

Tool cooling simulation for hot forming II. Experiments and Simulations

Takeshi KUROIWA

JSOL Corporation

1 Introduction

To fulfill recent regulations for automobile fuel economy great demand on saving weight of automobiles is growing. Since making a lighter car with conventional material loses occupant safety, at least stiffer materials with the same weight are needed. For example, use of CFRP (Carbon Fiber Reinforced Plastics), Aluminum, Magnesium or Titanium is attracting our attention in these days. But technique to handle these materials is still under developmental stage. High-tensile steels made by hot forming is one of the most promising candidate since it can realize better balance between cost and weight saving.

In hot forming technique, heated blank material is pressed by tools and then quenched by various methods to cause martensitic transition of the blank to obtain high tensile steel. It is not only stiff but also has good shape freezing property, causing smaller springback of the stamped materials. As another advantage of hot forming, steels as raw materials can be easily obtained all over the world, compared with other materials listed above. On the other hand its disadvantages is relatively large investment in plant and equipment such as chiller or cooling tower and costs for prototyping production of tools with pipes to run cooling water.

In order to cause the martensitic transition of the blank materials, one needs to quench it sufficiently fast. We, JSOL, think that a CAE tool to calculate and predict the stiffness of high-tensile materials contributes to ensure their strength in mass production stage. Important points in accurate prediction are following: (i) to calculate phase transition of the materials correctly (ii) to predict cooling performance of the tools to ensure (i). By making these uncertainties clear it is expected that CAE is capable of reducing trials-and-errors on prototyping, causing reduction of tool designing costs.

LSTC, Dynamore and JSOL have been working on formulating a manufacturing CAE solution to the hot forming techniques. For example, development of phase transition material models (*MAT_244, *MAT_254) will overcome the uncertainty (i) described above. We also have been investigating simulation technique for thermal-structural-fluid coupling calculation to demonstrate the behavior of cooling water flowing in pipes of the tools, corresponding to item (ii) above. In this paper we report the results of recent solution developments on the latter point.

2 Motivation

Our goal is twofold. First is to predict cooling ability of tools when their CAD data is given. Second is to provide a designer's CAE tool to predict the distribution of martensitic phase ratio of blank material for multi-shot process, as a result of accomplishment of the first goal. To achieve these, we have to calculate correctly deformation of work material, temperature of work, die and cooling water, together with velocity and pressure of the water. Such problem is classified to the multiphysics problem, which should be tackled by next generation CAE. As a first step toward our goal we here only consider tools and cooling water and so far ignore the existence of the workpiece.

In order for the simulation of tool cooling to reduce the number of trial-and-error in prototyping of the tools, its calculation time should be much shorter than the time needed to create tools. To this end we have to minimize the total hours from creation of CAD data to the estimation of cooling performance of the tools. It can be divided into the time to prepare the simulation data and calculation hours. Here we put our focus on the latter.

In our previous paper[1] we have showed that recent development in LS-DYNA enables us to simulate tool cooling in reasonable calculation time. In this paper we report the result of validation for modelling of tool cooling simulation using LS-DYNA.

3 Experiment

An experiment on tool cooling is carried out. A tool is heated by using an electric oven upto 80 degree and then cooled by water with 20 degree through the cooling pipe equipped inside of the tool. The geometry of tool with cooling pipe is shown below.

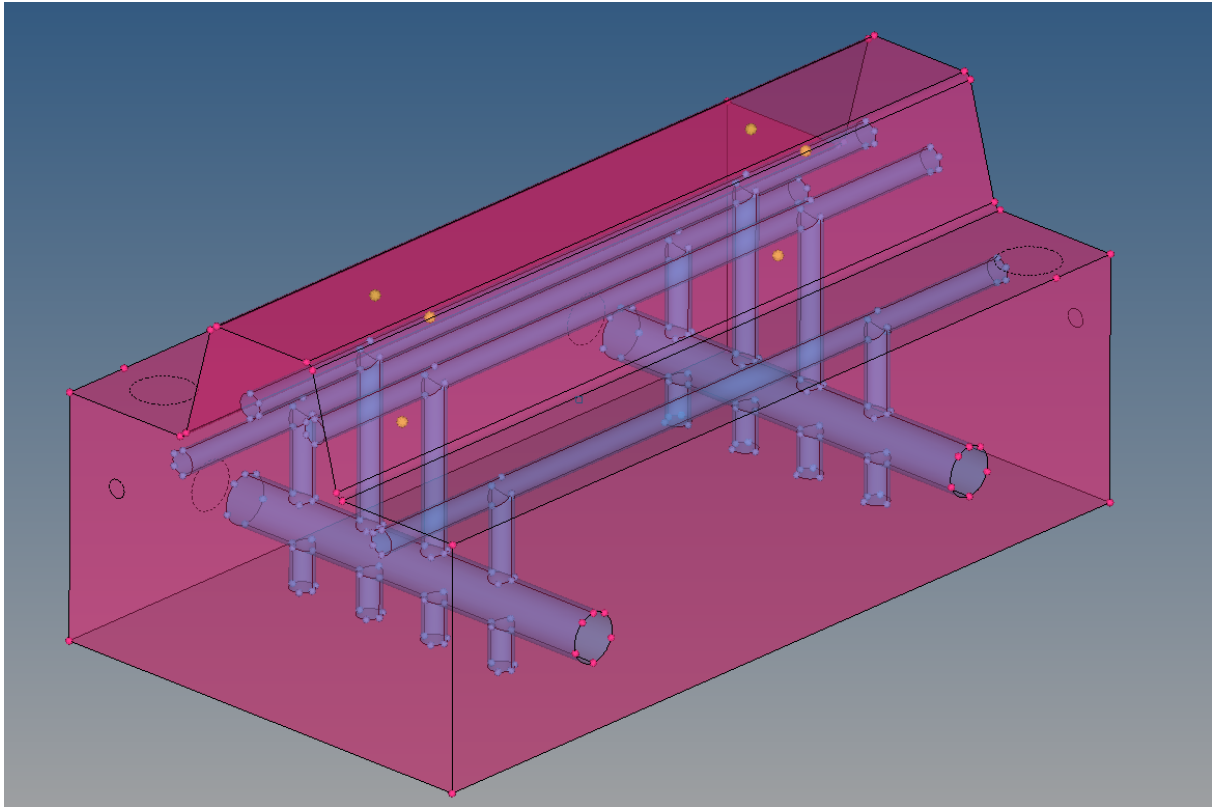


Fig.1: overveiw of tool with cooling pipe.

The size of the tool is 200mm in the longer side, 120mm in the shorter side. Height of the convex shape is 40mm. The diameter of the cooling pipe is 16mm. It branches to 4 narrow pipe with their diameter being 8mm. Then they join to one pipe together again and it goes out of the tool. Temperature of some points inside the tool is measured throughout the experiment to see how cooling water exchanges heat with hot tool. Measured points are shown in Fig.2:.. Inlet temperature is 20 degree and temperature of the atmosphere is around 20 degree throughout the experiment. Inlet volume flux is 5.4L/min.

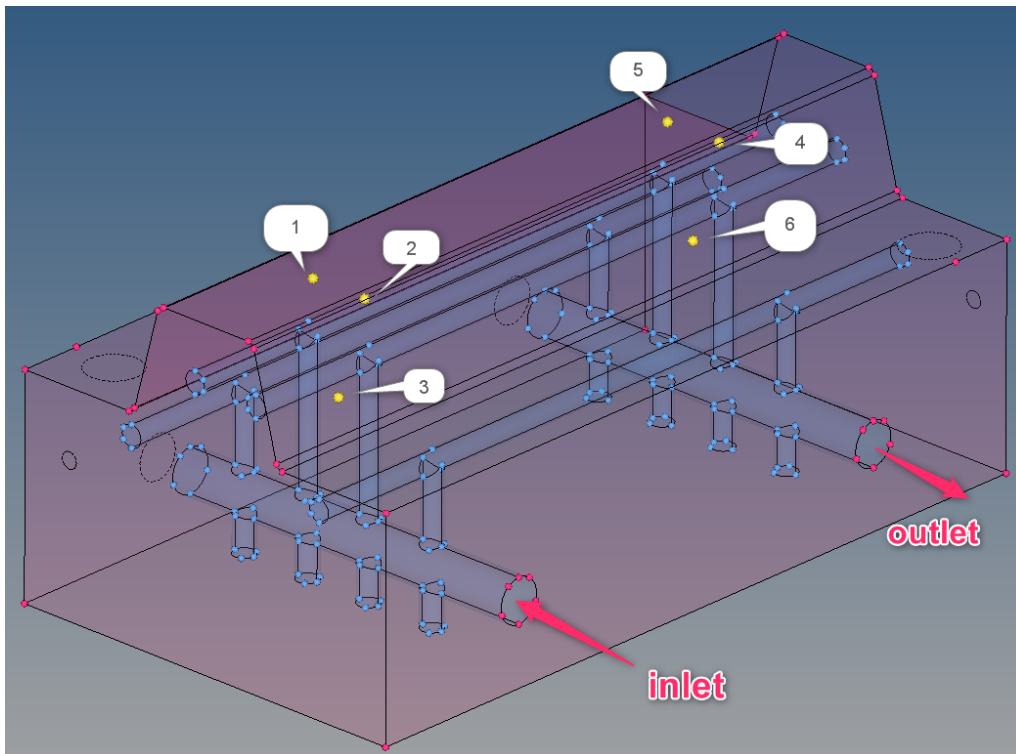


Fig.2: measured point are shown in yellow dots. Inlet and outlet of the cooling water are also shown.

The result of temperature measurement is shown in Fig.3:. Here $t=0$ is defined by the time when the cooling water starts flowing into the vacant pipe in the tool.

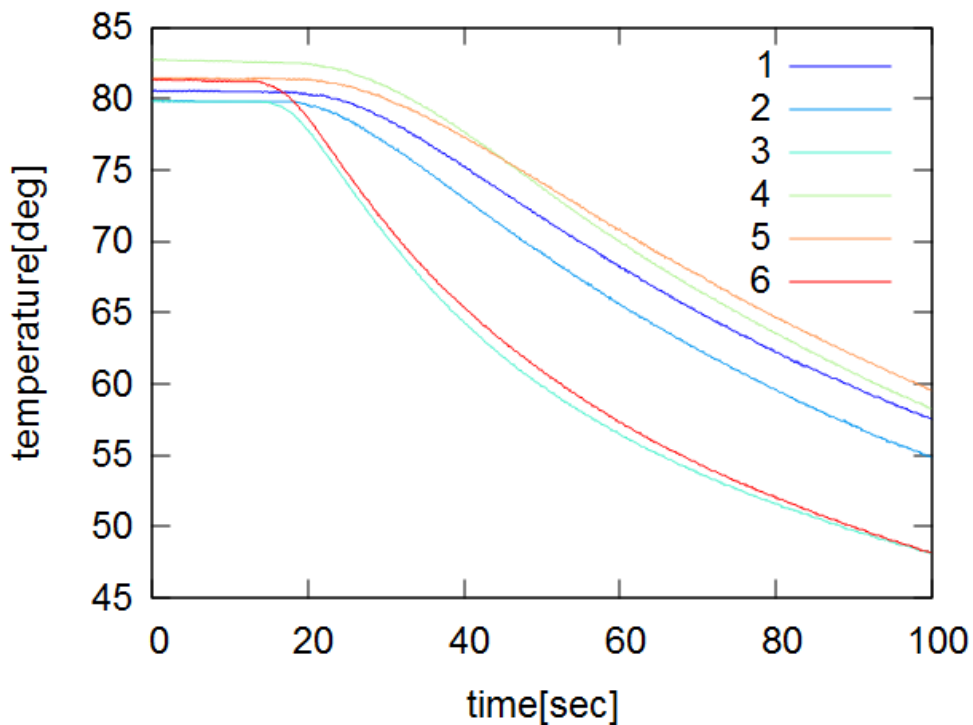


Fig.3: temperature of measured points obtained in the experiment.

4 Simulation

A finite element model for tool cooling simulation is constructed.

Table 1: Summary of number of nodes and elements for the benchmark model used. All elements are tetrahedra.

	# nodes	#elems
Structure	280,762	1,443,390
Fluid	441,305	2,423,487

Physical quantities used here are summarized in the table below. We here employed ***MAT_RIGID** for material of the solid.

Table 2: Summary of material constant employed throughout the simulation.

Physical quantity	unit	Value
Density of fluid	[ton/mm ³]	9.982e-10
Viscosity of fluid	[MPa s]	1.002e-9
Heat capacity of fluid	[mJ/(ton K)]	4.181e+9
Heat transfer coefficient of fluid	[W/(m K)]	0.58
Density of tool	[ton/mm ³]	8.0e-9
Heat capacity of tool	[mJ/(ton K)]	6.5e+8
Heat transfer coefficient of tool	[W/(m K)]	32.0

A fluid-structure interaction with conjugate heat transfer calculation is carried out. Flow of the cooling water is modeled by using ICFD function. Conjugate heat transfer between tool and water is available as a part of FSI capability in LS-DYNA. To save calculation time we first calculated the steady state of the flow velocity and pressure and then pass them to the conjugate heat transfer solver. By using this technique one has twofold advantage in calculation time:

1. save time on update fluid pressure and velocity after steady state is reached
2. be free from the Courant condition in solving Navier-Stokes equation and as a result larger timestep can be employed

These can be realized just using ***ICFD_CONTROL_STEADY** to turn on the steady state solver and ***ICFD_BOUNDARY_FSI** together with ***ICFD_BOUNDARY_CONJ_HEAT** to solve conjugate heat transfer across the tool and the water. Timestep of the conjugate heat transfer solver can be changed using lcts of ***CONTROL_THERMAL_TIMESTEP** so that it becomes gradually large as time passes.

In the simulation the cooling water is filled at t=0 with T=20 deg. The steady state of the flow is calculated without solving conjugate heat transfer and after the steady state is reached Navier-Stokes calculation is stopped and conjugate heat transfer calculation starts. Thus the CAE model does not completely reproduce the experimental condition. If one would like to model the physical process of filling in of the water one has to carry out unsteady simulation, which is lot more computationally cost consuming. This discrepancy will affects the behaviour around t=0 but after a long time it is expected to be negligible.

RANS standard k-epsilon model is adopted with standard wall law at first boundary layer node. The value of y+ is around 10.

The simulation is carried out at 64 parallel execution with 4 nodes cluster. CPU is Intel(R) Xeon(R) CPU E5-2670 0 @ 2.60GHz. Each nodes has 128GB RAM. Total calculation time is about 4 hours for 100sec simulation time.

5 Result

To see the effect of the FSI we first show a result of the calculation without FSI. Here cooling water is modeled as constant temperature boundary condition with $T=20$ deg. Thus flow of the water is completely neglected. The result is shown in Fig.4:. Since there is no effect of heating up of the coolant the difference in the temperature at upper stream point and downstream one is completely identical, while in the experiment temperature at the downstream point is relatively higher than that of upstream since temperature of the water is grows continuously through the downstream. Moreover it can be seen that temperature predicted by the simulation is always lower than experimental result. This is also due to the ignorance of the water. The temperature at the surface of the pipe is not equal to the temperature of the inflow water. The former is always higher as a result of conjugate heat transfer between tool and water.

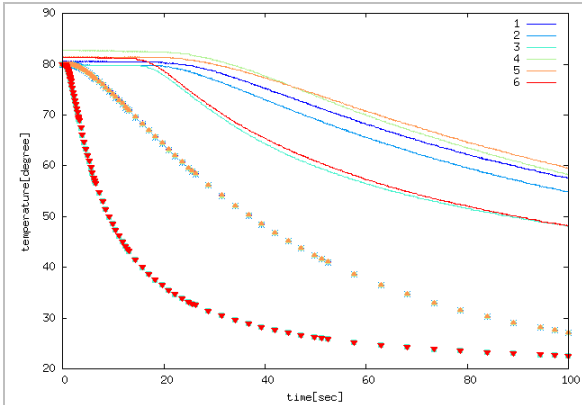


Fig.4: temperature of measured points. line represents experimental result and symbol with corresponding color is for simulation.

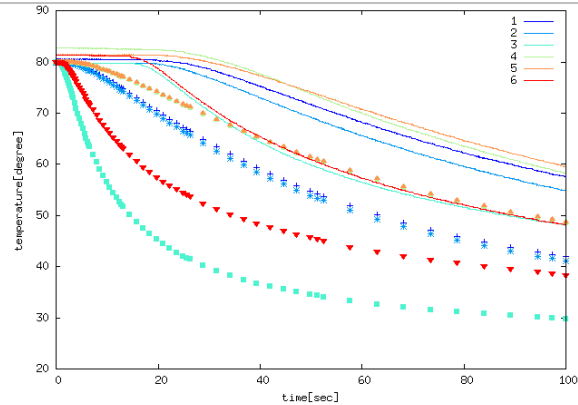


Fig.5: same as Fig.4: but with FSI calculation.

The result of the FSI calculation reviewed in Section 4 is shown in Fig.5:. One can see that temperature history for point 1 and 5, 2 and 4, 3 and 6, are now become different. This is due to the heat up of the water in the downstream. One can also see that simulation still underestimate the temperature overall.

Throughout the 2 cases a time when the temperature starts to drop is not reproduced. Simulation always predicts the time earlier than experiment. This comes from the difference in modelling in the inflow water at the beginning of the cooling,

6 Summary and discussion

A CAE study on tool cooling experiment is carried out with taking conjugate heat transfer between tool and water into account. The result becomes better than structure-only calculation, but still there are several things to be explored more. The most important point neglected in the FSI model is resolution of laminar boundary layer of temperature. The width of temperature boundary layer for water is about 7 times thinner than that of velocity. Thus one needs finer boundary layer mesh if laminar boundary for temperature is assumed, to predict heat transfer in the fluid accurately. It is almost impossible to carry out a calculation on such mesh since it needs much more computational resources. In order to overcome this difficulty wall function for temperature boundary layer is now being developed.

Accurate heat prediction of heat transfer at the wall without wall function for temperature is possible only in 2D. Here we carried out 2D heat transfer of water in a channel to show how fine mesh is needed. An overview of the model is described in Fig.6:.

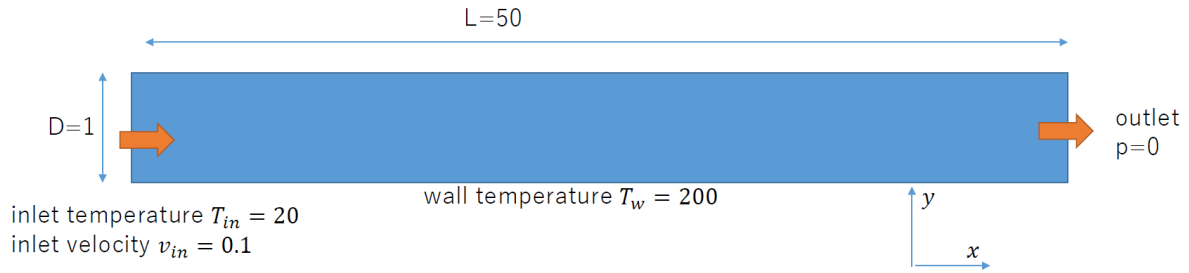


Fig.6: overview of 2d channel flow model

The model only contains fluid nodes. Fluid with temperature 20 deg flows into the channel with constant temperature 200 deg. Height of the channel is $D=1$ and length is $L=50$. Material constant of the fluid is same as that in Table 2. The Reynolds number of the flow is 10000, which is close to the case in section 4. 10 boundary layer mesh is inserted at the side wall. Dependence of heat flux at wall on mesh size is explored and is shown in Fig.7.:

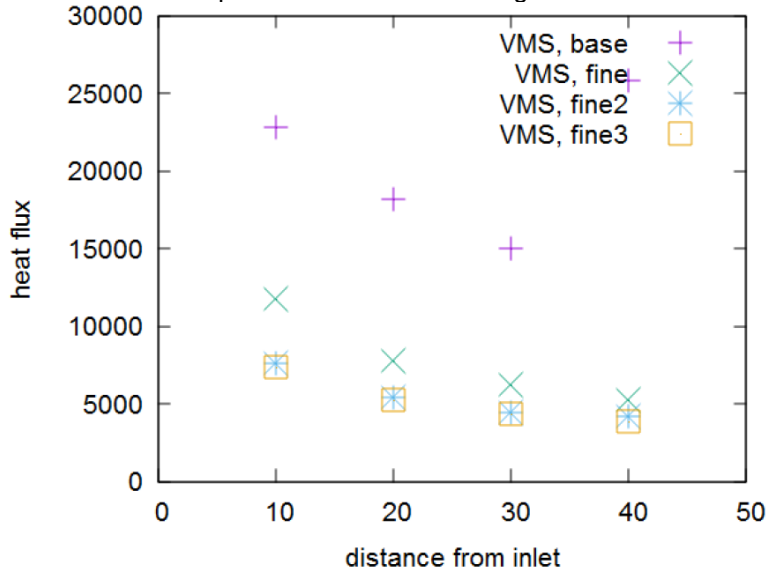


Fig.7: heat flux dependence on mesh size.

Here base, fine, fine2,... are identifier of the mesh. Mesh size for each names are described in Table 3.:

Table 3: Mesh identifier and their names. Values of y^+ for velocity realized is also shown

Identifier	Mesh size	y^+
base	0.05	0.08
fine	0.025	0.04
fine2	0.0125	0.02
fine3	0.00625	0.01

One can see that fine2 mesh is sufficiently fine and solution is converged. fine2 has mesh size 0.0125 thus inlet boundary is divided into 80 elements. We then assume that in 3D we need at least 1/80 of the characteristic length(=diameter of the pipe) for pipe flow case described in section 4. y^+ of the boundary layer to obtain accurate solution without wall function from the 2D simulation is shown as 0.02. These estimates show us that we need mesh size 0.1mm with boundary layer height 0.0007 in the 3D case, which leads to more than 300 million elements needed to resolve the conjugate heat transfer without using wall function for temperature. It is practically impossible to execute a simulation for such number of element in conventional computer cluster. This is the reason why wall function for temperature is needed to be implemented.

7 Literature

- [1] Kuroiwa, T: "Tool cooling simulation for hot forming", 11th European LS-DYNA conference, 2017,