Thermal Simulation of a Heat Pipe Tempered Injection-Mould Tool

S. Kartelmeyer[1*], L. Fromme[1], V. Hüttemann[1], E. Moritzer[2], C. Jaroschek[1]
[1]University of Applied Sciences, Faculty of Engineering and Mathematics, Bielefeld, Germany
[2]Paderborn University, Department of Plastics Engineering (KTP), Paderborn, Germany
*Corresponding author: stephan.kartelmeyer@fh-bielefeld.de

Abtract

The Bielefeld University of Applied Sciences [FH Bielefeld] developed an injection-moulding tool which is tempered by using heat pipes instead of common water based cooling channels. The performance of each heat pipe is precisely determined and displayed as a function. By using the individual heat transfer function of the heat pipe, these can be represented within a COMSOL based heat transfer simulation.

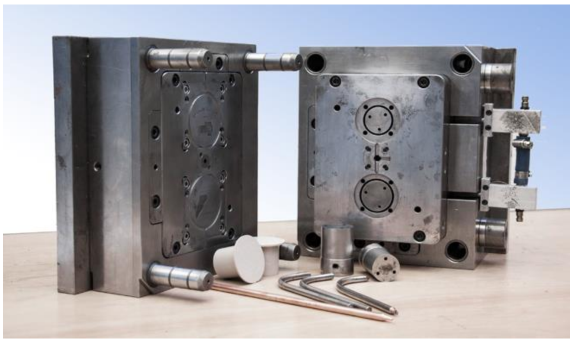

Figure 1 Heat pipe tempered mould

The COMSOL model has been validated by comparing real experiments and the simulation results which only show slight differences in the temperature distributions inside the injection-mould.

Keywords

Plastic, polymer, injection-moulding, tempering, heat pipe, water, cooling medium, heat transfer, temperature, validation

Introduction

In mass production of consumer goods, plastic injection-moulding is the method of choice. By injecting molten plastic into a steel cavity complex parts can be produced rapidly and in big quantities. In order to do so the molten plastic needs to be cooled down within a certain amount of time. The shorter the cooling time, the more economical the process. The conventional cooling method is to use cooling channels running with water. This method seems easy but has disadvantages like rust and corrosion slowly clocking the channels [1] [2]. Furthermore, the freedom of design is highly limited because of cooling channels only running perpendicular to each other. Modern additive manufacturing processes create a greater freedom of design but are not standard yet in every tooling department. The problem of rust and corrosion is still present.

To counteract these downsides, FH Bielefeld is researching a new approach of tempering injection-moulding tools [3]. The use of heat pipes can offer a huge optimization potential when used in a injection-moulding tool instead of common water channels.

Hermetically closed heat pipes were originally developed for aerospace purposes and are today a common way of cooling electronic devices like laptops or even smartphones.

These heat pipes are filled with a fluid which transfers the energy along the pipe. In order to do so heat is applied to one end of the heat pipe. This causes the fluid to phase change from liquid to vapour and to absorb the energy by the latent heat of evaporation. The vapour flows along the heat pipe until it reaches the other end of the heat pipe. This end is typically cooled by air or water. The vapour will then condensate on the cool inner walls of the heat pipe, submitting energy through the latent heat of condensation. To create a circular process, the liquid is transported either by a capillary structure or gravitational force, returning it back to the evaporator zone and starting the process again [**Fig. 2**] [4].

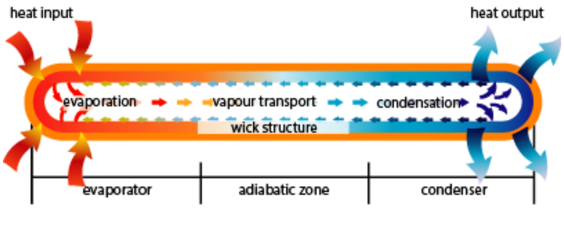

Figure 2 heat pipe working principle

Most heat pipes use water as fluid. So evaporation usually starts at 373 K or 100°C. To be applicable to lower temperatures the heat pipes use a low pressure (approx. 25mBar) allowing the water to evaporate at around 303 K or 30°C.

Theory and Experimental Set-up

Common simulation software for thermal analysis of plastic injection-moulding focuses mainly on conventional water based cooling. Todays CFD simulations can easily be applied to the cooling channels of injection-moulds. Due to the very complex inner phase changes and vapour transports, simulating a heat pipe is very difficult. The real performance of a heat pipe basically depends on length, diameter, pressure, amount and type of fluid and the capillary structure. So a lot of factors determine how a specific heat pipe works. Simulating this is theoretically possible but to time consuming and difficult for standard injection-mould manufacturer.

The approach of FH Bielefeld is to work with a heat pipe as a “black-box”. A test station was designed to measure the heat transfer capacity of heat pipes.

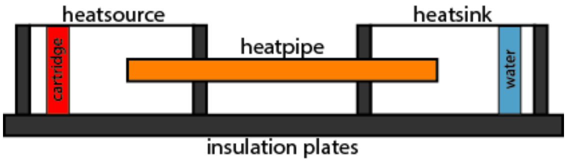

Figure 3 heat pipe test station

The test station consists of two blocks with the heat pipe mounted between them. One acts as a heat source, the other as a heat sink. Cold water is used to keep the heat sink at around 293 K (20°C) while a heat cartridge heats up the source side to a specific temperature (e.g. 500K). When both blocks have levelled their temperature, the cooling water is ejected from the system allowing the heat pipe to transfer energy from the heat source into the heat sink [5].

With the mass m and the specific heat capacity c_p, the change of the internal energy ΔE_i [J] is calculated.

$$\Delta E_i = mc_p\Delta T$$

The heat transfer is measured until a stationary temperature is reached. The result is the heat transfer capacity in $[\frac{J}{s} = W]$.

This process is repeated for several start temperatures of the heat source to develop a map of heat transfer capacity depending on the heat source temperature and the temperature difference between both blocks. A software specifically designed in the project resolves the measurements from environmental losses and creates a three dimensional plot of the heat transfer capacity [**Fig. 4**].

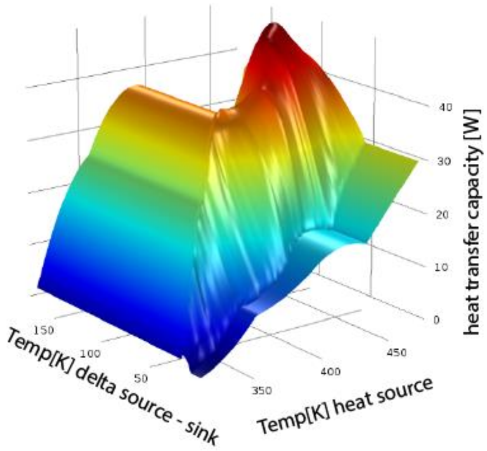

Figure 4 3D heat transfer capacity plot

Such a 3D plot represents the individual heat transfer capacity of each heat pipe. The plot shows the heat flux which is transferred by the heat pipe in dependence of the temperature of the heat source and the temperature difference between source and sink.

To validate the simulation two heat pipes were installed into the injection-moulding tool. For each heat pipe the individual 3D plot was created. The two cavities inside the tool have different dimensions, one small, one bigger [**Fig. 5**]. Small cavity cores tend to overheat over time, because a lot of hot molten plastic is exposed to fewer material. As the 3D plot shows the maximum heat flux a given heat pipe can transfer, it helps to determine which heat pipe is used for what purpose. In this case the more powerful heat pipe is installed in the smaller core.

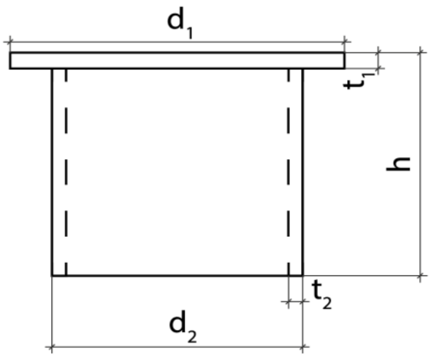

Figure 5 product dimensions

Calculations show that with each injection of plastic, the energy which has to be transferred in order to cool the plastic from 503 K to 353 K is approximately 600 J and 750 J in each core.

The product remains in the mould 16 seconds. This time includes the injection (1.5 s), the cooling time (6.5 s) and the holding pressure phase (7 s).

Calculations of the energy applied within a cycle time of 16 s show that the needed heat transfer capacity of the heat pipes is 37.6 W for the small core and 46.8 W for the large core.

The inserted heat pipes have a maximal heat transfer capacity of around 45 W depending on the temperature.

	Cavity – small [mm]	Cavity- big [mm]
d_1	60	60
d_2	39.8	45.8
t_1	2.5	2.5
t_2	1.4	1.4
h	40	40
Parameters		
Steel 1.2311		
rho	7830 kg/m^3	
k	34 W/(m*K)	
cp	460 J/(kg*K)	
Brinell hardness	970 MPa	
ABS		
rho	1050 kg/m^3	
k	0.18 W/(m*K)	
cp	1300 J/(kg*K)	

Table 1 material properties

Methods and Simulation with Heat Pipes

The 3D plot [**Fig. 4**] shows the effects that a heat pipe has on its surrounding. It's not necessary to simulate the heat pipe with its inner processes anymore thus avoiding the difficulty to accurately formulate the design parameter for e.g, the inner surface for the capillary reflux of condensed water. Only the representative heat flux is implemented into the COMSOL model without a solid model of a heat pipe present.

The boundaries where a heat pipe would be installed are applied with a heat flux node [**Fig. 6**].

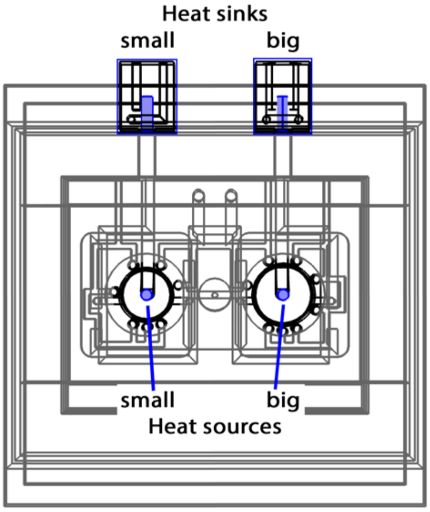

Figure 6 heat sources and sinks

In order for the heat flux to be simulated properly, the temperatures of the sources and sinks need to be measured. Every boundary has its own component coupling measuring the average temperature and providing them as a variable. The variables serve as inputs for the heat transfer function and COMSOL can then determine the exact heat flux for each source and sink.

As the heat flux can be directed in either direction, the setup can be used to transfer heat out of the mould as well as into the mould.

In a typical injection-moulding process, the mould itself keeps its dedicated temperature by a metered flow of water. So prior to start a new process the tool has to reach its operation temperature.
In a heat pipe tempered mould the heat pipes are used to get to the operation temperature. The cooling blocks are therefore equipped with electric heat cartridges which are implemented with a heat rate of 40W each.
To simulate the heating process, the heat flux at the heat pipe boundaries is inverted, transferring energy into the mould.

Heating the mould prior to the production takes 1300 s, after that timestep the heat fluxes are switched into their regular direction, drawing heat from the cores. The heat cartridges are also switched off after this time step.

To create the cyclic injections of molten plastic, an explicit event is used. The event reinitializes the temperature T of the cavity domain to 533 K each 25.5 s which resembles the circle time of the injection-moulding process.

Parallel to the re-initialization of the temperature the material properties of the domain change. This means that while in real life the cavity is empty, the domain has the material properties of air and while there is a product in the cavity the properties of the domain change to ABS plastics.

Simulation Results

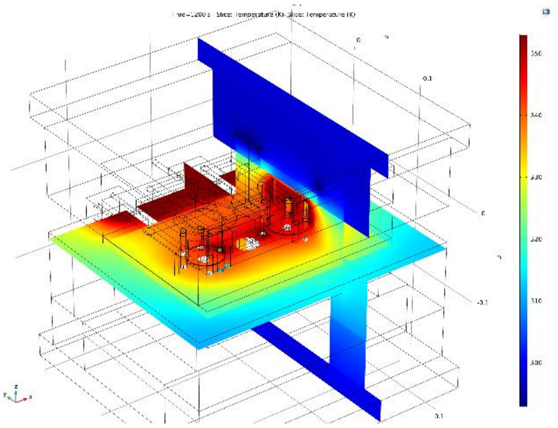

Figure 7 temperature distribution

The simulation shows a temperature distribution of the cavities that can be expected for such an injection-mould [**Fig 7**].

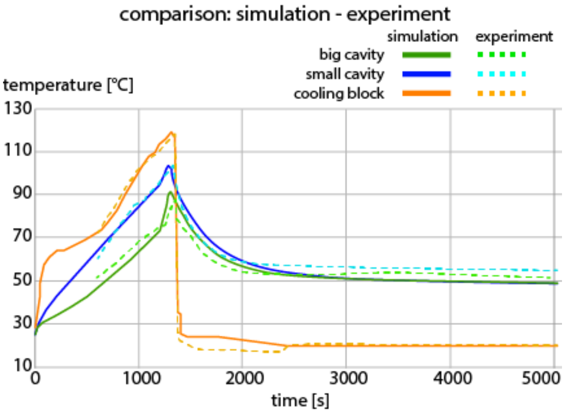

Figure 8 cavity temperature diagram

The diagram [**Fig. 8**] shows the measured cavity temperatures of the COMSOL simulation in comparison to the experimentally measured temperatures of the real mould. The heating process in the first 1300 s is properly simulated and so is the cooling process. After 1300s the cooling process starts and the temperatures drop till they reach a quasi-static level.

The experiment only allowed to measure the temperature at the end of each cycle when the product was ejected while COMSOL can monitor the temperature of the cavities over the whole time. To compare the measurements only the lowest temperature at the end of each cycle has to be taken into account.

Experimental Results

Inside the machine park of FH Bielefeld the real life injection-moulding tool is used to validate the COMSOL simulation results. An Engel victory injection-moulding machine is equipped with the heat pipes, a control unit for the heat cartridges and an infrared thermal camera. The experiment starts with a cold machine and mould. The infrared camera monitors the temperatures of the cavity; individual emission coefficients for each cavity have been estimated before. Starting the experiment, the heat cartridges and plasticizing unit of the machine are turned on to reach their operating temperatures. 19 heat cartridges heat up the cavities and the aluminum cooling blocks of the heat pipes. As mentioned before the process is now in reverse compared to the usual working process. The cold section of each heat pipe now is within the mould and so heat is transferred from the heated aluminum blocks into the mould. With this setup the mould and machine take 20 minutes to reach their operating temperatures: Mould cavity 353 K / 80 °C and plasticizing unit 503 K / 230 °C. This is a little faster than calculated with COMSOL.

Injection moulding starts while the heat cartridges are still switched on and the water cooling system is off. Four cycles are completed before switching from heating to cooling. The highly dynamic heat transfer capabilities of the heat pipes would otherwise start too early and cool the cavities to quickly.

With the process set up and the cooling fluid temperature leveled, the machine runs through 150 production circles while the thermal camera monitors the temperature of each cavity after ejecting the produced parts [**Fig. 8** – dotted lines].

The diagram shows the heating and cooling capabilities of the heat pipes, transferring heat into the cavity at first and transferring it out afterwards in the process cycles to maintain a stabilized temperature level over time. Technically the cavity temperature is slightly too low for ABS plastics. The reason here is due to a slightly long cycle time caused by a not optimized ejection process. Typically, the product should be ejected by a robot system in order to minimize the cycle time. This system was unfortunately not available in the experiment, so the cycle time was longer allowing the heat pipes to extract too much heat from the mould. In the future the process will be optimized, and an ideal temperature and cycle time can be achieved.

Discussion

The numerical simulation of a heat pipe with all its inner processes is very complicated and intense. With the "black-box" method it is possible to simulate a scenario with built-in heat pipes without designing the inner vapour processes. The simulated and experimentally measured temperatures of the cavities show only slight differences. The quasi static temperature only has a few degress offset which is still within a delta of 5 degress celcius. This deviation may be explained with inaccurate heat transfer coeficents entered into the software.

In addition all material properties are assumed as to be constant. Using temperature dependent properties for steel and ABS plastics could also help to increase the simulation precision.

Conclusion and Outlook

The experimental results show that a heat pipe based tempered plastic injection tool is possible and running. Heat pipes are a low cost and low energy alternative to conventional tempered systems.

Also the simulation of the experimental heat pipe capacity, with a focus on the heat transfer, is working as well. Practical results indicate that the the COMSOL simulation is close to the real process and that the variance is very small.

These results enable a thermal design of a heat pipe tempered tool before producing a prototype. It is possible to design the simulation parallel to the mechanical design.

Acknowledgements

The authors would like to acknowledge all the students, who worked inside this project.

Also great thanks are going to the project partner sk werkzeugbau GmbH from Extertal / Germany who supported the tool und and specific expertise regarding moulding technology.

Hasco Hasenclever GmbH + Co KG from Lüdenscheid / Germany and Quick-Ohm Küpper & Co. GmbH from Wuppertal / Germany provided a number of heat pipes for the studies.

SITRAPLAS GmbH from Bünde / Germany provided a number of ABS plastic for the studies.

This work is a project, funded by BMWi ("Development of a new generation of injection moulds without conventional media temperature control").

References

[1] C. Jaroschek, „Heiß oder Kalt, das ist hier die Frage?," *Kunststoffe,* pp. 92-98, 10 2012.

[2] O. Zöllner, „Optimierte Werkzeugtemperierung Bayer," *ATI 1104,* November 1997.

[3] S. Kartelmeyer, C. Jaroschek und E. Moritzer, „Temperieren ohne Wasser - Einsatz von Heatpipes in Spritzgießwerkzeugen," *VDWF im Dialog* , pp. 20-25, 04 2018.

[4] P. Stephan, „Wärmerohre," in *VDI-Wärmeatlas*, Berlin Heidelberg, Springer Verlag, 2013, pp. 1675-1686.

[5] C. Jaroschek, S. Kartelmeyer und V. Hüttemann, „No Cool Molds - Properly Simulated," *Kunststoffe*, pp. 30-33, 08 2017.