# Modeling of Time Dependent Failure of Injection Molded Parts using Multi-physics Approach

Nagarjun Jawahar, Saharash Khare

Hero MotoCorp Ltd., Dharuhera

*Abstract: This work demonstrates an approach to study and simulate the failure of an insert molded part which happened after few days of the part molding under idle condition. To simulate the above failure, an innovative approach coupling Moldflow and Abaqus software was derived. First, a flow simulation including phase change of plastic material was carried out with derived parameters, results of which were exported as input to the Abaqus structural solver. Secondly, a thermo-mechanical analysis of the model was then carried out considering the thermal and moisture effect on material property. A good correlation was achieved between the actual failure location and max stress location as predicted by said coupled approach.*

*Key Words: Insert mold, flow simulation, thermo-mechanical analysis*

## 1. Introduction

Plastic injection insert molding is a plastic injection molding process in which thermoplastic material is molded around an insert piece or pieces placed in the plastic injection molding cavity, resulting in a single strongly bonded, integrated assembly, with the insert or inserts encapsulated by the plastic. Inserts can be metal, another plastic, ceramic or just about any substance that can withstand the plastic injection molding process. It is an effective alternative to assemblies manufactured using soldering, adhesives or fasteners.

The reliability and strength of insert molded parts depends on the bond between the parts, shrinkage and warpage. Shrinkage is inherent in the injection molding process. Shrinkage occurs because the density of polymer varies from the processing temperature to the ambient temperature. During injection molding, the variation in shrinkage both globally and through the cross section of a part creates internal stresses. These so-called residual stresses act on the part with effects similar to externally applied stresses. If the residual stresses induced during molding are high enough to overcome the structural integrity of the part, the part will warp upon ejection from the mold or crack with external service load. Warpage is a distortion where the surfaces of the molded part do not follow the intended shape of the design. Part warpage results from molded-in residual stresses, which, in turn, is caused by differential shrinkage of material in the molded part. If the shrinkage throughout the part is uniform, the molding will not deform or warp, it simply becomes smaller. However, achieving low and uniform shrinkage is a complicated task due to the presence and interaction of many factors such as molecular and fiber orientations, mold cooling, part and mold designs, and process conditions.

## 2. Problem Definition

The function of fuel cap is to prevent leakage of vaporized fuel from fuel tank, thereby increasing the fuel economy and decreases the emission of hydrocarbon into atmosphere which is very harmful to the earth's ozone layer and it also causes smog. Most of the motorcycle use steel fuel caps. In order to save cost a plastic fuel cap with material X, was manufactured through insert injection molding process. The part is ejected at 60$^{o}$C from mold and allowed to cool at room temperature. The failure shown in figure1 was noticed after few days of part molding under idle condition.

**Figure 1. Failure in fuel cap**

## 3. Multi-Physics Approach

The task was to study the reason for the failure. The two important parameters which govern the injection molding process are temperature and pressure. The processing window for injection molding for defect less part is shown in figure2. If optimum condition is not maintained it leads to defect like short-mold (flow end freezes before completion of filling), melt (turbulent flow), flash (material leakage) and thermal degradation (silver streaks on surface). However, visual inspection of fuel cap shows none of the above defects. Thus the reason for failure could be weld line, air traps, shrinkage or warpage.

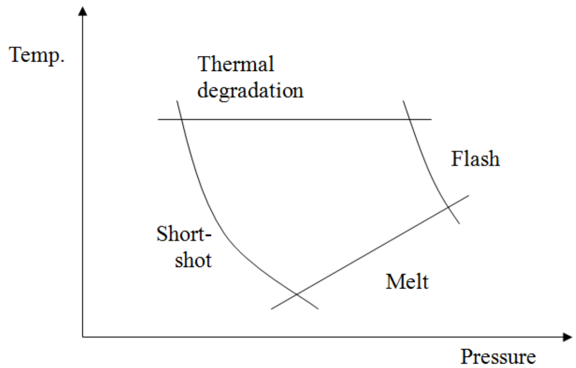


**Figure 2. Processing window for injection molding**

### 3.1 Flow Simulation

A flow simulation including phase change of plastic material was carried out with derived parameters. The gate location, process parameters and material properties were obtained from vendor. Fill Pack Cool and Warp analysis was performed using Moldflow software to study the part behavior. The analysis was governed by injection time.

Fill analysis shows whether pattern of flow, weld line, air traps and temperature of part at end of fill. Air traps are formed if part is not uniformly cooled or thickness is more. The Weld line displays the angle of convergence as two flow fronts meet as shown in figure3. The presence of weld lines may indicate a structural weakness and/or a surface blemish. From figure3a and 3b we can conclude that weld line and air traps shall not lead to failure as it is not in the region of crack propagation (figure1).

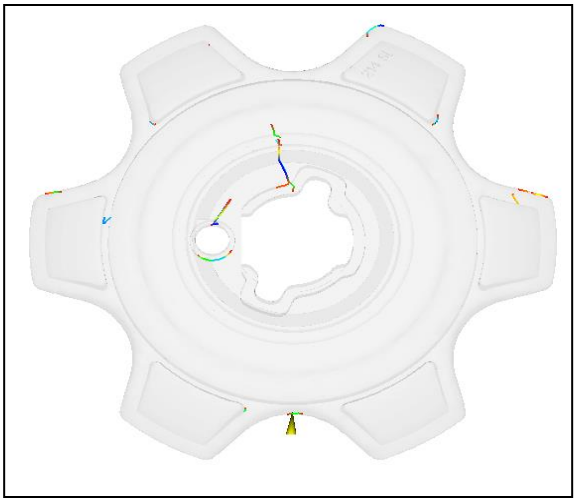

**Figure 3a. Weld line and weld line angle**

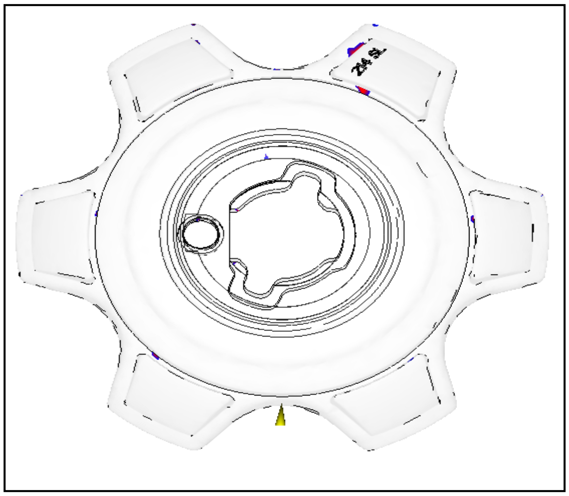

**Figure 3b. Air traps or blow holes**

The efficacy of thermoplastic packing has important effects on warpage, shrinkage, and the incidence of defects, such as sink marks. Figure4 shows the sink marks obtained at end of the process. The main output of a Pack analysis is volumetric shrinkage, and the distribution and magnitude of volumetric shrinkage play a key role in part quality.

The packing pressure is used to pack out a part and is often related to the fill pressure. As a rough guide, the packing pressure should be about 80 percent of the fill pressure; however, the packing pressure can vary significantly. Packing pressures are commonly between 20-100 percent of the fill pressure, and can be higher or lower. An important aspect of the packing pressure is that it cannot be so high that it exceeds the clamp limit of the machine.

The following formula is used to estimate the maximum pressure that should be used. This formula will determine a pressure, assuming a constant gradient across the part so that 80 percent of the machine capacity will be used. This is a conservative approach, but this can be used as a starting point.

$$P_{max} = \frac{F}{A}\ 100\ \times 0.8\ \mathrm{MPa}$$

Where, F is the clamp force (tonnes) and A is the total projected area of the model ($cm^2$)

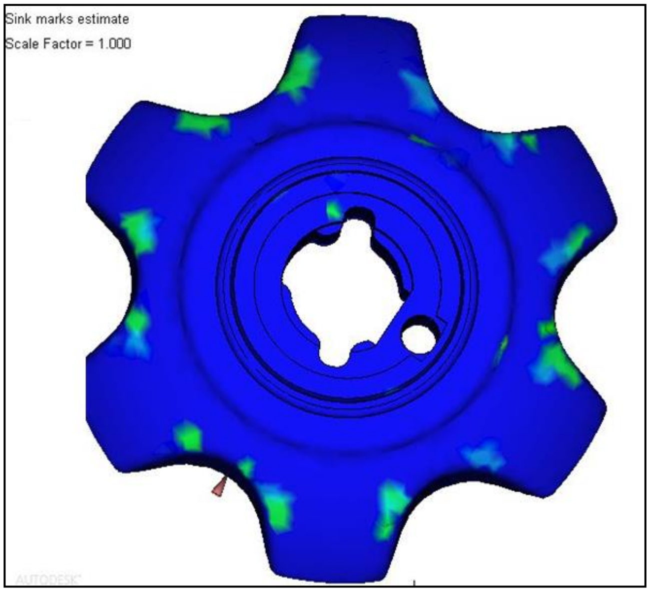


**Figure 4. Sink marks**

Material X, being a semi-crystalline material, the molding shrinkage is high. Volumetric shrinkage calculations begin once the cavity is filled, based on the difference between the current pvT state and the reference state (where the pressure p is zero and temperature T is the specified ambient temperature):

$$S(t) = \frac{p(t)}{p_{amp}}$$

As the mass of an element changes (for example, with polymer flow during packing), shrinkage continues to change with each change in the element's pvT state. Once the mass stops changing, the element's current pvT state is fixed in the shrinkage calculation as the reference state. The mass of an element stops changing when the cavity pressure has decayed to zero. After this, the volumetric shrinkage becomes a constant.

The shrinkage of injection molded parts depends on the thermodynamic behavior of the material during processing. For simplicity, we assume linear elastic behavior in the solidified part and purely viscous behavior in the melt.

It is reasonable to approximate the linear shrinkage using the formulation:

$$\varepsilon_i = \int_{T_r}^{T_0} \alpha_i(T)dT$$

Where, $\alpha$ is the linear thermal expansion coefficient (CTE) at temperature T in the $i^{th}$ principal direction.

$T_0$ is the temperature when the local cavity pressure reached the atmospheric condition. This value is obtained from the flow simulation. $T_r$ is the room temperature.

For a Warp analysis, constraints are applied to the model nodes by the software to prevent rigid body motion of the model, in response to the natural warpage of the part. The total deflection of the fuel cap was found higher (figure 5).

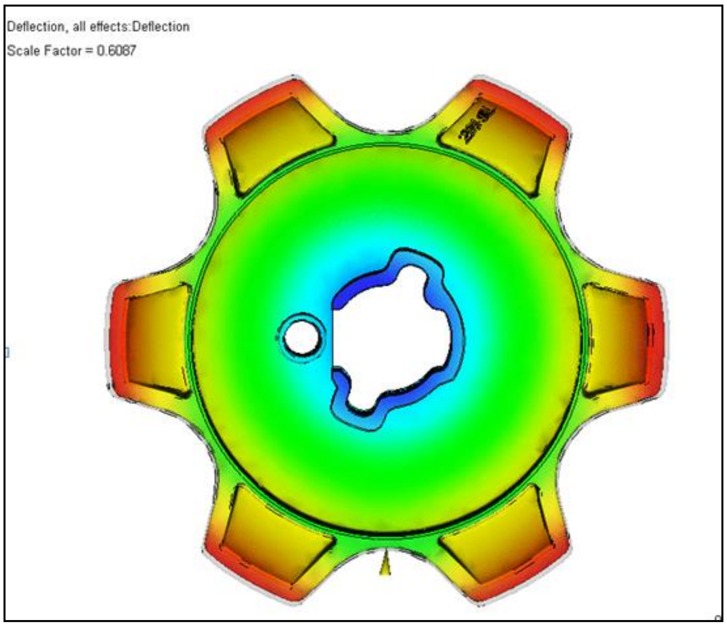


**Figure 5. Part Warpage**

### 3.2 Thermo-Mechanical Analysis

The molded part was ejected at 60°C from the mold and the part is allowed to cool at room temperature. The part is free to warp and shrink during the cooling process in any orientation as there are no constrain. In order to simulate the stresses during this process, the deformed geometry and pre-stresses obtained from moldflow software is taken as input in abaqus. The mesh is converted from first order to second order C3D10 elements. The connection between metal insert and plastic fuel cap are modeled using tie contact. The deck is then prepared to check the thermo-mechanical behavior of plastic part while cooling.

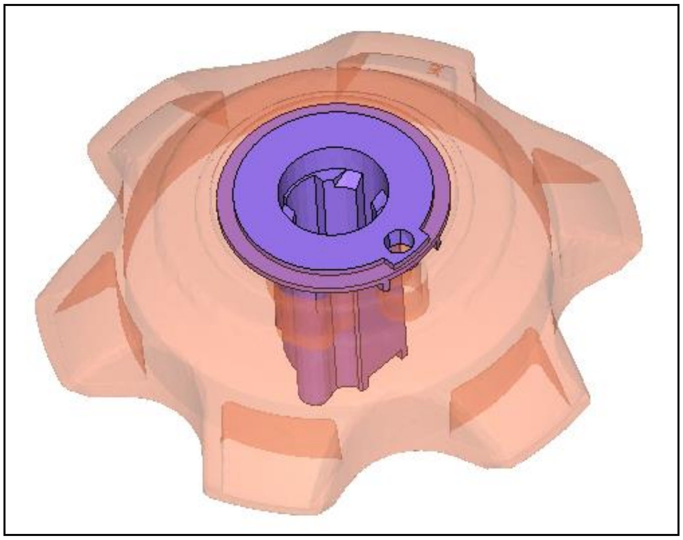

**Figure 6. Abaqus model of insert and fuel cap**

A Static thermal analysis was carried out to check the stress developed while the part cools from ejection temperature to room temperature. The max stress was found to be higher than the yield of material and exactly at crack initiation region (figure 7). Thus a good correlation was achieved between actual failure and virtual simulation.

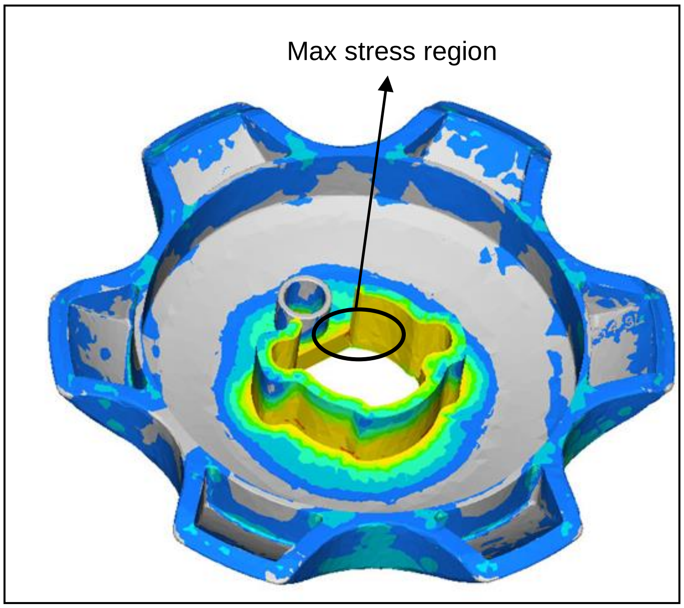


**Figure 7. Stress contour**

## 4. Conclusion

The multi-physics approach used to find the cause of failure can be summarized as below,

- Firstly, with 3D mesh, a Fill Pack Warp Cool simulation is performed in Moldflow. The stress file and mesh file are exported using mpi2abq.vbs script.
- Secondly, the model is converted to second order and connected with metal insert in Abaqus and is solved for stress to determine the thermo-mechanical behavior.

The molding shrinkage and warpage were found higher for material X, thus design modification may not be helpful. A different material shall be proposed for same design to save time, processing cost and die cost.

## 5. Future Scope

Similar analysis will be used for selection of suitable material and the performance shall be checked by physically manufacturing the part. For further correlation, a creep analysis would be done to generate crack propagation with material X.

## 6. Reference

Abaqus and Moldflow Plastics Insight help manual.