Autodesk® Simulation
Moldflow® Insight
2014

Transient Cool: Conformal cooling

Executive summary

The release of Autodesk Moldflow Insight 2014 enhances the 3D transient cool solver by increasing its functionality and versatility. This solver, known as Cool (FEM), supports conventional cooling and rapid heating and cooling processes on conformal cooling channels. Conformal cooling is a new technology being introduced for the first time in the Autodesk Moldflow Insight 2014 release. Conformal cooling channels are where the internal cooling channels are made to a shape that follows the precise geometry of the part in the mold. Modern injection mold manufacturing technologies allow conformal cooling channels to be easily manufactured.

Conformal cooling aids in creating a uniform temperature distribution in the part by targeting hot spots on the part surface with non-traditionally shaped cooling channels in the mold. Ultimately this process results in better quality parts, shortened cycle times, reduced waste and cost reductions. Simulation of these processes requires a fully transient, three-dimensional (3D), time dependent computational fluid dynamics (CFD) solution in the conformal cooling channels. Autodesk Simulation CFD 2014 is used for solving the (3D) flow and thermal solutions in the conformal cooling channels. The added conformal cooling functionality to the transient cool, Cool (FEM) solver is a worthwhile enhancement to Autodesk Moldflow Insight 2014 as more users are taking advantage of conformal cooling. The numerical results predicted by the conformal cooling capability of the transient cool solver agree with those obtained from conventional 1D solutions.

Contents

Introduction

Traditional injection molding design aims to maintain the mold at a constant temperature for the entire injection molding cycle. In order to achieve this, coolant is pumped through the mold cooling channels with constant inlet temperatures.

Traditionally these cooling lines are comprised of a network of holes drilled through the solid mold that are plugged at various positions forcing the coolant to follow a defined path. If more cooling is required closer to the part in a certain region of the mold, the traditional approach is to insert a baffle or a bubbler in this region [1].

Direct metal laser sintering (DMLS) is a new additive manufacturing technology that allows mold makers to manufacture mold inserts which contain conformal cooling channels directly from 3D CAD models. The mold insert is created by melting a layer of powder metal directly on to a previous metal layer with a fiber optic laser. The insert is grown layer by layer around the strategically placed conformal cooling voids. This results in a fully dense metal insert that is then assembled into an injection mold. Traditional cooling lines are used to supply and remove coolant from the insert containing the conformal cooling channel.

These conformal cooling channels replace the conventional cooling lines in the mold and no other change in the injection molding process is required. Conformal cooling channels are proving to also be very popular when used in conjunction with the Rapid Heating and Cooling injection molding process [2]. The use of conformal cooling channels can minimize the distance from the cooling channels to the part cavity, so allowing the mold in contact with the part to more quickly heat up and cool down during each cycle.

In order to utilize the full potential of conformal cooling it is very important that the coolant flow dynamics in the conformal cooling channel be understood. Large hollow voids in the mold insert may not be ideal conformal cooling channels for optimum heat removal. The coolant will always circulate along the path of least resistance through the conformal cooling channel. If the conformal cooling channel has large variations in cross sectional area it is highly likely that the flow will stagnate in some pockets with very little coolant circulating through those regions. Stagnant localized coolant will result in poor heat transfer in this region of the mold. The uneven cooling of the part will result in poor part quality.

When using the DMLS mold manufacturing technique, it is very important that the flow regime in the conformal cooling channel is understood before the mold manufacturing process begins. Simulation of the coolant flow allows the user to understand the consequences of the part and mold design decisions before mold manufacturing begins.

Computational Fluid Dynamics (CFD) is a robust technology that uses numerical methods and algorithms to analyze fluid flows by solving the Navier-Stokes equations. Commercial CFD packages have been in existence for many years and are used to solve these equations. The benefits of CFD technology can be leveraged to design optimum conformal cooling channels for plastic injection molds by customizing the CFD solvers to integrate it with plastic injection molding simulation technology requirements.

Simulation technology

In order to simulate the flow dynamics in the conformal cooling channels, the Autodesk Simulation CFD 2014 solver was used to provide the flow and temperature solutions in 3D conformal cooling channels [3]. Simulations of conformal cooling have been reported in the past, but these simulations have often employed one dimensional (1D) channel representations. The existing Autodesk Moldflow Insight 2013 Cool (FEM) transient mold cooling solution is able to simulate such 1D channel representations.

Autodesk Moldflow Insight 2014 introduces the conformal cooling capabilities of the transient mold cooling module on non-traditionally shaped 3D cooling channels in the mold. The incorporation and application of the Autodesk Simulation CFD 2014 in Cool (FEM) module is discussed. Results from the 3D CFD solution on a 1D cooling channel geometry are compared to those obtained from the current one-dimensional coolant flow solver. Finally, the capabilities of the conformal cooling solver are demonstrated on a fully 3D conformal cooling channel.

Due to the CFD solution being more complex and resource intensive than an ordinary 1D solution in the cooling channels, it is more efficient to use the 1D solution for drilled hole geometries and only use the CFD solution for complex cooling channels. For this reason only imported CAD bodies are used for the initial conformal cooling geometries to be analyzed with the CFD solution. If a 3D CFD solution is required for a 1D geometry cooling system, then the geometry needs to be modeled in 3D in CAD and then imported.

Autodesk Simulation CFD 2014 is used for simulating the fluid flow in the conformal cooling channel and is a finite element based solver requiring tetrahedral element meshes in the solution domain [3]. The tetrahedral element mesh of the cooling channel requires specific parameters, and so a standard tetrahedral element mesh used to model the part cannot be used. Figure 1 shows the new parameters required for conformal cooling meshing.

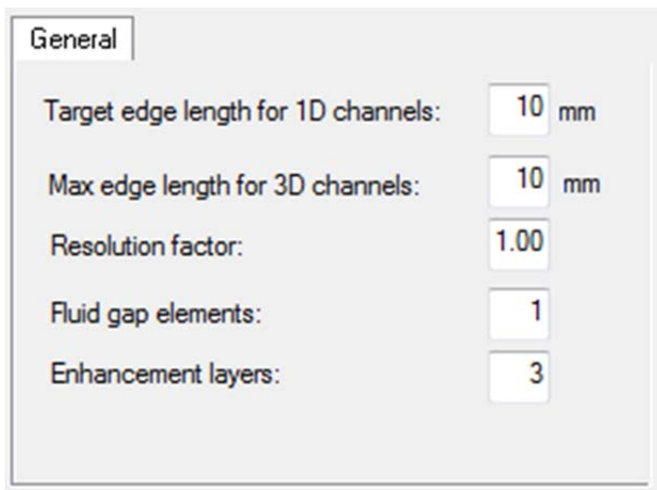

Figure 1: Meshing controls for Conformal Cooling

The “Target edge length for 1D channels” refers to 1D channels that may reside in the same model as the conformal cooling channels and refers to the maximum allowable length of the 1D channel elements. The implementation of Autodesk Simulation CFD 2014 in Autodesk Moldflow Insight 2014 allows the user to specify a maximum surface edge length for the tetrahedral elements on the conformal cooling channel wall by setting the “Max edge length for 3D channels”. This refers to the maximum facet edge length that the user allows on the surface facets of the conformal cooling channel figure 2. The user can also set the “Resolution factor”. The resolution factor sets the curvature control with a lower resolution factor placing more facets on the surfaces of high curvature. Figure 2 shows that the facet distribution is denser on the regions of sharp curvature.

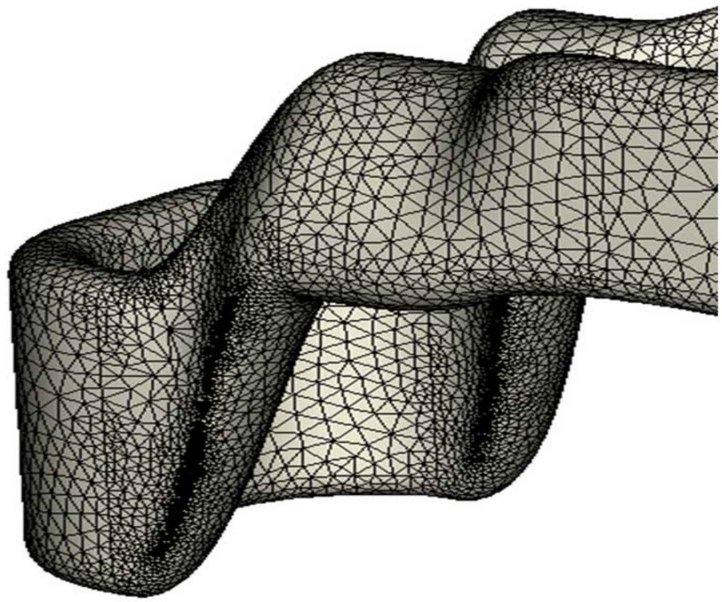

Figure 2: Conformal cooling mesh showing edge length and resolution factor

If the conformal cooling channel has a complex shape extra fluid gap elements for the complex regions can be specified through he “Fluid gap elements” control. This ensures that the mesh has more elements across the cross section of these regions to capture complex flow phenomena such as recirculation zones.

Autodesk Simulation CFD 2014 places enhancement layers close to the boundary wall of the channel, and allows the user the option of specifying the number of enhancement layers used. A maximum of twenty layers is allowed with the default being three.

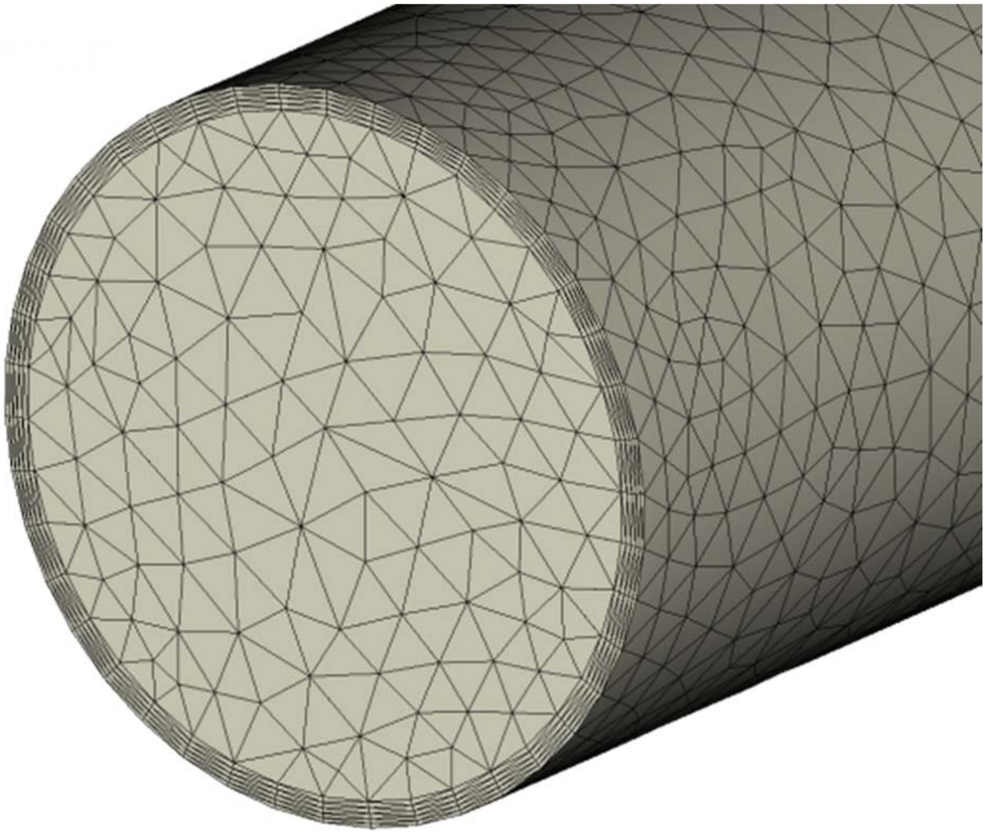

Figure 3: Illustration of enhancement layers along the fluid walls

Enhancement layers are element layers along all the fluid-walls, see figure 3. The enhancement layers augment the mesh to produce a smooth distribution along the walls, which is critical for accurate flow and temperature prediction in the conformal channel. Enhancement layers also ensure adequate mesh across small gaps. The more enhancement layers the model has, the more accurate the solution will be, especially close to the walls. Increasing the number of enhancement layers adds to the computation time.

The fluid flow solver used in Autodesk Simulation CFD 2014 has been fully integrated into the Cool (FEM) transient cooling module and the combined operation is fully automated. The CFD pressure and flow velocity solution is a steady state solution, as is used for the 1D coolant solution. This assumption is valid for thermoplastic injection molding as plastic injection molders are not known to dynamically vary the coolant flow rates during a cycle. This assumption is also correct for rapid heating and cooling, as molders may vary the medium flowing through the channel, i.e. cooling water, hot water or steam; however they will not vary the flow rate of the medium during an injection molding cycle.

The CFD temperature solution in the conformal cooling channels is a transient solution and is solved in conjunction with the mold and part temperature solutions. The cooling channel wall temperature is constantly updated by the mold temperature solution which is used as the boundary condition for the transient CFD temperature analysis of the coolant. The transient CFD temperature solution provides the transient mold temperature solution with a transient convection boundary condition. Once the transient coolant, mold and part temperature solutions have converged for a particular time step the computation advances to the next time step until the entire cycle time is completed.

The Cool (FEM) solver provides the same results set in the mold and part bodies as previous releases. In addition, new result plots have been implemented showing the pressure and velocity of the coolant in the conformal cooling channel. The velocity result needs to be viewed with a cutting plane because of a zero velocity boundary condition on the channel wall. Another option is to show the velocity vector as a dart with the magnitude shown by the color. The velocity can also be shown as a streamline plot. With the streamline plot, the nodes on the inlet face are used as seed points for the streamline and the path of the streamline is traced throughout the conformal channel. This plot tends to highlight recirculation zones quite well. A heat flux plot also shows the heat transfer between the coolant in the conformal cooling channel and the mold.

When viewing the velocity plot in conjunction with the temperature plot the correlation between the temperature on the surface of the conformal cooling channel and the flow inside the channel is evident. The velocity plot is used to identify areas of flow stagnation. The temperature and heat flux plot can therefore be used in conjunction with the velocity plot to guide design changes to the coolant channel within the mold.

In addition to the graphical results that the conformal cooling solver provides, the following detailed information on the conformal cooling channels is presented in the summary file. The solver uses a zero outlet pressure boundary condition in the solution. Hence the calculated inlet pressure is equivalent to the pressure drop across the cooling circuit. The inlet temperature is also specified as a boundary condition and the outlet temperature is calculated by the CFD solver. On the inlet and outlet faces the pressure and temperature quantities are area weighted in order to obtain the pressure drop and bulk temperature rise across the cooling circuits.

Autodesk Moldflow Insight 2013 Cool (FEM) introduced rapid heating and cooling for models with 1D cooling channels. This capability has been extended to include 3D conformal cooling channels in Autodesk Moldflow Insight 2014. A rapid heating and cooling inlet can be specified on the inlet face of the 3D conformal cooling channel. After setting this inlet boundary condition the full rapid heating and cooling cycle process information can be specified on this rapid heating and cooling inlet node. The Conformal cooling- rapid heating and cooling solver follows the same work flow as the 1D rapid heating and cooling solver on the cooling channels and supports both steam and hot water heating and cooling fluids as reported previously [2].

For the 1D rapid heating and cooling solver models, the air in the cooling channels during the air purge stage is treated as a compressible fluid taking the thermodynamic effects of the expanding air into account. Fully compressible CFD solutions can be very resource intensive, taking a long time to converge. These types of analyses can also be very unstable and sensitive to boundary conditions and mesh densities. It was decided that the disadvantages in simulating the air purge stage as a compressible fluid far outweigh the advantages. From tests performed using the 1D solution, it was concluded that the air purge stage did not affect the final temperature result on the part and mold significantly. Therefore, a convective boundary condition is applied to the conformal cooling channel during the air purge stage.

Similar results are provided from the conformal cooling rapid heating and cooling solver as are provided from the 1D channel rapid heating and cooling solver.

Results and Discussion

In order to validate the results from the conformal cooling solver, a conformal cooling channel model was made of the Autodesk Corner Mold. (Figure 4).

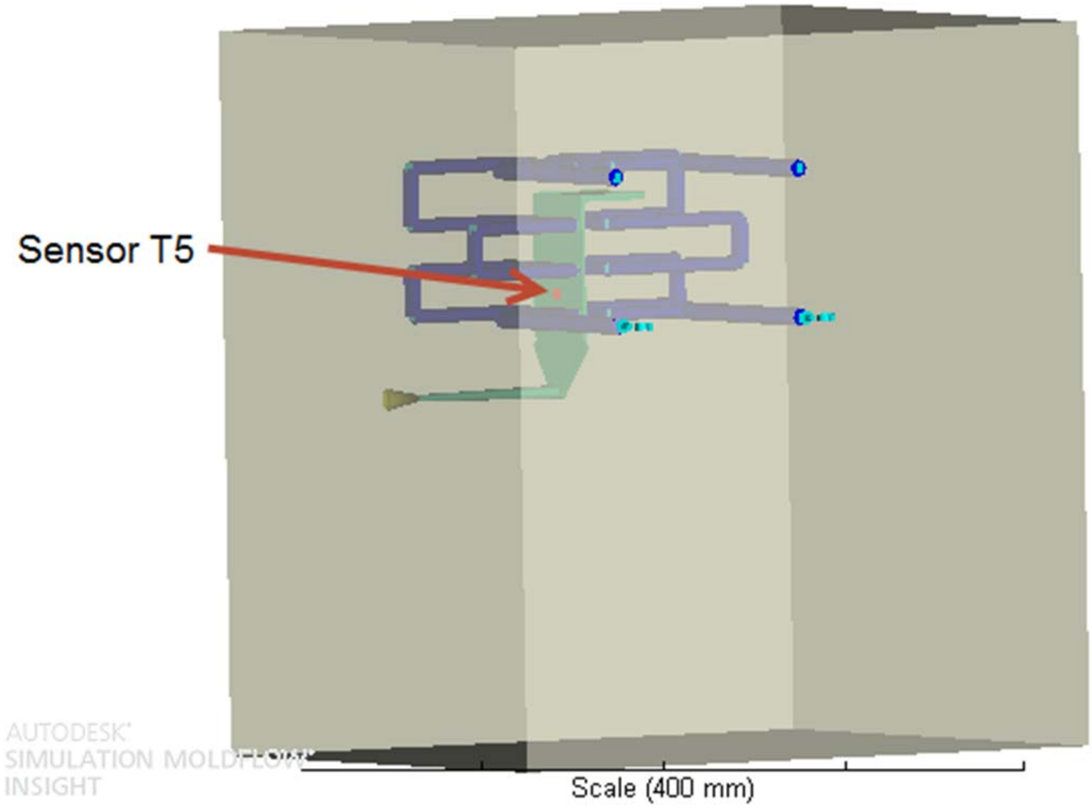

Figure 4: Autodesk Corner Mold

This is a fully instrumented mold and is used in the Autodesk Moldflow laboratory for validation and verification purposes. This mold was used to validate the results of the transient mold cooling module [4]. The cooling channels in this mold are of a 1D nature and normally it is advisable to mesh these channels using 1D channel elements because the 1D cooling channel analysis would be much faster and use much less computer memory than the 3D CFD solution for similar accuracy. However, for the purposes of validating the accuracy of the new conformal cooling solution it was decided to compare the results of these two solution methods.

In order to have an accurate solution, the conformal cooling channels were meshed using 3,619,008 tetrahedral elements. This mesh is approximately 10 times finer than the mesh made with default settings. Identical flow rate boundary conditions were applied to the 3D conformal cooling solution as were applied to the 1D cooling solution.

An accurate pressure solution requires a very fine mesh that is dependent on the conformal cooling channel geometry. This can be very resource intensive resulting in a long run time. If an accurate pressure solution is required, it is recommended that the user tries various mesh control settings and increased numbers of enhancement layers, to find the combination that best suits the conformal cooling channel being analyzed.

Figure 4 shows the position of a sensor (T5) in the Autodesk corner mold. This sensor was used to validate the temperature solution of the Transient cool solver when it was first released in Autodesk Moldflow Insight 2012 [5]. In order to validate the conformal cooling module this sensor position was used to compare the predicted temperature at sensor T5 from the 3D CFD solution to the 1D solution. Figure 5 gives the temperature history at sensor T5 for a stable molding cycle.

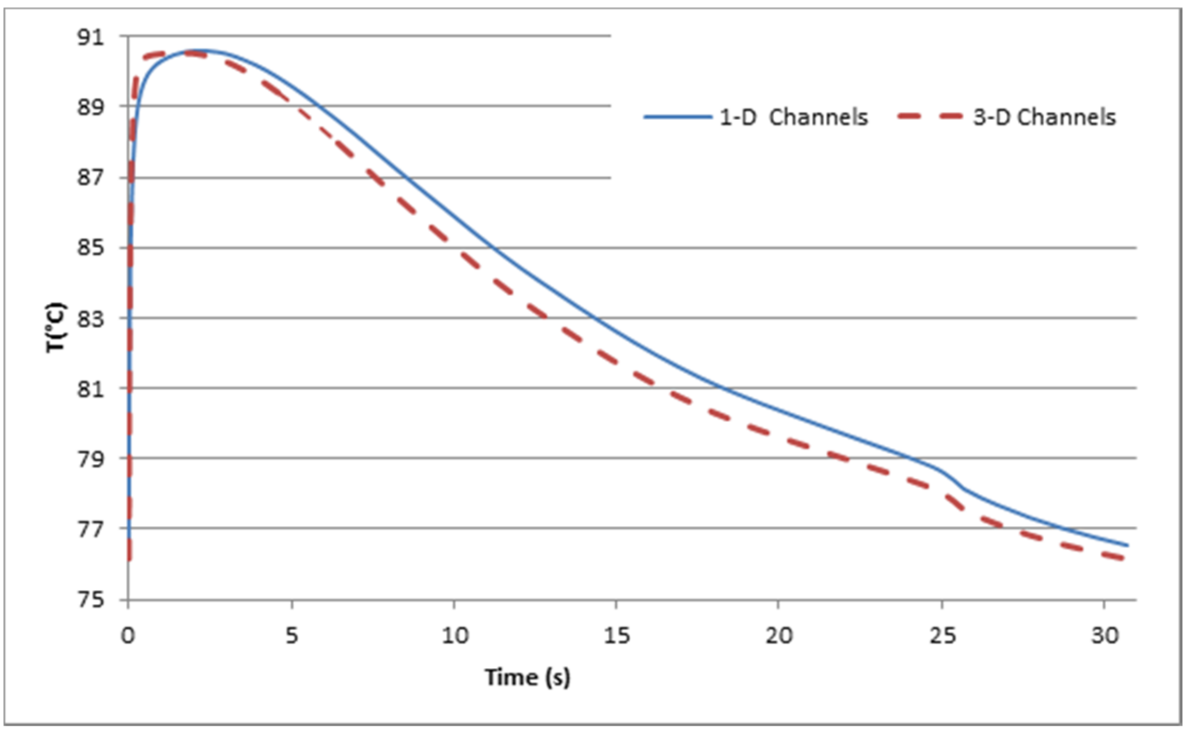

Figure 5: Predicted Sensor T5 results for Autodesk Corner Mold

Figure 5 shows the temperature prediction in the mold is slightly cooler when the 3D CFD solution is used in the cooling channels. Both the 1D solution and the 3D CFD solution maintain an energy balance when doing the analyses and this difference is attributed to the difference in numerical methodologies. In order to maintain an energy balance the 3D CFD solution is removing more heat out the mold than the 1D solution, hence the cooler mold temperature seen in Figure 5. This difference in the results between the two solutions is deemed to be acceptable. The comparison between the 1D solution and experimental data can be found in reference [5].

It should be noted that if the default mesh settings are used for the conformal cooling mesh then the overall solution time for the analysis performed on the Autodesk Corner Mold increased approximately by a factor of 4 when compared to the 1D solution. In order to get the accurate solution shown in figure 5, the conformal cooling mesh is meshed with 10 times more elements in the cooling channel than the default settings produce. This caused the analysis times to increase by at least a factor 10. Therefore, for 1D type channels it is preferable to use the 1D solver.

The real benefits of the 3D CFD solution become evident when arbitrary shaped conformal cooling channels need to be analyzed. The 1D solution cannot be used for these geometries. Figure 6 shows the conformal cooling channels which may be used in a mold used for manufacturing satellite dishes. The conformal cooling channels follow the contour of the part as closely as possible with solid mold sections obstructing the flow where either the feed system or ejector pins are placed. These obstructions have an effect on the flow pattern, which in turn has an effect on the coolant temperature distribution in the cooling channel. The temperature distribution then influences the heat transfer in the mold. Figure 7 shows the conformal cooling channel in isolation without being obscured by the part.

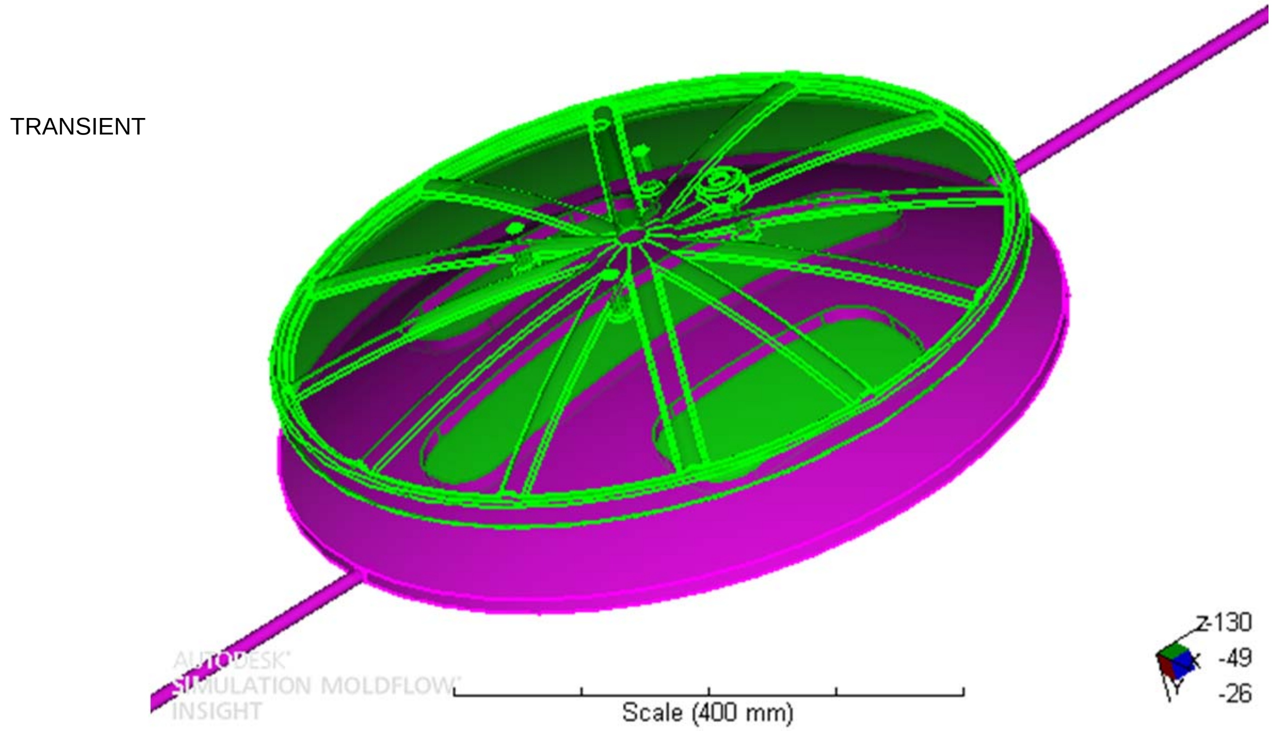

Figure 6: Satellite dish model with conformal channel.

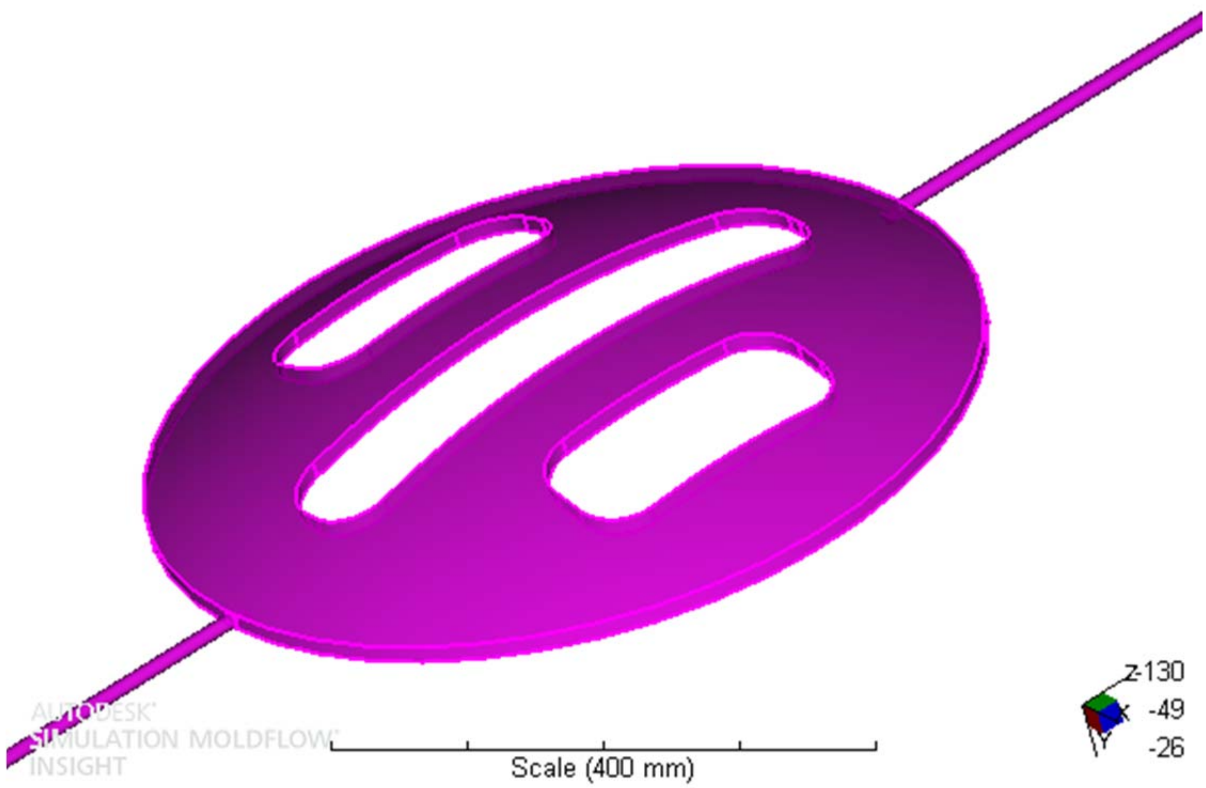

Figure 7: Conformal channel of the satellite dish model

Figure 8 shows the streamline plot of the velocity distribution in the conformal cooling channel. Streamlines close to together signify a higher flow rate though these regions, which means that the velocity is also generally higher. This is confirmed by the color of the streamlines. It can be seen that at the entrance of the conformal cooling channel, on the left of the plot, a large recirculation zone exists, while beyond the obstructions the velocities are very low and streamlines are absent.

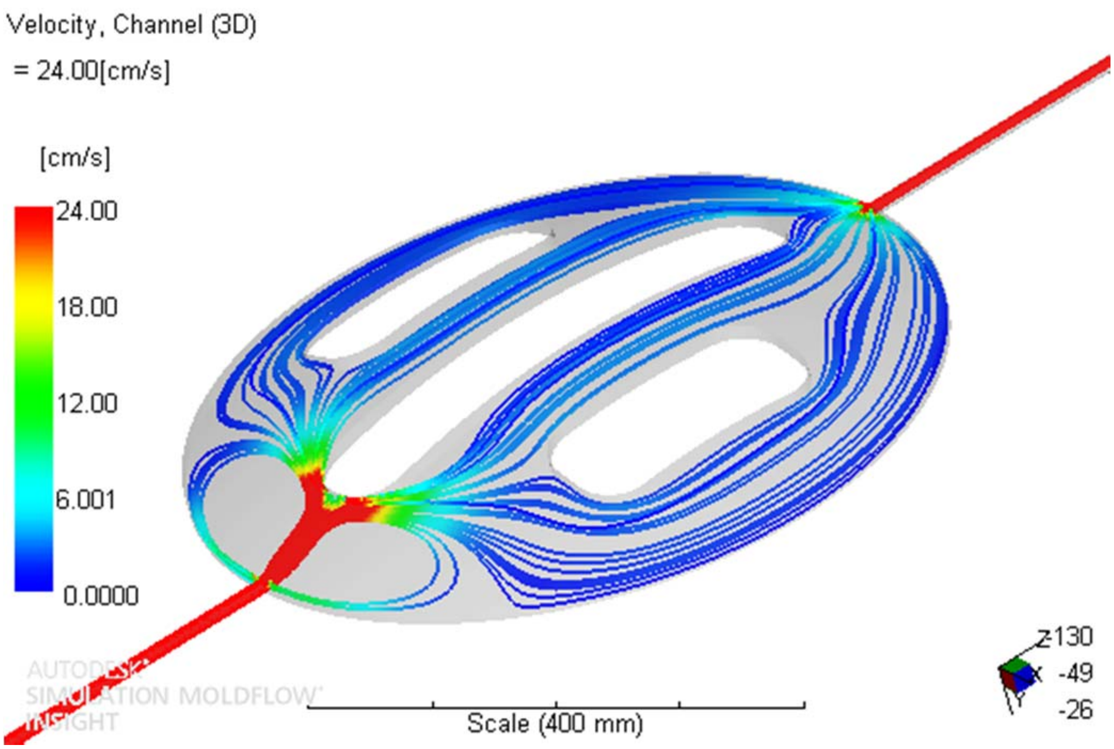

Figure 8: Streamline plot of velocity in lower channel

Figure 9 shows the coolant temperature in the conformal cooling channel. It can be seen that where the velocity is high the coolant temperature is low and vice versa. The coolant temperature is approximately 30C hotter beyond the obstruction than at the inlet of the conformal cooling channel.

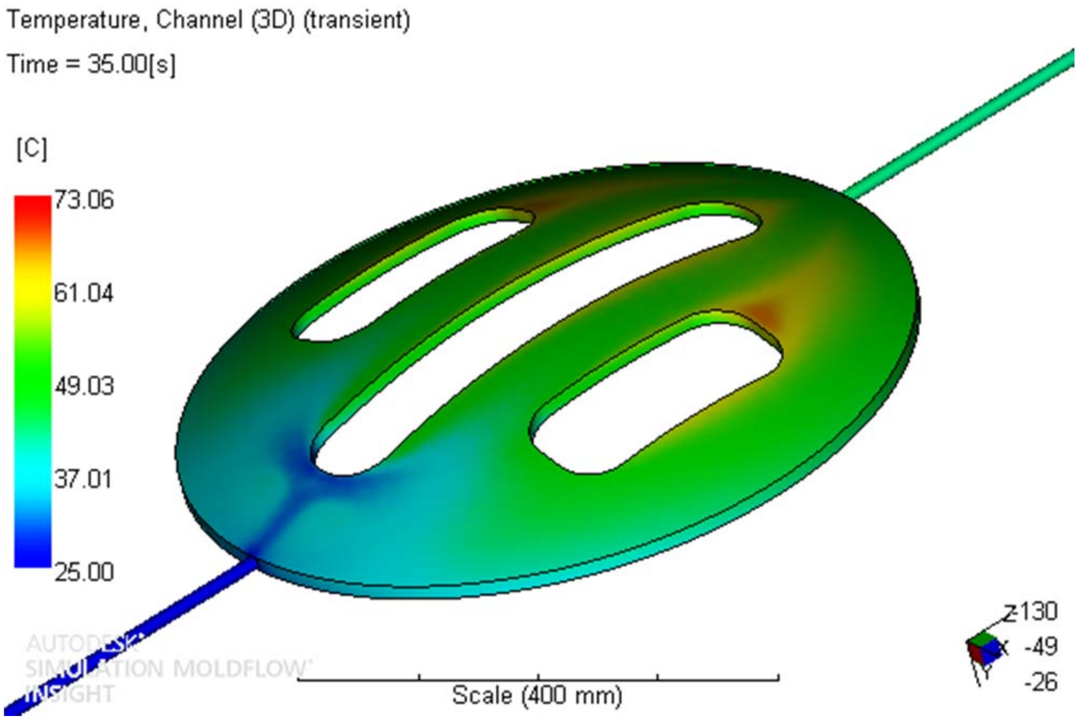

Figure 9: Coolant temperatures in the lower channel

Figure 10, showing the plot of the heat flux between the conformal cooling channel and the mold, shows that where the coolant stagnates there is very little heat transfer between the coolant and the mold. In the regions where the velocity is high there is much better heat transfer.

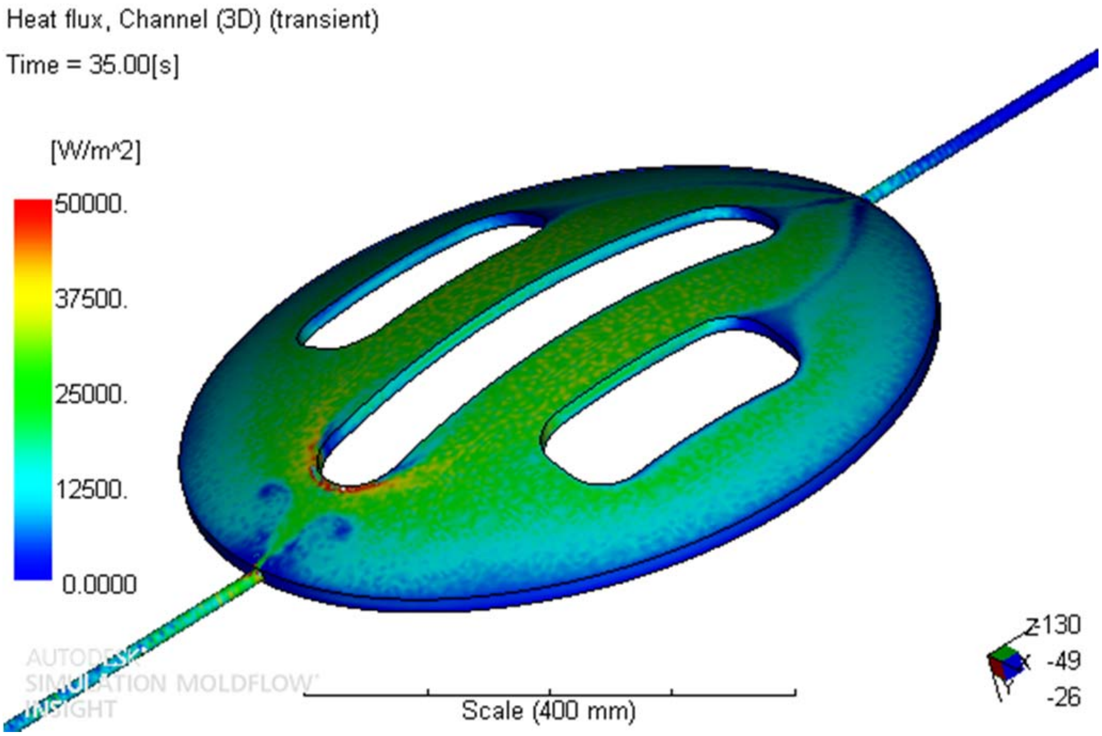

Figure 10: Heat flux at mold-coolant interface

Figures 8, 9 and 10 show that proper simulation of the fluid flow and heat transfer in the conformal cooling channel is vital in designing an appropriate conformal cooling system for the mold. Simulation can help the mold designer refine the design to create optimum cooling conditions. Without simulation a designer may arrive at a very complex and expensive mold design that will have very poor heat dissipation.

Figures 11 shows a very complex conformal cooling channel design used in a mold that manufactures lenses for eyeglasses, courtesy of Hofmann tool manufacturing [6]. As can be seen from figure 12 the conformal cooling channel follows the contour of the lens very closely with an elevated mesh channel. Figure 12 shows the convex nature of the lens that is being molded and also shows the proximity of the lens to the conformal cooling channel more clearly.

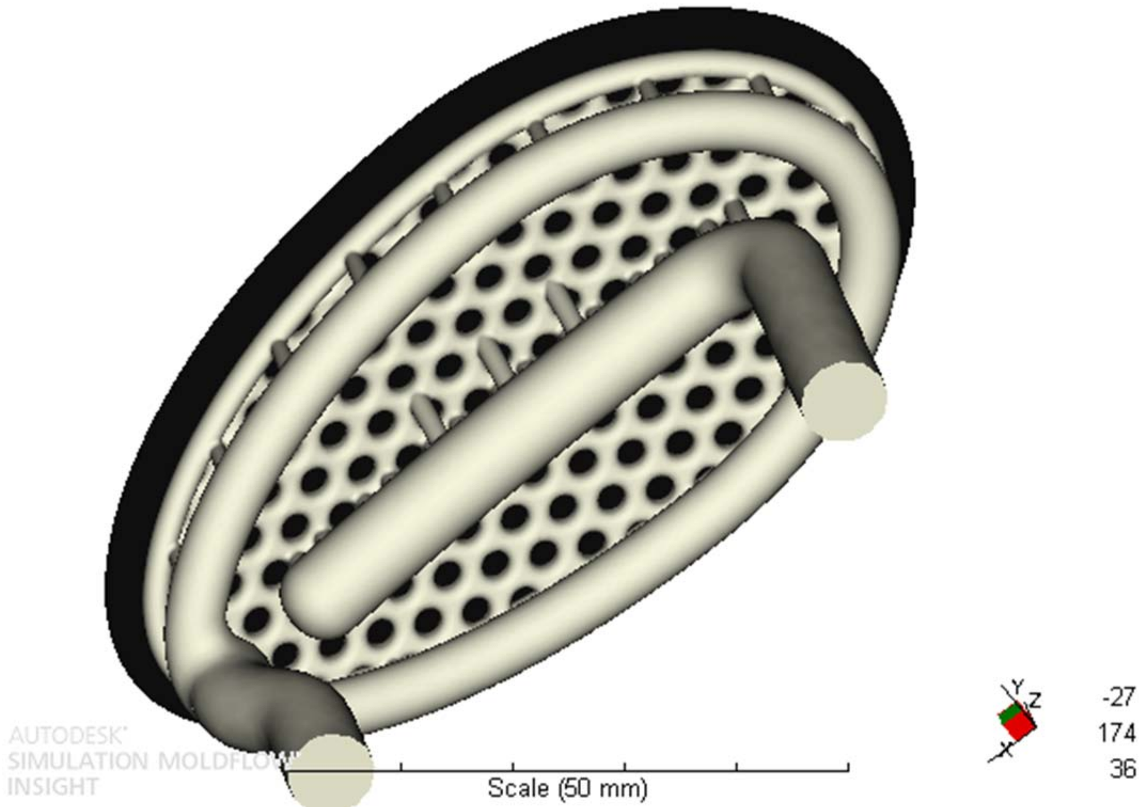

Figure 11: Conformal cooling channel for eyeglass lens

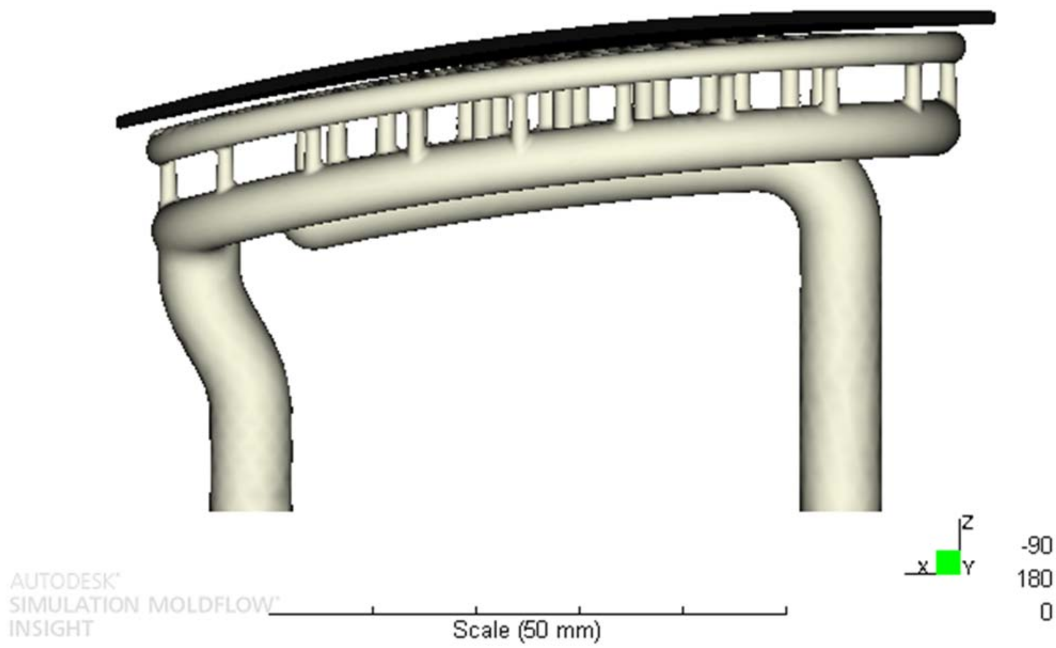

Figure 12: Side view of conformal cooling channel for eyeglass lens

Figure 13 shows the velocity streamline plot through the conformal cooling channel of the eyeglass lens model shown in Figure 11. From the plot it can be seen that the flow distribution in the conformal cooling section of the channel is very even, providing an even temperature distribution on the surface of the part Figure 14.

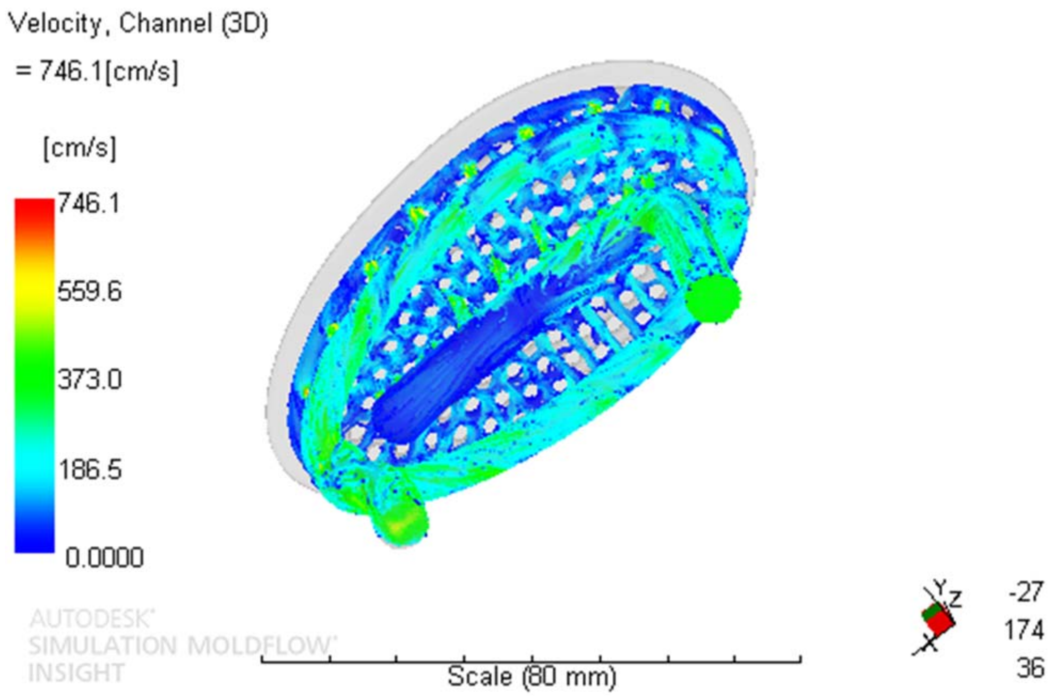

Figure 13: Velocity Streamline plot in eyeglass channel.

Figure 14 shows the temperature distribution on the surface of the lens. By looking at the legend of the result it can be seen that the temperature variation across the surface of the lens body is well within 1 C. The only hotter regions are around the rim of the lens. This shows that the conformal cooling channel is removing the heat evenly across the surface of the lens, which is vital in ensuring optimum optical properties in the lens.

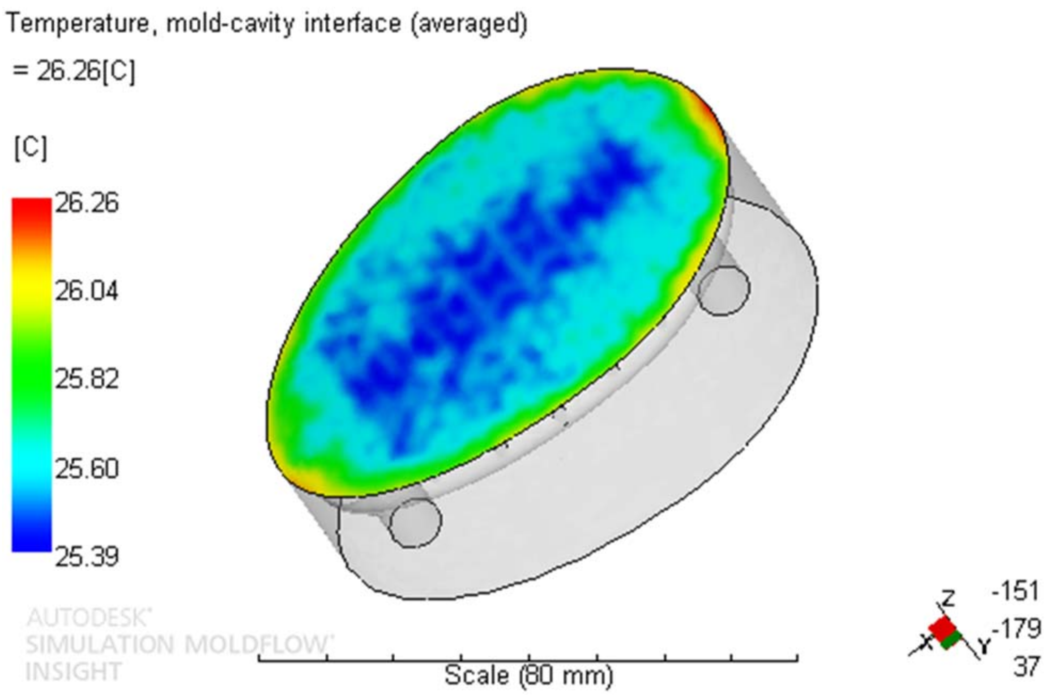

Figure 14: Temperature variations across the lens.

To demonstrate the rapid heating and cooling for conformal cooling, the component presented in Figure 15 is molded using rapid heating and cooling technology. The fixed side of the mold, the side with the feed system, is heated using the rapid heating and cooling channel shown close to the surface of the part. The moving half of the mold uses 4 conventional cooling channels using water set at 30 C flowing with a Reynolds number of 10000.

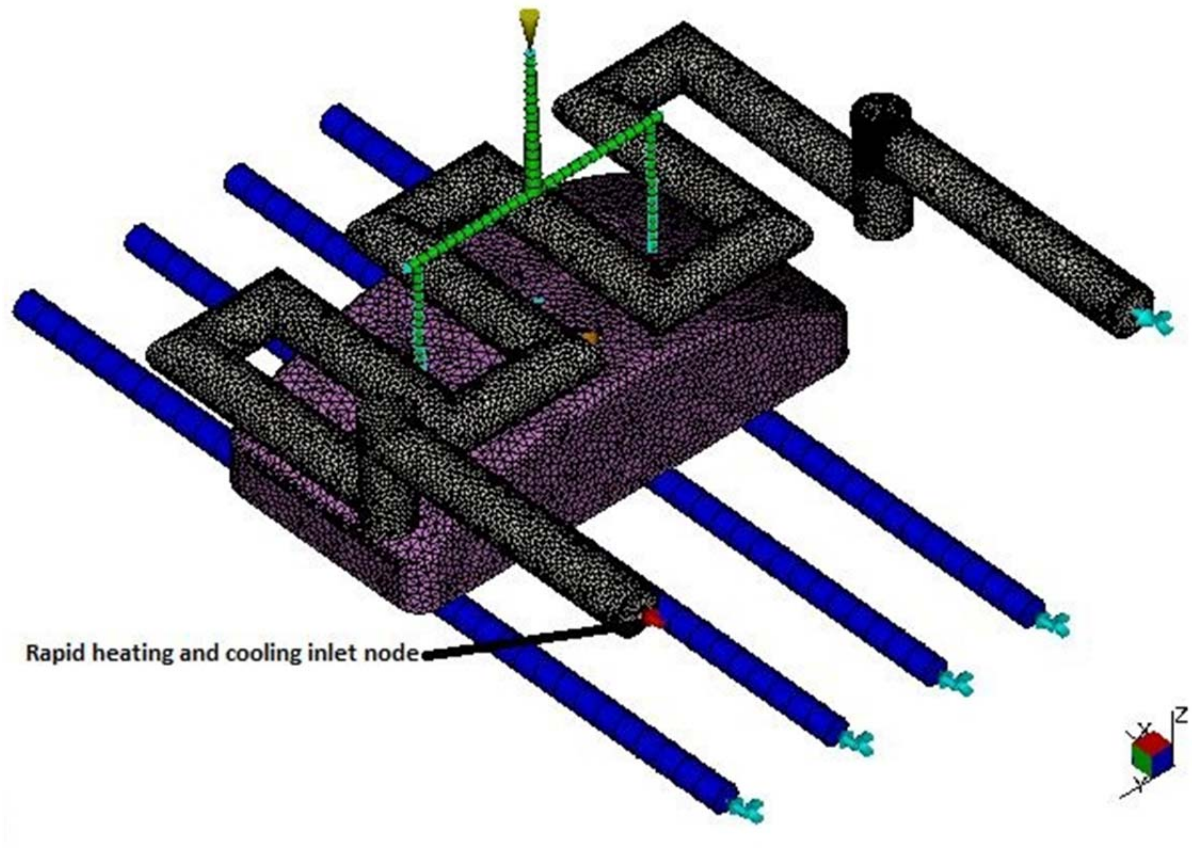

Figure 15: Rapid Heating and Cooling part and mold

The rapid heating and cooling boundary conditions are specified on the rapid heating and cooling boundary condition node as shown in Figure 15. This is the only place where rapid heating and cooling boundary conditions are specified. The model has the cooling channel in the fixed half modeled in 3D to simulate a conformal cooling analysis.

This case uses the *"Time controlled"* option for rapid heating and cooling. As the mold opens for part ejection the rapid heating and cooling cycle begins with a 2 second air purge to remove all the excess coolant in the channels. This air purge time corresponds to a 2 second mold open time set in the process settings. After the air purge, the channel is heated with saturated steam at 10 bar pressure for 15 seconds. The 15 seconds corresponds to the mold-close time before injection in the process settings. After the steam heating has finished, the filling process occurs during which the cavity is filled with a thermoplastic polymer at the manufacturer's recommended melt temperature with the cavity surface being at its highest temperature in the molding cycle. The start of filling also corresponds to the beginning of the primary air purge where the cooling channels are cleared of excess steam and condensate from the steam heating phase. After the primary air purge has completed the rapid heating and cooling channel is cooled down with chilled water set at 25 C for 13 seconds flowing with a Reynolds number of 10000.

To see the effect of the rapid heating and cooling cycling on the surface of the mold in contact with the part, a sensor node N1930 (see Figure 16) is chosen to monitor the temperature change during the cycle. These same conditions and part were used to validate the one dimensional rapid heating and cooling solver that was release in Autodesk Moldflow 2013 [2].

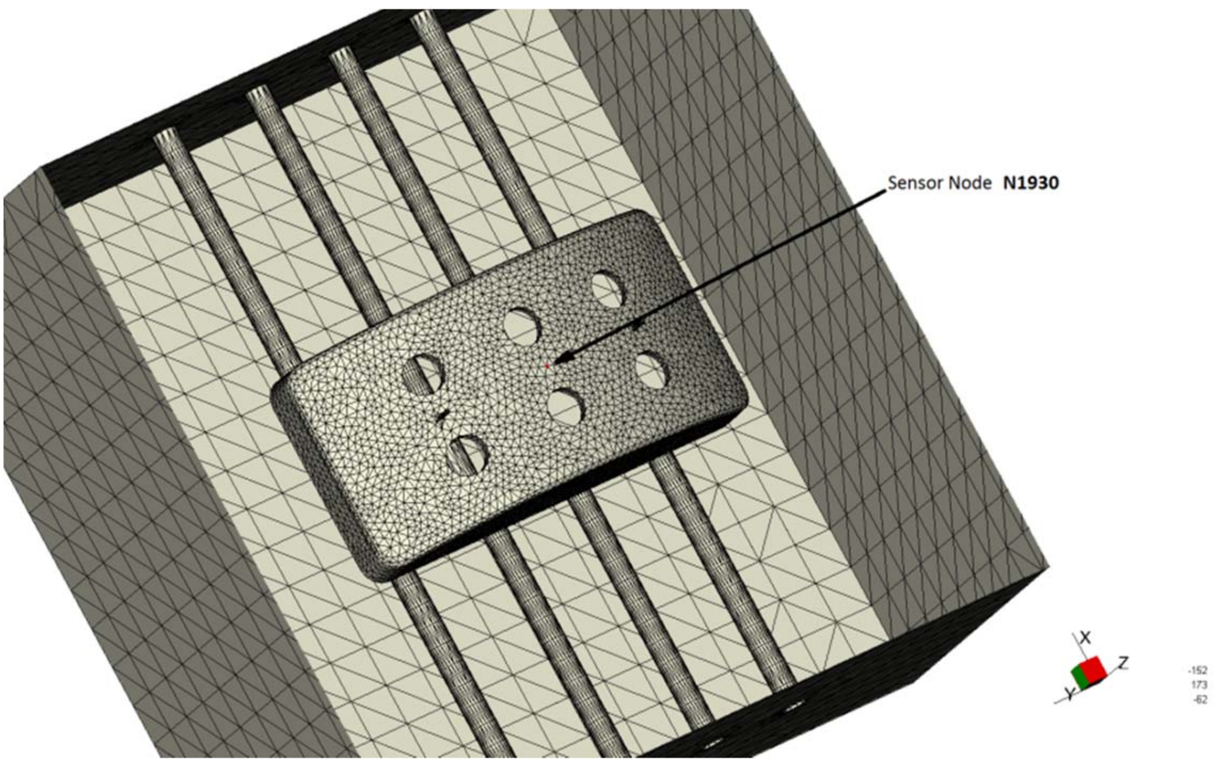

Figure 16: Sensor node on mold in contact with the part

Figure 17 shows the temperature change of that node during the injection molding cycle. It can be seen from Figure 17 that at the time of injection the surface temperature of the mold is at 137.5 C and is still heating up even though the rapid heating and cooling channel is being purged with air. This phenomena is called “thermal inertia” when the effects of the steam heating are still being transmitted through the mold whilst the steam heating has ended. The surface reaches a maximum temperature of 147.8 C after 2 seconds which corresponds to the start of the cooling phase in the rapid heating and cooling channel. During the cooling phase chilled water is circulated through the cooling channel and it can be seen that the surface temperature decreases to 83.7 C after 18.8 seconds which corresponds to the end of the mold open time, which is also the end of the secondary air purge. This temperature profile is as expected. After 18.0 seconds the mold surface starts to heat up again very quickly. This corresponds to the steam heating phase of the next cycle and is as expected. At the end of the cycle the temperature has reached 137.5 C again which corresponds to the temperature at the start of the cycle.

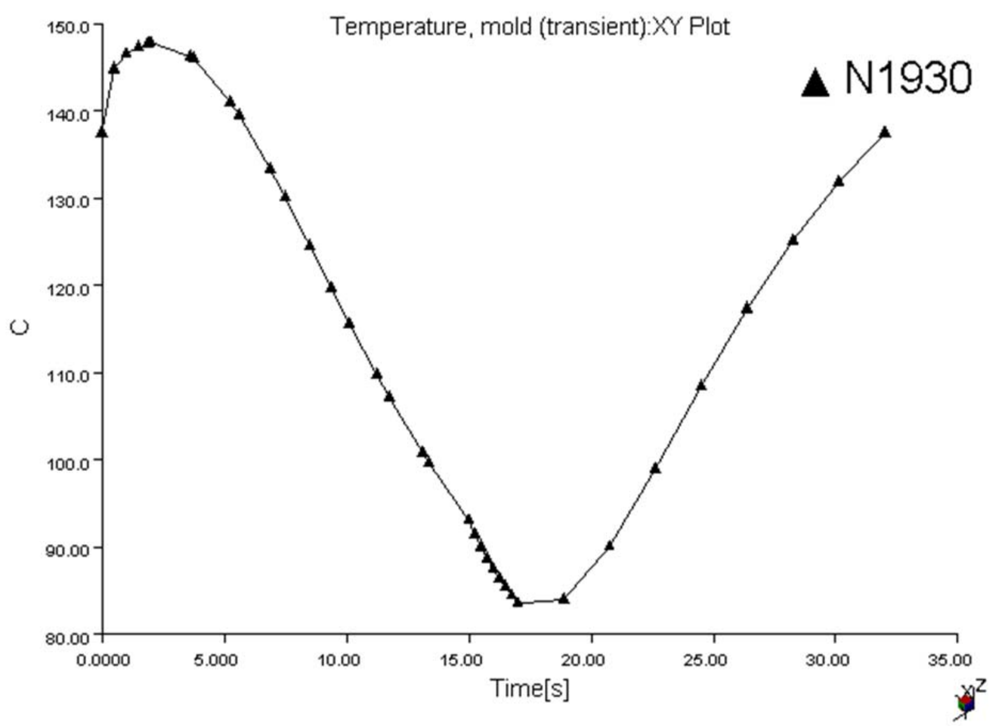

Figure 17: Temperature plot of the sensor node in the mold

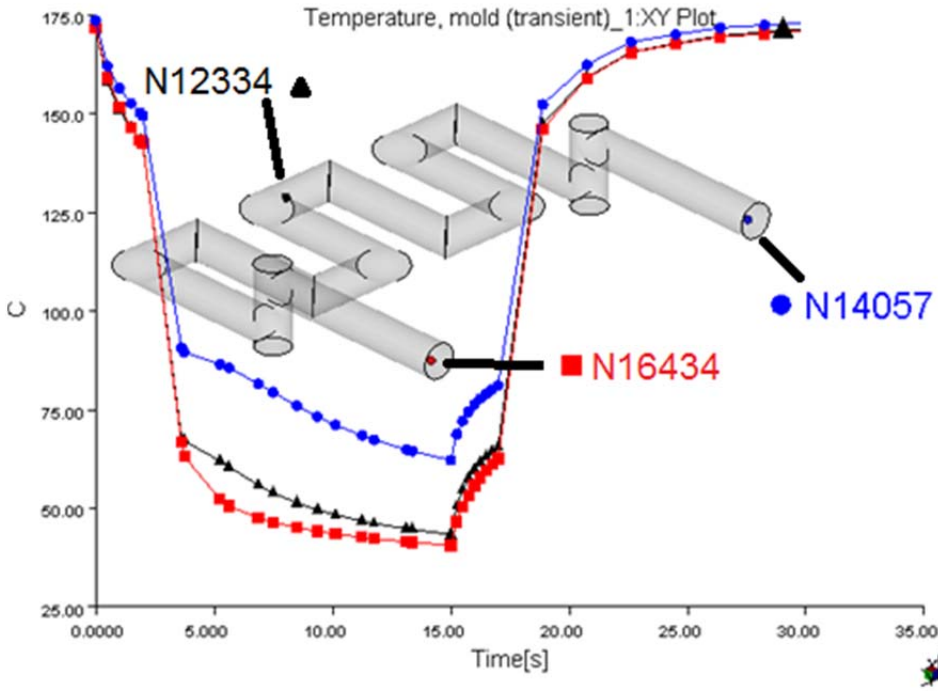

Figure 18: Time dependent temperature plot of the rapid heating and cooling channel surface

Figure 18 is a plot showing the surface temperature of the rapid heating and cooling channel that is in contact with the steam, air or coolant. It can be seen that at the end of the steam heating cycle the temperatures are around 175 C which are the hottest during the cycle. During the air purge the temperature reduces on the channels. However, after 2 seconds the coolant starts flowing in the circuit and the surface temperature decreases significantly very quickly. The node near the inlet N16434 cools to 40.74 C and the node near the exit N14057 cools to 62.69 C during the cooling stage. The mid channel node N12334's temperatures are between the inlet and exit elements temperature's, which is as expected. After 15 seconds the primary air purge begins and the temperature starts to rise again, as air does not remove heat as well as the coolant. Once the primary air purge is complete, the steam heating begins and the surface temperature of channel heats up quickly to 175 C. This is as expected.

Figures 19 and 20 show the mold cavity temperature in contact with the part, at the time of the flow front arrival. This result is a Flow solver result which takes advantage of the transient mold temperature solution. Figure 19 shows the top of the part temperature exposed to the rapid heating and cooling channel and Figure 20 shows the part temperature on the bottom side that is exposed to conventional cooling. By looking at these temperatures it can be seen that the top part temperature is much higher than the bottom temperature. This would be desirable if the user would prefer a better surface finish on the top of the part. However, the temperature distribution is not very uniform across the top surface which may necessitate a mold re-design so that the rapid heating and cooling channel can be a consistent distance from the part. A conformal cooling channel can be designed. Subsequent warp analysis to predict the final part shape as a consequence of these uneven cavity surface temperatures is also possible. Figure 19 is the most useful plot for a part and mold designer as the ultimate goal of rapid heating and cooling is to obtain a mold surface temperature as hot as possible during the filling stage. Figure 19 provides the user with this information.

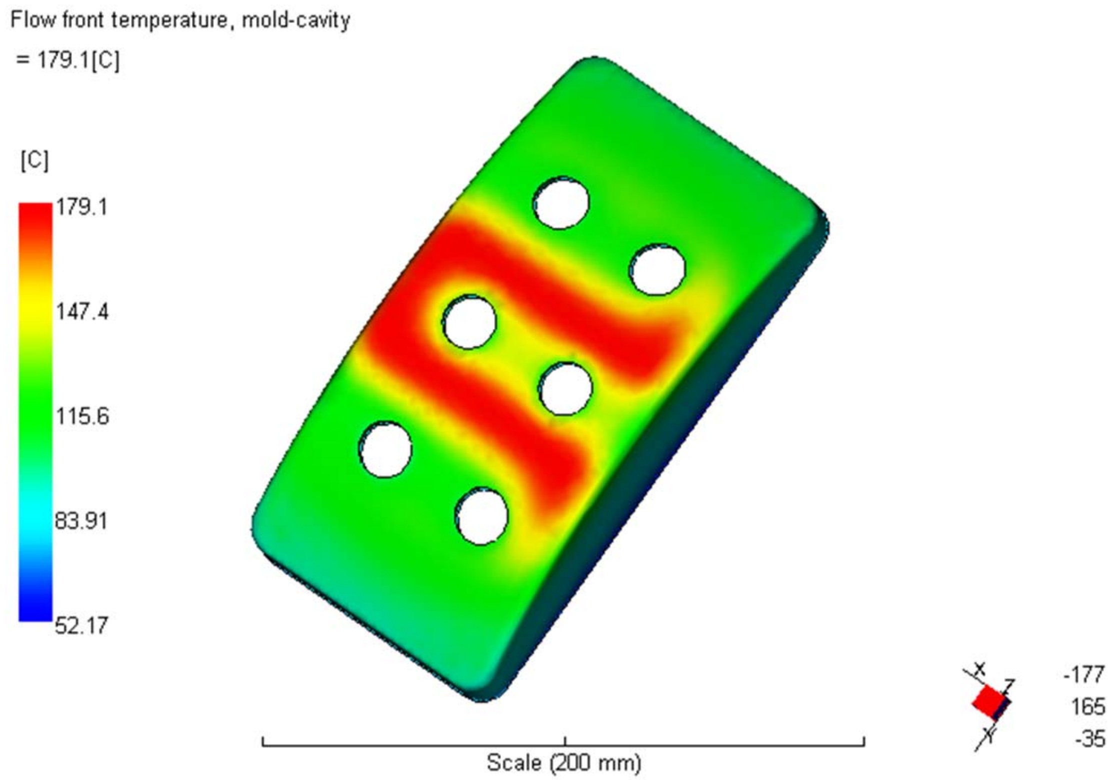

Figure 19: Flow front temperature, mold cavity result-Top side

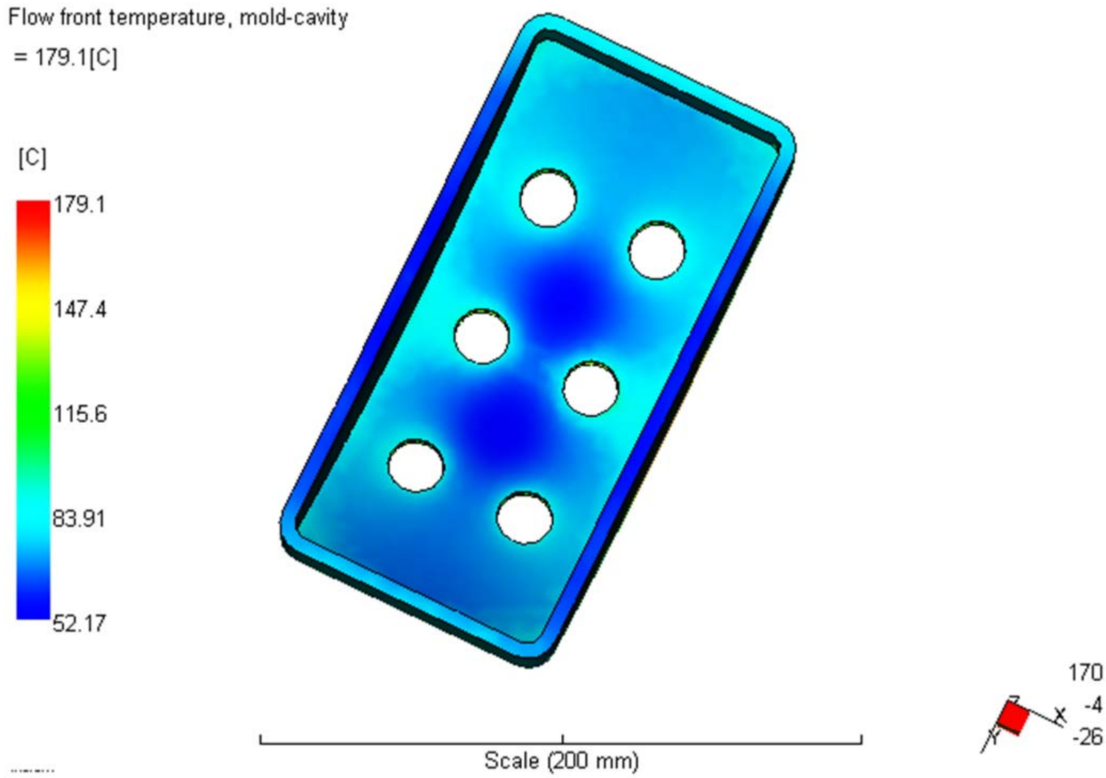

Figure 20: Flow front temperature, mold cavity-Bottom side.

When comparing the results from the rapid heating and cooling conformal cooling channels to those from the 1D solution [2] it can be seen that the conformal cooling channel tends to predict higher average temperatures when the mold is hot and colder temperatures when the mold is cold. However the maximum temperature reached with the 1D solver on node 1930 is 150.2 C whereas with the 3D solver the temperature is 147.8 C, Figure 17. However on average the 3D solver predicts hotter temperatures than the 1D solver. Figure 18 shows that the maximum temperature in the circuit is 175 C for the 3D solver whereas it is 166 C for the 1D solver [2]. It should be noted that the conformal cooling channel does not solve the air purges in the channel and only uses a heat transfer coefficient boundary condition in that region. This could be the cause of the differences. Another reason could be the way in which the CFD solution applies the boundary conditions on the mold surfaces as opposed the way the 1D solution does. The 1D solution uses a Dittus-boelter correlation whereas the CFD solution calculates the heat flux directly on the boundary and this could be the cause for the discrepancy.

However when looking at figures 19 and 20 it can be seen that the flow front temperatures on the cavity match those from the 1D solution very well, refer to reference [2]. A good flow front temperature on the cavity affects the filling patterns and warpage of the part and a very good correlation is achieved in this regard.

Conclusions

The conformal cooling functionality added to the transient cool simulation is a worthwhile enhancement to the existing simulation technology when considering that more mold makers are taking advantage of direct metal laser sintering (DMLS) manufacturing technologies to make conformal cooling molds.

The numerical results predicted by the conformal cooling module of the transient cool solver agree with results obtained from the existing one dimensional coolant solver that has been previously validated experimentally.

The conformal cooling solver can be used on cooling channels that are unsuitable for 1-D analyses. The conformal cooling solver is able to predict complex flow patterns in complex channels.

The conformal cooling solver can deal with rapid heating and cooling cycles on conformal cooling channels, providing acceptable results.

Acknowledgement

The authors wish to thank *Hofmann tool manufacturing* of Lichtenfels, Germany for kindly providing the eyeglass lens model shown in Figure 11 and for granting permission to publish the results shown in Figures 13 and 14 that were obtained on their model.

References

[1]. Jay Shoemaker (Editor), *Moldflow Design Guide. A Resource for Plastics Engineers.* Hanser (2006)

[2]. Autodesk Moldflow Insight 2013, Transient Cool Validation report.

[3]. Schnipke, R.J, *Streamline upwind finite-element method for laminar and turbulent flow*, Thesis, Virginia Univ., Charlottesville (USA), 1986 Jan 01.

[4]. C. Kietzmann, L Chen, H Lin-Feng, F. Costa, R Le Goff, Transient Mold Cooling Simulation for the Injection Molding Process, *SPE ANTEC Proceedings*, Boston(2011).

[5]. Autodesk Moldflow Insight 2012, Transient Cool Validation Report.

[6]. Hofmann tool manufacturing, www.hofmann-innovation.com.

AUTODESK.