simplified version of this mesh.

x- and y-vector point in the horizontal and vertical direction, respectively and the boundary to the plunger is on the left hand side of the rectangle. The two strategies that shall be explained here, briefly, are mesh compressing and layer addition and removal, while here only layer removal is of practical use as the fluid domain decreases in size over the course of the simulation.
After a certain interval of $\Delta t$ has elapsed, the mesh looks as in figure 5 if the compressing mesh motion strategy is applied. The entire domain has shrunk to half of its original length and volume. One can see that all cells themselves are also shrunk to half of their original horizontal length as shown in figure 4.

As seen in figure 5, the first strategy compressed the mesh, i.e. incrementally shortening the length of each cell in the x-direction. The other strategy (layer removal) compresses only the outermost layer of the mesh at the same time. The basic principle is further illustrated in figure 6.

The thickness of the outermost layer is reduced further and further until a user-defined minimum layer thickness is reached (figure 6(a), here $\frac{L}{2}$). Figure 6 shows the situation just before that last layer is removed. If the thickness of this last layer is about to fall below the user-specified limit, the mesh motion solver removes that layer and adds its residual thickness to the next layer by...
adjusting the points of that layer (see figure 6(b)). In this paper the strategy of layer addition and removal was used as it proved to be more stable. Also, the changing of the mesh happens at a location mostly quite far away from the melt-air interface which is of course the part, that the engineer is in most cases particularly interested in. An additional speed-up of the simulations was also see towards the end using the layer removal strategy as the solver had less and less cells to handle during the later time steps.

4. Solver development and testing

Given the complexity of the physics involved and also the niche nature of the application, the C++ toolbox OpenFOAM [22–24] was used to implement the model as it is freely available, rather suitable for being extended by the user and very well scalable for industrial application as extra cores in parallelisation do not require additional licenses. Talking about the solver one has to distinguish between the physics solver that solves the differential equations noted in section 2 and the mesh motion solver. The physics solver is the solver *interDyMFoam* or *compressibleInterDyMFoam* which exist inside the standard foundation release of OpenFOAM. Please note that the equations that were noted in section 2 are the more complex ones for the solver *compressibleInterDyMFoam*. The solver *interDyMFoam* assumes incompressibility for both phases and is therefore able to simplify the model equations significantly. All time derivatives of $\rho$ disappear and no equation of state is needed and therefore also no additional PDE for temperature.

The solver that was used for the mesh motion in this paper exists in the OpenFOAM foundation release and in a slightly altered form also inside the
foam-extend package; it had, thus, not to be implemented as outlined in the previous section 3 for the mesh motion solver. The solving procedure of the code is shown in figure 8. The reader can see that it follows basically the same procedure of the classic PIMPLE algorithm – which is a combination of the SIMPLE [25] and PISO [26] solutions procedures –, with the addition that at the beginning of each loop the PDE for the phase fraction $\gamma$ is being solved.

As outlined in section 2 by the presented equations, already the melt-air system is a rather complex fluid flow problem. However, such a solver exits inside the foundation release of OpenFOAM as well as in the release of the foam-extend package. The implementation of the two, however, is different. While the compressible solver `compressibleInterDyMFoam` inside the foundation release uses the equations as stated in section 2, the solver inside the foam-extend release uses the simplification of modelling the gas as a barotropic fluid, i.e. the equations of state (equation (6)) changes to

$$\rho_g = \frac{p}{R_s T} = p\psi,$$  \hspace{1cm} (17)

by omitting the influence of temperature and modelling the density of air dependent on pressure via a compressibility factor $\psi$. Due to the simplified equation of state, it is not necessary for the foam-extend solver, to solve the PDE for temperature (equation (7)).

Upon successful compilation, the solver was benchmarked against existing data by Korti and Aboudi [21]. Figure 7 shows the result of this benchmarking study. It is important to note that Korti and Aboudi themselves compared their simulation result with previously published data obtained from the shallow-water model and a PHOENICS model by López et al. [5]. They found good agreement. By comparing our results with the results from the Korti and Aboudi paper, we were thus able to validate it against the data from their benchmark studies as well. The mesh for which the solver was validated with the results in [21] was a 220 × 51 mesh. The same mesh as the one that the previous authors used with (60 × 30 elements) also yielded good results. However, even with the finer mesh
the 2D simulations ran in less than 10 minutes and a mesh of these dimensions was supposed to be used for the later simulations.

In figure 7 the x- and y-axes show the respective x- and y-coordinates of the interface of the propagating melt inside the shot sleeve chamber at various time step. The interface velocity was prescribed to follow that law stated by Korti and Aboudi [21], i.e.

$$U_{\text{movingWall},x} = \begin{cases} \alpha \beta (e^{\alpha t} - 1), & \text{if } t \leq t_h \\ 2(\sqrt{gH} - \sqrt{gh}), & \text{if } t > t_h \end{cases}$$ \tag{18}$$

Please note that while $U_{\text{movingWall}}$ in equation (18) is of course a vector, only its x-component is prescribed. All the other components were set to be 0. This follows from the fact that the mesh was arranged in a way that the plunger moved horizontally in positive x-direction. In fact the solver was programmed in a way that the user only specifies the patch normal velocity as a scalar and the solver itself transforming it into a vector.

The reader sees that equation (18) introduces the unknowns $\alpha$, $\beta$, $H$ and $h_0$. Those values are all constants; in case of $H$ and $h_0$ they describe the initial geometrical setup of the shot sleeve. All values for these constants and other necessary geometrical constraints are given in table 2.

Going back to the result of the benchmark study in figure 7, the reader can see that at all 4 documented time steps, the tokens are on top of each other.

Figure 8: Solving process scheme of the two phase VOF-solver
One slight deviation is observable. The OpenFOAM result seems to indicate a slightly higher result in terms of the melt height at the plunger interface and also the melt momentum sees to propagate slightly faster through the melt domain.

The authors, however, have reason to believe that this result is actually more correct. The first time that is evaluated is the time for a parameter of $\alpha = 1.8$ where $t = t_h$, which is according to the definition by López et al. [5] the physics of the system change as the melt cannot accumulate in front of the plunger anymore as it has reached the height of the shot sleeve’s ceiling and the plunger from now on propagates with constant velocity according to the second case in equation (18). We therefore conclude that here our solution is slightly more accurate.

In addition to this graphical comparison the solver has also been tested for mass conservation, volume conservation and conservation of $\gamma$ in the process on simple geometries and benchmarked with analytical models. One analytical model for this purpose is for example the model by Reikher and Barkhudarov [27]. The formula shown below gives the height of the melt at the piston boundary

$$h_p(t) = \frac{1}{g} \left( c_0 + \frac{1}{2} v_p(t) \right)^2. \quad (19)$$

Equation (19) is ultimately derived from the shallow-water differential equations [28]. As the authors were interested in benchmarking the solver on a geometry closer to the one they were ultimately interested in, a geometry with the constraints shown in table 3 was created using the blockMesh utility inside the OpenFOAM toolbox. The mesh spacing for this study was 2 mm in horizontal direction and 1 mm in vertical direction. The result of this study can be seen in figure 9. $t_h$ for this configuration is 0.865 and the critical velocity according to equation (18) is 0.44 ms$^{-1}$. The reader also sees this in figure 9 that after the value $t_h$ is reached, the increase in melt height stops at this point as $t_h$ is by definition the time when then melt-air interface hits the ceiling of the shot sleeve chamber. It follows from figure 9 that the numerical simulation of the OpenFOAM model is in excellent agreement with the analytical formula in equation (19).

Table 2: Geometrical and process constants for benchmarking the introduced model with data from [5, 21]

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$H$</td>
<td>$5.08 \times 10^{-2}$ m</td>
</tr>
<tr>
<td>$L$</td>
<td>$45.72 \times 10^{-2}$ m</td>
</tr>
<tr>
<td>$h_0$</td>
<td>$1.67 \times 10^{-2}$ m</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>1.8</td>
</tr>
<tr>
<td>$\beta$</td>
<td>$0.5 \times H$</td>
</tr>
</tbody>
</table>
\[ H = 130 \times 10^{-3} \text{ m} \]
\[ L = 1.015 \text{ m} \]
\[ \varphi = 65 \% \]
\[ \alpha = 1.8 \]
\[ \beta = 0.5 \times H \]

Table 3: Geometrical and process constants for benchmarking the introduced model with the analytical law for melt height (equation (19)) at the piston boundary in [27]

5. Results and discussion

5.1. Parameter studies with 2D-shot sleeve

As this study was conducted by manufacturers out of an interest to previously define the correct process parameters for their casting process. It was hence looked for what kind of experimental data was available in order to attend to the given problem. As early as 1986 an experimental study of the optimal velocity for the slow phase in the die casting machine was conducted by Garber [29] with the critical velocity being plotted against the show sleeve diameter for several filling degrees.

The first task for the newly developed solver was therefore to redo these results and determine why this was the particular value for the critical velocities and what would happen if other parameters were chosen instead. For this purpose, a simple 2D shot sleeve mesh was constructed that resembles a shot sleeve with the diameter of 130 mm. Fill fraction was 65 % and the sleeve was 1.015 m long.

With this shot sleeve a parameter study was conducted and the results evaluated. An array of 100 different slow phase velocities was tested and the residual air in the sleeve at the point where the melt front reaches the ingate was evaluated. The result of these simulations is shown in figure 10. The abscissa features the different velocities in ms\(^{-1}\) and the ordinate shows the corresponding trapped air in the sleeve. The time for the evaluation differs as the melt
The tested velocity in the slow phase was incrementally increased starting from 0 to 1 m\(s^{-1}\) with an increment of 0.01 m\(s^{-1}\). One sees that in the beginning the differences between the several tested velocities can be quite large. The reason for this pattern is believed to be the lottery nature of the wave proceeding inside the chamber. If the peak of the wave is somewhere near the piston when the first melt front leaves the chamber, the residual air inside the shot sleeve will be quite neatly pushed out in front of the metal. This is a desirable state also in the serial process as the air can then be vented out of the die via the venting system.

However, as figure 10 indicates, already small changes in the velocity can turn the result towards the other side where the wave peak is near the outlet when the melt front propagates through the inlet of the casting. By doing so, it trapped a rather big bubble of air between the piston and the melt front. This process state is not desirable as this melt is basically stuck there as it can not be vented via the de-airing system of the die either. The melt will flow into this venting system first and seal it for the residual air in the process. This process state is therefore to be avoided.

At about 0.13 m\(s^{-1}\) one observes a stable low in the trapped air. This is due to a state of the system where plunger speed is still much slower than the natural wave propagation speed in the chamber. But given the system constraints such as the length of the shot sleeve, the wave can now more or less travel to and fro in the chamber thus resulting in only a minimal amount of air being trapped behind the melt front with the majority being ejected in front of the melt-air interface. This stable period lasts until a plunger velocity of about 0.17 m\(s^{-1}\) is reached.

After this value the trapped air increases again very steeply until again a stable plateau is reached starting from the velocity value of about 0.23 m\(s^{-1}\). There one observes the opposite effect. The piston advances now too fast for allowing the wave to travel back and forth inside the shot sleeve. Contrarily, it is interrupted on this journey while the peak is still in a front location. Thus, a large amount of melt is sealed between the wave peak that is now near the
outlet and the piston front. This state is more or less stable until a velocity value of 0.32 ms$^{-1}$ is reached.

After that the piston enters into the regime where the optimal velocity is found. It starts at about 0.38 ms$^{-1}$ (see figure 10). There one sees a constantly low level of trapped air close to zero. The flow pattern that corresponds with this result is a wave peak that builds up in front of the piston until the ceiling of the shot sleeve is reached. From then on the piston has more or less the same speed as the wave of that height difference would have where it to proceed naturally and without the piston. By applying this pattern the operator ensures that all of the air that is inside the shot sleeve gets pushed out in front of the melt and hence no melt is sealed behind the outlet. The optimal range continues up to a plunger speed of 0.46 ms$^{-1}$. These values match existing experimental data very well as Garber [29] and Brunnhuber [2] report 0.46 ms$^{-1}$ as the critical velocity for the given configuration.

After this value the processes that are happening change completely. While the amount of air that was trapped behind the wave peak previously determined how much air was left in the chamber, there is no longer a wave travelling through the shot sleeve from this point on. The piston now propagates much faster than the natural wave speed inside the shot sleeve so melt accumulates continuously in front of the piston. This process goes on until a critical limit is reached and the wave plunges, thus entraining air in the process that is later measured as residual air in the CFD model. It is an interesting observation that the amount of air that is entrained by this process only increases not very steeply. The recommendation for process engineers would therefore be that it is less harmful to let the plunger move a bit faster than the exact critical velocity as the entrainment process is much less critical compared to the other process of locking an air pocket behind the melt peak. An additional observation further underscores this thought. The contaminated melt is injected into the casting in front of the clean melt. It is therefore a part of the melt that will end up in the de-airing channels of the die. On the other hand the air bubble that is behind the melt front will most likely end up somewhere inside the casting.

5.2. Testing the solver on a 3D shot sleeve

The solver was then also applied to a real world geometry, i.e. the shot-sleeve parameters of the crank case for the EA211 crank case at Volkswagen. The diameter of the shot sleeve in this case is again 130 mm. The sleeve is 1.015 m and the fill fraction is 65 %. The velocity was here increased by increments of 0.05 m/s starting from 0.15 m/s. The amount of trapped air that was evaluated by a function object is plotted in figure 11.

The data shown in figure 11 was taken out of a 3D simulation that also included the particular ingate geometry of the casting. Its shape can be seen in the following figure (figure 12). One sees here that as soon as the melt frontier approaches towards the inlet of the casting, the amount of trapped air behind that front constantly decreases with increasing velocity of the plunger in the first phase.
(a) trapped air in shot-sleeve

(b) distance-velocity law for plunger

Figure 11: Trapped air for the shot-sleeve in the EA211 crank case application and the corresponding tested plunger law.
The reader should also note that this is normally critical for the casting’s quality as this amount of air can not be extracted from the die. Neither the vacuum or other venting system can carry this out as they are typically automatically closed as soon as the melt front approaches.

![Image](image.png)

Figure 12: The fraction occupied by air after the meltfront has propagated into the ingate.

6. Conclusions

The research in this paper showed that it is possible even with freely available open-source software to model the computationally rather complex flow problem of shot sleeve dynamics in the slow phase without the need for costly commercial software tools. The documented results are well in line with previous CFD simulations on the matter and also analytical models derived from the shallow-water equations. Two different mesh motion strategies were presented with a preference for the layer-removal approach. The solver was also ultimately capable of solving the melt flow of 3D shot sleeves with regular industrial ingate systems attached. It was in general possible to reproduce also the results expected for 3D geometries in practical tests. The economical benefits of increasing the velocity in the slow phase of the piston motion not only for the sake of less air entrapment were also pointed out.

References


