Heat Transfer in LS-DYNA

Authors:

Arthur B. Shapiro
Livermore Software Technology Corporation

Correspondence:

Arthur B. Shapiro
Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, CA 94550, USA

Telephone: +1 925 449 2500
Email: Shapiro@lstc.com

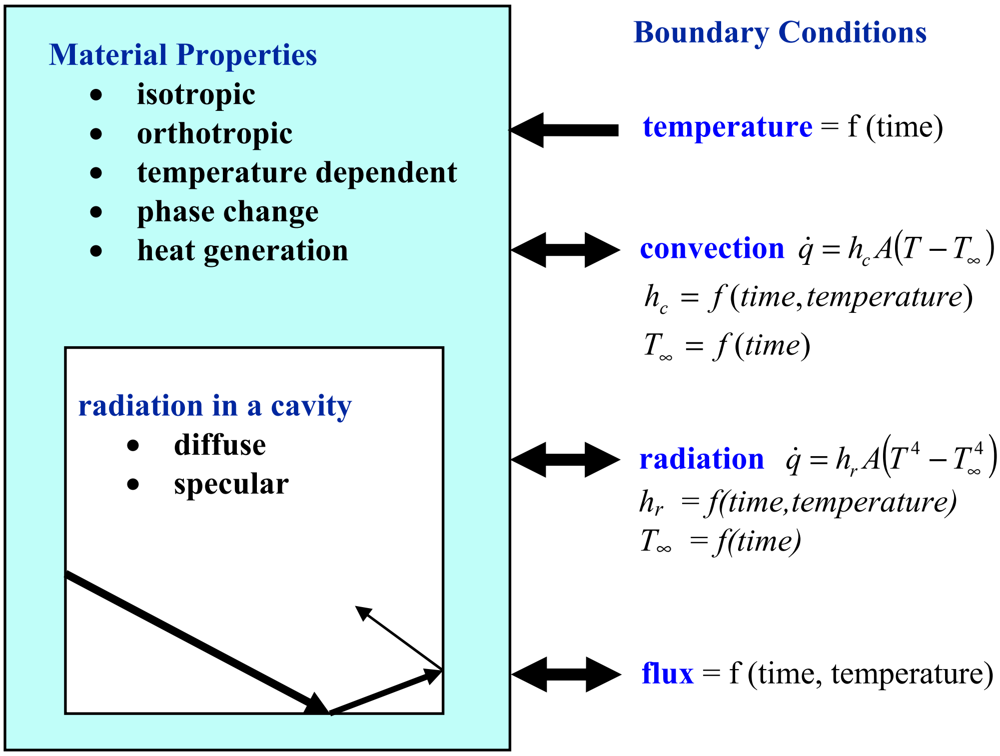
Keywords:
Heat transfer, thermal stress, welding, casting, metal forming

ABSTRACT

LS-Dyna can solve steady state and transient heat transfer problems on 2-dimensional parts, cylindrical symmetric parts (axisymmetric), and 3-dimensional parts. Heat transfer can be coupled with other features in LS-DYNA to provide modeling capabilities for thermal-stress and thermal-fluid coupling. This paper presents several examples using LS-DYNA for modeling manufacturing processes (e.g., metal forming, welding, casting).

INTRODUCTION

Heat transfer modeling options in LS-DYNA are summarized in the following figure.



Boundary conditions include temperature, flux, convection, and radiation. Material properties can be isotropic or orthotropic. Thermal conductivity and heat capacity can be functions of temperature. The material can undergo solid-liquid phase change and be defined with a heat generation rate that can be a function of time and temperature. Enclosure radiation heat transfer can be modeled using diffuse or specular surfaces.

Other features are described in the boxes below:

Element Types

- 8 node brick & 4 node quad
- 4 node shell (membrane)
- 12 node quadratic shell

Element Integration

- 1 point quadrature => best for coupled thermal-stress problems
- 8 point quadrature => best for thermal only problems

Analysis Type

- steady state
- transient
  - fully implicit scheme
  - Crank Nicolson scheme

Equation Solver

- direct (Gauss) solver
- conjugate gradient iterative
  - diagonal preconditioned
  - incomplete Choleski

The box below shows the minimum number of keywords required for a thermal analysis. Other keywords are available to define initial conditions, boundary conditions, contact surfaces, heat generation, material properties, and special features (e.g., weld heat source, thermostat). The LS-Dyna Keyword User Manual [1] should be consulted for a description of these keywords.

```
*CONTROL_SOLUTION
          1
*CONTROL_THERMAL_SOLVER
          1
*CONTROL_THERMAL_TIMESTEP
          0         1.         .1
*MAT_THERMAL_ISOTROPIC
          1      2700.          0   2.44e+07
       904.       222.
```

0 = mechanical
1 = thermal
2 = mech + therm

1 = Gauss
3 = iterative
>11 MPP

Thermal material properties

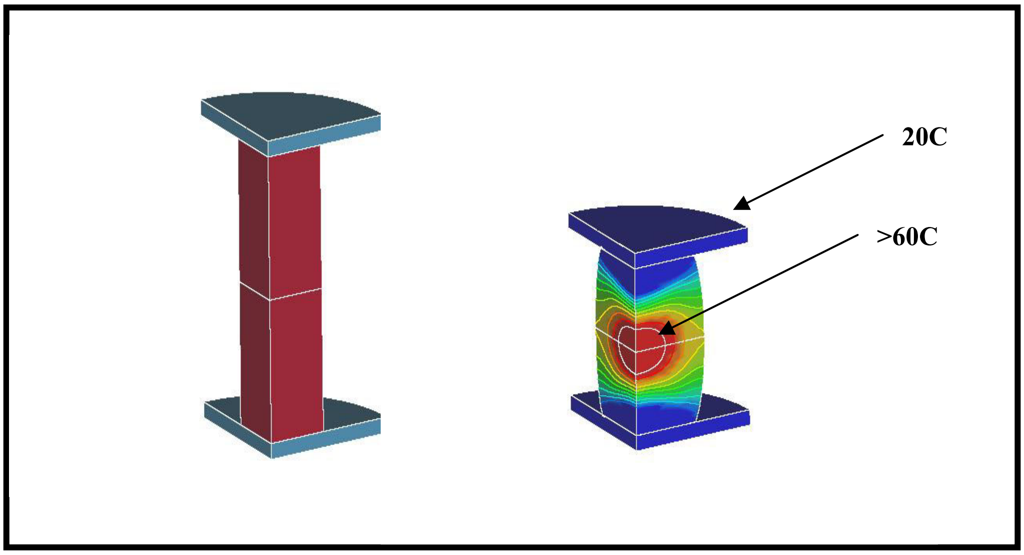
The following mechanical constitutive models have the thermal coefficient of expansion in their definition and calculate thermal strains.

- Type 4 *MAT_ELASTIC_PLASTIC_THERMAL
- Type 21 *MAT_ORTHOTROPIC_THERMAL
- Type 106 *MAT_ELASTIC_VISCOPLASTIC_THERMAL

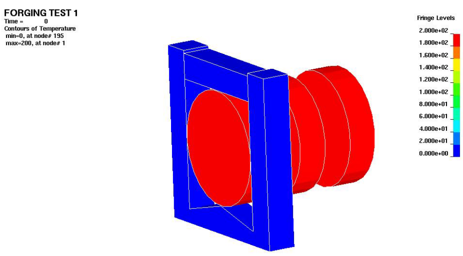
Plastic Work to Heat

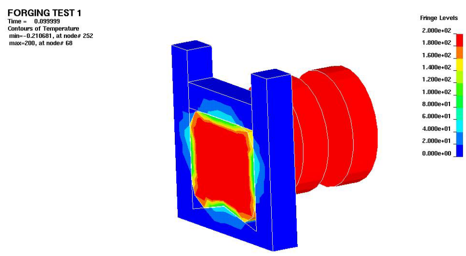
In metal forming processes such as rolling, extrusion, and stamping, large plastic deformation of the metal occurs. The temperature of the body changes due to the conversion of mechanical work into heat through plastic deformation. The upsetting process is defined as the axial compression of a cylinder between two perfectly rough, insulated plates. The process is fast enough such that there is no heat transfer with the environment.

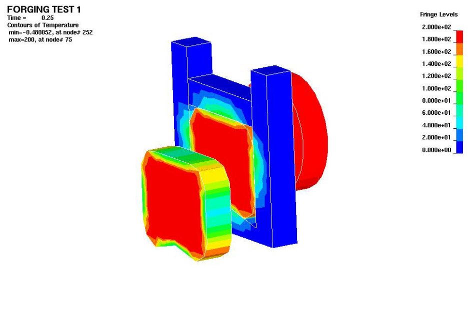
The upsetting problem shown in the figure below is described in detail in a publication by J. van der Lugt [2]. The low carbon steel sample has an initial height of 36 mm, radius of 9 mm, and initial temperature of 20 C. The total compression is 44% (i.e., $\Delta h/h$) in 1.6 seconds. Note that the top and bottom surfaces of the cylindrical specimen are constrained in the radial direction. The temperature increase of the cylinder is completely due to the conversion of plastic work to heat

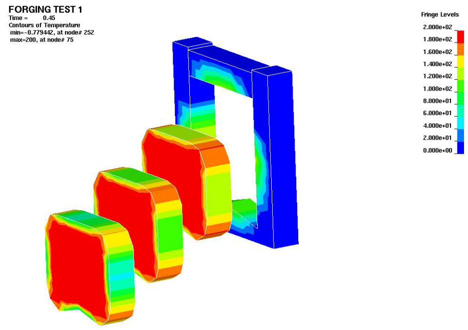


The following simplified forging problem demonstraes thermal-mechanical coupling across contact interfaces. This problem models the start-up time of a forging process in which many hot blanks pass through the dies. Each hot blank raises the temperature of the dies until a steady state process is achieved.







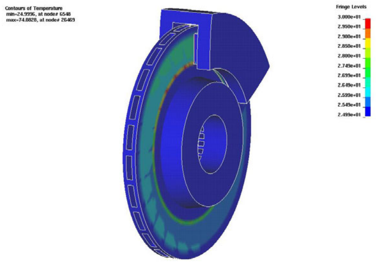


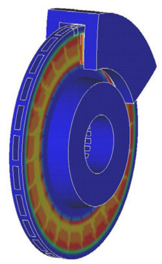
Friction to Heat

LS-DYNA can solve for the frictional energy due to sliding at the interface between 2 parts**.** The parameter FRCENG on the *CONTROL_CONTACT keyword is used to activate the calculation of frictional sliding energy. The frictional energy is divided equally between the surfaces. The frictional energy is converted into heat in a coupled thermal-mechanical simulation.

Modeling the conversion of sliding frictional energy to heat is important in the design of automotive disc brake systems. As the rotor temperature increases, the temperature gradient can cause rotor deformation due to thermal expansion. This can lead to banded hot-spotting, tapered wear, and noise.

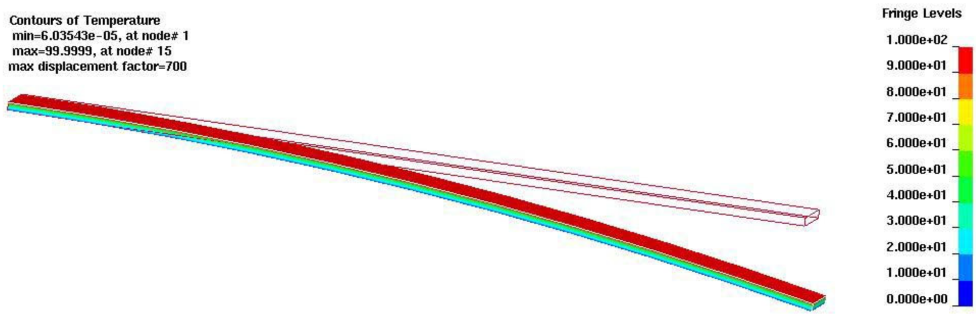
The following figures show the disk brake at two different times during the simulation. The rotor is given an initial rotational velocity and a force is applied to the pads. All parts start with an initial temperature of 25C. The figure on the left shows temperature fringes after a few rotations of the rotor. The rotor is heating up due to frictional heating. In this problem, the pads are making greater contact with the rotor near the rotor hub. This is indicated by the higher temperature pattern (i.e., red). The figure on the right shows the temperature pattern when the rotor has come to rest. The cooler (i.e., yellow) radial lines on the rotor are due to the rotor ribs acting as a heat sink. A movie showing the results can be found on www.feainformation.com. Click the “AVI Library” link in the left hand menu and then go to AVI movie 609 under the heading “Contact”.

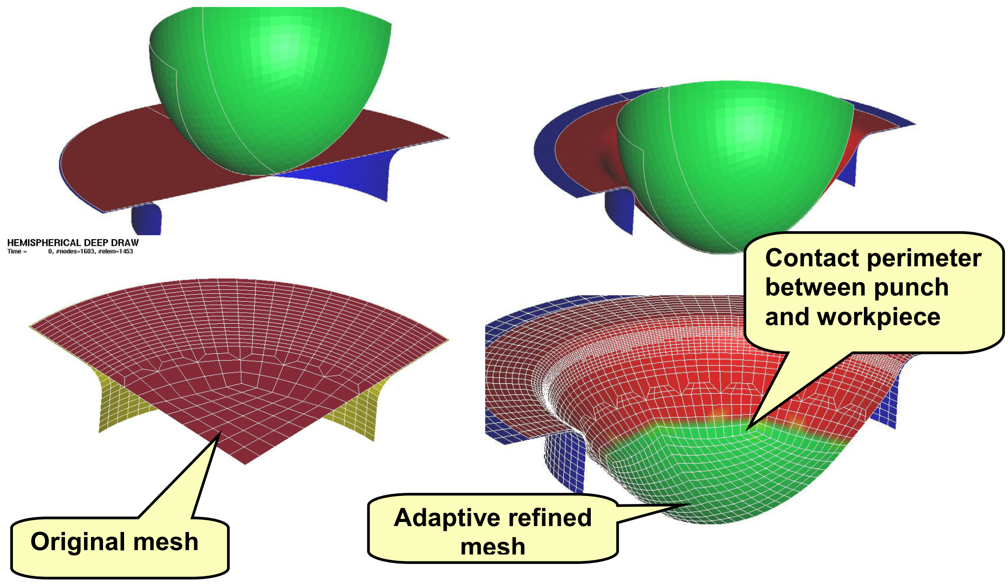




Thick Thermal Shell

The through thickness temperature gradient may be important during non-isothermal forming of sheet metal parts. The sheet metal part is modeled using 4-node shell elements. If the parameter TSHELL=1 is set on the *CONTROL_SHELL keyword, then this 4-noded shell is treated as a 12-node thermal shell [3] by LS-DYNA. Another way of thinking about this is that each of the 4 nodes has 3 degrees of freedom: (1) a mid-plane temperature, (2) a bottom surface temperature, and (3) a top surface temperature. Quadratic shape functions are used to interpolate temperature through the thickness of the shell and bi-linear shape functions are used in the plane of the shell. The through thickness temperature gradient causes thermal strains. The following figure shows a cantilever beam modeled using 4 node shell elements that are flagged as thick thermal shells. All nodes (including the fictitious bottom and top surface nodes) are given an initial temperature of 50. The word “fictious” is used because the user does not define node numbers for the top and bottom surface nodes. The user only defines the 4 mid-plane nodes. The temperature of the top surface of the beam is ramped up to a temperature of 100 and the bottom surface is ramped down to a temperature of 0. The top of the beam expands and the bottom of the beam contracts. The thermal strains created by this temperature gradient causes the beam to bend as shown in the figure below.





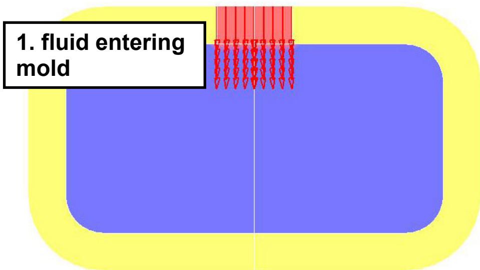
A more complex metal forming problem is shown in the preceding figure. The initially flat work piece is modeled using 4-node mechanical shells that are flagged to be treated as thick thermal shells. The LS-DYNA contact algorithm notes that thick thermal shells are present. This is very important because the driving potential for energy transport is the temperature difference between the dies and the shell top or bottom surface temperature and not the mid-plane temperature. This problem also demonstrates mesh adaptivity.

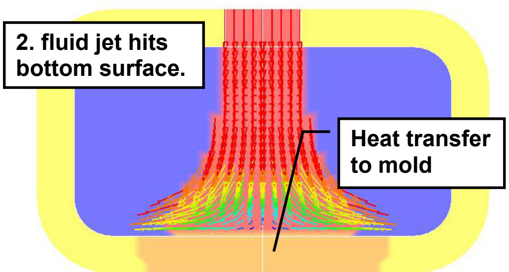
Coupled thermal-fluid-mechanics

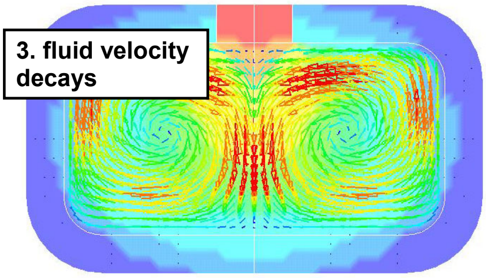
This hypothetical plate casting problem demonstrates features in LS-DYNA to model filling a mold with liquid metal and heat transfer during solidification and cool down to room temperature. The mold is modeled as a Lagrangian solid. The volume within the mold can be modeled in two ways: (1) as a single material Euler element plus void which models metal filling a cavity under vacuum; or (2) multi-material ALE element which models metal filling a cavity initially filled with air. The element formulation parameter (ELFORM) on the *SECTION_SOLID keyword card is used to distinguish between these two options. Three modeling stages were defined to solve this problem. Switching between stages is accomplished by birth and death times defined by Keyword input and specified by load curves.

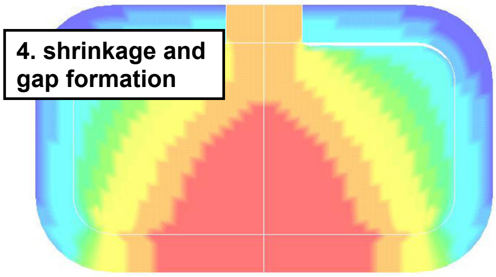
1. **Liquid metal inflow** is specified by the ambient element type (AET) parameter on the *SECTION_SOLID keyword card. The ambient element nodes are given a velocity boundary condition to model the inflow velocity of metal into the mold. Liquid metal inflow is turned off when the mold is full
2. **Fluid velocity decays** as the fluid transitions from liquid properties to solid properties during solidification. The Eulerian fluid mesh is switched to a Lagrangian mesh when the fluid circulation velocity is low enough. This switch point was determined by trial-and-error. If the velocities were to high when switching to a Lagraingian mesh, the nodes with high velocity tended to tear the mesh apart. The contact surface between the Eulerian fluid mesh and Lagrangian mold was also switched at this time from tied to sliding with voids.
3. **Part shrinkage** occurs during cool down to room temperature. Shrinkage causes gaps to open between the cast part and mold.

The following figures show these 3 modeling stages of the process.









Conclusion

MPP thermal and thermal-stress coupling is being implemented in LS-971. Both Gauss type sparse solvers and iterative methods are available to solve the FEM equations.

References

1. ,LS-DYNA Keyword User's Manual, Version 960, Vol. 1-2, March 2001.

2. J. Van der Lugt, "Thermal Mechanically Coupled Finite Element Analysis in Metal Forming Processes", Computer Methods In Applied Mechanics and Engineering, Vol. 54 (1986) p145-160.

3. G. Bergman & M. Oldenburg, "A Finite Element Model for Thermo-mechanical Analysis for Sheet Metal Forming", Dept. of Mechanical Engineering, Lulea Univeristy of Technology, Lulea Sweden.