Temperature Distributions on Mold Surface by Induction Heating

Soohong Kim
University of Massachusetts, Amherst
MIE 605
Finite Element Analysis
Spring 2003

ABSTRACT
A sequential field coupling of induction heating analysis was performed to detect the temperature distribution on mold surface. The induction coils were placed in close to mold surface and coils excited by a large alternating current at high frequency. The AC current induced heat in the mold surface and the surface temperature was raised quickly. The analysis showed that the results of surface temperature distributions with different number of the coils. This leads to the conclusion that increasing the number of coils and input current would be able to get uniform surface temperature distribution as well as rapid temperature increase.

INTRODUCTION
Injection molding is the most popular plastic processing in the entire plastic industry for cost-effective mass production of plastic components. In injection molding process, a hot and molten polymer flow through a runner system and gate, then rapidly injected into a cold mold, packed under high pressure and cooled until solid shape reflecting the cavity. While the molten polymer fills the mold cavity, it starts to cool and freezes from mold surface. This reduces the quality of molded parts such as bad surface finish, causing residual stress, etc. and also reduces flow length of the molten polymer. To improve the quality and the flow length of the molten polymer, the mold cavity needs to be hot during the filling process. However, this will increase the cycle time of injection molding process. To minimize the cycle time due to heating up the mold, new technologies have been developed. One of new technologies is the induction heating method.

The theory of induction heating method can raise the temperature of mold surface rapidly, and the Rapid Thermal Response System (RTR), which developed by the Injection Molding Laboratory (IML) at University of Massachusetts based on induction heating theory, allows to control of the mold surface over time. By heating up the mold surface, quality of the molded parts can be improved.

TEMPERATURE DISTRIBUTION ON MOLD SURFACE
Although the mold surface could be heated up to improve the quality of molded parts and increase flow length of the molten polymer, the temperature distribution of the mold surface would be concerned. If the mold surface could not be heated up uniformly, the problems will still remain. For this reason, the induction heating analysis on the mold surface has been performed using Ansys® FEA software. The induction heating analysis is a sequential field coupled analysis of an electromagnetic and a transient thermal analysis. An electromagnetic analysis calculates Joule heat generation data, and a transient thermal analysis uses to predict a time-dependent temperature solution. The sequential coupled means that results of one analysis become loads for the second analysis, and results of the second analysis will change some input to the first analysis. Figure 1 shows the solution flow diagram of analysis.

![Figure 1. Solution Flow Chart](image-url)

The dimension of the mold was chosen to be 4 inches by 1/4 inches and the mold material was assigned as a 420
stainless steel. The diameter of the coil was chosen to be 1/4 inches and the coil material was assigned as copper. With assigned dimensions, material, and air region, 2D models were created. First model, shown in Figure 2, had 3 coils (model 1) and second model had 5 coils (model 2). Two models had same geometries except for number of coils and their locations. The material properties were assigned to each of these materials.

Figure 2. Created 2D model with 3 coils.

Initial and boundary conditions were assigned to the model and an assumption was made that outer boundary of the mold was insulated. For the transient thermal analysis part, the radiation effects from mold surface to air was considered. Initial mold temperature and ambient temperature was assigned to be 25ºC. Initial input of current density could be calculated with the current and the cross sectional area of the coil using equation 1.

$$I = \int_S J \cdot da$$  \hspace{1cm} (Eq. 1)

Since the coil size has been selected, the cross sectional area of the coil is fixed, and the current could be a variable. The amount of the current input into the system is depended on the power station capability. In this analysis, the current input was assumed to be 36[A]. With this condition, the calculated current was applied to the coils and same amount of current density was applied to two models. Since same amount of energy was applied to two different cases, the approximated temperature distribution could be plotted. The maximum temperature with model 1 will be higher than the maximum temperature of model 2, but the temperature distribution on surface will be more uniform in case of model 2.

Appropriated elements were selected for the electromagnetic analysis and the transient thermal analysis. For the electromagnetic analysis, plane 13 was selected for mold, coils, and air. This element is a 4 nodes element and has nonlinear magnetic capability. The mold surface, which shown as a line in Figure 2, was assigned to null element The null type element is not considered for analysis. For the thermal analysis, plane 55 was selected for the mold. This element is a 4 nodes element and has a two-dimensional thermal conduction capability. The coils and air were assigned to null type element. Surface 151 was selected for the mold surface. The element is applicable to two-dimensional thermal analysis with surface effect application.

Skin depth effect was considered in meshing step. Skin depth could be approximated with equation 2.

$$\delta = \frac{1}{\sqrt{\pi f \mu \sigma}}$$  \hspace{1cm} (Eq. 2)

Where $\delta$ is the skin depth (meter), $f$ is the frequency (Hertz), $\mu$ is the absolute permeability (Henries/meters) and $\sigma$ is conductivity (siemens/meter). In ANSYS simulation, to calculate accurate the electromagnetic field and joule heat losses, finite element mesh size is required to be fine enough to capture the surface phenomena. Near the mold surface, the finite element was meshed smaller than the skin depth. At this point model 3 was created. Model 3 was modified from model 1 but mesh size was different.

Analysis type, time step, final run time and convergence tolerance for nonlinear solution were assigned. With large number of time step, the analysis will be long process.

Ansys® FEA analyses were performed the three different models. The temperature responses of model 1 and model 2 were compared to the effect of mesh sizes.

Figure 3 shows the temperature responses of model 1 and model 3 at the nodes where the maximum temperatures were detected. Model 1 and model 2 had same conditions and same current density input, but only difference was the mesh size. Model 1, which had mesh size smaller than the skin depth, responded very rapidly. The maximum temperatures of two models were found to be 510.99ºC and 45.20ºC. For the sequential coupled analysis of the induction heating, the mesh size must be smaller than the skin depth size. Otherwise the element would not be able to capture the surface phenomena properly.

Three nodes were selected: the first selected node located at the middle of mold surface and second and third nodes were selected near the first node for the model 1 and model 2. Since two models had same node number for the mold region, the locations of the nodes were the same for model 1 and model 2.

Figure 4, shows the temperature responses at the three selected nodes in model 1 and model 2. The small difference of the temperature indicates that the mold surface was heated up...
uniformly. This indicates the model 2 was more uniformly heated up the mold surface. However, the maximum temperature was much higher in case of model 1.

The temperatures on three nodes at the end of the time steps were tabulated at Table 1 and standard deviation was tabulated at Table 2.

<table>
<thead>
<tr>
<th>Node Number</th>
<th>Model 1 Temp. (°C)</th>
<th>Model 2 Temp. (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>352</td>
<td>257.75</td>
<td>230.95</td>
</tr>
<tr>
<td>357</td>
<td>510.75</td>
<td>250.11</td>
</tr>
<tr>
<td>362</td>
<td>259.25</td>
<td>230.45</td>
</tr>
</tbody>
</table>

Table 1. Temperatures at the end of time step

<table>
<thead>
<tr>
<th></th>
<th>Model 1</th>
<th>Model 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard Deviation</td>
<td>120.9</td>
<td>11.2</td>
</tr>
</tbody>
</table>

Table 2. Standard Deviation

Figure 5 shows temperature distributions on the mold surface in both cases. The coils were located in front of where the maximum stresses were detected. The region of heated mold surface was small in case of model 1, and the region of heated mold surface was wilder in case of model 2. However, total of heat input on to the system were same. Total heat input could be calculated by using Equation 3, which known as Joule’s Law.

\[ Q = Pt = I^2 R t. \quad \text{Eq. 3} \]

**CONCLUSION**

2D induction heating analysis, a sequential field coupled analysis of an electromagnetic and a transient thermal analysis was performed to study of mold surface temperature. The temperature distribution shape and the points of maximum temperatures of model were roughly estimated with given conditions and input data previously. However, the results from Ansys® FEA analyses show the exact temperature values on each node on the mold surface and temperature distributions on mold surface. These results conclude that using large number of coils and applying greater heat input would be needed to solve the problems, which were related in filling process of injection molding process.

To get the best uniform temperature on the mold surface, the cross sectional area of the coil must be small. If the area is small, the current density will be increased as well as heat input. If the cross sectional area is too small the coil might be deformed because of the large energy passing through the coil.
To design the best coil size, this fact and input current should be concerned.

In the Ansys® FEA analysis, mesh procedure should be carefully done with some element such as surface 151. However, large scaled model might have difficulty in meshing the element due to limitation of the maximum number of element that Ansys® provide. For the transient analysis, applying time step and sub step was one of the issues of the performed induction heating analysis. If time step is too big, the accuracy of the result will be dropped. However, the time step is too small the analysis will be arduous.

REFERENCES
http://www.matweb.com