

Flow and Heat Transfer in a Radially Spreading Liquid Metal Jet Related to Casting of Ferroalloys

Harald Haaland

Elkem Shared Services, Technology and Projects,
P.O.Box 4376, 0402 Nydalen, Norway

Jon Arne Bakken

Norwegian University of Science and Technology,
Dept. of Materials Technology and Electrochemistry,
7491 Trondheim, Norway

ABSTRACT

In the past more and more advanced and fine-tuned processes for steel production have resulted in increased demands for new and more cost-effective ferroalloys used as constituents in the processes. Casting techniques and equipment are continually studied for potential improvements. Research on free metal flows, as in casting into open moulds, is scarce and to remedy this a doctoral study at the Norwegian University of Science and Technology was initiated.

The study was limited to the region around the impingement point of the metal jet, because this is the critical area for both heat and mass transfer. Only steady-state conditions on a flat plate (without accumulation of fluid) are studied. This system is believed to be a good approximation for the initial conditions during the filling of an open mould

Numerous practical and mathematical simulations have been carried out and a relatively simple analytical model depicting the surface profile of the liquid metal including heat transfer to the surroundings has been developed. The computational fluid dynamics code FLUENT was also used to compute the surface profile with the Volume-of-Fluid technique, but with little success. The code was instead used to determine the flow and temperature fields inside the surface profile established by other methods. Various laminar and turbulent flow models (variations of the $k - \varepsilon$ model) were compared. Experiments with water were carried out for studying the flow field. Tin was used for heat transfer studies. Finally, the simulations were compared with results from practical experiments.

Measured heat flow usually exceeded predicted values, particularly in the stagnation region. On the other hand, measured temperatures in the plate were noticeably lower than computed. Good agreement is shown between the results from the FLUENT simulations and the new analytic model, which shows good promise of acting as a useful alternative to the much more demanding numerical simulations.

1. INTRODUCTION

More and more advanced and fine-tuned processes for steel production have resulted in increased demands for new and more effective ferroalloys used as constituents in the processes. Elkem, as one of the world's leading suppliers, is continuously seeking to improve its competitive position. In order to achieve this Elkem had to increase its efforts in research and development and introduce more cost-effective technology at the smelting plants; thus making products with higher value and quality for the customers. Some important parameters for metal quality is chemical composition, level of impurities, intermetallic structure and grain size.

In 1991 Elkem started a five year development program (Post Taphole 1995) with the aim to find a new process for casting ferroalloys and silicon metal.

2. PROCESS DEVELOPMENT

Without going into detail, a bar casting technique was developed. The bars are cast directly to their final size in two dimensions and need only to be cut in the longitudinal direction to meet customer

specifications. In this way the fines generation is reduced by two thirds. Numerous mathematical and practical simulations of flow and heat transfer in open channels were performed and a pilot plant was built. The system worked as planned, but thermocouples in the mould registered temperatures that were somewhat higher than calculated, especially beneath the impingement point of the metal jet. This is a critical area for both heat and mass transfer. A decision was made to investigate more closely the heat transfer and metal flow around such an impingement point. This doctoral study was initiated as a result.

3. EXPERIMENTS

The flow analyzed is *axisymmetric* with an idealized vertical jet *impinging on a horizontal flat plate*. In order to simplify matters as best as possible, the fluid is left to flow over the edge of the plate. In this way there is no filling of a receptacle as in traditional casting, but the simplified flow is believed to reliably reflect the conditions during the initial stages of a casting process. This period, before the mould has started filling, is when the thermal loads are at their greatest. Moulds have then been known to crack.

Figure 1 shows the simplified jet. It impinges on a horizontal circular plate and will develop a radial flow pattern, spreading evenly over the plate. If the flow spreads far enough it will eventually increase quite abruptly in thickness, known as a hydraulic jump.

For studying the flow pattern, water can be used, as its viscous properties are relatively similar to that of liquid metals. When studying heat transfer, water, however, cannot be used. The thermal properties of these fluids are vastly different and therefore a liquid metal must be used. Tin has a comfortably lower melting point than ferroalloys and was therefore chosen for this - simplifying the experiments greatly.

Similar experimental set-ups were used for water and tin. Water was directed on a horizontal glass plate, while a cooled stainless steel plate 150 mm in radius and equipped with 51 thermocouples, was used for tin. These were placed on the top and lower surfaces of the plate so that temperatures and heat flow to the

plate could be measured across the whole plate. Figure 2 shows the experimental set-up.

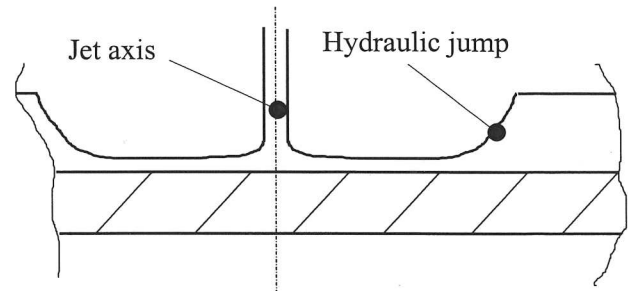


Figure 1 Axisymmetric jet on circular plate.

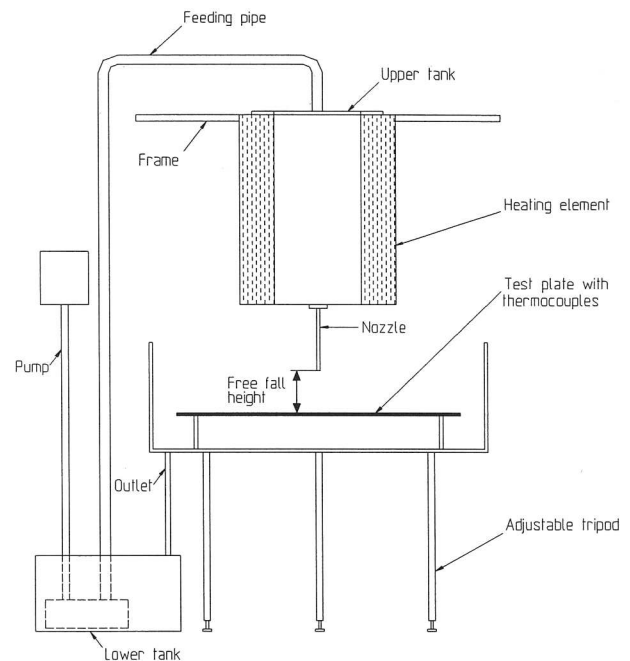
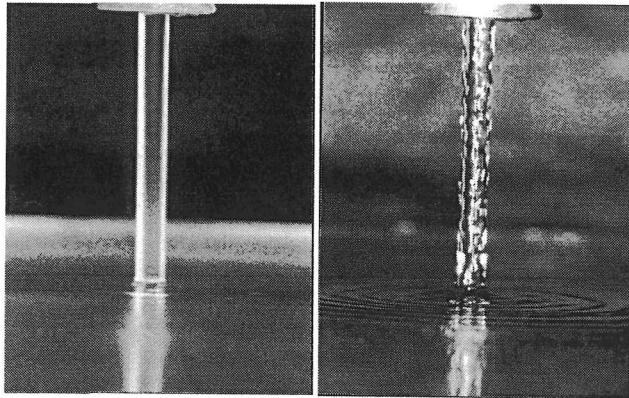


Figure 2 Set-up for tin experiments.

A laser-Doppler instrument was used to measure velocities in the water film. This was not specifically successful as ripples on the water surface disturbed the measurements. In addition, the measuring volume of the instrument proved to be too large in relation to the film thickness. This thickness is estimated to be 0.1 mm before the hydraulic jump and approximately 3.5 mm after for water, and 5.5 mm for tin.

It is not known if the surface ripples are generically inherent to such a flow or if they are caused by some external source. The water jet from the nozzle was photographed to study how smooth and ideal it is in

reality. Figure 3 shows the result. The first photograph illustrates what is perceived by the human eye. In the second a higher shutter speed is used and one can clearly see that the jet is uneven. This will in all probability lead to disturbances in the surface of the radially spreading flow.



a)

b)

Figure 3 Photographs of impinging water jet. Nozzle diameter is 10 mm.

a) low shutter speed and resembles what is perceived by the human eye

b) high shutter speed.

4. COMPUTATIONS

The finite difference program FLUENT was originally intended to simulate the flow and heat transfer problem. FLUENT uses the *Volume-of-Fluid* technique for free-surface computations. However, numerous simulations proved that reliable results could not be obtained. One possible way of solving the problem is to apply some other method to depict the surface profile of the flow, and then instruct our finite-difference code to perform a more exact calculation of velocities and heat transfer within this flow profile.

From a thorough literature survey three analytic models were elected and studied more closely: Watson (1964), Buyevich and Ustinov (1994) and Alekseenko et al. (1994). A new model was also developed in the course of this work. It is based on the Bernoulli equation including friction. A force balance is used for predicting the position of the hydraulic jump. Table 1 compares some analytically derived jump positions with experimental results.

Flow rate [$10^{-4} \text{ m}^3/\text{s}$]	Free fall height [m]	Re_j [m]	Observed jump position [m]	Watson [m]	Critical velocity method [m]	Buyevich And Ustinov [m]	Bernoulli [m]
1.000	0.081	79560	0.11	0.107/0.077	0.152	0.105	0.096/0.082
1.044	0.090	83150	0.11-0.12	0.112/0.080	0.150	0.110	0.100/0.086
0.959	0.028	72630	0.11	0.100/0.073	0.147	0.099	0.092/0.080
1.132	0.089	98140	0.10	0.120/0.086	0.164	0.118	0.106/0.092
1.093	0.048	83620	0.10-0.11	0.115/0.083	0.160	0.113	0.104/0.088
1.070	0.088	84800	0.10	0.114/0.082	0.159	0.112	0.102/0.088
1.100	0.089	86950	0.10-0.11	0.116/0.084	0.161	0.115	0.104/0.090
1.046	0.092	83490	0.10	0.112/0.080	0.152	0.110	0.100/0.086
1.051	0.037	79830	0.10-0.11	0.110/0.080	0.156	0.108	0.100/0.086

Table 1 Predicted jump positions for tin flow through 8 mm nozzle. Plate radius 150 mm. Where two values separated by slashes are given, the first is based on a laminar model and the second on a turbulent model. The critical velocity method is based on the turbulent theory of Alekseenko et al. The Buyevich and Ustinov model is laminar.

The film thickness at the plate edge is a critical parameter for the jump position. The theories of

Watson and Alekseenko et al. require that this value is by some means *known and stays constant* all the

way in to the jump. At the same time no consideration is taken of the plate size and its possible influence on the jump position. The theory of Buyevich and Ustinov and the Bernoulli model are *superior in this manner*. Both these models, however, predict only slight variation in the film thickness after the jump - *agreeing well with practical observations*.

Various methods for predicting the film thickness at the edge of the plate have been found, but these are substantially smaller than what was experimentally observed. Based on the idea that the minimum thickness a film can have is equal to thickness of a sessile drop, a new useful correlation for the film thickness at the plate edge, h_R is proposed:

$$h_R = \left(\frac{3}{2} \frac{\nu V}{\pi g R} \right)^{\frac{1}{3}} + \sqrt{\frac{2\sigma}{\rho g} (1 - \cos \theta)}$$

ν is the kinematic viscosity of the fluid, σ its surface tension, ρ its density and θ the wetting angle between the fluid and the surface underneath. V is the volumetric flow rate and R the plate radius. g is the acceleration of gravity.

The Bernoulli model predicts jump positions in the same region of accuracy as the other models.

Analytical Heat Transfer Model

Studies regarding heat transfer in radially spreading flows due to an impinging jet have been published before. These findings, however, relate to fluids with Prandtl numbers (Pr) in the region of unity and higher, and are thus *not* applicable to liquid metals. In this chapter a *new theory* for such flows with $Pr \ll 1$ is proposed.

The high thermal conductivity of liquid metals account for their very low Prandtl number, since the viscosity and specific heat do not differ greatly from other common fluids. The high thermal conductivity is the basis for their unusual heat transfer characteristics. Molecular conduction is the dominant heat transfer mechanism, even in fully turbulent flows. The effect of eddy diffusivity becomes smaller as Pr approaches zero. As a result

of this, heat transfer in liquid metals has many characteristics of a laminar flow.

This means that the effective conductivity is fairly constant throughout a thermal boundary layer, even though the effective viscosity may vary considerably. This again leads to only slight variations in the temperature gradient throughout this boundary layer.

As in laminar flows, the Nusselt number for a liquid metal is often low, but due to the high molecular thermal conductivity, heat transfer coefficients tend to be high.

With such high heat transfer coefficients involved, surface contamination and wetting problems can easily and noticeably affect the heat transfer. These facts have reportedly led to difficulties and large uncertainties in experiments. As is fairly commonly known, theoretical expressions do not always match the values found in experiments. It is not improbable that such disagreements are augmented with liquid metals. Theory in this work, of course, applies to clean systems with ideal wetting of the surfaces involved.

Only *laminar* conditions are considered in the analytical heat transfer models.

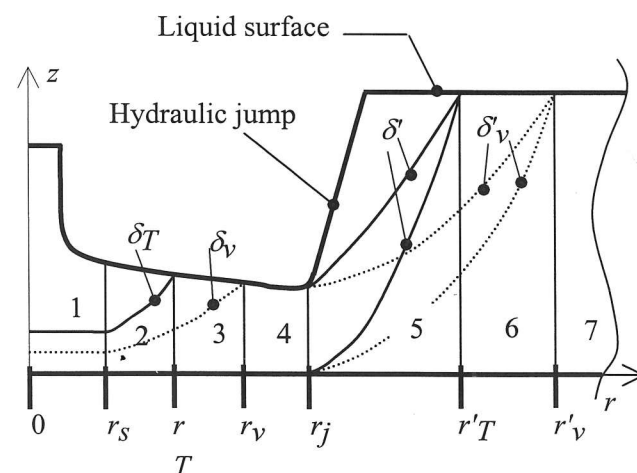


Figure 4 Regions for theoretical heat transfer analysis for liquid metals.

A jet impinging on a horizontal surface leading to a radially spreading flow incorporating a hydraulic jump, is for clarity divided into *seven regions* for

heat transfer analysis, see Figure 4. r_s marks the boundary of the stagnation region. r_T and r_v are where the thermal and velocity boundary layers reach the fluid surface respectively. r_j is the position of the hydraulic jump. Symbols marked ' refer to conditions after the jump. δ_v and δ_T (the viscous and thermal boundary layers) are envisioned to develop in two different manners after the jump. In the first, they develop from the value they have immediately before the jump. In the second, the jump is thought to disturb the flow so much that the boundary layers are completely broken down and must be built up from scratch.

Region 1, $0 < r < r_s$, includes the free jet and its deflection into a radial jet. Here there is a stagnation point, whose dimensions are of the order of the jet radius a . The flow velocity grows thereafter rapidly from zero to a value often assumed equal to the undisturbed jet velocity just before impact, U_∞ . Because of the acceleration (from nozzle orifice to plate) due to gravity, this velocity may be greater than that in the nozzle itself. It is generally assumed that the jet flow is at a high enough Reynolds number so that the stagnation region is inviscid and that viscous effects occur near the lower boundary as the radial flow is retarded. The fluid temperature is equal to the bulk jet temperature, T_∞ .

The thermal and viscous boundary layers are constant in thickness.

A detailed solution of the flow problem (Schlichting, 1979 and Shach, 1934) and neglecting radial convection and conduction in the energy equation results in a Nusselt number, Nu , of the form

$$Nu = \frac{0.9381 \sqrt{Re_j}}{\int_0^\infty \exp \left[-2 Pr \int_0^\eta \phi(\eta) d\eta \right] d\eta}$$

This equation can be solved numerically, but a new, more practical correlation for $Pr \ll 1$ was developed in this work:

$$Nu = 0.904 Re_j^{0.5} Pr^{0.48}$$

This expression is found to deviate from the "exact" solution by maximum 2.5% for $0.001 < Pr < 0.05$. It is noted that the result is independent of radial location.

Region 2, $r_s < r < r_T$. The heat flow and viscous stresses cause the two boundary layers to grow until they eventually reach the film surface. For $Pr < 1$ the thermal boundary layer develops fastest, while the viscous boundary layer develops fastest for fluids with $Pr > 1$. r_T and r_v are the radial positions where the thermal and viscous boundary layers reach the surface.

Based on assumed third order polynomials for the temperature and velocity profiles through the film thickness, we find from the integral energy equation:

$$Nu_r \approx 0.769 Re_r^{0.5} Pr^{0.48}$$

$$\text{with } Re_r = \frac{U_\infty r}{\nu}.$$

In addition

$$r_v \approx 0.305 \left(\frac{Va^2}{\nu} \right)^{\frac{1}{3}}$$

$$\text{and } r_T \approx \left(\frac{5}{5.824 Pr^{-0.48} - 3} \right)^{\frac{2}{3}} r_v$$

The maximum deviation from the exact solution is less than 8% for $0.00015 < Pr < 0.1$ and less than 2% for $0.00015 < Pr < 0.02$. Alternatively, several, piecewise more accurate correlations could be composed for various Prandtl number intervals.

Region 3, $r_T < r < r_v$. Here the thermal boundary layer occupies the whole cross-section of the flow, while the viscous boundary layer is still developing.

Employing a third order polynomial for the temperature profile, we find an approximate expression for the convective heat transfer coefficient, h_c , between the fluid and the plate beneath

$$h_c = 3.46 \frac{k}{h}$$

and the local Nusselt number, defined in the terms of the local fluid film thickness, h and its thermal conductivity, k .

$$Nu = \frac{h_c h}{k} \approx 3.46$$

Region 4, $r_v < r < r_j$. Here both boundary layers have reached the fluid surface and occupy the whole film thickness.

$$h_c = \frac{105}{31} \frac{k}{h}$$

and the local Nusselt number

$$Nu = \frac{h_c h}{k} = \frac{105}{31} \approx 3.39$$

Region 5 $r_j < r < r_T'$. In connection with the jump the fluid flow must reorganize, involving new development of the boundary layers. One possible scenario is that both boundary layers have reached the surface of the flow before the jump and will now develop further from this thickness. Matters are, however, simplified if the boundary layers are assumed to have been broken down completely and start developing from zero thickness at the jump. Since the film thickness is an order of magnitude greater after the jump than before, this simplification is considered to have little effect on the final results.

The smoothing out of the flow gradients (over the cross-section of the flow) with respect to velocity and temperature cannot of course occur abruptly at r_j , but must in reality take place over a finite interval Δr . The theory outlined below will therefore not be applicable at exactly r_j (just as in the case of free-stream boundary layer flow over a flat plate, where the theory breaks down at the tip of the plate, i.e. $x=0$).

The conditions after the jump are somewhat simplified by the fact that the film thickness in this region varies little. It can with no great error be

regarded as constant, equal to the thickness at the plate edge, h_R . The reasoning and line of action is otherwise almost identical to that carried out prior to the jump. The velocity outside the viscous boundary layer is assumed equal to the bulk velocity of the fluid immediately before the jump, u_{mj} .

We find
$$h_c \approx 0.769k \sqrt{\frac{u_{mj}}{\nu(r-r_j)}} Pr^{0.48}$$

$$Nu \approx 0.769h_R \sqrt{\frac{u_{mj}}{\nu(r-r_j)}} Pr^{0.48}$$

Region 6, $r_j < r < r_T'$. This corresponds to region 3 before the jump. Analysis gives

$$h_c = 3.46 \frac{k}{h_R}$$

$$Nu = 3.46$$

Region 7, $r > r_T'$. This corresponds to region 4 before the jump and

$$h_c = \frac{105}{31} \frac{k}{h_R}$$

$$Nu \approx 3.39$$

Figure 5 shows a plot of the theoretical heat transfer coefficient computed from the preceding formulas. After the *constant value* in the stagnation region, it *increases* to a local peak where a new correlation takes over. The peak arises because these two correlations do *not exactly match* at the point r_s . Outside of this point the convection coefficient *decreases* to a local minimum where the thermal boundary layer reaches the fluid surface, r_T . This *reduction* in the heat transfer coefficient due to the *increasing* thickness of the thermal boundary layer is a *general trend*, also attributable to other flows. After r_T this boundary layer is equal to the film thickness, which in this region ($r_T < r < r_v$) *decreases*, manifesting itself as an *increase* in the convection coefficient. In this way there is a sudden change in the development of the thermal boundary layer at r_T and the curves for h_c before and after this

point cannot (again) be evenly merged. After r_v the film thickness continues to decrease, reaches a *minimum* and *increases* towards the hydraulic jump. This is reflected as a continuing *increase* in h_c till it reaches a *maximum*. Towards the hydraulic jump it then *decreases* due to the increased thermal boundary layer (following the film thickness). At the jump it *drops sharply*, for so to *decrease* inversely proportional to the square root of its radial location, up to $r_{T'}$. In this theory there is no influence from any possible increase of turbulence associated with the hydraulic jump. Finally, at $r_{T'}$ it adjusts to an almost *constant value* (since the film thickness is almost constant) which changes minutely at r_v .

The value of the local convective heat transfer coefficient is in the same order of magnitude as experienced in rapid solidification processes. Contrary to what was initially believed, it is *not maximum in the stagnation region, but at the point of minimum film thickness*. Mathematically it can be shown that this lies *between* r_v and r_j when a third order polynomial is assumed for the velocity profile. This feature remains true for all practical purposes. Analysis yields that the stagnation heat transfer coefficient is largest only in flows with extremely high Reynold numbers, which one in practice doubtfully will achieve. Hypothetically, for tin flow through an 8 mm nozzle the heat transfer coefficient is greatest in the stagnation region when the nozzle velocity increases to about 16000 m/s ($Re_j = 5.5 \cdot 10^8$).

Wang et al. (1993) have also confirmed that the heat transfer coefficient can be greatest outside the stagnation region. This was, however, for gas jets at low flow rates ($Re_j < 10$) where natural convection becomes preponderant.

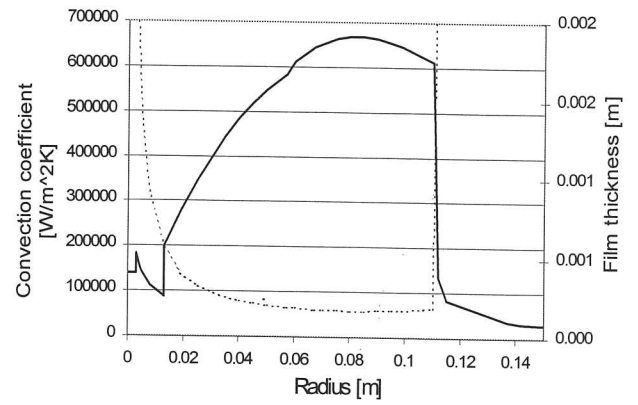


Figure 5 Example of analytically computed local convective heat transfer coefficient (solid line) for tin at 341 °C impinging on a flat horizontal plate and corresponding film thickness. Flow through 8 mm nozzle, $1.0464 \cdot 10^{-4} \text{ m}^3/\text{s}$, 92 mm free fall. Values are related to the *local* bulk tin temperature (a function of radial location).

Comparison of Heat Transfer in Water and Liquid Metal

For sake of comparison, some heat transfer coefficients for *water* and *liquid tin* will be calculated.

First we consider an impingement flow on a flat, horizontal plate and restrict the study to *region 4*, $r_v < r < r_j$, in Figure 4, i.e. after both boundary layers have reached the fluid surface. The wall temperature is assumed constant and there is no heat transferred from the liquid surface. The jet has a diameter of 10 mm and a velocity of 2 m/s. The temperature profile through the cross-section of the fluid is described by a third order polynomial.

For *water* at 50 °C we have

$$h_c = \frac{105}{31} \frac{k}{h} = \frac{105}{31} \frac{0.641}{\frac{175}{136} \frac{5.54 \cdot 10^{-5}}{0.01^2} r^2} \approx \frac{6.1}{r^2} \text{ W/m}^2\text{K}$$

For *liquid tin* at 300 °C we accordingly have:

$$h_c = \frac{105}{31} \frac{k}{h} = \frac{105}{31} \frac{32.6}{\frac{175 \cdot 2.41 \cdot 10^{-7}}{136 \cdot 0.01^2 \cdot 2} r^2} \approx \frac{7120}{r^2} \text{ W/m}^2\text{K}$$

In order to display definite values, these coefficients can for example be evaluated at the location $r = r_v$. For water we then get

$$r_v = 0.032 \text{ m}, \quad h_c = 6020 \text{ W/m}^2\text{K}$$

and for tin

$$r_v = 0.195 \text{ m}, \quad h_c = 187560 \text{ W/m}^2\text{K}$$

For a fully developed laminar flow in a pipe (2 m/s, 10 mm diameter pipe), we have $Nu = 3.66$ (regardless of diameter and velocity).

With liquid tin under the same conditions, the flow is turbulent and the following correlation can be used (Seban and Shimazaki, 1951):

$$Nu = 5 + 0.025(Re Pr)^{0.8}$$

In both cases $h_c = \frac{Nu \cdot k}{d}$ (d is the pipe diameter), which gives

$$h_c = 235 \text{ W/m}^2\text{K} \quad \text{for water}$$

$$h_c = 38170 \text{ W/m}^2\text{K} \quad \text{for tin}$$

This clearly shows that liquid metal flows are endowed with qualities that lead to unsurpassed convective heat transfer coefficients. It is also clear that impinging jets, regardless of fluid, exhibit high heat transfer coefficients compared to other flows (at least before a hydraulic jump).

Another point worth mentioning is that for fully developed *laminar* flows - region 4 (the boundary layers occupy the whole cross-section of the flow) - liquid metals *show the same characteristics as other fluids*. From a practical point of view this means that their heat transfer coefficients can be calculated using *general correlations* ($Nu = f(Re, Pr)$) for the

specific flow type hopefully found in a heat transfer handbook. *Turbulence*, however, influences the flow in a manner heavily dependent on fluid properties. The turbulent contribution to the effective thermal conductivity, for example, is different in flows with water and liquid metal, even if the Reynold numbers are the same.

5. COMPUTATIONAL AND EXPERIMENTAL RESULTS

A finite-difference mesh was constructed inside the four analytical surface profiles. The final mesh comprised 25 nodes evenly spaced across the film thickness before the jump and around 50 after (refined towards the plate). Around 300 nodes were used in the radial direction. Refining this mesh did not seem to influence any results. The plate beneath the liquid was not modelled. Based on the simulated heat flux, the temperature anywhere in this plate can easily be computed manually. Both laminar and various turbulence models were used, see Table 2. The isothermal flow was first computed, and afterwards, "on top" of this converged flow field, the calculation of the energy equation was performed. This is permissible in our case, since the temperature variation is small (maximum 20 °C) and the physical properties influencing the flow are not heavily temperature dependent. Thus there is a coupling from the flow field to the temperature field, but not vice versa. Constant properties simplify both the numerical and analytical calculations. Viscous heating is considered negligible and omitted in all simulations.

The flow simulations could take up to 40 CPU hours on a Silicon Graphics Indigo 2 computer, whereas thermal simulations used up to 15 CPU hours.

There was in general some difficulty involved in making the simulations converge. Even though steady-state conditions are analyzed, both the flow and thermal simulations had to be performed time-dependently in order to converge satisfactory, often with small time steps involved. Manipulation of time steps and underrelaxation factors was also necessary. The computer's response to these manipulations was fairly slow, making this a tedious job.

The computational difficulties are believed to hail from two different sources. First, the mesh used is not optimal, but a compromise between accuracy and CPU time. A rather fine mesh through the thickness of the film was chosen at the sacrifice of more cells in the radial direction. This led to long thin cells in some areas and subsequently anisotropic coefficients in the discretization equations, which in turn impedes convergence.

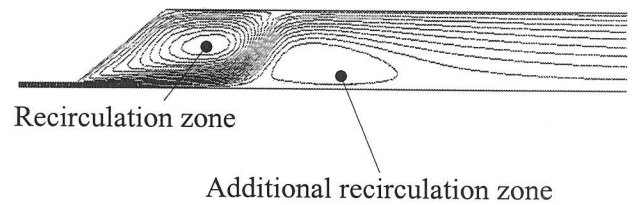
Secondly, the very nature of the flow itself leads to numerical problems. The abrupt increase of the flow's cross-section at the jump results in eddies and complete reorganization of the flow. This reorganization may need to take place over a longer distance than that available between the jump and the plate edge (outlet boundary). Radial gradients can in such a case still be large at the outlet and fluid can also be pulled into the FLUENT domain, which greatly affects the results. In some situations the calculation will not converge at all.

Computational model	
Flow	Heat Transfer
laminar	laminar
standard $k-\varepsilon$	standard $k-\varepsilon$
RNG, nonequilibrium	RNG, nonequilibrium
RNG, nonequilibrium	laminar
RNG two-layer	RNG, two-layer
RNG, standard wall function	RNG, standard wall function
RNG, standard wall function	RNG, no wall function
RNG, nonequilibrium Differential viscosity model <i>off</i> (on in all other cases)	RNG, nonequilibrium

Table 2 Summary of test computations performed in FLUENT for tin flow through 8 mm nozzle, $1.046 \cdot 10^{-4} \text{ m}^3/\text{s}$, 92 mm free fall. Surface profile is from the *turbulent Bernoulli model*.

The laminar computations did in general not converge sufficiently well. One noticeable difference

could be observed between the various flow patterns in the turbulent simulations, namely the occasional existence of *two* recirculation zones after the jump. All laminar and turbulent computations predict *one* such zone in the jump region between the incoming "jet" and the fluid surface. All laminar flows show an additional zone after the breakdown of the jet, extending to the outlet and pulling fluid of an undefined temperature into the domain. *Some* turbulent simulations exhibit a similar zone, but more locally confined - not extending to the outlet. See Figure 6 for an example. This feature is not always existent, but seems to depend on the surface profile and jump position.



a) Watson turbulent surface profile



b) Bernoulli turbulent surface profile

Figure 6 Example of stream functions from FLUENT simulations of prescribed surface profiles with tin. Computations are with the $k-\varepsilon$ turbulence model.

The FLUENT simulations inside the prescribed surface profiles showed little difference in temperatures between the various thermal computations performed on the same flow field.

It can be concluded that different procedures for calculating heat transfer on an already established flow field *do not* result in significantly different heat flows from the fluid to the surface beneath. The temperatures in the system are more dependent on the technique used for calculating the *flow field*. Supercritical flows (before the jump) seem to be the exception, yielding practically no variation in temperature, *regardless* of flow or heat transfer model chosen.

Simple analytical calculations for temperature and heat flux can easily be performed with help of the theory from section 4. The convective heat transfer coefficients are known from the plate center to the edge and the problem is solved by iteration. The major difficulty is to find an expression for the fluid's bulk temperature as a function of radial distance from the plate center. One simple suggestion is to use a *linear* variation of the bulk temperature with radius, i.e. from the known inlet temperature (valid up to r_T) to the initially unknown outlet temperature. The outlet temperature is the only unknown and is found by equating the total heat flow to the plate with the heat loss in the tin. This simplifies analytical calculations and it is assumed that it does not greatly affect the correctness of the final results. The total heat flow to the plate can be found by dividing the plate into annular regions, computing the heat received by these and finally adding them together.

Figure 7 shows an example of computed and measured surface temperatures in the plate.

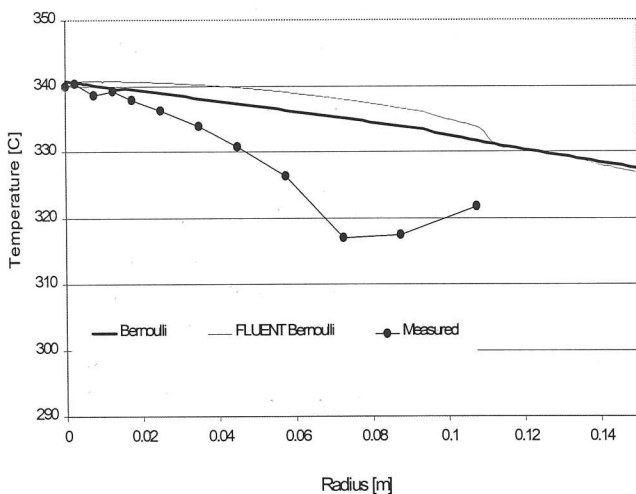


Figure 7 Tin flow through 8 mm nozzle, $1.046 \cdot 10^{-4} \text{ m}^3/\text{s}$ flow rate.

The FLUENT simulations always predict a slightly higher wall temperature than the analytical calculations, except near the plate edge. This can be associated with the fact that the latter are based on an *imposed linear temperature drop* in the fluid. Such a simple assumption is most likely not correct, and will

affect the local heat flow and resultant wall temperatures. A slacker temperature drop will increase the wall temperature (remember that the convective coefficients are temperature independent). If the computed convection coefficients from the fluid to the plate are too low, the wall temperatures will also be too low. Downstream of the jump, the analytical temperatures tend to cross the "FLUENT temperatures". The overall heat loss from the fluid can for this reason turn out very similar in all computations.

Table 3 compares experimental and computed temperatures.

Surface profile	Computation al model for flow and heat transfer	Temperature [°C]		Heat loss in tin [W]
		Inlet	Outlet	
Bernoulli, turbulent	laminar	339	325.3	2898
	standard $k - \varepsilon$		325.1	2940
Watson, turbulent	laminar		325.4	2877
	standard $k - \varepsilon$		325.2	2919
Buyevich and Ustinov, laminar	analytical		325.2	2918
Bernoulli, turbulent	analytical		325.2	2912
experimental measurements		342	323.0	4019

Table 3 Comparison of results from FLUENT simulations, analytical model computations and measurements. Tin flow through 10 mm nozzle, $1.1928 \cdot 10^{-4} \text{ m}^3/\text{s}$, 28 mm free fall. Calculations do not include heat transfer from fluid surface. Calculations do not include heat transfer from fluid surface. Cooling air temperature is 140 °C with an overall heat transfer coefficient at the underside of the fluid of 218 W/m²K. The inlet tin temperature varied somewhat in the experiments and the instantaneous value shown here does not exactly

match the time-averaged value used in the computations.

6. DISCUSSION

The discrepancy between measured and computed temperature can rely on various factors:

- *Positioning of thermocouples.* A thermocouple lacking proper contact with the plate, will on the top face report temperatures exceeding the plate temperature. The opposite will be the case on the bottom face. Inspection of the thermocouples' location, combined with the fact that the topside temperatures are lower than calculated, indicates that this is not a likely explanation
- *Cooling air temperature and flow rate.* The error in measured values is considered small and irrelevant.
- *Radial heat flow.* The computations assume a one-dimensional heat flow through the plate. Based on the experiments, the radial heat flow is maximum 0.1% of the transverse heat flow, i.e. the one-dimensional simplification does not introduce significant error.
- *Oxide on tin surface.* It is not completely understood how this affects the flow of the tin. In any case it was only visible after the hydraulic jump and can not explain the discrepancies before this point.
- *Flow pattern.* It is possible that the thermocouples disturb the flow. A possibility is that the tin somehow splits up and "goes around" some thermocouples, afterwards merging into one flow again. This can explain the local dips and peaks in the wall temperature. Insulating air pockets and/or oxide formation can theoretically also obstruct the heat transfer to the thermocouples.

Warping in the plate can disturb the flow and measurements. Inspection of the plate after the experiments revealed it was slightly concave - about 1 mm out of plane. The horizontal position of the plate during experiments was checked with a level and is considered satisfactory.

- The calculations assume a perfectly symmetric and even tin jet. In reality it was uneven and sometimes sputtering. This extra turbulence may

increase the heat transfer in the stagnation region. The coinciding results from turbulent and laminar FLUENT simulations in the whole region before the jump, indicate that liquid metals are practically insensitive to these parameters.

- *Material properties.* These are fairly well documented for the materials in question.
- *Cooling air temperature, flow rate and humidity.* The error in temperature measurements is considered to be small. Leakage of cooling air (about 10%) was evident in some experiments. A decrease in air flow will reduce the heat transfer through the plate. Humidity variations in the air may affect heat transfer.
- *Heat exchange with surroundings.* Some heat will certainly be exchanged, leading to a slight increase in the plate and cooling air temperature.

7. CONCLUSION

As mentioned in the introduction, the primary aim of this work was to find a reliable tool for simulating casting of ferroalloys in open moulds, with special focus on the impingement region. This is believed to be where heat and mass transfer is greatest. FLUENT's *Volume-of-Fluid* model meant for such free surface flows proved incapable of simulating such a flow reliably.

It was therefore necessary to attack the problem from a different angle. Theoretical models for describing the surface profile of the flow, including a hydraulic jump, were studied. In addition, a new simple model based on Bernoulli's equation was developed. Results based on this model were in good agreement with the other theories and practical experiments.

FLUENT simulations of the flow inside these profiles showed that the maximum velocity in the liquid film near the stagnation region is not at the free surface. This invalidates the assumptions made for the velocity profile in many analytical models and corroborates the measurements of Stevens and Webb.

Flow simulations with FLUENT must be performed *turbulently* in order to get sensible flow patterns *after* the jump. Laminar simulations predicted a

recirculation zone extending to the plate edge where fluid was pulled into the computational domain - resulting in erroneous temperature calculations. Laminar results before the jump, however, are believed to be realistic. There is in general *little difference* between the thermal results in this region *regardless of flow and heat transfer model used*. Thus, for liquid metals, only obvious *large-scale* variations, such as recirculation zones, seem to affect the final temperature field. There is no great difference in the FLUENT simulations of the four surface profiles analyzed here.

Measured temperatures in experiments with liquid tin were consistently lower than simulated. Measured heat flow usually exceeded predicted values, particularly in the stagnation region. Similar results have been widely reported in systems with gas and water, and are found to vary significantly with free stream turbulence intensity. As mentioned, this does not seem to be valid for liquid metal simulations. The reasons for the deviations between experiments and computations are not fully understood. The flow may be too complex for a two-dimensional model to handle, which additionally of course is an idealization/simplification of a real flow. Systems involving liquid metal heat transfer are vulnerable to disturbances and if such exist, measurements can easily deviate from calculations.

An analytical model for heat transfer was also developed. Wall temperatures calculated in this way were a little lower than the numerically simulated temperatures, but still higher than those predicted by FLUENT. The comparison of results shows good promise that this analytical method can act as a useful alternative to the much more demanding numerical simulations. The model was developed on the basis of laminar flow, but numerical simulations show it is just as applicable to turbulent flows. It can be further developed to include inclined jets and jets impinging on moving surfaces, as in strip casting.

The analytical models shows that in all practical flows with liquid metals the heat transfer coefficient is largest where the film thickness is at a minimum and *not* in the stagnation region. The heat flow rate is all the same greatest in this region, due to the bulk temperature of the fluid being highest there. In traditional casting methods (not rapid solidification)

the resistance to heat flow from the liquid metal will be in the mould and from the mould to the surroundings or a cooling system and not from the metal to the mould. The convective heat transfer coefficients from the metal to the mould are very high, it will thus not be critical which theoretical flow model is used for thermal computations.

Regions with high convective heat transfer coefficients tend to generally also have high shear stress and mass transfer coefficients, i.e. the highest thermal, mechanical and chemical loads usually occur in the same areas.

Only steady state impingement flows were considered, and transient conditions, such as filling a mould, must (at least temporarily) be simulated numerically.

Measurements of an impinging water jet with a laser-Doppler instrument were performed. This is an unacceptable method due to the roughness of the free surface and the large measuring volume in relation to the film thickness before the jump.

Experiments show that the hydraulic jump develops before the flow reaches the edge of the plate and remains quite steady at this position even after steady-state conditions are established. This indicates that plate size does not heavily influence the jump position, agreeing well with the theory of Buyevich and Ustinov and the Bernoulli model developed in the course of this study.

The system studied in this work is steady state impingement flow on a flat plate without accumulation of the fluid, as in the filling of a mould. It is, however, believed to resemble the conditions at the start of a casting in an empty mould, when the strain on the mould is greatest. The heat flow in such transient conditions is probably a little different.

ACKNOWLEDGEMENTS

The development of this new process has been partly financed by the Norwegian Research Council under the EXPOMAT program and this has been of major importance for such a long term and innovative

project. We would also like to thank the personnel at SINTEF for their help with the experiments.

LIST OF SYMBOLS

a	radius of impinging jet [m]
d	diameter [m]
g	acceleration of gravity [m^2/s]
h	film thickness [m]
h_c	convective heat transfer coefficient [$\text{W}/\text{m}^2\text{K}$]
h_R	film thickness at plate edge [m]
k	thermal conductivity [W/mK]
Nu	Nusselt number
Pr	Prandtl number
r	radial location [m]
R	plate radius [m]
r_j	radial location of hydraulic jump [m]
r_T	radial location where thermal boundary layer reaches fluid surface before hydraulic jump [m]
r_v	radial location where viscous boundary layer reaches fluid surface before hydraulic jump [m]
r_T'	radial location where thermal boundary layer reaches fluid surface after hydraulic jump [m]
r_v'	radial location where viscous boundary layer reaches fluid surface after hydraulic jump [m]
Re	Reynolds number
Re_j	Reynolds number in impinging jet, $\frac{2au_j}{v}$
Re_r	Reynolds number in radially spreading film
T_∞	bulk temperature of fluid [$^\circ\text{C}$]
u	velocity [m/s]
u_j	jet velocity [m/s]
$\dot{V}$	volumetric flow rate [m^3/s]
δ_T	thermal boundary layer thickness before hydraulic jump [m]
δ_v	viscous boundary layer thickness before hydraulic jump [m]
δ_T'	thermal boundary layer thickness after hydraulic jump [m]
δ_v'	viscous boundary layer thickness after hydraulic jump [m]
$\phi(\eta)$	transformation function
η	dimensionless variable
ν	kinematic viscosity [m^2/s]
ρ	density [kg/m^3]
σ	surface tension [N/m]
θ	wetting angle [degrees]

REFERENCES

- Alekseenko, S. V., Nakoryakov, V. E., Pokusaev, B. G.: Wave Flow of Liquid Films, Begell House, New York, 1994.
- Bakken, J. A.: Metallurgiteknikk 2, Norwegian University of Science and Technology, Trondheim, 1993 (in Norwegian).
- Brdlik, P.M., Savin, V.K.: Heat transfer between an axisymmetric jet and a plate normal to the flow, Inzhenerno-Fizicheskii Zhurnal, Vol. 8, pp. 156-155 (1965).
- Buyevich, Y. A., Ustinov, V. A.: Hydrodynamic Conditions of Transfer Processes Through a Radial Jet Spreading Over a Flat Surface, Int. J. Heat Mass Transfer, Vol.37 No. 1, pp. 165-173 (1994).
- Fluent (computational fluid dynamics code), various versions up to version 4.5, Fluent Incorporated, New Hampshire, USA.
- Haaland, H., Mühlbradt, T., Bakken, J.A.: Flow and Heat Transfer in Elkem's New Continuous Cast-to-Shape Process, Proceedings of the 54th Electric Furnace Conference, Dallas, pp. 169-175, 1996.
- Schach, W.: Umlenkung eines Flüssigkeitsstrahles an einer ebenen Platte, Ing.-Arch., Vol. 5, pp. 460-467 (1934).
- Schlichting, H.: Boundary-Layer Theory, McGraw-Hill, New York, 7th edition, 1979.
- Seban, R.A., Shimazaki, T.T.: Heat Transfer to a Fluid Flowing Turbulently in a Smooth Pipe with Walls at Constant Temperature, Transactions of ASME, Vol. 73, p.803, 1951.
- Watson, E. J.: The radial spread of a liquid jet over a horizontal plane, Journal of Fluid Mech. Vol. 20, pp 481-495 (1964).