

# **LS-DYNA**

## **Thermal Analysis User Guide**

**August 1999**

Copyright © 1999  
**LIVERMORE SOFTWARE**  
**TECHNOLOGY CORPORATION**  
All Rights Reserved

**Mailing Address:**

**Livermore Software Technology Corporation  
2876 Waverley Way  
Livermore, California 94550-1740**

**Support Address:**

**Livermore Software Technology Corporation  
7374 Las Positas Road  
Livermore, California 94550**

**TEL: 925-449-2500**

**FAX: 925-449-2507**

**EMAIL: [sales@lstc.com](mailto:sales@lstc.com)**

**Copyright © 1999 by Livermore Software Technology Corporation**

**All Rights Reserved**

## Introduction

LS-DYNA can solve steady state and transient heat transfer problems on 2-dimensional plane 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 document presents several "very simple" examples in using LS-DYNA for heat transfer, coupled thermal-stress, and coupled fluid-thermal problems.

The input files presented below use the `KEY-WORD` structure for clarity. The structured LS-DYNA input file requires that thermal control parameters be defined on control cards 27-30, followed by thermal material definition cards and thermal boundary condition cards. LS-DYNA distinguishes between a structural, thermal, or coupled structure-thermal analysis by the `"CONTROL_SOLUTION"` keyword or by specifying "thermal" or "couple" on the LS-DYNA execution line.

The following problems demonstrate using LS-DYNA for heat transfer and coupled thermal-stress.

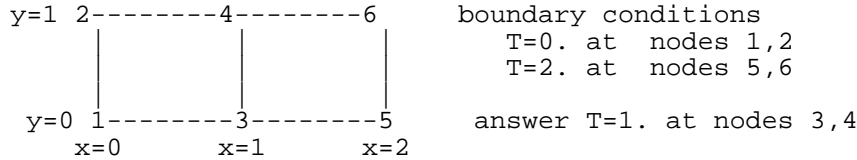
1. **Steady State Heat Transfer** - steady state heat transfer in a slab using shell elements
2. **Transient Heat Transfer** - transient heat transfer in a rod using 8-node brick elements
3. **Thermal Stress** - unconstrained expansion of a block due to heating

The following problems demonstrate using LS-DYNA for manufacturing problems where modeling coupled fluid, thermal, and mechanical effects are important.

4. **Welding** – demonstrates the use of a thermal-mechanical slide surface
5. **Metal Forming** – demonstrates modeling of the conversion of plastic work to heat

**Problem 1:** Steady State Heat Transfer in a Slab Using Shell Elements

This problem demonstrates using LS-DYNA to solve a steady state, 2-dimensional, heat transfer problem with temperature boundary conditions. The figure below defines the geometry, mesh, and boundary conditions.



The analytical solution for the temperature distribution in a slab of thickness  $l$  with prescribed temperatures  $T_0$  at  $x=0$  and  $T_l$  at  $x = l$  is

$$\frac{T - T_0}{T_l - T_0} = \frac{x}{l}$$

The answer to this problem is  $T=1$  at  $x=1$ .

Note the following model parameters defined in the input file that follows:

1. The input is defined using shell elements with a shell formulation of 12 for a plane geometry (note: use shell formulation 14 for an axisymmetric geometry).
2. Direct and iterative solvers are available to solve the system of equations for the heat transfer calculations. The solver to be used is specified in (`*CONTROL_THERMAL_SOLVER`). This problem uses the direct solver ACTCOL, which is a Gauss type profile solver. The diagonal scaled preconditioned conjugate gradient solver, DSCG, uses much less memory and is faster on most problems than the direct solvers.

**LS-DYNA Input File:**

```
*KEYWORD
$
$      1      2      3      4      5      6      7      8
$
$=====CONTROL DEFINITIONS =====
$
*TITLE
steady state conduction in a slab
*CONTROL_SOLUTION
  1
*CONTROL_THERMAL_SOLVER
  0      0      1
*CONTROL_TERMINATION
  1.
*DATABASE_TPRINT
  1.
```

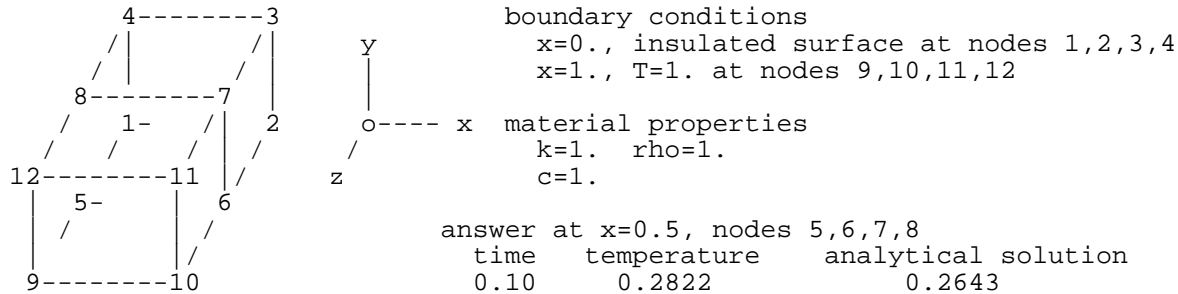
```

*DATABASE_BINARY_D3PLOT
  1.
$
$===== PART DEFINITIONS =====
$
*PART
$      PID      SECID      MID      TMID
slab      1      1
$
$===== SECTION PROPERTIES =====
$
*SECTION_SHELL
$      SECID      ELFORM
      1      12
      0.001      0.001      0.001      0.001      0.
$
$===== THERMAL MATERIAL PROPERTIES =====
$
*MAT_THERMAL_ISOTROPIC
      1      1.
      1.      1.
$
$===== NODE DEFINITIONS =====
$
*NODE
      1      0.      0.
      2      0.      1.
      3      1.      0.
      4      1.      1.
      5      2.      0.
      6      2.      1.
$
$===== ELEMENT DEFINITIONS =====
$
*ELEMENT_SHELL
      1      1      1      3      4      2
      2      1      3      5      6      4
$
$===== THERMAL BOUNDARY CONDITIONS =====
$
*BOUNDARY_TEMPERATURE_NODE
      1      0      0.
      2      0      0.
      5      0      2.
      6      0      2.
*END

```

**Problem 2:** Transient Heat Transfer in a Rod Using 8-Node Brick Elements

This problem demonstrates using LS-DYNA to solve a transient, 3-dimensional, heat transfer problem with temperature boundary conditions. The figure below defines the problem.



The analytical solution to this problem is:

$$T(x,t) = T_{BC} + (T_{IC} - T_{BC}) \frac{2}{\pi} \sum_{n=0}^{\infty} \exp \left[ -F\pi^2 \left( n + \frac{1}{2} \right)^2 \right] \frac{(-1)^n}{(n + 1/2)} \cos(n + 1/2)\pi \frac{x}{L}$$

With the Fourier number defined as,  $F = \alpha t/L^2$ , and the thermal diffusivity defined as,  $\alpha = k/\rho c$ .

Parameters for this problem are:

thermal conductivity	$k = 1 \text{ W/m C}$
density	$\rho = 1 \text{ kg/m}^3$
heat capacity	$c = 1 \text{ J/kg C}$
temperature initial condition	$T_{IC} = 0.$
temperature boundary condition	$T_{BC} = 1. \text{ at } x = 1.$

Note the following model parameters defined in the input file that follows:

1. Transient thermal problems are solved using implicit time integration. Therefore, there is no stability condition on the thermal time step. Much larger time steps can be used for the thermal solution as opposed to the mechanical solution, which uses explicit time integration. Time step size is set in (\*CONTROL\_THERMAL\_TIMESTEP).
2. Transient thermal problems are solved using a generalized trapezoidal time integration algorithm. Two special cases are the Crank Nicholson method,  $\alpha = 0.5$ , and the fully implicit method,  $\alpha = 1.0$ , which are defined in (\*CONTROL\_THERMAL\_TIMESTEP). Although, the Crank Nicholson method is second order accurate in time, it can introduce oscillations in the solution. These oscillations can cause nonlinear problems to diverge. Therefore, we suggest using the fully implicit method for nonlinear problems.

3. Insulated boundary conditions, such as exist in this problem for the surface at  $x=0$ ., are the default and don't need to be specified in the input file.

**LS-DYNA Input File:**

```

*KEYWORD
$
$          1          2          3          4          5          6          7          8
$-----CONTROL DEFINITIONS -----
$
*TITLE
transient conduction in a slab
*CONTROL_SOLUTION
    1
*CONTROL_THERMAL_SOLVER
    1          0          3
*CONTROL_THERMAL_TIMESTEP
    0          1.          .01
*CONTROL_TERMINATION
    .1
*DATABASE_TPRINT
    .02
*DATABASE_BINARY_D3PLOT
    .01
$
$-----PART DEFINITIONS -----
$
*PART
$      PID      SECID      MID      TMID
slab      1          1
$
$-----SECTION PROPERTIES -----
$
*SECTION_SOLID
$      SECID      ELFORM
    1          1
$
$-----THERMAL MATERIAL PROPERTIES -----
$
*MAT_THERMAL_ISOTROPIC
    1          1.
    1.          1.
$
$-----NODE DEFINITIONS -----
$
*NODE
    1          0.          0.          0.
    2          1.          0.          0.
    3          1.          1.          0.
    4          0.          1.          0.
    5          0.          0.          .5
    6          1.          0.          .5
    7          1.          1.          .5
    8          0.          1.          .5
    9          0.          0.          1.
   10         1.          0.          1.
   11         1.          1.          1.
   12         0.          1.          1.
$
$-----ELEMENT DEFINITIONS -----

```

```
$
*ELEMENT_SOLID
      1      1      1      2      3      4      5      6      7      8
      2      1      5      6      7      8      9     10     11     12
$
$===== THERMAL BOUNDARY CONDITIONS =====
$
*BOUNDARY_TEMPERATURE_NODE
      9      0      1.
     10      0      1.
     11      0      1.
     12      0      1.
*END
```



**Problem 3:** 3D Thermal Stress

This problem demonstrates using LS-DYNA to solve for the unconstrained expansion of a block due to heating. The model consists of one 8 node brick element at an initial temperature of 10C. The brick material is given a volumetric thermal generation rate. The rise in temperature can be calculated by equating the thermal generation rate to an increase in internal energy.

$$qVt = mc\Delta T$$

Where:

volumetric heat generation rate	$q = 10 \text{ W/m}^3$
volume	$V = 1 \text{ m}^3$
mass	$m = \rho V = (1.)(1.) = 1. \text{ kg}$
heat capacity	$c = 1 \text{ J/kg C}$
time for heat generation	$t = 3 \text{ sec}$

The block increases in temperature by 30 C. The final block temperature is 10C + 30C = 40C.

A tangent coefficient of thermal expansion is defined by

$$\lambda = 2.0e - 07 * T$$

The thermal expansion can be calculated by

$$\Delta l = \int_{10}^{40} 2.0e - 07 * T dT = [1.0e - 07 * T^2]_{10}^{40} = 1.5e - 04$$

Note the following model parameters defined in the input file that follows:

1. The problem is specified as a coupled structural thermal analysis in the (\*CONTROL\_SOLUTION) section.
2. Any LS-DYNA material model can be used. It is not necessary to use the temperature dependent constitutive models 4, 21, 23 and 60. However, only these 4 models account for the thermal coefficient of expansion. LS-DYNA uses a tangent coefficient of thermal expansion, which is defined as the slope of the thermal strain versus temperature curve for the material. This should not be confused with the secant coefficient of thermal expansion, which is the change in thermal strain between the current temperature and a reference temperature.
3. The mechanical mass scaled time step is set to 0.01 seconds (\*CONTROL\_TIMESTEP) and the thermal time step is set to 0.1 seconds (\*CONTROL\_THERMAL\_TIMESTEP). Explicit time integration is used for the structural calculations and implicit time integration is used for the thermal calculations. Implicit time integration is unconditionally stable and, therefore, a larger thermal time step can be taken. (\*CONTROL\_THERMAL\_TIMESTEP).

**LS-DYNA Input File:**

```

*KEYWORD
$
$      1      2      3      4      5      6      7      8
$
$=====CONTROL DEFINITIONS =====
$
*TITLE
thermal expansion of a block
*CONTROL_SOLUTION
  2
*CONTROL_THERMAL_SOLVER
  1      0      1
*CONTROL_TIMESTEP
                                           .01
*CONTROL_THERMAL_TIMESTEP
  0      1.      .1
*CONTROL_TERMINATION
  3.
*DATABASE_BINARY_D3PLOT
  .01
$
$===== PART DEFINITIONS =====
$
$
*PART
$      PID      SECID      MID      TMID
slab
      1      1      1      1
$
$===== SECTION PROPERTIES =====
$
*SECTION_SOLID
$      SECID      ELFORM
      1      1
$
$===== MECHANICAL MATERIAL PROPERTIES =====
$
*MAT_ELASTIC_PLASTIC_THERMAL
      1      1.
      0.      10.      20.      30.      40.      50.
      1.e+10      1.e+10      1.e+10      1.e+10      1.e+10      1.e+10
      .3      .3      .3      .3      .3      .3
      0.e-06      20.e-07      40.e-07      60.e-07      80.e-07      100.e-07
      1.e+20      1.e+20      1.e+20      1.e+20      1.e+20      1.e+20
      0.      0.      0.      0.      0.      0.
$
$===== THERMAL MATERIAL PROPERTIES =====
$
*MAT_THERMAL_ISOTROPIC
      1      1.      0      10.
      1.      1.
$
$===== NODE DEFINITIONS =====
$
*NODE
      1      0.      0.      0.      7
      2      1.      0.      0.      5
      3      1.      1.      0.      3
      4      0.      1.      0.      6
      5      0.      0.      1.      4
      6      1.      0.      1.      2
      7      1.      1.      1.      0
      8      0.      1.      1.      1

```

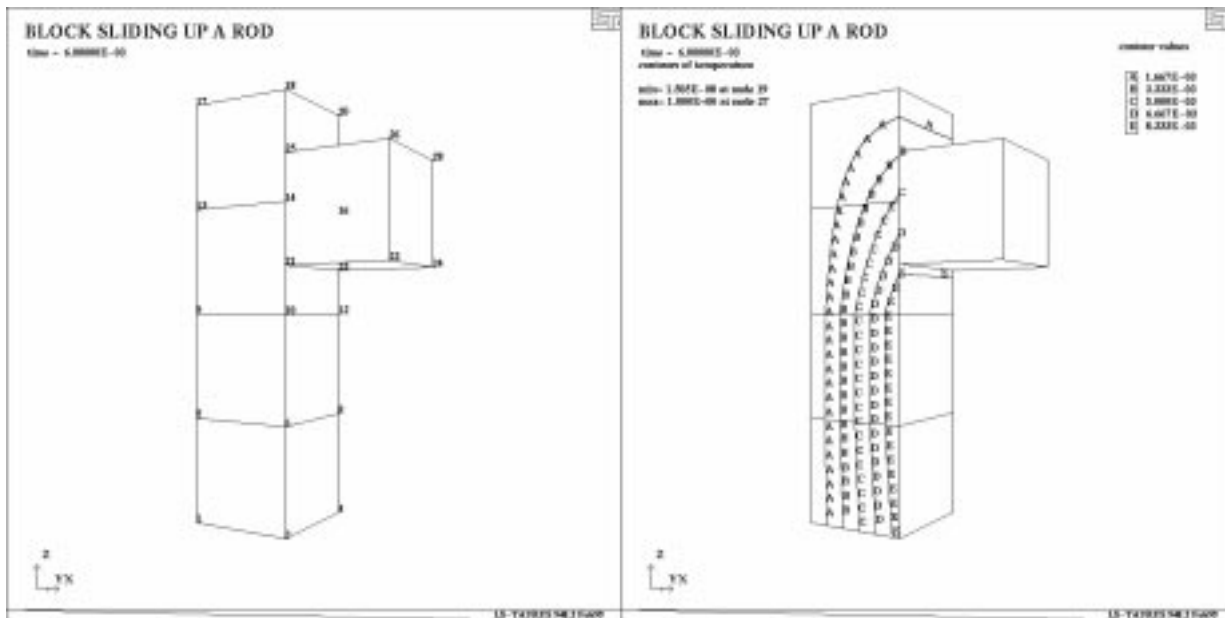
```

$
$===== ELEMENT DEFINITIONS =====
$
*ELEMENT_SOLID
      1      1      1      2      3      4      5      6      7      8
$
$===== THERMAL BOUNDARY CONDITIONS =====
$
*INITIAL_TEMPERATURE_NODE
      1      10.
      2      10.
      3      10.
      4      10.
      5      10.
      6      10.
      7      10.
      8      10.
*END

```

**Problem 4:** Welding, 3D Thermal Stress With Slide Surfaces

This problem models a hot block sliding along a colder block. This is a way to model welding. The hot block is set at a fixed temperature to model a heat source (e.g., torch, e-beam, or laser). The velocity of the moving hot block and contact resistance between the blocks determines the energy transfer to the work piece.



Shown are the node numbers defining the elements for this problem. A thermal-mechanical slide surface is defined between the 1 block on the right and the 4 blocks on the left.

The hot block on the right is sliding up the 4 element block on the left. Shown are temperature contours.

Note the following model parameters defined in the input file that follows:

1. The problem is specified as a coupled structural thermal analysis in the (\*CONTROL\_SOLUTION) section.
2. The mechanical mass scaled time step is set to 0.0001 seconds (\*CONTROL\_TIMESTEP) and the thermal time step is set to 0.001 seconds (\*CONTROL\_THERMAL\_TIMESTEP). Explicit time integration is used for the structural calculations and implicit time integration is used for the thermal calculations. Implicit time integration is unconditionally stable and, therefore, a larger thermal time step can be taken. (\*CONTROL\_THERMAL\_TIMESTEP).
3. A thermal-mechanical slide surface between the blocks is used to define thermal resistance parameters (\*CONTACT\_SURFACE\_TO\_SURFACE\_THERMAL).

4. The cold block has an initial temperature of 0 at time=0. The hot block has a temperature boundary condition of 10 degrees and a displacement boundary condition to move it along the cold block.
5. Any LS-DYNA material model can be used. It is not necessary to use the temperature dependent constitutive models 4, 21, 23 and 60. However, only these 4 models account for the thermal coefficient of expansion.

**LS-DYNA Input File:**

```

*KEYWORD
$
$      1      2      3      4      5      6      7      8
$
$=====CONTROL DEFINITIONS =====
$
*TITLE
block sliding up a rod
*CONTROL_SOLUTION
  2
*CONTROL_THERMAL_SOLVER
  1      0      1
*CONTROL_TIMESTEP
                                           .0001
*CONTROL_THERMAL_TIMESTEP
  0      1.      .001
*CONTROL_TERMINATION
  .007
*DATABASE_BINARY_D3PLOT
  .001
$
$===== PART DEFINITIONS =====
$
*PART
$      PID      SECID      MID      TMID
slab
      1      1      1      1
$
$===== SECTION PROPERTIES =====
$
*SECTION_SOLID
      1      1
$
$===== MECHANICAL MATERIAL PROPERTIES =====
$
*MAT_ELASTIC
      1      7830.      3.e+07      .3
$
$===== THERMAL MATERIAL PROPERTIES =====
$
*MAT_THERMAL_ISOTROPIC
      1      7830.
      460.      46.
$
$===== NODE DEFINITIONS =====
$
*NODE
      1      .0      .0      .0      7
      2      .1      .0      .0      7
      3      .0      .1      .0      7
      4      .1      .1      .0      7

```

5	.0	.0	.1	7
6	.1	.0	.1	7
7	.0	.1	.1	7
8	.1	.1	.1	7
9	.0	.0	.2	7
10	.1	.0	.2	7
11	.0	.1	.2	7
12	.1	.1	.2	7
13	.0	.0	.3	7
14	.1	.0	.3	7
15	.0	.1	.3	7
16	.1	.1	.3	7
17	.0	.0	.4	7
18	.1	.0	.4	7
19	.0	.1	.4	7
20	.1	.1	.4	7
21	.1	.0	.0	4
22	.2	.0	.0	4
23	.1	.1	.0	4
24	.2	.1	.0	4
25	.1	.0	.1	4
26	.2	.0	.1	4
27	.1	.1	.1	4
28	.2	.1	.1	4

\$\$\$\$  
 \$===== ELEMENT DEFINITIONS =====  
 \$\$\$

\*ELEMENT\_SOLID  

1	1	1	2	4	3	5	6	8	7
2	1	5	6	8	7	9	10	12	11
3	1	9	10	12	11	13	14	16	15
4	1	13	14	16	15	17	18	20	19
5	1	21	22	24	23	25	26	28	27

\$\$\$\$  
 \$===== DEFINE NODE SET FOR BOUNDARY CONDITIONS =====  
 \$\$\$

\*SET\_NODE\_LIST\_GENERATE  

1	
21	28

\$\$\$\$  
 \$===== MECHANICAL BOUNDARY CONDITIONS =====  
 \$\$\$

\*BOUNDARY\_PRESCRIBED\_MOTION\_SET  

1	3	2	1
---	---	---	---

\*DEFINE\_CURVE  

1	0.	0.
	.01	.4

\$\$\$\$  
 \$===== THERMAL BOUNDARY CONDITIONS =====  
 \$\$\$

\*BOUNDARY\_TEMPERATURE\_SET  

1	0	1.
---	---	----

\$\$\$\$  
 \$===== THERMAL MECHANICAL CONTACT DEFINITION =====  
 \$\$\$

\*SET\_SEGMENT  

1			
21	25	27	23

\*SET\_SEGMENT  

2			
2	6	8	4
6	10	12	8
10	14	16	12
14	18	20	16

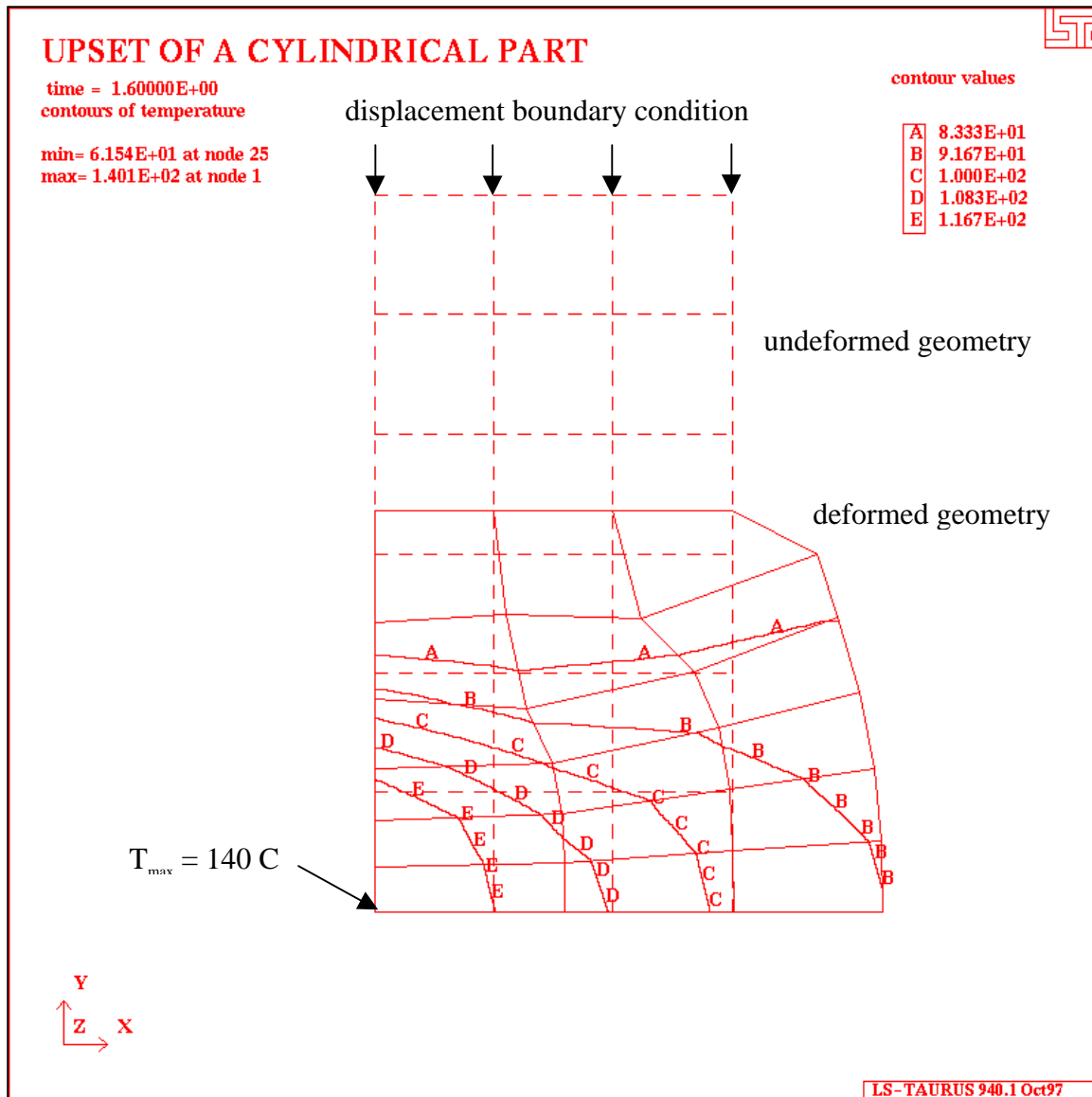
\*CONTACT\_SURFACE\_TO\_SURFACE\_THERMAL

```
      1      2
      0.
      1.      1.
      0.      0.      1.e+06      1.e-02      1.e-02
*END
```

**Problem 5:** Metal Forming, Compression of a Cylinder With Conversion of Plastic Work to Heat

This problem models the metal forming process known as upsetting. The upsetting process is defined as the axial compression of an axisymmetric body between two perfectly rough, insulated plates. The material used is low carbon steel. The initial temperature is 20C, the initial height is 0.036m and the initial radius is 0.009m. There is no heat transfer to the environment. The total imposed reduction is  $(\Delta h/h) = 44\%$  over 1.6 seconds (i.e., displacement boundary condition = -0.0792m). Only a quarter of the problem is modeled because of symmetry.

The problem geometry and results are shown in the following figure. The temperature change of the body is from conversion of mechanical work into heat through plastic deformation.





Note the following model parameters defined in the input file that follows:

1. A mechanical mass scaled time step (dt=1.e-04) is used (\*CONTROL\_TIMESTEP).
2. A thermal time step (dt=0.1) is used which is 1000 times larger than the mechanical time step (\*CONTROL\_THERMAL\_TIMESTEP). Heat transfer takes place on a longer time scale than the mechanical deformation. Volumetric adiabatic heating is from conversion of mechanical work into heat through plastic deformation. 100% of the mechanical work is converted to heat by setting the conversion factor to 1 in the (\*MAT\_THERMAL\_SOLVER) input section.
3. The hourglass control is set to 4 (\*CONTROL\_HOURLASS). This helps control but does not eliminate hourglassing on this very coarse mesh.
4. Shell formulation 15 is used (\*SECTION\_SHELL). This 2D axisymmetric shell formulation with volume weighting works better than shell formulation 14 with area weighting for this problem.
5. Any LS-DYNA material model can be used. It is not necessary to use the temperature dependent constitutive models 4, 21, 23 and 60. However, only these 4 models account for the thermal coefficient of expansion.

**LS-DYNA Input File:**

```
*KEYWORD
$
$      1      2      3      4      5      6      7      8
$
$===== CONTROL DEFINITIONS =====
$
*TITLE
upset of a cylindrical part
*CONTROL_SOLUTION
    2
*CONTROL_TIMESTEP
    0.      0.      0      0.      1.e-04
*CONTROL_THERMAL_SOLVER
    1      0      1      1.
*CONTROL_THERMAL_TIMESTEP
    0      1.      .1
*CONTROL_TERMINATION
    1.6
*CONTROL_HOURLASS
    4
*DATABASE_TPRINT
    .1
*DATABASE_BINARY_D3PLOT
    .1
$
$===== PART DEFINITIONS =====
$
*PART
cylinder
    1      1      1      1
$
```

SECTION PROPERTIES

```
*SECTION_SHELL
    1      15
  0.001  0.001  0.001  0.001  0.
```

MECHANICAL MATERIAL PROPERTIES

```
*MAT_PIECEWISE_LINEAR_PLASTICITY
    1      7830.  210.e+09  .28
    0.      0.      0.      0.
  0.000E+00  1.000E-02  2.000E-02  5.000E-02  1.000E-01  2.000E-01  4.000E-01  1.000E+00
  2.500E+08  2.800E+08  3.000E+08  3.500E+08  4.200E+08  5.000E+08  5.200E+08  5.400E+08
```

THERMAL MATERIAL PROPERTIES

```
*MAT_THERMAL_ISOTROPIC
    1      7830.
  460.    46.
```

NODE DEFINITIONS

```
*NODE
  1          0.          0.          4.          7.
  2          .003        0.          2.          7.
  3          .006        0.          2.          7.
  4          .009        0.          2.          7.
  5          0.          .003        1.          7.
  6          .003        .003        0.          7.
  7          .006        .003        0.          7.
  8          .009        .003        0.          7.
  9          0.          .006        1.          7.
 10         .003        .006        0.          7.
 11         .006        .006        0.          7.
 12         .009        .006        0.          7.
 13         0.          .009        1.          7.
 14         .003        .009        0.          7.
 15         .006        .009        0.          7.
 16         .009        .009        0.          7.
 17         0.          .012        1.          7.
 18         .003        .012        0.          7.
 19         .006        .012        0.          7.
 20         .009        .012        0.          7.
 21         0.          .015        1.          7.
 22         .003        .015        0.          7.
 23         .006        .015        0.          7.
 24         .009        .015        0.          7.
 25         0.          .018        1.          7.
 26         .003        .018        1.          7.
 27         .006        .018        1.          7.
 28         .009        .018        1.          7.
```

ELEMENT DEFINITIONS

```
*ELEMENT_SHELL
  1      1      1      2      6      5
  2      1      2      3      7      6
  3      1      3      4      8      7
  4      1      5      6     10      9
  5      1      6      7     11     10
  6      1      7      8     12     11
  7      1      9     10     14     13
  8      1     10     11     15     14
  9      1     11     12     16     15
 10     1     13     14     18     17
 11     1     14     15     19     18
```

```

12      1      15      16      20      19
13      1      17      18      22      21
14      1      18      19      23      22
15      1      19      20      24      23
16      1      21      22      26      25
17      1      22      23      27      26
18      1      23      24      28      27
$
$===== DEFINE NODE SET FOR BOUNDARY CONDITIONS =====
$
*SET_NODE_LIST_GENERATE
  1
 25      28
*SET_NODE_LIST_GENERATE
  2
  1      28
$
$===== TEMPERATURE INITIAL CONDITON =====
$
*INITIAL_TEMPERATURE_SET
  2      20.
$
$===== MECHANICAL BOUNDARY CONDITIONS =====
$
*BOUNDARY_PRESCRIBED_MOTION_SET
  1      2      2      1      1.
*DEFINE_CURVE
  1
      0.      0.
      2.      -.0099
*end

```

