Direct Investment Casting Numerical Study for ABS P400 FDM Materials

O.M.F. Marwah.¹,a, S. Sharif²,b, S.Sulaiman³,c M.Ibrahim.⁴,d, E.J. Mohamad⁵,e

¹,³,⁴,⁵Faculty of Mechanical & Manufacturing Eng., Universiti Tun Hussein Onn Malaysia, 86400 Batu Pahat, Johor, Malaysia
²Faculty of Mechanical Eng., Universiti Teknologi Malaysia, 81300, Skudai, Johor, Malaysia.
⁵Faculty of Electrical and Electronic Eng., Universiti Tun Hussein Onn Malaysia, 86400 Batu Pahat, Johor, Malaysia

¹mdfaizan@uthm.edu.my, ²safian@fkm.utm.my, ³suhaimisulaiman01@gmail.com, ⁴mustaffa@uthm.edu.my, ⁵elmy@uthm.edu.my

Keywords: Rapid prototyping, Investment casting, Multijet Modeling, Projet

Abstract. The purpose of this paper is to study the collapsibility of internal structure for Acrylonitrile butadiene styrene (ABS) P400 materials as a sacrificial patterns in investment casting (IC). Two different internal patterns structure which consist hollow and square 90° were performed by Insight software. Experiment were perform by applying 3D coupled thermal mechanical finite element analysis on cube ceramic shell during burnout stage. As numerical observation the deformation stress for cube square 90° and hollow models have been evaluated by increasing the temperature during heating up to 700°C. It is found that both pattern start to buckling over the autoclave temperature through simulation observations and shows no cracking defect occured at ceramic mould.

Introduction

Since the early days of stereolithography (SL), investment casting foundries have been using prototype patterns built on SL apparatus (SLA) machines. These early attempts at using the models as sacrificial pattern were very haphazard. The models were built using a solid build style, and so the process of burning the patterns out of the ceramic shell caused a considerable amount of thermal expansion, thus causing the shell to crack (because the coefficient of thermal expansion – CTE – for thermoset epoxy material is higher than the CTE of ceramic material)[1]. There have been some prior studies on what causes shell cracking and how to prevent cracking during the pattern burnout process [2]. Ceramic shell cracking can be prevented if the epoxy pattern buckles at a temperature lower than the ceramic shell cracking temperature and webbed pattern would collapse inwards[3]. Shell cracking occurred at temperatures below the glass transition temperature of epoxy resin, and no cracking if the glass transition temperature of the epoxy resin is lower than the ceramic shell cracking temperature[4].

To ensure the patterns built on the SLA machines had yield rate, the patterns had to be much weaker to allow the burnout to occur more easily and to eliminate the expansion of the pattern. This would mean that creating a new build style, which would create a series of interlinked voids, thus making a quasihollow part, and with low viscosity, which would burn away easily. This new build style, named QuickCast 1.0. This involved changing the geometry of the build style, the first one being QuickCast 1.1 that changed the build style from triangular to square, the second one changing the build style from square to an offset hexagon, this was called QuickCast 2.0 [5,6]. These paper describes a numerical model that can be used to determine by computer simulation, for investigate internal structure form will effect on ceramic cube shell during burnout process in investment casting. The objective of this research to compare internal structure between open and closed ceramic cube with research aims to introduce a new internal structure for investment casting FDM pattern.
Methodology

**Part Design and Material Selection.** The prototype model has a simple geometry based mainly in straight shapes. The CAD modeling was undertaken on Solidwork 2011 software as shown at Fig. 1. From CAD a STL format has been generated for RP technology with consist of a mesh of connected triangular planar facets, each with a directed surface normal, which represents the pattern shape. The “sparse” that is used to “hollow” a solid structure. The dimensions of the internal links and thickness of ceramic shell (7mm) are the same for both of the hollow and square 90° structures. The finite element analysis (FEA) simulations was performed with ANSYS 12.1. The temperature was kept uniform during continuous heating which to understand the pattern collapsibility and shell cracking at burnout stage. The material properties of epoxy as ABS which know as thermoset resin changes with temperature. It is essential to have the diagrams of changing the mechanical properties during heating. The ceramic shells are formed during layer coatings in IC practice. The mechanical properties in this analysis are based on the SiO₂. The properties for ABS materials and ceramic shells are shown in Table 1.

---

**Table 1 ABS and Ceramics Materials Properties**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Units</th>
<th>ABS</th>
<th>Ceramic</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>Kg/m³</td>
<td>1060</td>
<td>2300</td>
</tr>
<tr>
<td>Elastic Modulus</td>
<td>MN/m²</td>
<td>2275</td>
<td>2.2059x10⁵</td>
</tr>
<tr>
<td>Poison ratio</td>
<td>N/A</td>
<td>0.394</td>
<td>0.22</td>
</tr>
<tr>
<td>Coefficient Thermal Expansion</td>
<td>/K</td>
<td>9.0x10⁻⁵</td>
<td>1.08x10⁻⁵</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>W/(m.K)</td>
<td>12710.5</td>
<td>1.4949</td>
</tr>
</tbody>
</table>

---

(a) Hollow                             (b) Square 90°

**Fig. 1 Internal Structure development**
Meshing Method. ANSYS provides a mesh generation features that creates initial element sizes operations. It gives the mesher a better chance of creating reasonably shaped elements during automatic shape generation[7]. For easing analysis, isothermal thermal condition loading represent the heating process from 0℃ to 700℃ with 5℃ rise for each step. The coarse, medium and fine meshing element are performed in all comparison work during simulation analysis. Furthermore, the Conta 174, Solid 186, Solid 187 and Solid 190 elements used for meshing the models. Fig. 2 shows the elements used for meshing models.

![Meshing Method](image)

Results and Discussion

Simulation Analysis. The purpose of this analysis to study the behavior in FEA numerical analysis and to see the fragility what happen in the inside, which in this part will remove during the process of investment casting. Deformation behavior was compared between these different internal structures. Fig. 3 shows three different type of mesh were including coarse, medium, and fine type after temperature ramp to 700℃. This hollow on the internal size of 37mm x 44mm shape closed on the outside of the internal structure, while the outside is set of ceramic with size 7mm wide. According to result, the maximum amount of deformation is 5.03e+01MPa, while the minimum is 1.08e+01 and the fragility facing up at a specific time in assume for course meshing. For mesh medium type, the maximum value and minimum 5.02e+01MPa and 3.29e-01MPa. In terms of the fragility shows higher than course mesh. Fine mesh shows better than others which maximum value and minimum value of 5.08e+01MPa and 2.33e-01MPa respectively. The FEA numerical analysis shows very clearly the ABS materials will burn out and no cracking defect occurred when employed 7mm of ceramic thickness. It’s shown the expansion towards the top and red marked area visible at model analyzed. For guidance on colour on the model are 51 imposed temperature, explained through the colours representing high values, while the bottom representing the lowest value in each analysis run.

Fig. 4(a)-(c) shows the obtained result for square 90° internal structure. The maximum amount of deformation is 1.13E+03MPa, while the minimum is 5.74E-02MPa and the fragility facing up at a specific time in assume as shown at Fig. 4(a). The internal structure shows burnout up constant until deformation. The results were even better than coarse as shown at Fig. 4(b) which is the maximum value and minimum 1.13E+03MPa and 6.69E-02 MPa, in terms of the fragility of his higher than coarse type. The fine mesh type shows better deformation which maximum value and minimum value of 1.13E+03MPa and 1.70E-01MPa at Fig. 4(c). FEA results shows very clearly the square 90° will be buckling when heating up temperature and will be burn out. The expansion towards the top in condition constant until the red colour shown on model. On the left model show , for guidance on
colour on the model are imposed temperature, explained through the colours representing high values, while the bottom representing the lowest value in each analysis run.

Hollow and Square 90° internal structure was deformed in satisfying on all mesh condition. Deformation phenomenon shows the internal pattern Square 90° were buckling when temperature increased. When an investment casting shell with a FDM epoxy pattern inside is in an autoclave or flash-furnace oven during the burnout process, it is subjected to high temperature rise, thermal expansion, and large strains. Since the difference of the CTEs of epoxy and investment ceramic is more than one order of magnitude, the epoxy pattern can exert considerable stress on the ceramic shell. During the pattern burnout process, the inner epoxy pattern expands more than the outer ceramic shell. As a result, there exists contact pressure between the ceramic shell and epoxy pattern shown 7mm thickness can prevent a compressive load contributed from ABS material expansion in burnout stages.

![Fig.3](image1.png)

**Fig.3.** Total ABS FDM hollow pattern deformation with different meshing type

![Fig.4](image2.png)

**Fig.4.** Total ABS FDM square 90° pattern deformation with different meshing type

**Conclusions**
The used of FEA to determine the internal pattern deformation and shell cracking hollow and square 90° pattern build style for ABS P400 based materials. The experimental has been developed which address to be burnout in investment casting process as a sacrificial pattern when heated up temperature up to 700°C. The simulation of the pattern burnout process results indicate both RP pattern will be totally deformation and burnout from ceramic shell. This means that applying hollow
based internal structure will significantly reduce the stress when pattern expansion during heating up the temperature. Result shows no cracking defect occurred at low and higher temperature according to simulations analysis. Therefore, it is suggested the ceramic shell thickness develop with thicker to avoid any excessive load on part expansion especially when reach glass temperature.

Acknowledgement

The authors wish to thank Ministry of Education Malaysia for funded Fundamental Research Grant Scheme (FRGS), Faculty of Mechanical & Manufacturing Engineering in Universiti Tun Hussein Onn Malaysia (UTHM) and Faculty of Mechanical Engineering, Universiti Teknologi Malaysia (UTM) for facilitating and financial support to this research programme.

References


