ARCHIVES OF METALLURGY AND MATERIALS
Volume 60 2015 Issue 2
DOI: 10.1515/amm-2015-0154

K.S. KIM*,♯, J.Y. KIM*

ANALYSIS OF THE EXTRUSION PROCESS OF A SQUARE TUBE MULTI-CHANNEL HEAT PIPE

ANALIZA PROCESU WYTŁACZANIA WIELOKANAŁOWYCH CIEPŁOWODÓW O KWADRATOWYM PRZEKROJU

Heat pipes have been recently in use for cooling purposes in various fields, including electronic circuit boards and vehicle parts that generate large amounts of heat. In order to minimize the loss of heat transferred, there is a need to maximize the contact area of the working fluid. This study produced a square tube multi-channel heat pipe to replace the existing circular pipe type to maximize the internal surface area thereof. This expands the surface, allowing the working fluid to come into contact with a wider area and enhancing thermal radiation performance. A mold for the production for such a product was designed, and finite element simulation was performed to determine whether production is possible.

Keywords: Aluminum hot extrusion, FE simulation, stress analysis, extrusion experiment

1. Introduction

Heat pipes are generally used in electronic circuit boards, vehicle lamps, and electronics, etc., for the purpose of transferring heat. Heat pipes are developing into various shapes in order to transport increasing amounts of heat. In order to enhance the heat transfer performance of such devices, it is important to maximize the cross sectional area of the wall inside the heat pipe, which is in contact with the liquid refrigerant. For mass production, heat pipes are extruded in a continuous process. Difficulties experienced here include the long length of the heat pipes, and the collapsing of the shape of the pipes due to the small dimensions of the space inside the pipes, which is designed to increase the cross-sectional area.

Accordingly, this study proposes a multi-channel heat pipe employing square tubes instead of the traditional circular pipes, so as to increase the contact area and to allow for easy mounting on vehicles and electronic products.

Also, in order to reduce collapsing of the cross section, the mold is not produced based on engineer experience but according to the results of extrusion simulation performed before production. Extrusion simulation is performed to minimize the time and cost required for production. First, in process design and mold modeling, a structure that maximizes the internal cross section to increase the contact area with the liquid refrigerant is designed. In the following simulation, plastic flow, speed, and residual stress according to compressive load were measured, and the stresses applied to the mold were found. Later, using the actual product, the experimental and simulated results for microscopic test, hardness testing, and component analysis were compared, and verification was performed using the significant values.

2. Finite element simulation of the extrusion process

2.1. Materials used

The material used for heat pipe production was Al-1100 with 99% aluminum content, and good non-scaling properties and heat conducting properties. Al-1100 is characterized by its relatively low strength and ease of forming and extruding.[5,6] Table. 1 and Figure 1 show the chemical composition of the material as determined through a high-resolution scanning electron microscope.

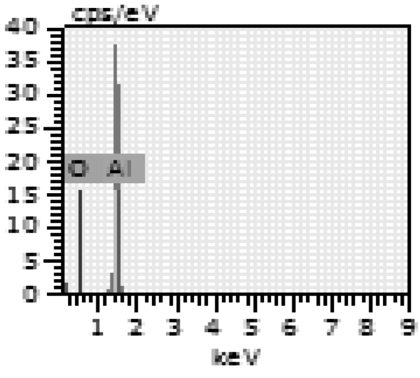


Fig. 1. Graph of Al-1100 chemical composition analysis

TABLE 1
Al-1100 Chemical composition

Element	Norm [wt.%]	C Atom [at.%]	C Error (3 Sigma) [wt.%]
Aluminum	96.84	94.78	13.03
Oxygen	3.16	5.22	2.75

---

* DIVISION OF MECHANICAL & AUTOMOTIVE ENGINEERING, KONGJU NATIONAL UNIVERSITY, CHEONAN-SI, REPUBLIC OF KOREA
♯ Corresponding author: keysun@kongju.ac.kr

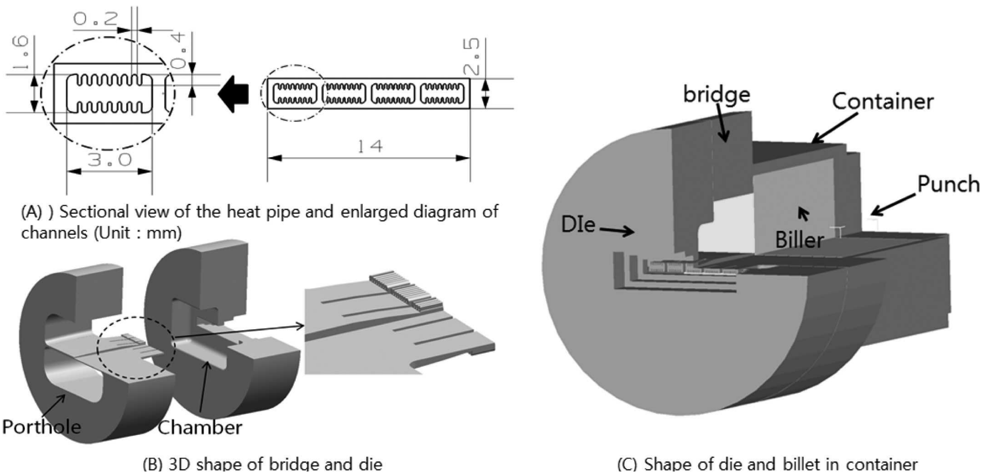
2.2. Product shape and die structure

The heat pipe model to be used in this study is shown in Figure 2(A). The thickness of the test specimen was 2.5 mm with a width of 14 mm, containing 4 square channels. The insides of these channels were designed with a very small pitch and a height not exceeding 0.4 mm in order to maximize the surface area in contact with the working fluid. Figure 2(B) is a 3D representation of the die set used for the extrusion process to produce the heat pipe, and Figure 2(C) is a 3D representation of the shape of the die and billet inside the container for extrusion simulation. For molding analysis, the commercial plastic finite element analysis code DEFORM-3D was used. (2) As for the shape of the product mold, the top and bottom and right and left faces were symmetrical, and, to expedite simulation, simulation was performed on 1/4 of the cross section. In molding analysis, the billet was assumed to be a rigid plastic body, while the mold was assumed to be a rigid body without deformation. The initial temperature of the mold and material was set to 550°C, and the material of the mold was SKD6, which is the normal material for hot tool steel. The friction constant (m) was set to 0.6m, which is the value normally applied to un-lubricated hot extrusion.(3) Here, the speed of the material is 6 mm/s, and the external ambient temperature was set to 20°C. Table 2 shows the simulation conditions.

TABLE 2

Condition of FE simulation

Extrusion condition	Value
Material of tools	SKD 11
Material of billet	Al 1100
Initial temperature of tools (°C)	550
Initial temperature of billet (°C)	550
Punch speed (mm/s)	6
Room temperature (°C)	20

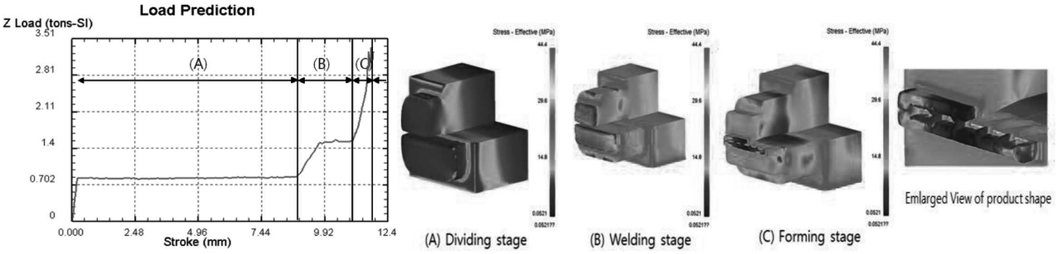


(A)) Sectional view of the heat pipe and enlarged diagram of channels (Unit : mm)

(B) 3D shape of bridge and die

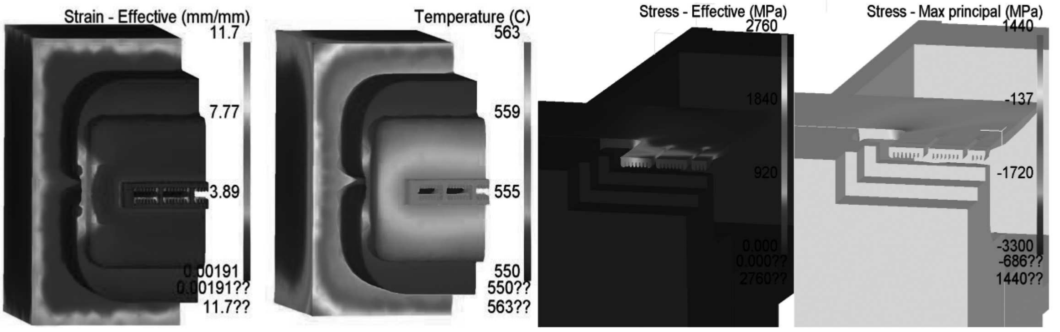
(C) Shape of die and billet in container

Fig. 2. Sectional view of the heat pipe and 3D shape of the mold



(A) Dividing stage (B) Welding stage (C) Forming stage

Fig. 3. Graph of load according to stroke, and stress distribution according to shape change



(A) Strain Ratio (B) Temp. Distribution (C) Effective Stress (D) Max. Principle Stress

Fig. 4. Temperature distribution and strain ratio in the product forming stage

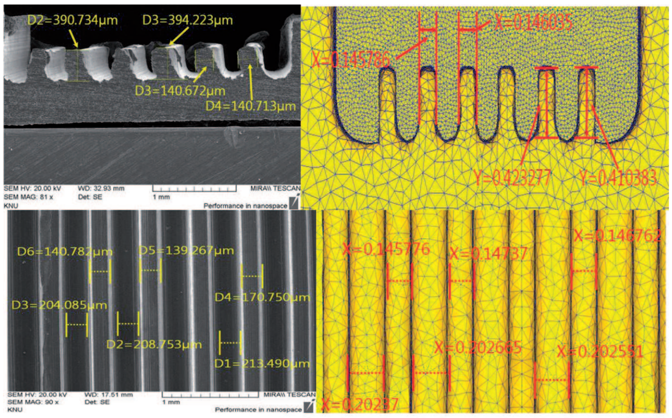
3. Simulation results and performance experiment

3.1. Results of extrusion simulation

Figure 3 shows the load of the punch and change in shape of the billet as occurring during the extrusion process. The changes in shape can largely be categorized into three types. Stage (a) in Figure 3 (Dividing Stage) is the initial stage wherein the billet passed from the container through the port hole inside the bridge to fill the inside. Here, the load increased by approximately 0.7 tons. Stage (b) in Figure 3 (Welding Stage) is the stage where the billet having filled the port hole moved to the chamfer part of the bridge, then filled the chamfer completely. As in the initial stage of movement from the container to the port hole, pressure rose slightly, and the average extrusion pressure increase here was 1.53 tons. Lastly, stage (c) in Figure 3 (Forming Stage), the billet was just about to exit the forming section of the bridge. Here, load again increased rapidly, reaching the maximum value, and the final shape of the heat pipe was formed. Here, the maximum load was 3.3 tons. When procuring the extruder required for production, the selection can be made based on the load values as determined in simulation. It is advisable that the selected extruder has a tolerance that exceeds the load required for production. It is usually advised that the extruder be able to withstand a load at least 1.5 times the load required for the product.

Figure 3 (A) and (B) show the temperature distribution and strain ratio distribution of the billet in the product forming stage. As for the overall strain ratio of the billet, the highest strain was shown in the section where the billet entered the chamber through the port hole, and the section where the shape of the heat pipe was formed. A high strain ratio means that billet flow was relatively fast in a given section, and that deformation resistance was high. As for the temperature distribution, a maximum increase of approximately 16°C from the initial container temperature was observed. It was seen that the temperature increase had a similar distribution to strain ratio. [4] Figure 3 (C) and (D) show the effective stress applied to the mold by the billet, and the maximum principle stress. It can be seen that high stress occurred in the shape forming section of the bridge, overloading the mold. The mold in this study had an average stress distribution of around 1440MPa. As the tensile strength of normal die steel is 1500~2000MPa, this stress corresponds to an average value. The maximum principle stress value is the maximum value of effective stress. This identifies which part of the mold stress was concentrated in. Damage can easily occur near the location where stress is concentrated.

Figure 4 shows the SEM photo of the product cross section, and the dimensions thereof. It can be seen that the cross section as viewed through a high-resolution scanning electron microscope photo and the cross section as expressed through simulation are similar without significant differences. The width of the groove work of the actual product was measured at 140.672 μm and 140.713 μm, while the values gained through simulation, at 0.146035 mm and 0.145786 mm equaling 140.6035 μm and 140.786 μm when converted, were very close to these dimensions. There was some error in height, however, with the actual measured values being 394.223 μm and 390.734 μm while the simulation values were 0.423277 mm and 0.410383 mm, which when converted corresponds to 423.227 μm and 410383 μm. This represents an error of around 1.5%. However, as values within 3% are considered to be within the design tolerance, the simulation values can be considered to be good.



(A) Product cross section photo and dimensions (Unit :µm)

(B) Simulation cross section photo and dimensions (Unit:mm)

Fig. 5. Product cross section SEM photo, and simulation photo and dimensions

4. Conclusion

This study performed mold design through a finite element simulation of the die extrusion process required for the design of a multi-channel heat pipe. Through a comparative analysis of the actual product with simulation results, the following conclusions were reached.

1. The maximum extrusion load of heat pipe production was measured at around 3.3 tons, and, for safe production, an extruder with at least 1.5 times the measured stress value must be used.
2. The speed of the billets moving from inside the port hole to the chamber was the same at the top and bottom. They moved to the forming stage at a constant speed, with the maximum speed of movement being 54.5mm/sec.
3. As for the overall strain ratio and temperature distribution of the billet, the highest strain and temperature change was shown in the section where the billet entered the chamber through the port hole, and the section where the shape of the heat pipe was formed.
4. Stress of about 1440mpa was shown in the section where the shape of the heat pipe was formed, and the mold can be relatively easily deformed or damaged where stress is concentrated.
5. Examination of the dimensions of the groove work with a high resolution scanning electron microscope showed that the widths were very similar. While there was around a 1.5% error in height, as values within 3% are considered to be within the design tolerance, the result can be considered to be good.
6. Through the finite element simulation performed in this study, data allowing for the shortening of time required for die design, production and modification in the field, as well as contributing to the enhancement of product quality, was acquired.

REFERENCES

[1] S.P. Jang, S.J. Kim, K.Y. Paik, Sci. Physical **105**, 211 (2003).

[2] Lee, Jung Min, Kim, Byung Min, Kang, Chung Gil, Jo, Hyung Ho, Kic **21**, 53 (2004).

[3] Lee, Jung Min, Kim, Byung Min, Kang, Chung Gil, Jo, Hyung Ho. Kic **22,** 49 (2005).

[4] J.M. Lee, B.M. Kim, C.G. Kang, Kic **13**, 422 (2004).

[5] K. Plela, L. Blaz, M. Jaskowski, Sci. **58**, 638 (2013).

[6] K. Piela, L. Blaz, Z. Sierpinski, T. Forys, Sci. **57**, 703 (2012).

Received: 20 November 2014.