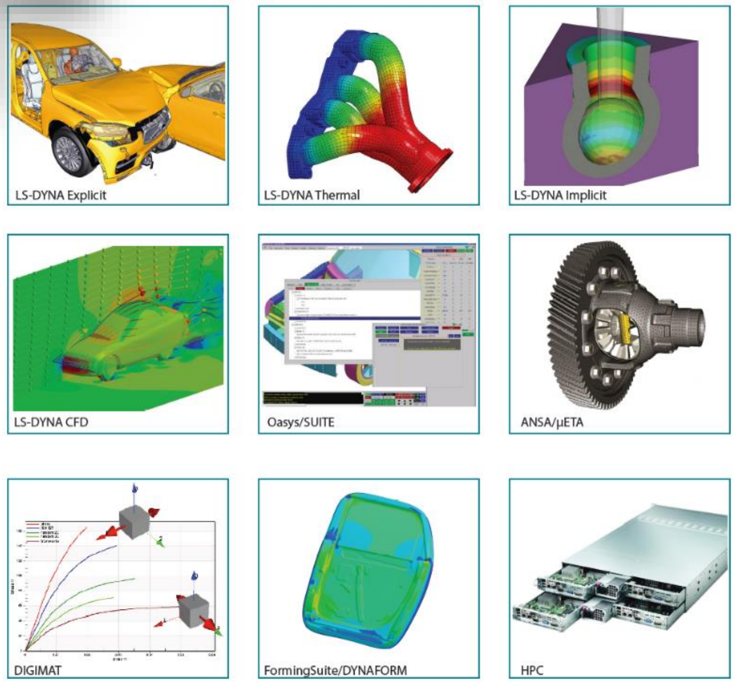
Thermal & thermo-mechanical simulations using LS-DYNA

Marcus Lilja, DYNAmore Nordic AB

Contact – DYNAmore Nordic

- Support
  - Sales: sales@dynamore.se
  - E-mail: support@dynamore.se
    - Target to answer in 4 hours
    - Call: +46 13 236680
- Training & seminars including on-line & on-site: www.dynamore.se
- Secure file server: files.dynamore.se
  - Software and license download, client area with guidelines and more
- www.dynamore.se - information on LS-DYNA, Seminars, Conferences
- www.dynalook.com – Papers from international LS-DYNA conferenses
- www.dynasupport.com – General support for LS-DYNA
- www.dynaexamples.com – LS-DYNA example models from crash to DEM



Thermal simulations in LS-DYNA

Intention

The intention is to give the audience basic knowledge of

- What is included in LS-DYNA
- How to select between different options and features
- How to set-up and run a thermal and thermo-mechanical simulation

Thermal simulations in LS-DYNA

Heat transfer mechanisms

There are basically three ways to transfer **heat energy**

Conduction

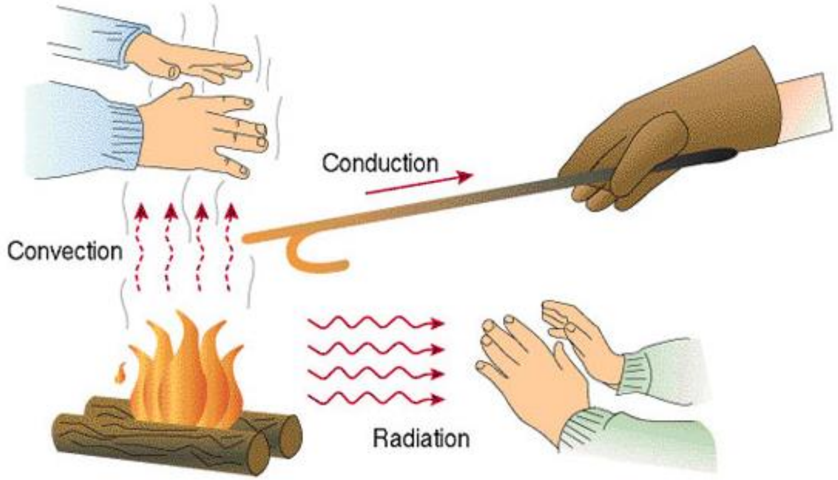
- Transfer of heat energy between particles of objects in contact.
- Needs a media.

Convection

- Transfer of heat energy by the movement of a fluid.
- Needs a media.

Radiation

- Transfer of heat energy by electromagnetic waves
- No movement of matter.
- No media needed.



Thermal simulations in LS-DYNA

Basic knowledge

Thermal solution

- Solution type
- Solver settings

Thermal conditions

- Heat source
- Heat energy transfer mechanisms
- Initial conditions

Thermal contacts

- Correct heat transfer at interfaces

Thermal material models

- What is it?
- Different types
- How to choose

Thermal elements

- Definition in LS-DYNA
- Supported types
- Special case

Thermal solution types

Thermal simulations in LS-DYNA

Thermal solution types

There are different thermal solution type options available in LS-DYNA

- Steady state thermal simulation
- Transient thermal simulation
- Un-coupled thermal and mechanical simulation
- Coupled transient thermo-mechanical simulation

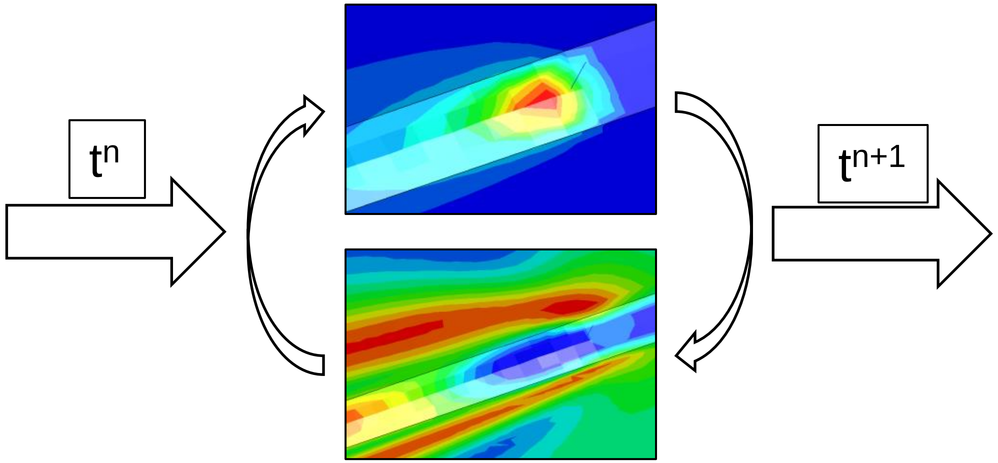
Thermal simulations in LS-DYNA

Thermal solution types

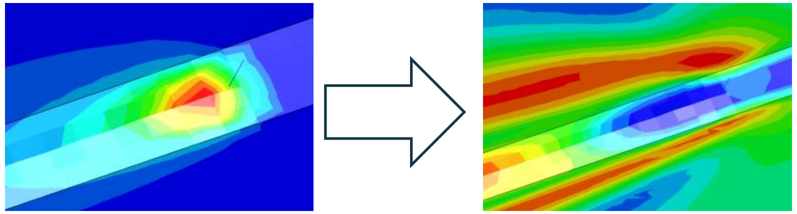
Solution type options, explanation

- **Steady state thermal simulation**
  - Very fast!
  - Linear assumption
  - Heat transfer problem solved in one single step
  - No stress/strain output
- **Transient thermal simulation**
  - Linear of non-linear
  - For path dependent problems
  - No stress/strain output
- **Un-coupled thermal and mechanical simulation**
  - No mechancal-thermal dependency
  - *LOAD_THERMAL_D3PLOT
- **Coupled transient thermal and mechanical simulation**
  - Linear or non-linear
  - Needed if the mechanical behavior affects the thermal properties
  - Complete output

Coupled thermo-mechanical simulation



Un-coupled thermo-mechanical simulation



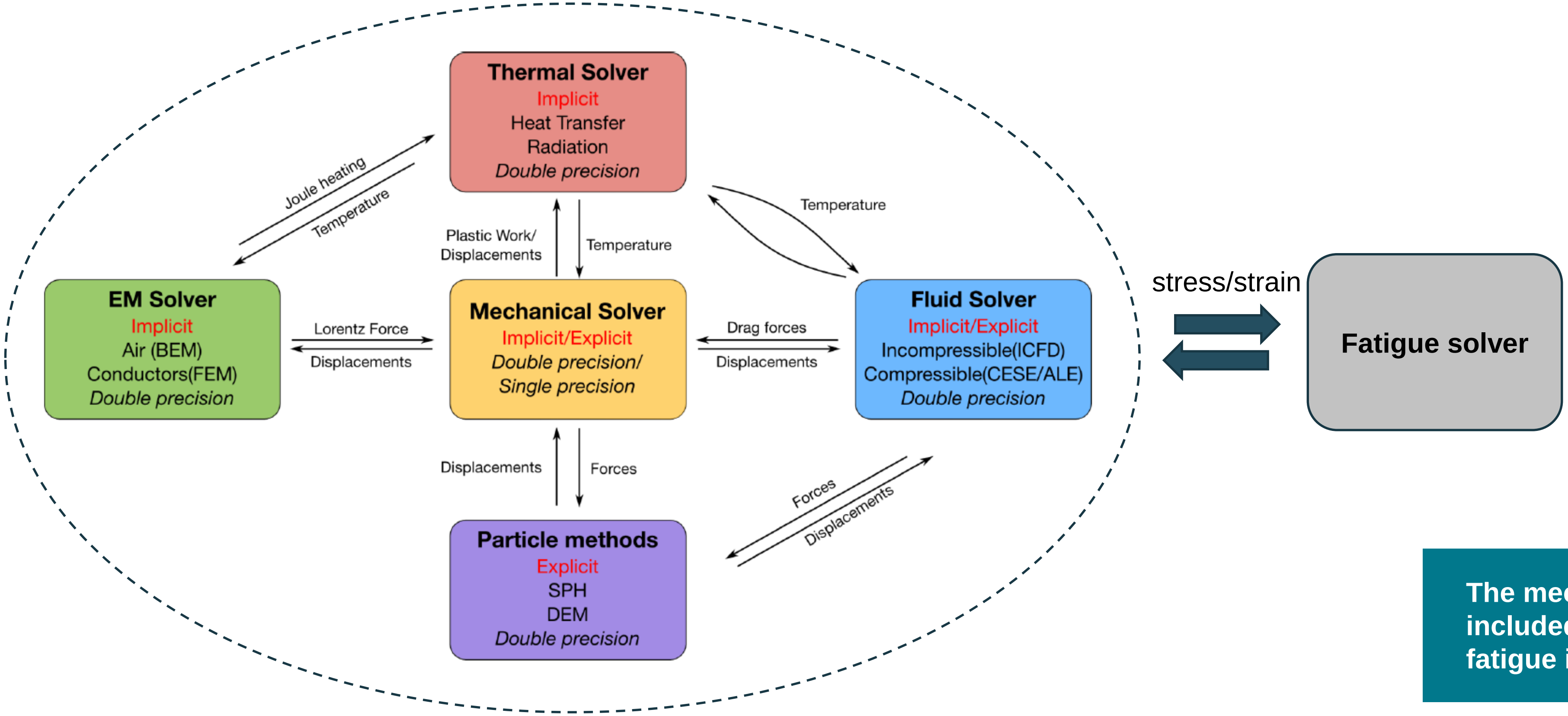
Temperature input

LS-DYNA thermal solver

Thermal simulations in LS-DYNA

LS-DYNA thermal solver

- The thermal solver can be coupled to any any other solver in the LS-DYNA-package or be used as a “stand-alone“-solver for unclupled thermal problems.

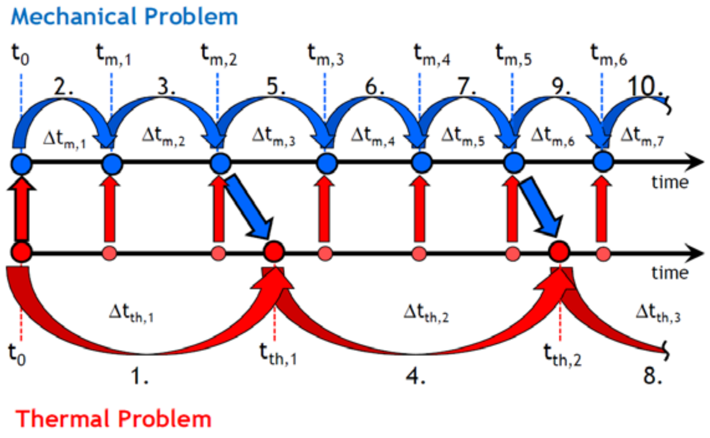


The mechanical solver must be included if thermo-mechanical fatigue is of interest.

Thermal simulations in LS-DYNA

LS-DYNA thermal solver

- The thermal solver is an implicit solver and the solution scheme for a coupled thermo-mechanical problem is stagered.



- The mechanical time-step (implicit or explicit) should not exceed the thermal time-step size.
  - If automatic thermal time step size is used, then: $t_{therm} = 100 * t_{mech}$
- The thermal time step must be small enough to capture the temperature/time behavior
- A convergence study of the results may be neccessary.

Mechanical solution (explicit)

$$t_{mech} = \frac{L_e}{c} \qquad c = \sqrt{\frac{E}{\rho}}$$

Thermal solution (implicit)

$$t_{therm} = \frac{\rho c_p L_e^2}{2k}$$

Thermal simulations in LS-DYNA

LS-DYNA thermal solver

- Activate the thermal solver by setting SOLN = 1 or 2 on *CONTROL_SOLUTION

Card 1	1	2	3	4	5	6	7	8
Variable	SOLN	NLQ	ISNAN	LCINT	LCACC	NCDCF	NOCOPY	
Type	I	I	I	I	I	I	I	
Default	0	0	0	100	0	1	0	

VARIABLE	DESCRIPTION
SOLN	Analysis solution procedure: EQ.0: Structural analysis only EQ.1: Thermal analysis only EQ.2: Combined structural, multiphysics, and thermal analysis

Thermal simulations in LS-DYNA

LS-DYNA thermal solver

- ***CONTROL_THERMAL_SOLVER**
  - Define analysis & thermal problem type
- *CONTROL_SOLUTION need to be specified!

Card 1	1	2	3	4	5	6	7	8
Variable	ATYPE	PTYPE	SOLVER		GPT	EQHEAT	FWORK	SBC
Type	I	I	I		I	F	F	F
Default	0	0	↓		8	1.	1.	0.
Remark				11				

Analysis type (ATYPE) · Solution type (PTYPE) · Solver (SOLVER)

VARIABLE	DESCRIPTION
ATYPE	Thermal analysis type: EQ.0: Steady state analysis (see Remark 4) EQ.1: Transient analysis (see Remark 3)
PTYPE	Thermal problem type: (see *CONTROL_THERMAL_NONLINEAR if nonzero) EQ.0: Linear problem, EQ.1: Nonlinear problem with material properties evaluated at gauss point temperature. EQ.2: Nonlinear problem with material properties evaluated at element average temperature.

Thermal simulations in LS-DYNA

LS-DYNA thermal solver

- ***CONTROL_THERMAL_SOLVER**
  - Optional card 2
  - Direct input of time scaling parameter
    - Instead of applying it manually to material properties

Card 2	1	2	3	4	5	6	7	8
Variable	MSGLVL	MAXITR	ABSTOL	RELTOL	OMEGA			TSF
Type	I	I	F	F	F			F
Default	0	500	1.0e-10	1.0e-04	1.0 or 0.			1.

Time scale factor

VARIABLE	DESCRIPTION
MSGLVL	Output message level (For SOLVER > 10) EQ.0: No output (default), EQ.1: Summary information, EQ.2: Detailed information, use only for debugging.
MAXITR	Maximum number of iterations. For SOLVER > 11. EQ.0: Use default value 500,
ABSTOL	Absolute convergence tolerance. For SOLVER > 11. EQ.0.0: Use default value 10^{-10}
RELTOL	Relative convergence tolerance. For SOLVER > 11. EQ.0.0: Use default value 10^{-6}
OMEGA	Relaxation parameter omega for SOLVER 14 and 16. EQ.0.0: Use default value 1.0 for Solver 14, use default value 0.0 for Solver 16.
TSF	Thermal Speedup Factor. This factor multiplies all thermal parameters with units of time in the denominator (e.g., thermal conductivity, convection heat transfer coefficients). It is used to artificially time scale the problem. EQ.0.0: Default value 1.0, LT.0.0: |TSF| is a load curve ID. Curve defines speedup factor as a function of time.

Thermal simulations in LS-DYNA

LS-DYNA thermal solver

- Time scaling
  - Time saving
  - The thermal problem is scaled in time in the same manner as the structural problem.
  - Typical thermal properties are:

Thermal capacity	J/kgK
Heat transfer	W/m^2K
Thermal conductivity	W/mK

  e.g. using N, mm, s and a time scale factor of X yields

  Thermal capacity=Thermal capacity*1000.000
  Heat transfer=Heat transfer/1000*X
  Thermal conductivity=Thermal conductivity*X

  - The time scale factor can be input directly on the second card of the *CONTROL_THERMAL_SOLVER keyword. Then LS-DYNA scales the time dependent thermal input

Thermal simulations in LS-DYNA

LS-DYNA thermal solver

- ***CONTROL_THERMAL_TIMESTEP**
  - Specify time step options for the thermal solver.
  - *CONTROL_SOLUTION need to be specified!

Card 1	1	2	3	4	5	6	7	8
Variable	TS	TIP	ITS	TMIN	TMAX	DTEMP	TSCP	LCTS
Type	I	F	F	F	F	F	F	I
Default	0	1.0	none	↓	↓	1.0	0.5	0

- TS: Fixed or variable timestep size
- TIP: Time Integration parameters
- ITS: Initial thermal timestep
- TMIN: Min. thermal timestep
- TMAX: Max. thermal timestep.
- DTEMP: Max. temperature difference / timestep

Defaults
- TMIN = stuctural time step
- TMAX = 100*strucural time step

Thermal simulations in LS-DYNA

LS-DYNA thermal solver

- ***CONTROL_THERMAL_NONLINEAR**
  - Set convergence related conditions for the non-linear thermal solver
  - *CONTROL_SOLUTION need to be specified!

Card 1	1	2	3	4	5	6	7	8
Variable	REFMAX	TOL	DCP	LUMPBC	THLSTL	NLTHPR	PHCHPN	
Type	I	F	F	I	F	I	F	
Default	10	10^{-4}	1.0 or 0.5	0	0.	0	100.	

VARIABLE	DESCRIPTION
REFMAX	Maximum number of matrix reformations per time step. EQ.0: Set to 10 reformations.
TOL	Convergence tolerance for temperature. EQ.0.0: Set to 1000 × machine roundoff.

Thermal elements

Thermal simulations in LS-DYNA

Thermal elements

- Supported element types
  - Solid, shells, thick shells, beams, discrete beams

- When a thermal material id (TMID) is assigned to the part definition (*PART) the elements in that part becomes thermal elements.
- To run a thermal simulation all elements in the model must have a thermal material assignment.

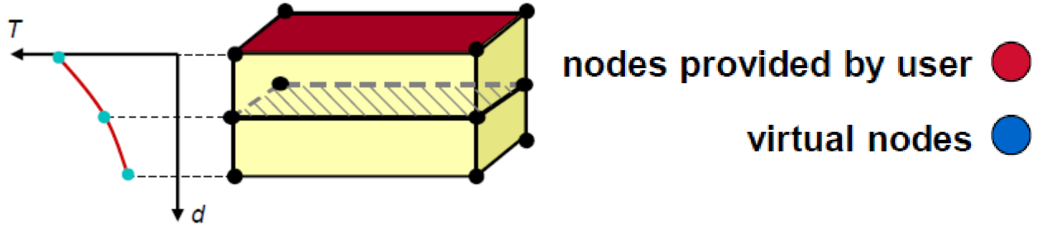
Card 2	1	2	3	4	5	6	7	8
Variable	PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
Type	I/A	I/A	I/A	I/A	I/A	I	I	I/A
Default	none	none	none	0	0	0	0	0

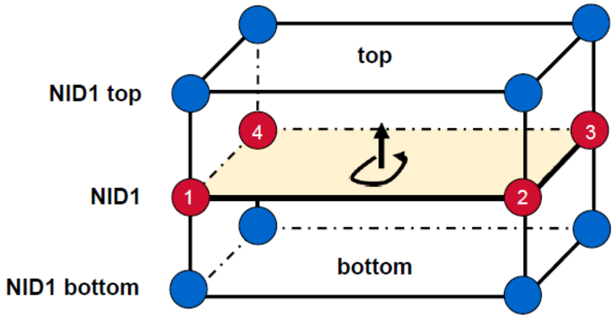
Thermal simulations in LS-DYNA

Thermal elements

- Thick thermal shells
  - THSHEL=1 on *CONTROL_SHELL will trigger that all 4-noded elements are treated as 12-noded.
  - The through thickness nodes will use quadratic shape functions and this will give a highly resolved temperature gradient through the shell element thickness.

Card 2	1	2	3	4	5	6	7	8
Variable	ROTASCL	INTGRD	LAMSHT	CSTYP6	THSHEL			
Type	F	I	I	I	I			
Default	1.	0	0	1	0			





Thermal simulations in LS-DYNA

Thermal contacts

- Thick thermal shell elements
  - Thermal contact should be defined at the outer surfaces only. This is achieved by setting ITHOFF=1 on the 5th card *CONTROL_CONTACT

Card 5	1	2	3	4	5	6	7	8
Variable	ISYM	NSEROD	RWGAPS	RWGDTH	RWKSF	ICOV	SWRADF	ITHOFF
Type	I	I	I	F	F	I	F	I
Default	0	0	0	0.	1.0	0	0.	0

ITHOFF — Flag for offsetting thermal contact surfaces for thick thermal shells:

EQ.0: No offset. If thickness is not included in the contact, the heat will be transferred between the mid-surfaces of the corresponding contact segments (shells).

EQ.1: Offsets are applied so that contact heat transfer is always between the outer surfaces of the contact segments (shells).

Thermal simulations in LS-DYNA

Thermal bounday conditions

- Thick thermal shell elements
  - The shells have a lower, middle and upper surface.
  - Initial conditons and boundary conditions must be applied to all surfaces. This is done by using the LOC-parameter which is found on all neccessary input-cards.

***INITIAL_TEMPERATURE**

Card 1	1	2	3	4	5	6	7	8
Variable	NSID/NID	TEMP	LOC					
Type	I	I	I					
Default	↓	0.	0					
Remarks	1							

***BOUNDARY_TEMPERATURE**

Card 1	1	2	3	4	5	6	7	8
Variable	NID	TLCID	TMULT	LOC	TDEATH	TBIRTH		
Type	I	I	F	I	F	F		
Default	none	0	0.	0	10^{20}	0.		

LOC	For a thick thermal shell, the temperature will be applied to the surface identified by LOC. See parameter, THSHEL, on the *CONTROL_SHELL keyword. EQ.-1: lower surface of thermal shell element EQ.0: middle surface of thermal shell element EQ.1: upper surface of thermal shell element

Thermal material models

Thermal simulations in LS-DYNA

Thermal material models

- Thermal material models

Thermal material models

- *MAT_THERMAL
  - *MAT_T01/*MAT_THERMAL_ISOTROPIC
  - *MAT_T02/*MAT_THERMAL_ORTHOTROPIC
  - *MAT_T03/*MAT_THERMAL_ISOTROPIC_TD
  - *MAT_T04/*MAT_THERMAL_ORTHOTROPIC_TD
  - *MAT_T05/*MAT_THERMAL_DISCRETE_BEAM
  - *MAT_T06/*MAT_THERMAL_CHEMICAL_REACTION
  - *MAT_T07/*MAT_THERMAL_CWM
  - *MAT_T08/*MAT_THERMAL_ORTHOTROPIC_TD_LC
  - *MAT_T09/*MAT_THERMAL_ISOTROPIC_PHASE_CHANGE
  - *MAT_T10/*MAT_THERMAL_ISOTROPIC_TD_LC
  - *MAT_T11-T15/*MAT_THERMAL_USER_DEFINED
  - *MAT_T17/*MAT_THERMAL_CHEMICAL_REACTION_ORTHOTROPIC

- Heat capacity and conductivity may be temperature dependent.
- Mandatory in a thermal simulation
- Input curve data is never extrapolated

Thermal simulations in LS-DYNA

Thermal material models

- Temperature dependent material models

Coupled material models

- *MAT_004/*MAT_ELASTIC_PLASTIC_THERMAL
- *MAT_021/*MAT_ORTHOTROPIC_THERMAL
- *MAT_106/*MAT_ELASTIC_VISCOPLASTIC_THERMAL
- *MAT_113/*MAT_TRIP
- *MAT_188/*MAT_THERMO_ELASTO_VISCOPLASTIC_CREEP
- *MAT_189/*MAT_ANISOTROPIC_THERMOELASTIC
- *MAT_249/*MAT_REINFORCED_THERMOPLASTIC
- *MAT_249_CRASH/*MAT_REINFORCED_THERMOPLASTIC_CRASH
- *MAT_249_UDFIBER/*MAT_REINFORCED_THERMOPLASTIC_UDFIBER

- Temperature dependent material properties.
- Not mandatory in a thermal simulation
- Curve input is extrapolated

Thermal simulations in LS-DYNA

Thermal material models

- *MAT_ADD_...

Add-on thermal strain effects

*MAT_ADD_THERMAL_EXPANSION

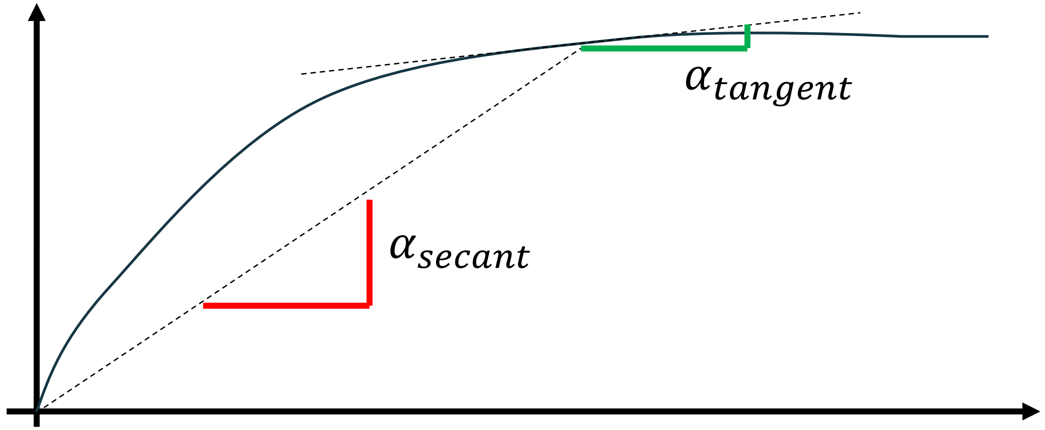
Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCID	MULT	LCIDY	MULTY	LCIDZ	MULTZ	
Type	I	I	F	I	F	I	F	
Default	none	none	1.0	LCID	MULT	LCID	MULT	

- Add thermal expansion coefficient to any material model.
- The simulation may be thermal only, coupled thermo-machanical or structural only

Thermal simulations in LS-DYNA

Thermal material models

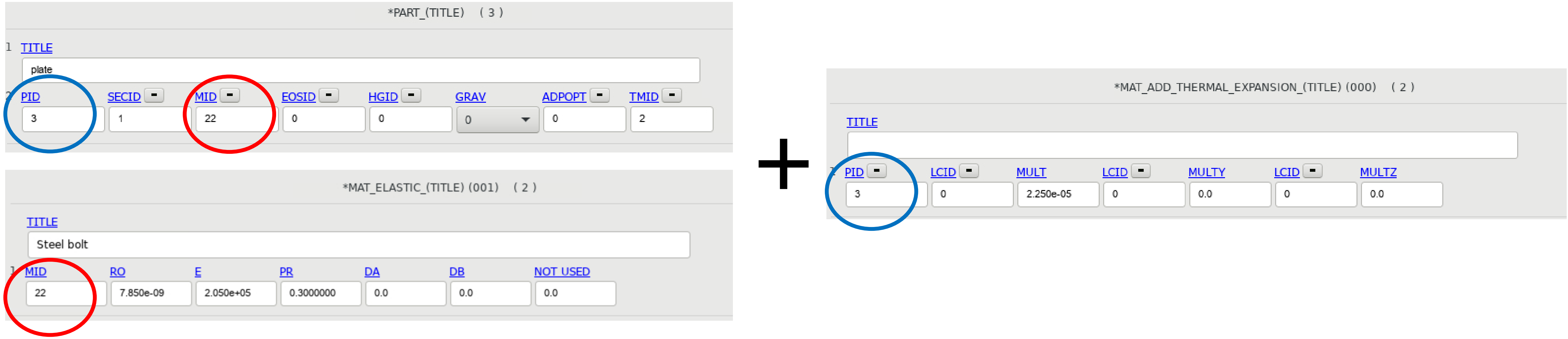
- Thermal expansion
  - Given as a tangential input, not secant input. Hence, no reference temperature is needed.



Thermal simulations in LS-DYNA

Thermal material models

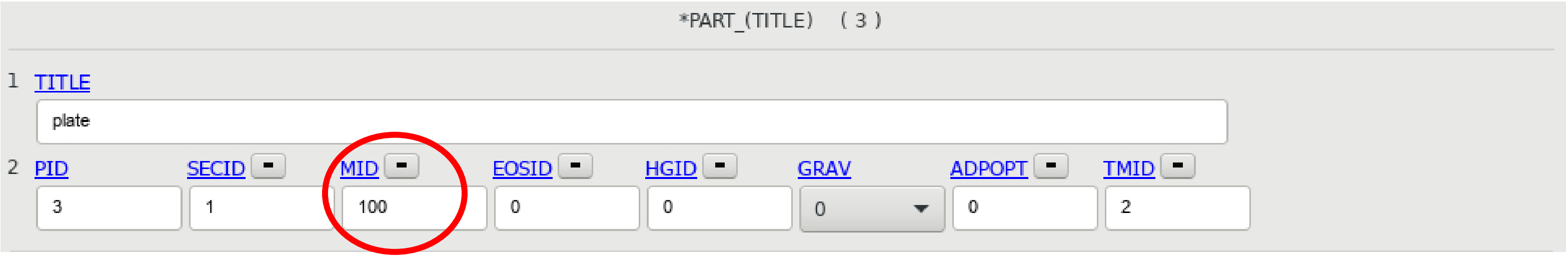
- Thermal expansion
- Included in the coupled thermo-mechanical material models.
- For a non-coupled material model, use *MAT_ADD_THERMAL_EXPANSION to include the effect of thermal strains.



Thermal simulations in LS-DYNA

Thermal material models

- Thermal expansion is included in a coupled material model.

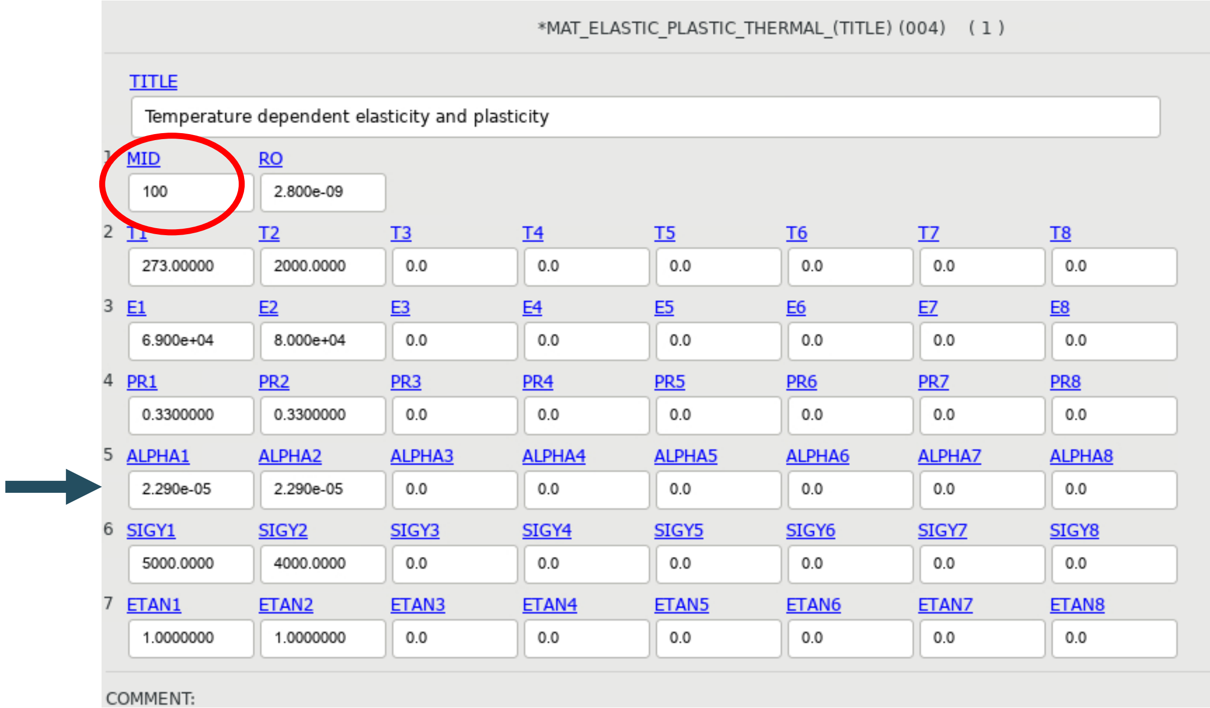


*PART_(TITLE) (3)

1 TITLE

plate

2	PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
	3	1	100	0	0	0	0	2



*MAT_ELASTIC_PLASTIC_THERMAL_(TITLE) (004) (1)

TITLE

Temperature dependent elasticity and plasticity

1	MID	RO
	100	2.800e-09

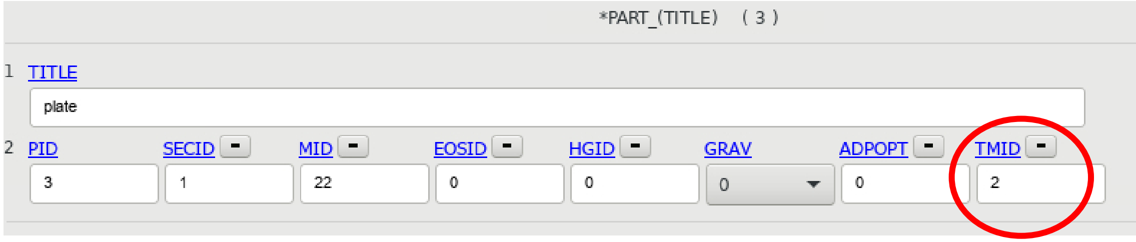
2	T1	T2	T3	T4	T5	T6	T7	T8
	273.00000	2000.0000	0.0	0.0	0.0	0.0	0.0	0.0
3	E1	E2	E3	E4	E5	E6	E7	E8
	6.900e+04	8.000e+04	0.0	0.0	0.0	0.0	0.0	0.0
4	PR1	PR2	PR3	PR4	PR5	PR6	PR7	PR8
	0.3300000	0.3300000	0.0	0.0	0.0	0.0	0.0	0.0
5	ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8
	2.290e-05	2.290e-05	0.0	0.0	0.0	0.0	0.0	0.0
6	SIGY1	SIGY2	SIGY3	SIGY4	SIGY5	SIGY6	SIGY7	SIGY8
	5000.0000	4000.0000	0.0	0.0	0.0	0.0	0.0	0.0
7	ETAN1	ETAN2	ETAN3	ETAN4	ETAN5	ETAN6	ETAN7	ETAN8
	1.0000000	1.0000000	0.0	0.0	0.0	0.0	0.0	0.0

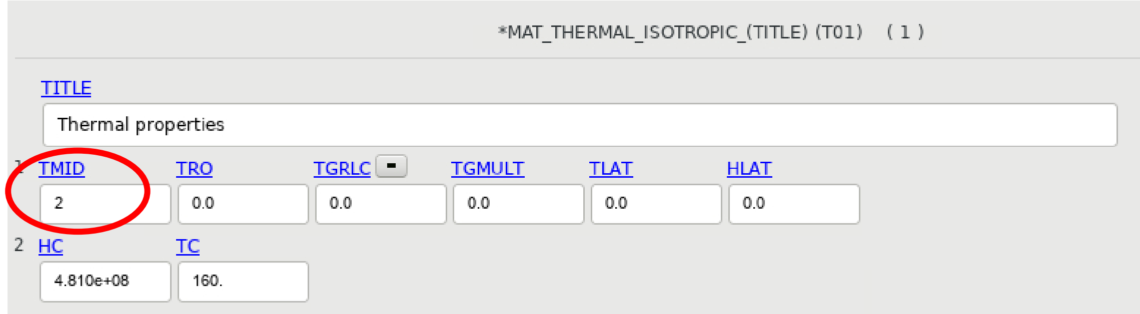
COMMENT:

Thermal simulations in LS-DYNA

Thermal material models

- A thermal material model, TMID, need to be defined.
  - Linear material properties





TMID	Thermal material identification. A unique number or must be specified (see *PART).
TRO	Thermal density: EQ.0.0: Default to structural density
TGRLC	Thermal generation rate (see *DEFINE_CURVE). GT.0: Load curve ID giving thermal generation rate as a function of time EQ.0: Thermal generation rate is the constant multiplier, TGMULT. LT.0: |TGRLC| is a load curve ID defining thermal generation rate as a function of temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: No heat generation
TLAT	Phase change temperature
HLAT	Latent heat
HC	Specific heat
TC	Thermal conductivity

Thermal simulations in LS-DYNA

Thermal material models

- A thermal material model, TMID, need to be defined.
  - Non-linear material properties

*PART_(TITLE) (3)

1 TITLE: plate

2	PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
	3	1	22	0	0	0	0	2

*MAT_THERMAL_ISOTROPIC_TD_(TITLE) (T03) (1)

TITLE: Temperature dependent thermal properties

1	TMID	TRO	TGRLC	TGMULT	TLAT	HLAT		
	2	0.0	0.0	0.0	0.0	0.0		
2	T1	T2	T3	T4	T5	T6	T7	T8
	200.00000	1500.0000	0.0	0.0	0.0	0.0	0.0	0.0
3	C1	C2	C3	C4	C5	C6	C7	C8
	2.610+08	4.810e+08	0.0	0.0	0.0	0.0	0.0	0.0
4	K1	K2	K3	K4	K5	K6	K7	K8
	9.300e+03	1.600e+04	0.0	0.0	0.0	0.0	0.0	0.0

TMID	Thermal material identification. A unique number or must be specified (see *PART).
TRO	Thermal density: EQ.0.0: Default to structural density
TGRLC	Thermal generation rate (see *DEFINE_CURVE). GT.0: Load curve ID giving thermal generation rate as a function of time EQ.0: Thermal generation rate is the constant multiplier, TGMULT. LT.0: |TGRLC| is a load curve ID defining thermal generation rate as a function of temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: No heat generation
TLAT	Phase change temperature
HLAT	Latent heat
T1, …, T8	Temperatures: T1, ..., T8
C1, …, C8	Specific heat at: T1, …, T8
K1, …, K8	Thermal conductivity at: T1, …, T8

Thermal contact

Thermal simulations in LS-DYNA

Thermal contacts

- Thermal contact is activated by adding THERMAL or THERMAL_FRICTION to the contact type name
- Applicable to AUTOMATIC and FORMING contact types.

THRM 1	1	2	3	4	5	6	7	8
Variable	K	FRAD	H0	LMIN	LMAX	FTOSA	BC_FLG	ALGO
Type	F	F	F	F	F	F	I	I
Default	none	none	none	none	none	0.5	0	0

- K: Conductivity of fluid between faces
- FRAD: Radiation
- H0: Heat transfer coefficient
- LMIN: Min. gap of thermal contact
- LMAX: Max. gap of thermal contact

About LMIN & LMAX

- Gap > LMAX:
  - No thermal contact
- Gap < LMIN:
  - Thermal contact using H0

Thermal simulations in LS-DYNA

Thermal contacts

- By adding _THERMAL_FRICTION instead of _THERMAL the user may define friction and heat conductance as a function of temperature and interface pressure.

THRM 1	1	2	3	4	5	6	7	8
Variable	K	FRAD	H0	LMIN	LMAX	FTOSA	BC_FLG	ALGO
Type	F	F	F	F	F	F	I	I
Default	none	none	none	none	none	0.5	0	0

_THERMAL

THERMAL Card 2.

THRM 2	1	2	3	4	5	6	7	8
Variable	LCFST	LCFDT	FORMULA	A	B	C	D	LCH
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

_THERMAL_FRICTION

Thermal simulations in LS-DYNA

Thermal contacts

- Defining h(p) (conductance as a function of interface pressure) with _THERMAL_FRICTION

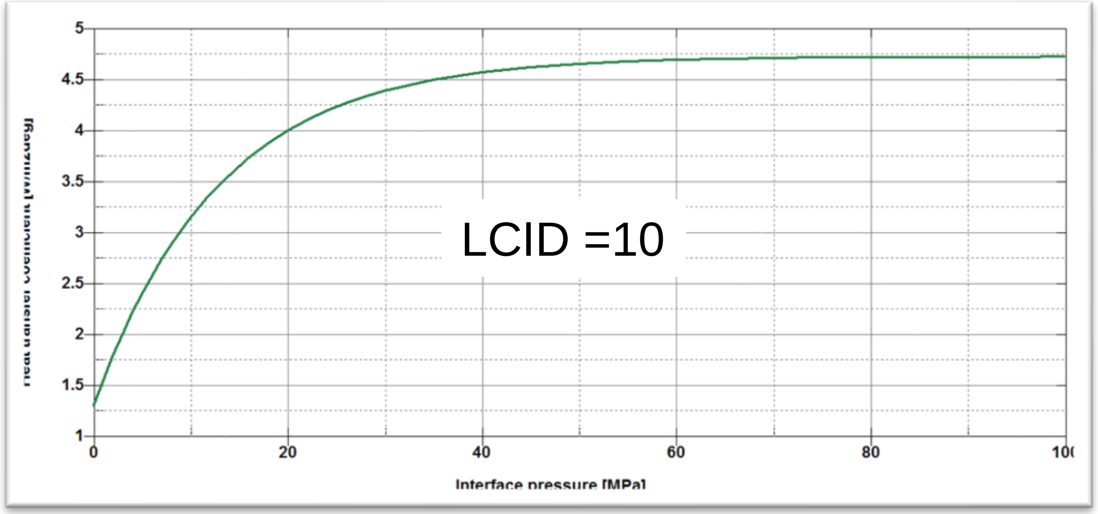
THERMAL Card 2.

THRM 2	1	2	3	4	5	6	7	8
Variable	LCFST	LCFDT	FORMULA	A	B	C	D	LCH
Type	I	I	I	I	I	I	I	I
Default	0	0	1	10	0	0	0	0

FORMULA — Formula that defines the contact heat conductance as a function of temperature and pressure.

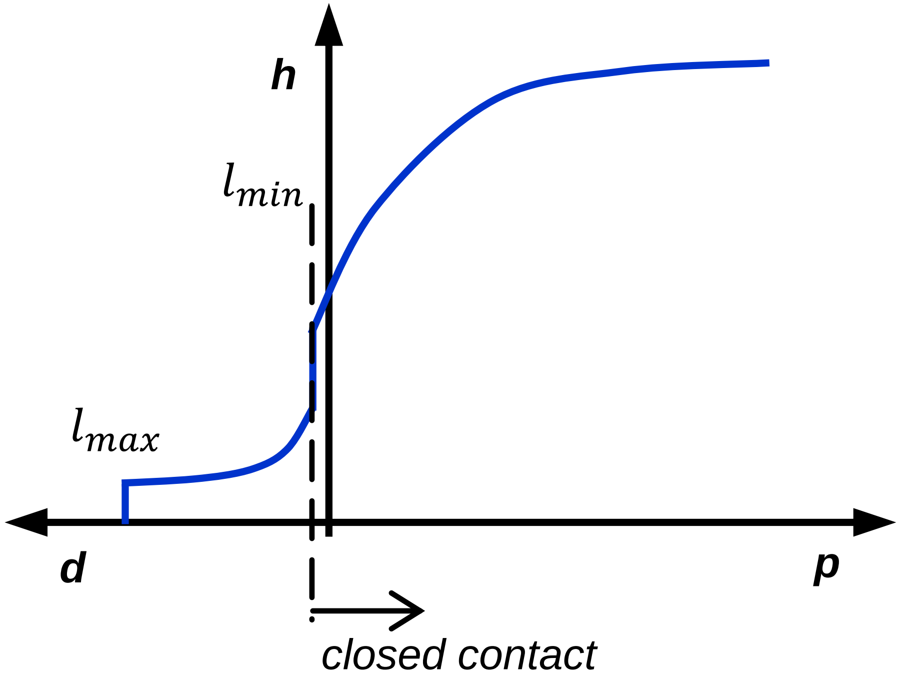
EQ.1: $h(P)$ is defined by load curve A, which contains data for contact conductance as a function of pressure.

A — Load curve ID for the a coefficient used in the formula



Thermal contacts

- Contact gap & Heat transfer



$$0 \leq gap \leq l_{\min} \qquad h = h0(p)$$

$$l_{\min} \leq gap \leq l_{\max} \qquad h = h_{cond} + h_{rad}$$

$$gap \geq l_{\max} \qquad h = 0$$

$$h = \frac{k}{L_{gap}} + f_{rad}\left(T + T_\infty\right)\left(T^2 + T_\infty^2\right)$$

Thermal boundary conditions

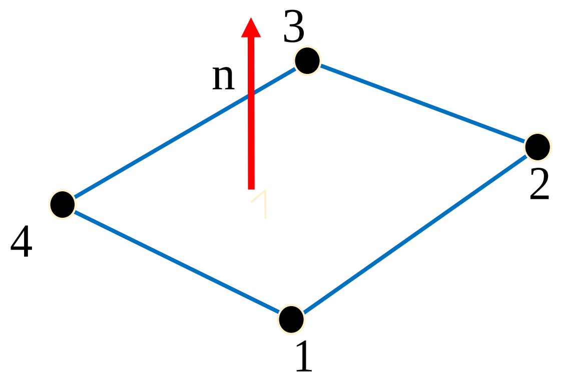
Thermal simulations in LS-DYNA

Thermal bounday conditions

- *BOUNDARY_CONVECTION
  - Air cooling
  - Applied on single segment or on segment set(s)
  - Input convection heat transfer coeficient, α, and surrounding temperature, T_∞
  - α may be given as a constant or as a curve for time or temperature dependency
  - T_∞ may be given as a constant or as a curve for time dependency

Card 2	1	2	3	4	5	6	7	8
Variable	HLCID	HMULT	TLCID	TMULT	LOC			
Type	I	F	I	F	I			
Default	none	0.	none	0.	0			

$$\dot{q} = \alpha(T)(T - T_\infty)$$



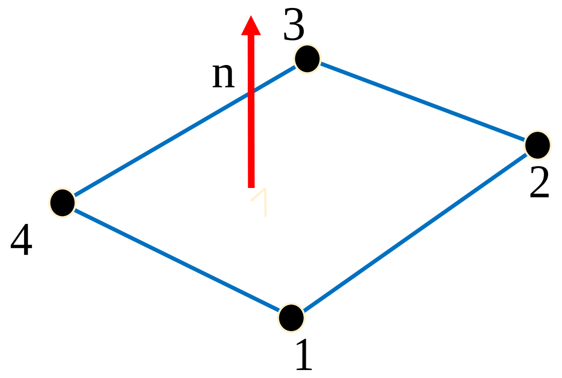
Thermal simulations in LS-DYNA

Thermal bounday conditions

- *BOUNDARY_RADIATION
  - Heat transfer between parts
  - Several options… but for the SEGMENT_SET-option:
    - Applied on single segment or on segment set(s)
    - Input Radiation heat transfer coeficient, f, and surrounding temperature, T_∞
    - f may be given as a constant or as a curve for time or temperature dependency
    - T_∞ may be given as a constant or as a curve for time dependency

Card 2	1	2	3	4	5	6	7	8
Variable	FLCID	FMULT	TLCID	TMULT	LOC			
Type	I	F	I	F	I			
Default	none	0.	none	0.	0			

$$\dot{q}'' = \sigma \varepsilon F \left(T_{surface}^4 - T_\infty^4\right) = f\left(T_{surface}^4 - T_\infty^4\right)$$



Thermal simulations in LS-DYNA

Thermal bounday conditions

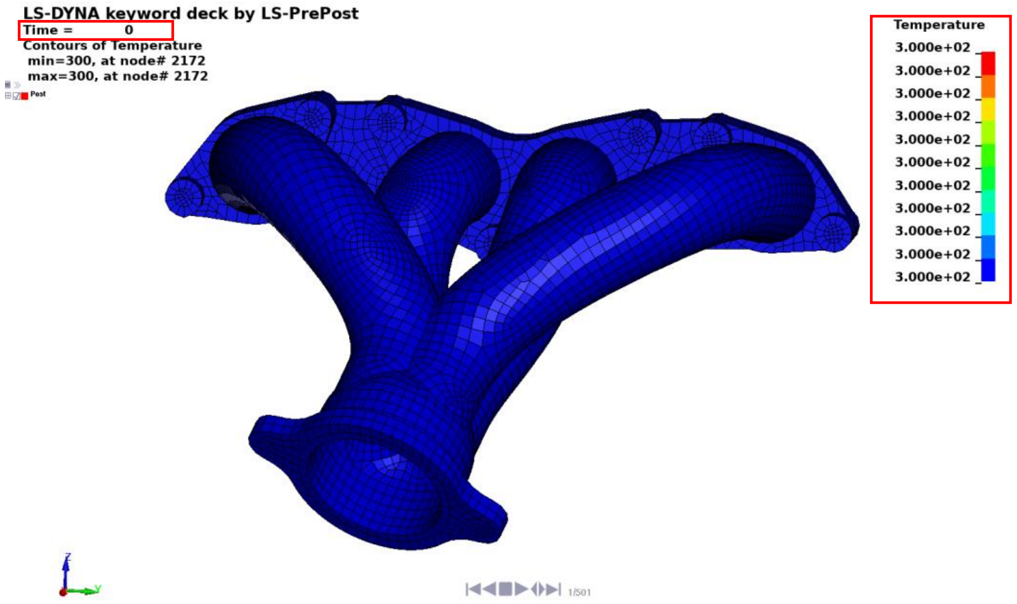
- *INITIAL_TEMPERATURE
  - Define an initial temperature for the simulation
  - Applied to a node or set of nodes.

NSID/NID = 0 => All nodes are included (default)

Card 1	1	2	3	4	5	6	7	8
Variable	NSID/NID	TEMP	LOC					
Type	I	I	I					
Default	↓	0.	0					
Remarks	1							

Node or node set id

Temperature



Thermal simulations in LS-DYNA

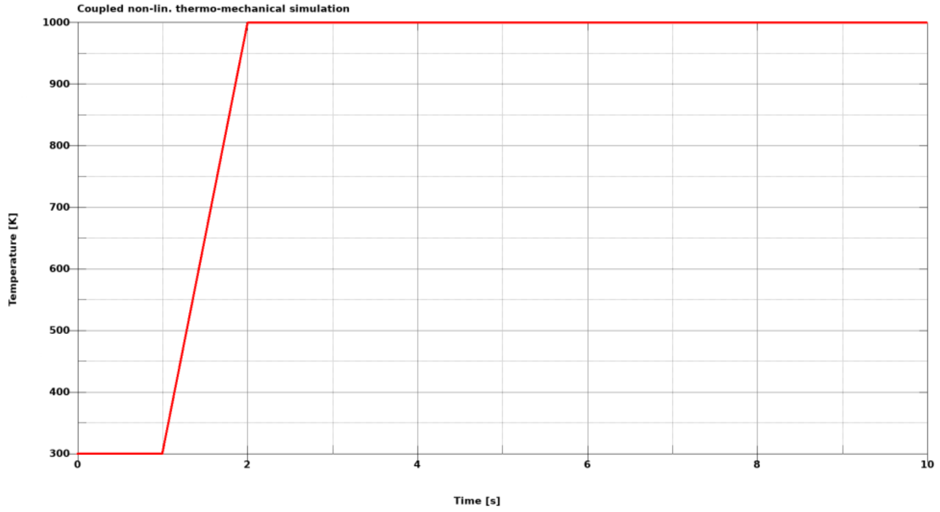
Thermal bounday conditions

- *BOUNDARY_TEMPERATURE
  - Heat source
  - Define boundary temperature for a thermal simulation
  - Several tailored options available (PERIODIC_SET, RSW, TRAJECTORY, WELD,….)

Card 1	1	2	3	4	5	6	7	8
Variable	NID	TLCID	TMULT	LOC	TDEATH	TBIRTH		
Type	I	I	F	I	F	F		
Default	none	0	0.	0	10^{20}	0.		

Node or node set id

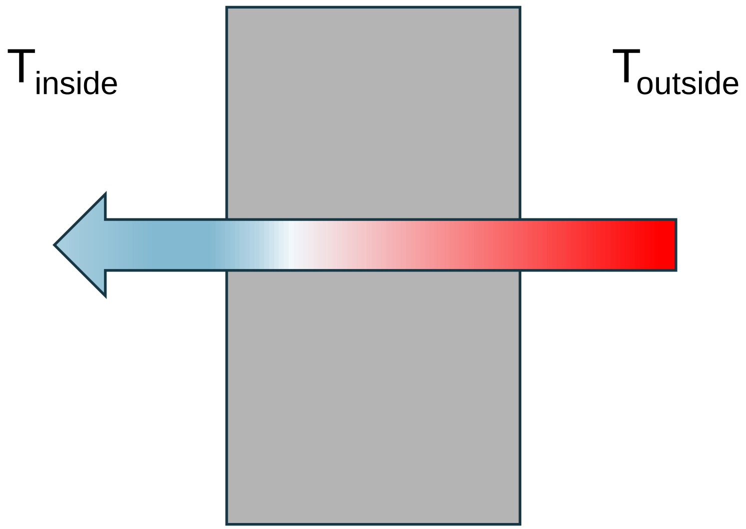
Load curve Temperature vs. time



Thermal simulations in LS-DYNA

Thermal bounday conditions

- *BOUNDARY_FLUX
  - Define boundary conditions on the amount of heat energy passing through a certain surface.
  - Given as heat flux density, q, in units of power/area.



$$q = -k\nabla T$$

Thermal strain output

Thermal simulations in LS-DYNA

Thermal strain output

- Output of thermal strain specified on *DATABASE_EXTENT_BINARY

Card 1b	1	2	3	4	5	6	7	8
Variable	NEIPH	NEIPS	MAXINT	STRFLG	SIGFLG	EPSFLG	RLTFLG	ENGFLG
Type	I	I	I	I	I	I	I	I
Default	0	0	3	0	1	1	1	1
Remarks			1	10				

VARIABLE	DESCRIPTION
STRFLG	Flag for output of strain tensors. STRFLG is interpreted digit-wise STRFLG = [*NML*],

$$STRFLG = L + M \times 10 + N \times 100$$

L.EQ.1: Write strain tensor data to d3plot, elout, and dynain. For shell and thick shell elements two tensors are written, one at the innermost and one at the outermost integration point. For solid elements a single strain tensor is written.

M.EQ.1: Write plastic strain data to d3plot.

N.EQ.1: Write thermal strain data to d3plot.

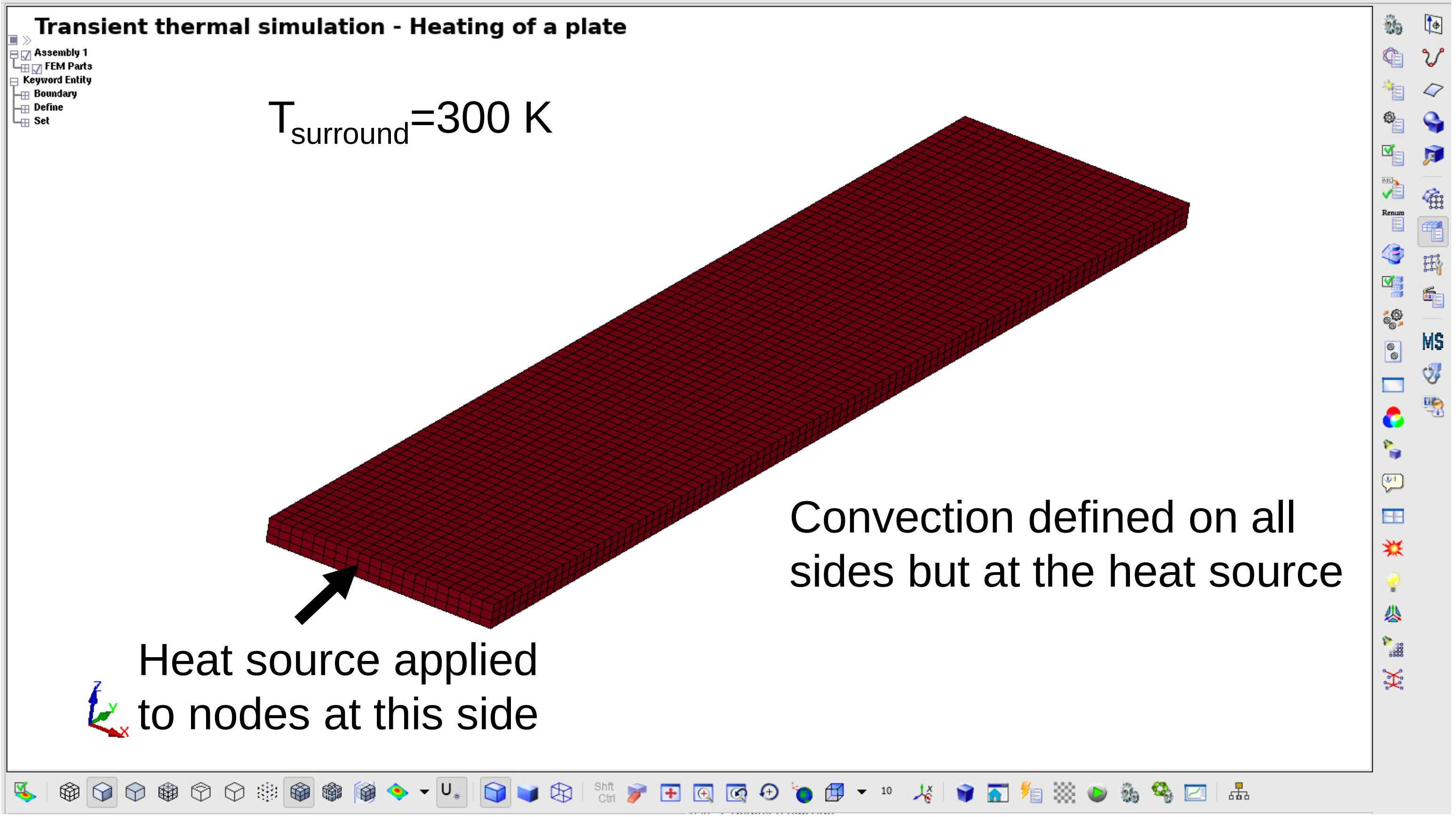
Examples. For STRFLG = 11 (011) LS-DYNA will write both strain and plastic strain tensors, but no thermal strain tensors. Whereas for STRFLG = 110, LS-DYNA will write plastic and thermal strain tensors but no strain tensors. For more information and supported elements and materials, see Remark 10.

Example 1 – Heating of a steel plate

Thermal simulations in LS-DYNA

Heating of a steel plate

- Thermal only analysis of a plate
  - Material: Steel
  - Environment: Air
  - Surrounding temp.: Room-temp.
  - Heat source temp.: 600 K

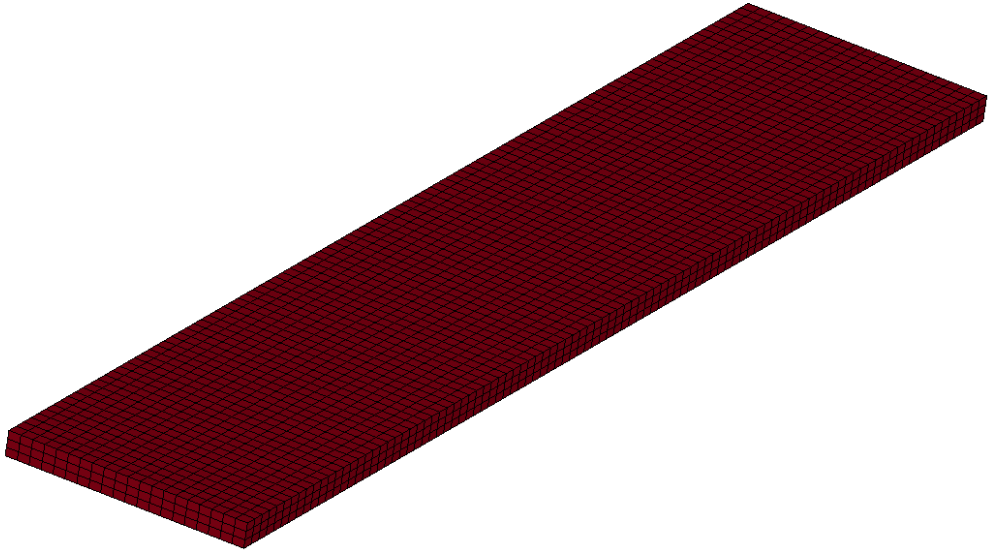


Thermal simulations in LS-DYNA

Heating of a steel plate

- Load case set-up

```
*INITIAL_TEMPERATURE_SET
$#    nsid       temp       loc
         0     300.0          0

*BOUNDARY_CONVECTION_SET
$#    ssid     pserod
         1
$#   hlcid      hmult      tlcid      tmult        loc
         0        1.0          0      300.0          0

*BOUNDARY_TEMPERATURE_SET
$#    nsid       lcid      cmult        loc     tdeath     tbirth
         6        500        1.0          0

*DEFINE_CURVE_TITLE
plate_temp
$#    lcid       sidr        sfa        sfo       offa       offo     dattyp      lcint
       500          0        0.0        0.0        0.0        0.0          0          0
$#                 a1                    o1
                  0.0                 300.0
                 10.0                 600.0
                100.0                 600.0
```

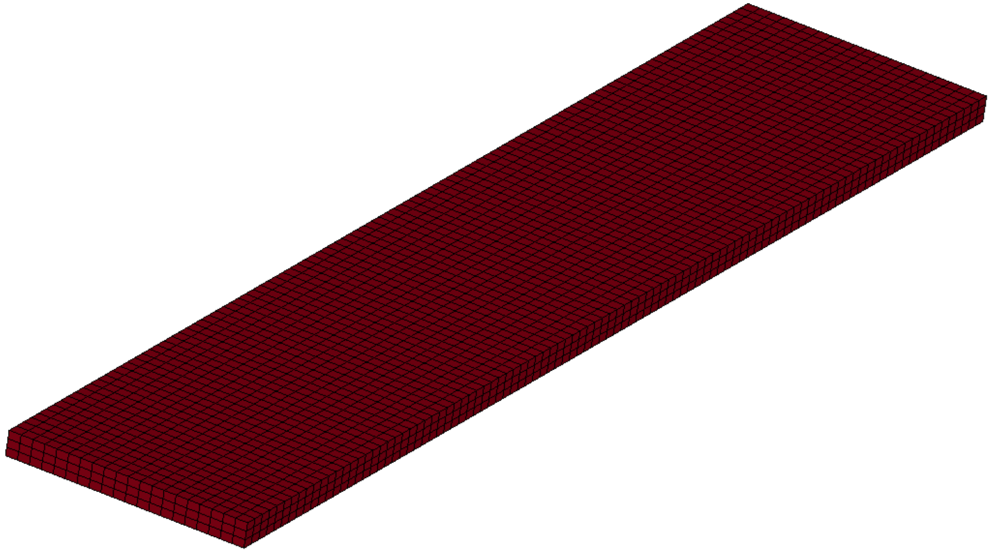


Thermal simulations in LS-DYNA

Heating of a steel plate

- Material modeling

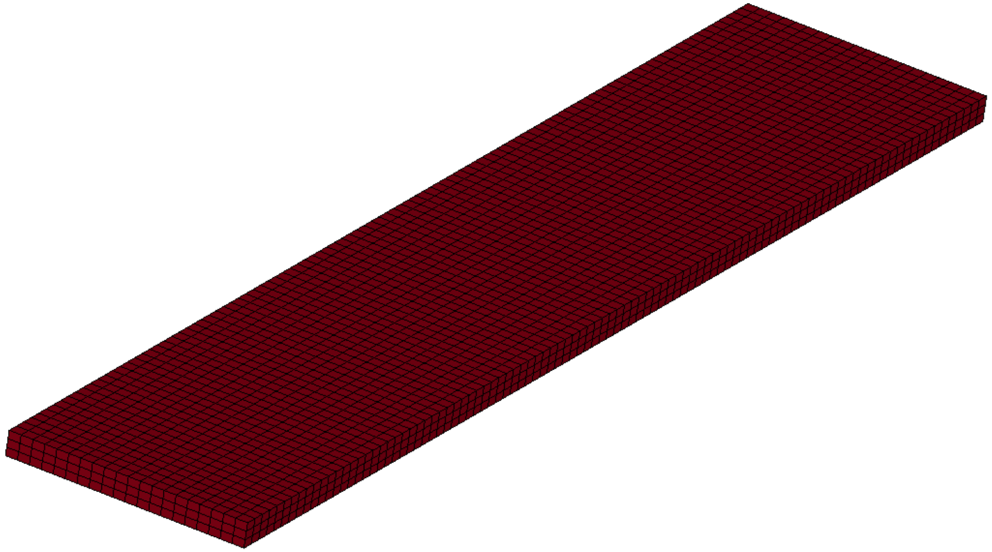
```
*PART
$#                                                                         title
Plate
$#      pid     secid       mid     eosid      hgid      grav    adpopt      tmid
          1         1         1         0         0         0         0         2

*MAT_ELASTIC
$#      mid        ro         e        pr        da        db  not used
          1 7.8500E-9  205000.0       0.3       0.0       0.0       0.0

*MAT_THERMAL_ISOTROPIC
$#     tmid        ro     tgrlc    tgmult      tlat      hlat
          2       0.0       0.0       0.0       0.0       0.0
$#       hc        tc
4.810000E8   16000.0
```



Thermal simulations in LS-DYNA

Heating of a steel plate

- Solution & solver definition



```
*CONTROL_SOLUTION
$#    soln       nlq     isnan     lcint     lcacc     ncdcf     nocop
         1         0         0      1000         0         0         0
```

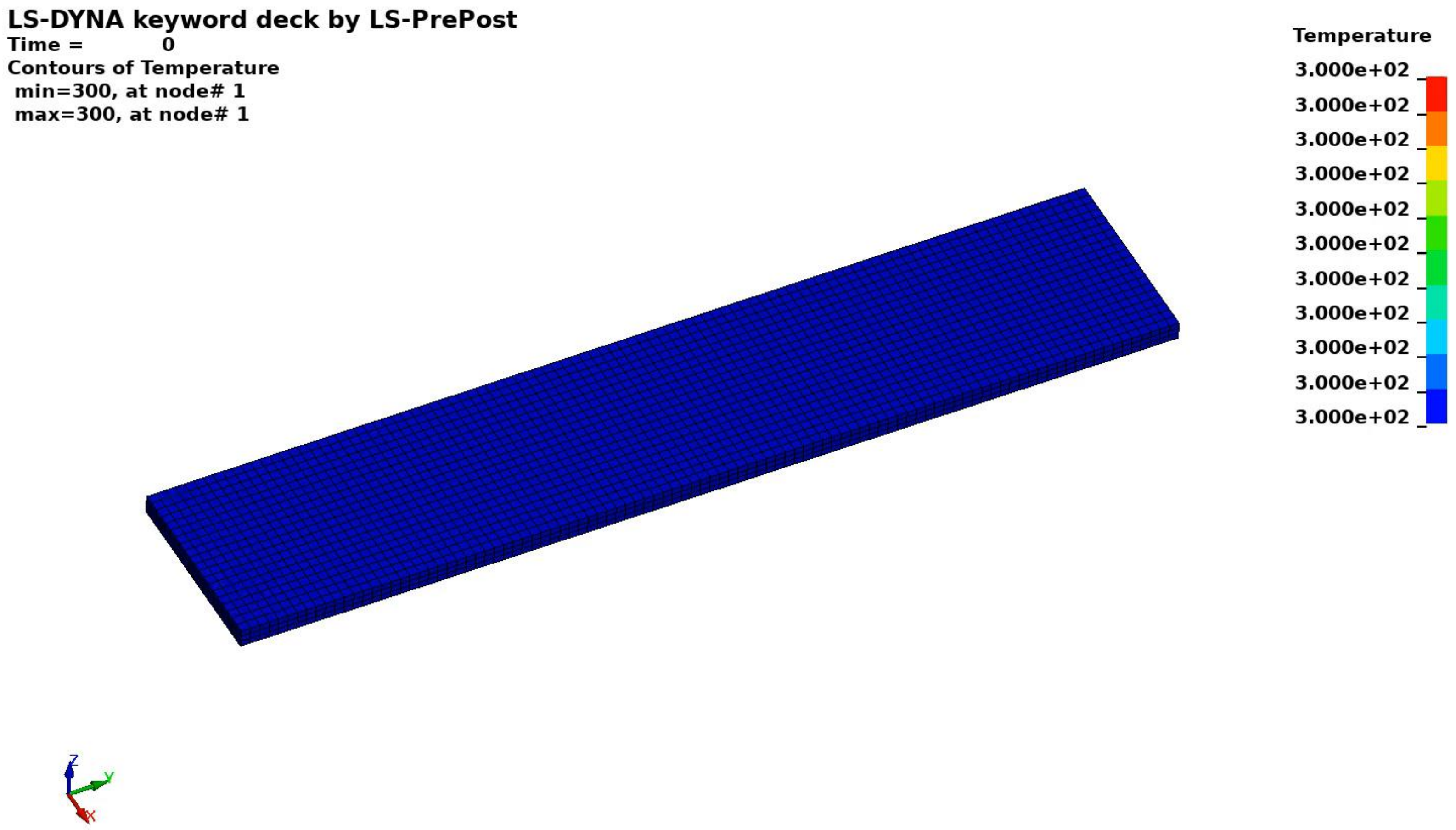
```
*CONTROL_THERMAL_SOLVER
$#   atype     ptype    solver         -       gpt    eqheat     fwork       sbc
         1         0        11                   8       1.0       1.0       0.0
```

```
*CONTROL_THERMAL_TIMESTEP
$#      ts       tip       its      tmin      tmax     dtemp      tscp      lcts
       1.0       1.0     1.e-3       0.0       0.0       1.0       0.5         0
```

Thermal simulations in LS-DYNA

Heating of a steel plate

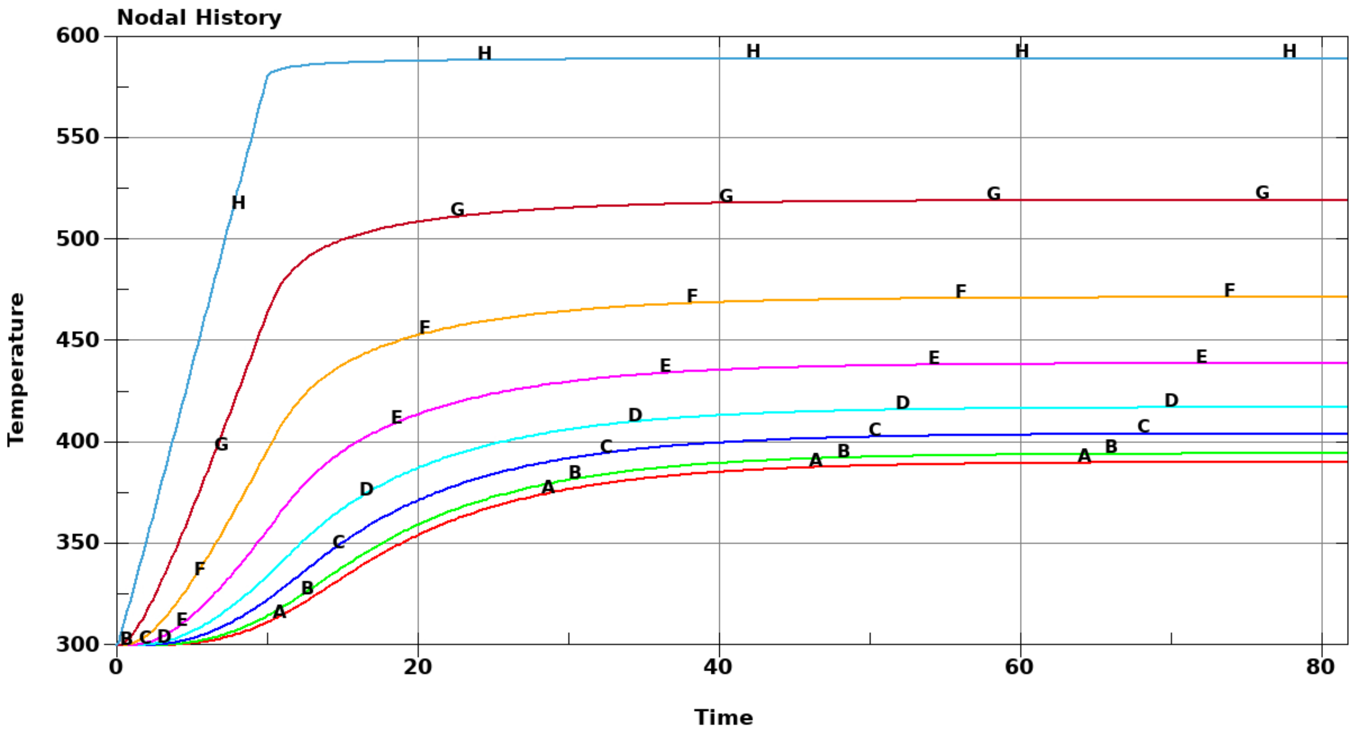
- Results

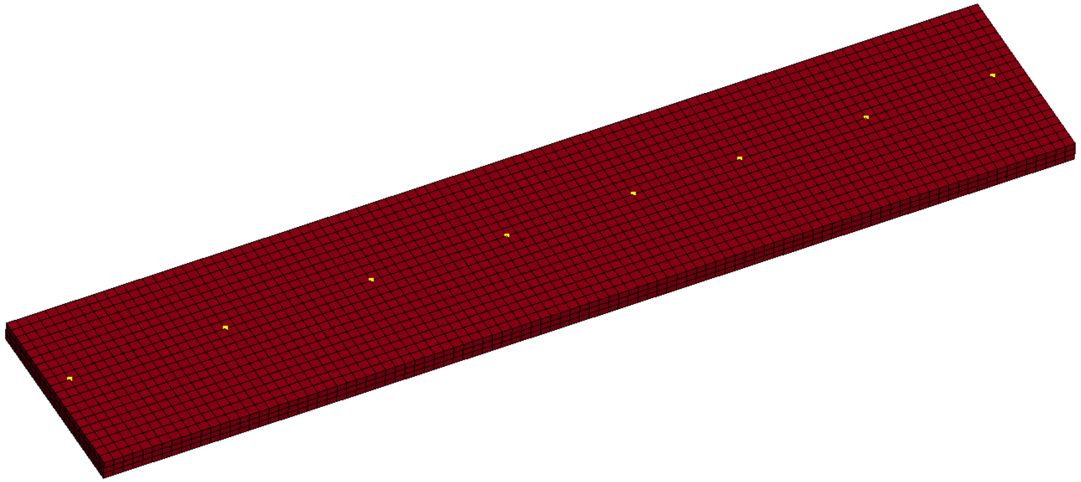


Thermal simulations in LS-DYNA

Heating of a steel plate

- Results
  - Steady-state found at t > 60s => Time consuming?
  - The problem is not path-dependent and linear => try Steady-state

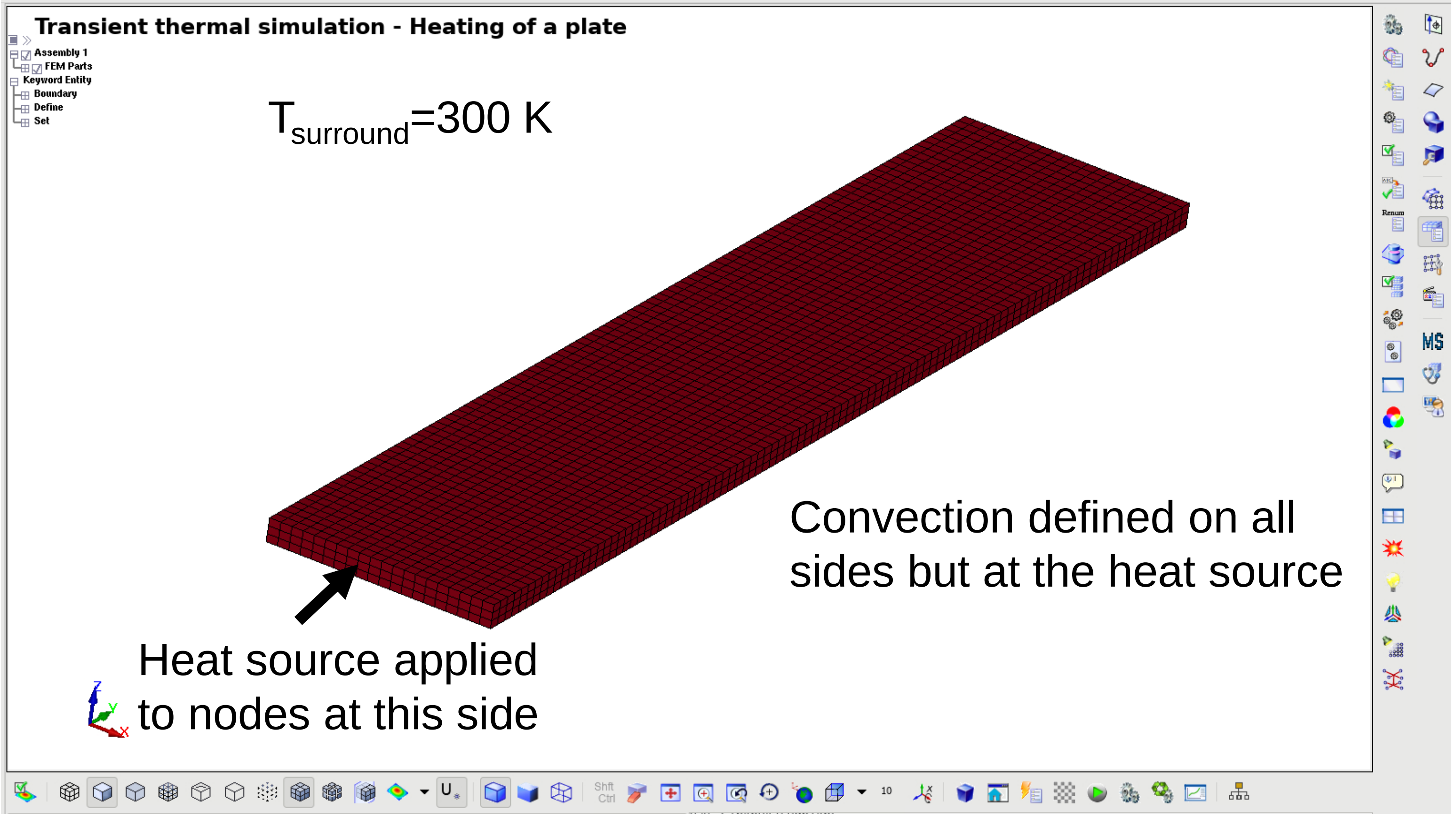




Thermal simulations in LS-DYNA

Heating of a steel plate

- Steady-state thermal analysis of a plate
  - Material: Steel
  - Environment: Air
  - Surrounding temp.: Room-temp.
  - Heat source temp.: 600 K



Thermal simulations in LS-DYNA

Heating of a steel plate

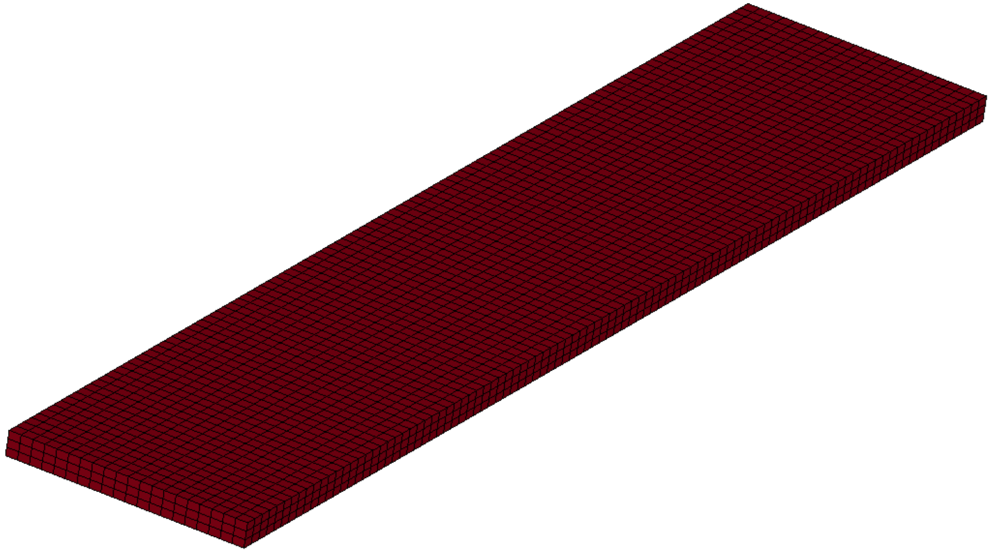
- Changes

```
*DEFINE_CURVE_TITLE
plate_temp
$#     lcid      sidr       sfa       sfo      offa      offo    dattyp
        500         0       0.0       0.0       0.0       0.0         0
$#                 a1                  o1
                  0.0               600.0
                 10.0               600.0
                100.0               600.0
```

```
*CONTROL_THERMAL_SOLVER
$#   atype     ptype    solver         -       gpt    eqheat     fwork       sbc
         0         0        11                   8       1.0       1.0       0.0
```

```
*CONTROL_THERMAL_TIMESTEP
$#      ts       tip       its      tmin      tmax     dtemp      tscp      lcts
         0       1.0       1.0       0.0       0.0                 0.5         0
```

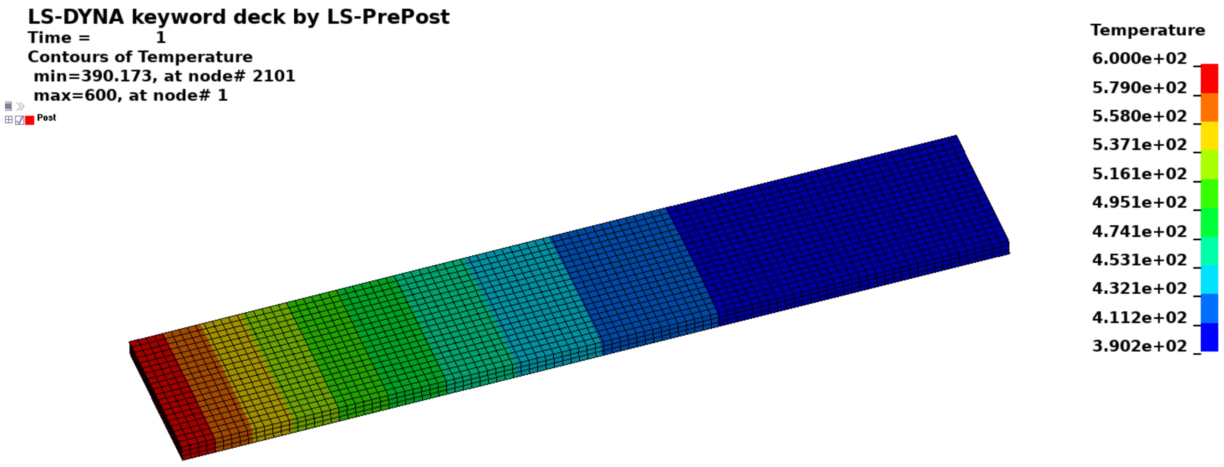
```
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas     nosol
       1.0         0       0.0       0.0       0.0
```



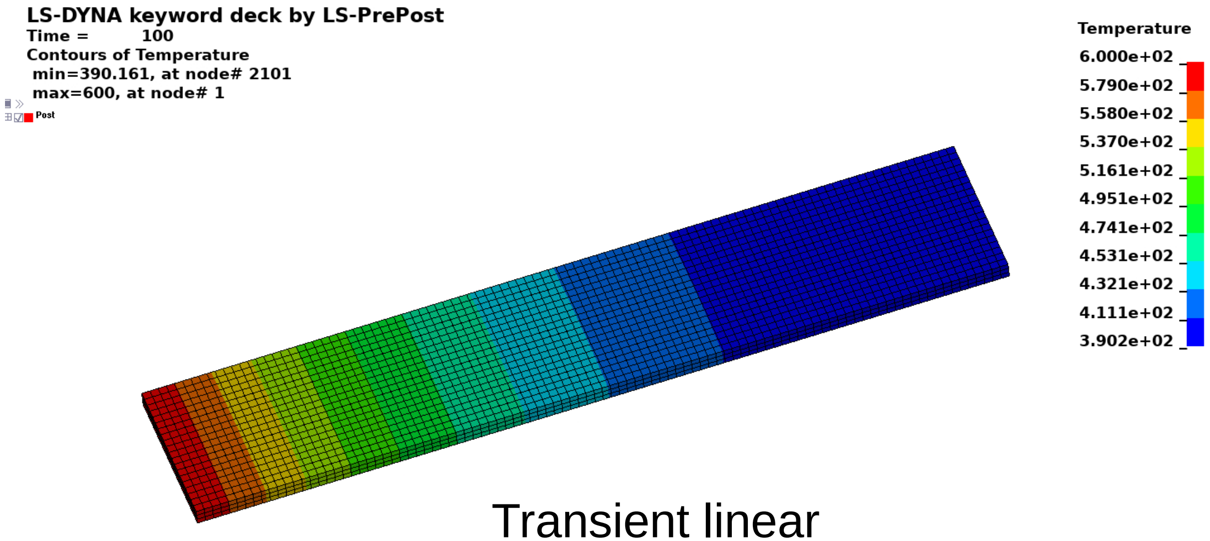
Thermal simulations in LS-DYNA

Heating of a steel plate

- Results
  - Solution time for Steady state is, in this case, 24 times faster than for the transient linear solution!



Steady state



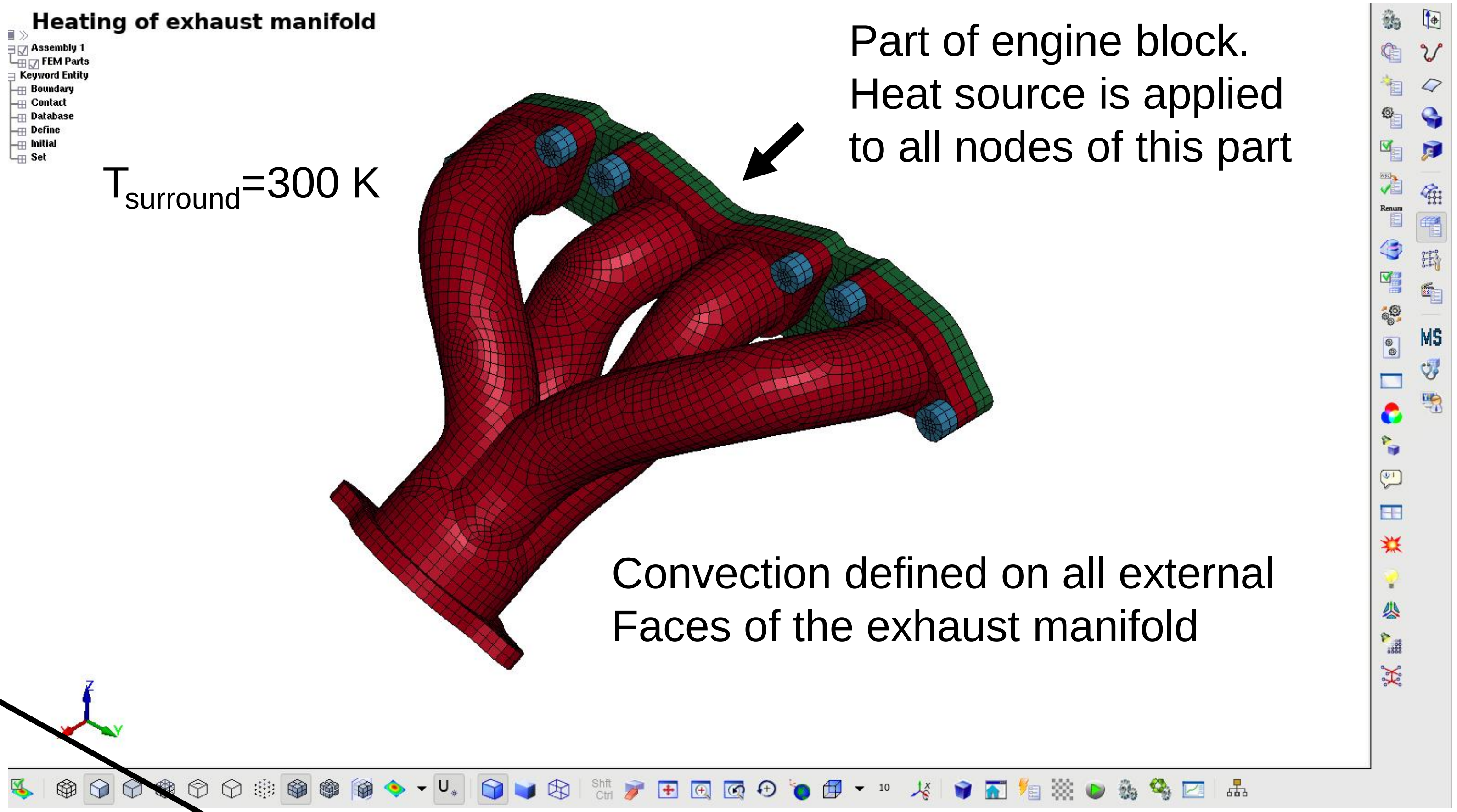
Transient linear

Example 2 – Exhaust manifold

Thermal simulations in LS-DYNA

Exhaust manifold

- Thermo-mechanical simulation
  - Material: Steel
  - Environment: Air
  - Surrounding temp.: Room-temp.
  - Heat source temp.: 600 K
  - Pre-stressed bolts
  - Thermal contacts
    - Conductance is dependent on interface pressure
  - Stress output requested
  - Non-linear thermal properties
    - Conductivity & specific heat are temperature dependent



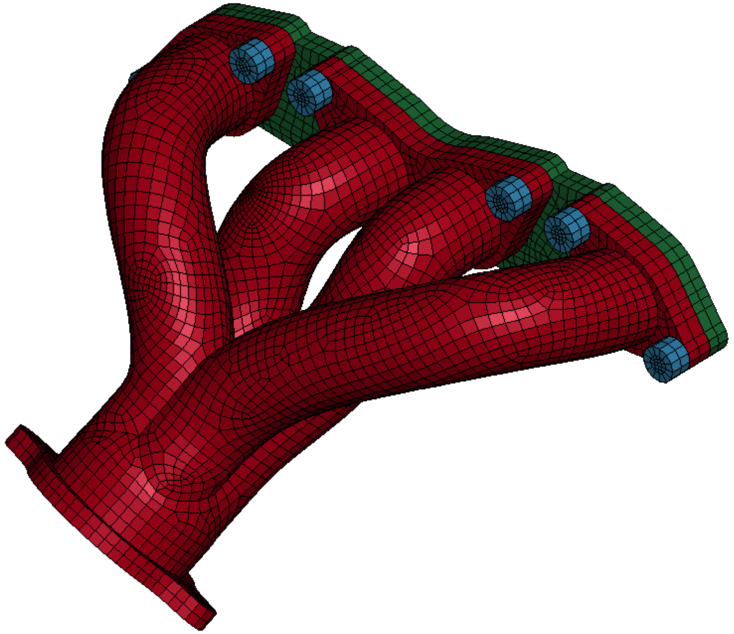
Need to use a non-linear, fully coupled approach

Thermal simulations in LS-DYNA

Exhaust manifold

- Load case set-up

```
*INITIAL_TEMPERATURE_SET
$#     nsid       temp        loc
          0      300.0          0

*BOUNDARY_CONVECTION_SET
$#     ssid     pserod
          1
$#    hlcid      hmult      tlcid      tmult        loc
          0        1.0          0      300.0          0

*BOUNDARY_TEMPERATURE_SET
$#     nsid       lcid      cmult        loc     tdeath     tbirth
          6        500        1.0          0


*DEFINE_CURVE_TITLE
engine_temp
$#     lcid       sidr        sfa        sfo       offa       offo     dattyp      lcint
        500          0        0.0        0.0        0.0        0.0          0          0
$#                  a1                    o1
                   0.0                 300.0
                  10.0                 600.0
                 100.0                 600.0
```

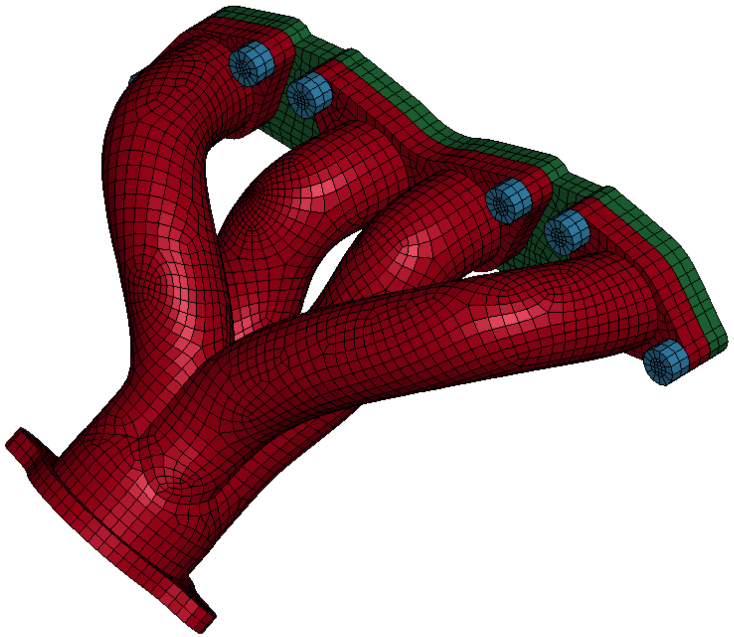


Thermal simulations in LS-DYNA

Exhaust manifold

- Material modeling

```
*PART
manifold
$#     pid     secid       mid     eosid      hgid      grav     adopt      tmid
         1         1         1         0         0         0         0         2
*PART
bolts
$#     pid     secid       mid     eosid      hgid      grav     adopt      tmid
         2         1         1         0         0         0         0         2
*PART
engine
$#     pid     secid       mid     eosid      hgid      grav     adopt      tmid
         3         1         1         0         0         0         0         2

*MAT ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1 7.8500E-9  205000.0       0.3       0.0       0.0       0.0


*MAT_THERMAL_ISOTROPIC_TD
$#    tmid       tro     tgrlc    tgmult      tlat      hlat
         2       0.0       0.0       0.0       0.0       0.0
$#      t1        t2        t3        t4        t5        t6        t7        t8
     200.0     400.0    1500.0       0.0       0.0       0.0       0.0       0.0
$#      c1        c2        c3        c4        c5        c6        c7        c8
 4.81000E8 4.81000E9 4.81000E9       0.0       0.0       0.0       0.0       0.0
$#      k1        k2        k3        k4        k5        k6        k7        k8
   16000.0  160000.0  160000.0       0.0       0.0       0.0       0.0       0.0
```

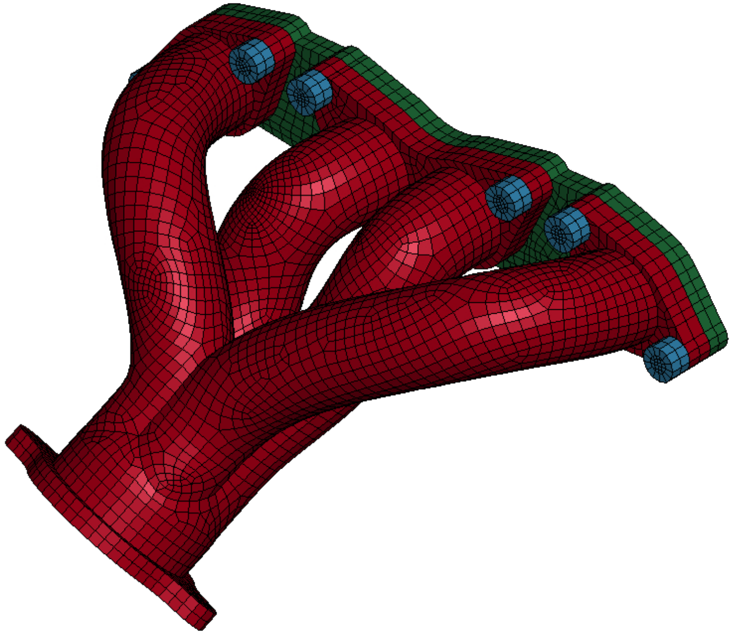


Thermal simulations in LS-DYNA

Exhaust manifold

- Solution & solver definition

```
*CONTROL_SOLUTION
$#    soln       nlq     isnan     lcint     lcacc     ncdcf     nocop
         2         0         0      1000         0         0         0

*CONTROL_THERMAL_SOLVER
$#   atype     ptype    solver         -       gpt    eqheat     fwork       sbc
         1         2        11                   8       1.0       1.0       0.0

*CONTROL_THERMAL_TIMESTEP
$#      ts       tip       its      tmin      tmax     dtemp      tscp      lcts
         1       1.0     1.e-3       0.0       0.0       1.0       0.5         0

*CONTROL_THERMAL_NONLINEAR
$#  refmax       tol       dcp    lumpbc    thlstl    nlthpr    phchpn
       100     1.e-6       0.5         0       0.0         1     100.0
```

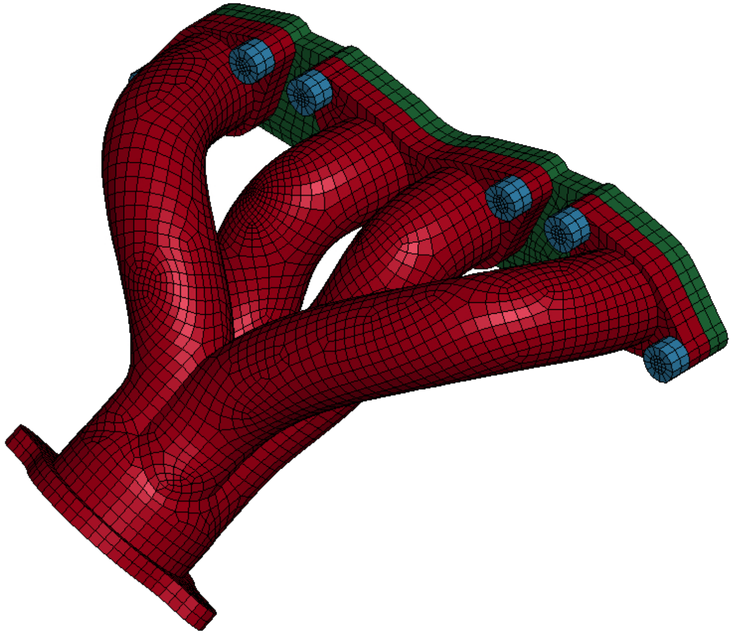


Thermal simulations in LS-DYNA

Exhaust manifold

- Thermal contact
  - All contacts are defined the same in this example

```
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_THERMAL_FRICTION_ID
$#     cid                                                                 title
         1bolts and engine
$#   surfa     surfb  surfatyp  surfbtyp   saboxid   sbboxid      sapr      sbpr
         3         2         3         3         0         0         0         0
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
       0.2       0.2       0.0       0.0       0.0               0.01.00000E20
$#    sfsa      sfsb      sast      sbst     sfsat                 fsf       vsf
       1.0       1.0       1.0       0.0       1.0       1.0       1.0       1.0
$#       k      frad        h0      lmin      lmax     ftosa    bc_flg      algo
                        3000.0     0.001      0.01       1.0         0         0
$#   lcfst     lcfdt   formula         a         b         c         d       lch
         0         0         1        10         0         0         0         0
$#    soft    sofscl    lcidab    maxpar     sbopt     depth     bsort    frcfrq
         0       0.1         0     1.025       2.0         2         0         1
$#  penmax    thkopt    shlthk     snlog      isym     i2d3d    sldthk    sldstf
       0.0         0         2         0         0         0       0.0       0.0
$#    igap    ignore    dprfac    dtstif    edgek    unused    flangl    cid_rcf
         2         1       0.0       0.0       0.0                 0.0         0
```



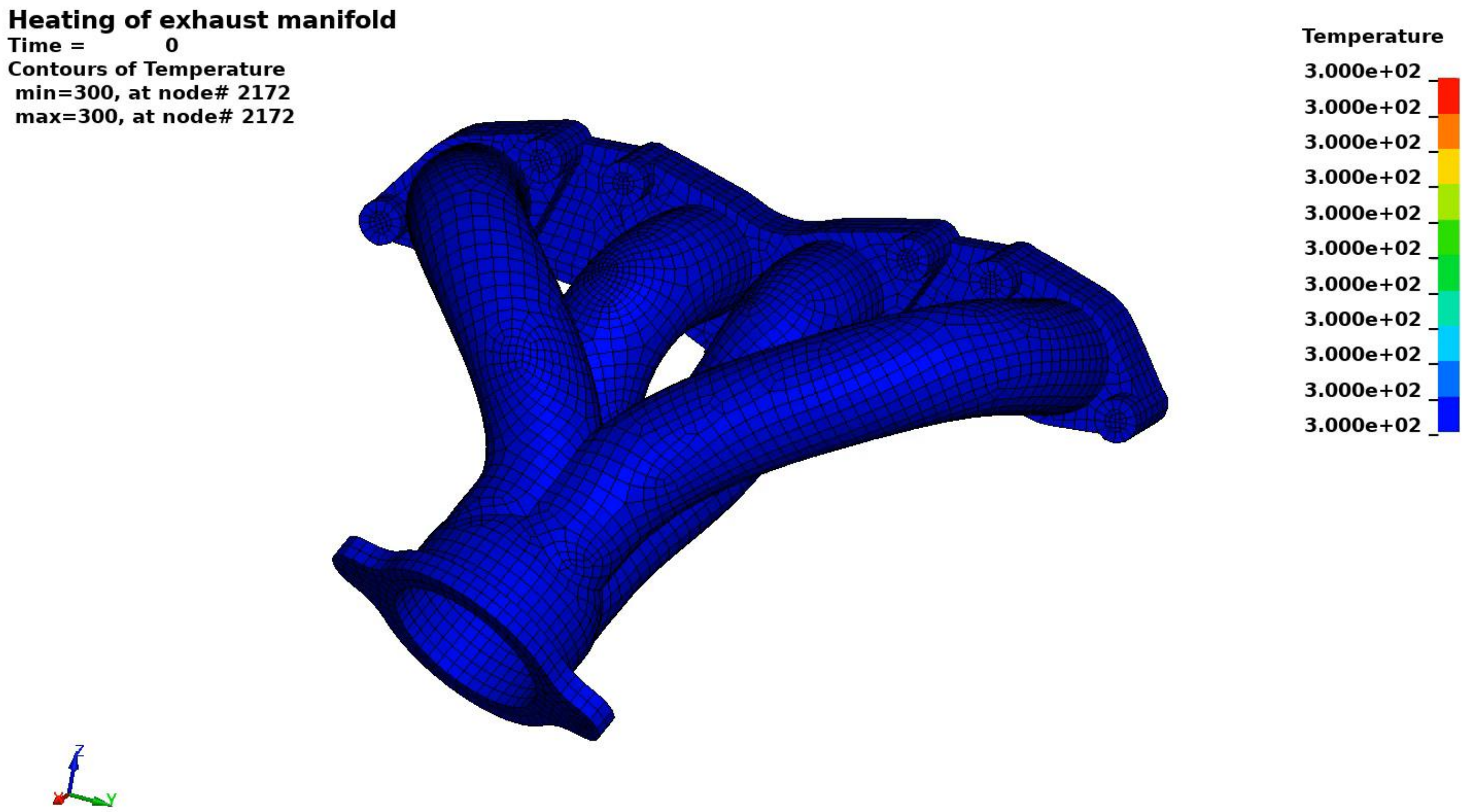
k and frad are not defined.

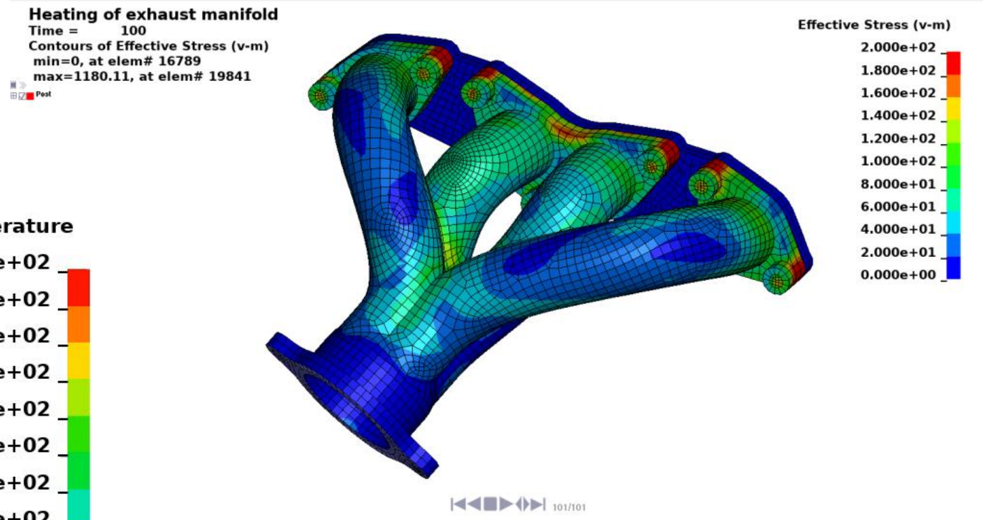
The bolts are prestressed so there is no gap in the model.

Thermal simulations in LS-DYNA

Exhaust manifold

- Results

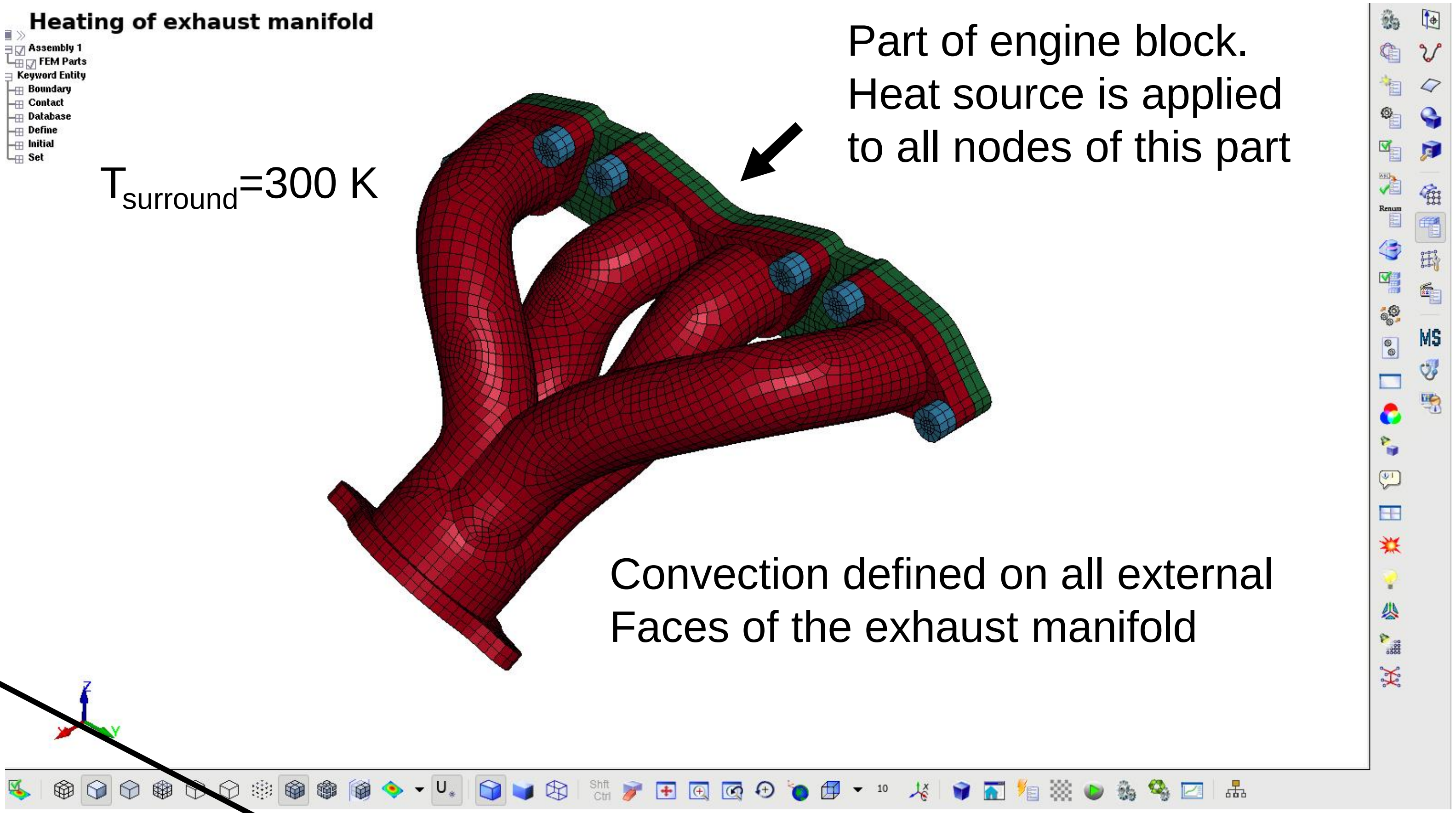




Thermal simulations in LS-DYNA

Exhaust manifold

- Thermo-mechanical simulation
  - Material: Steel
  - Environment: Air
  - Surrounding temp.: Room-temp.
  - Heat source temp.: 600 K
  - Pre-stressed bolts
  - Thermal contacts
    - Conductance is **not** dependent on interface pressure
  - Stress output requested
  - Non-linear thermal properties
    - Conductivity & specific heat are temperature dependen



$T_{surround}$=300 K

Part of engine block. Heat source is applied to all nodes of this part

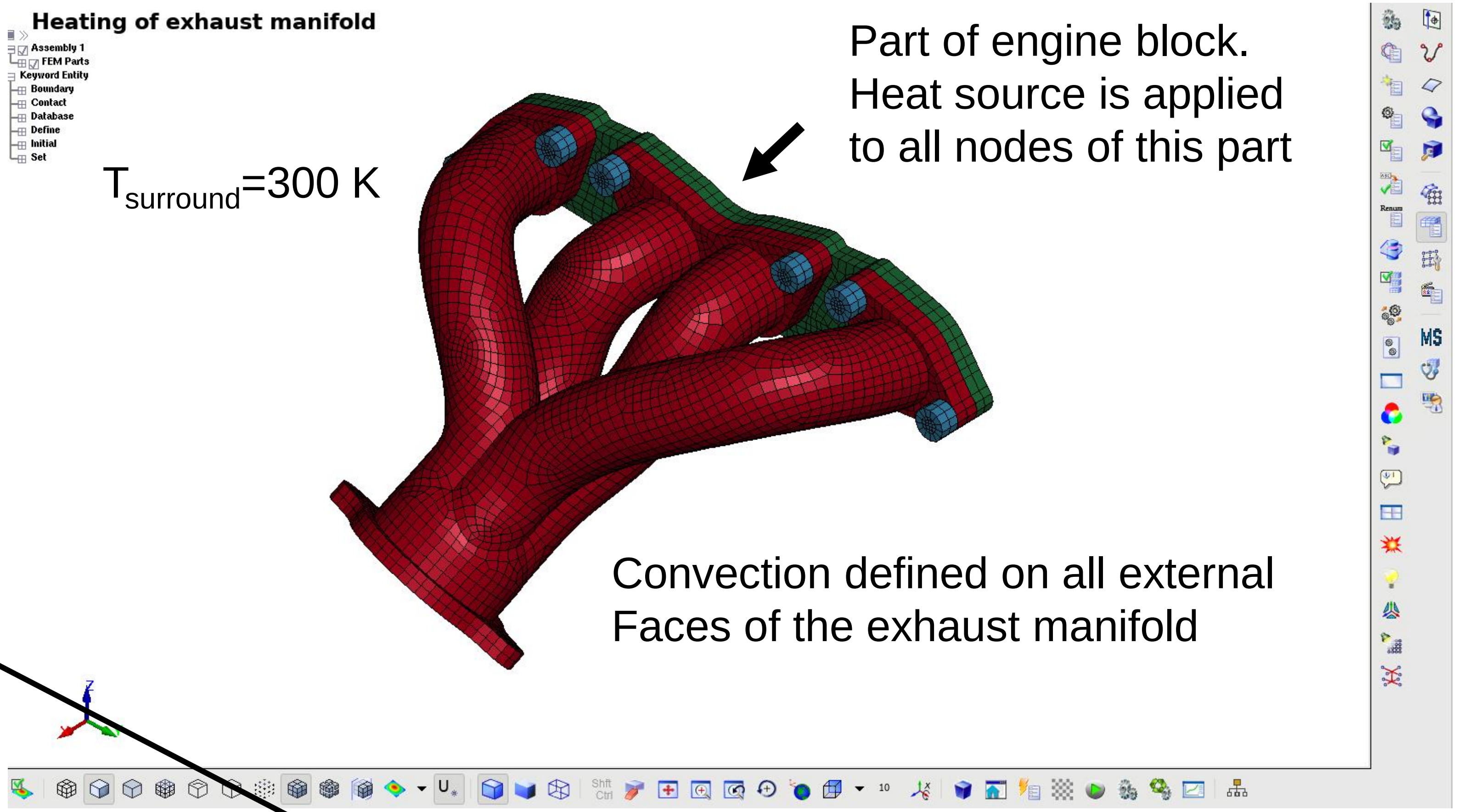
Convection defined on all external Faces of the exhaust manifold

May use a non-linear, un-coupled thermo-mechanical approach

Thermal simulations in LS-DYNA

Exhaust manifold

- Thermo-mechanical simulation
  - Material: Steel
  - Environment: Air
  - Surrounding temp.: Room-temp.
  - Heat source temp.: 600 K
  - Pre-stressed bolts
  - Thermal contacts
    - Conductance is **not** dependent on interface pressure
  - Stress output requested
  - Linear thermal properties
    - Conductivity & specific heat are **not** temperature dependent

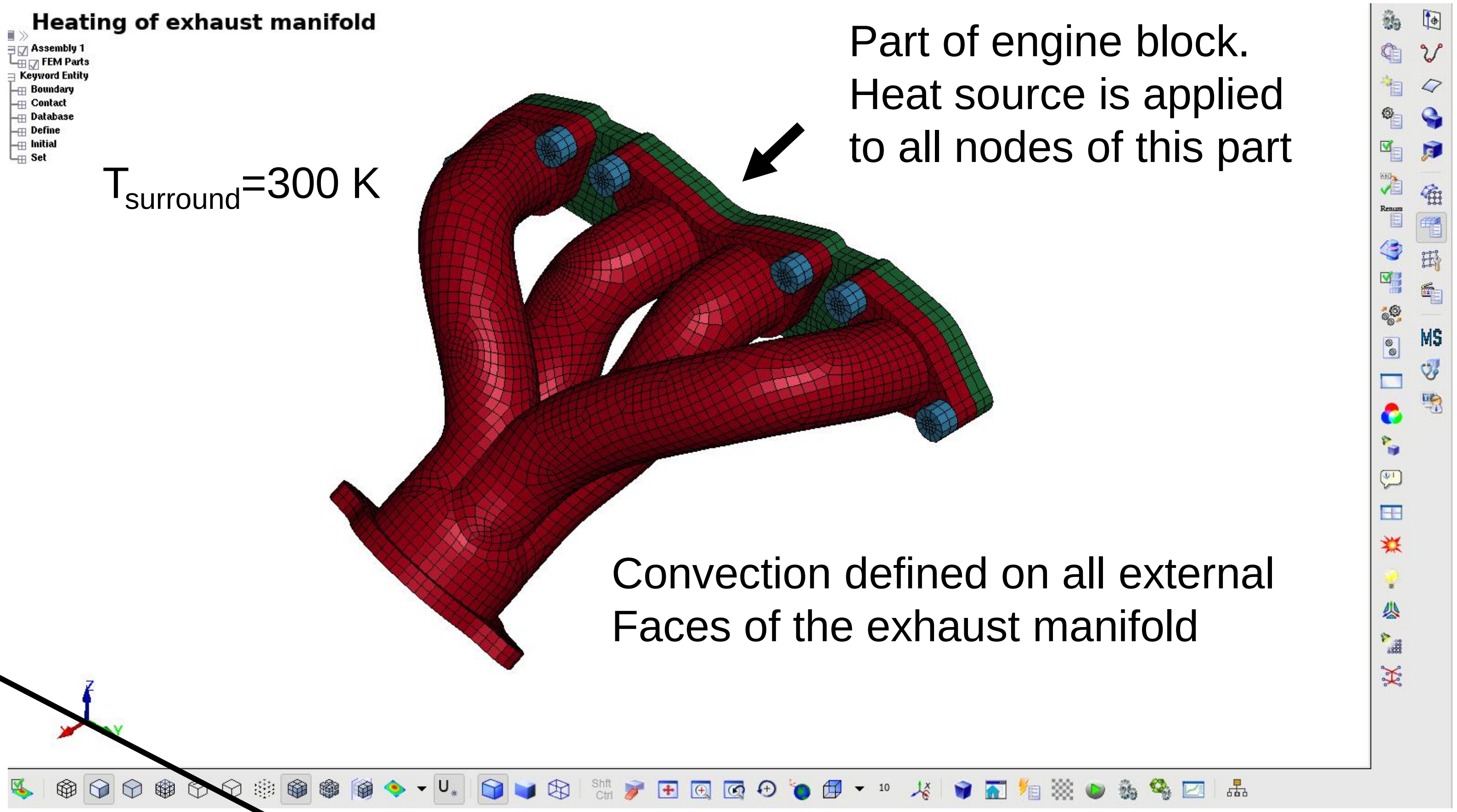


May use a linear, un-coupled thermo-mechanical approach

Thermal simulations in LS-DYNA

Exhaust manifold

- Thermo-mechanical simulation
  - Material: Steel
  - Environment: Air
  - Surrounding temp.: Room-temp.
  - Heat source temp.: 600 K
  - Pre-stressed bolts
  - Thermal contacts
    - Conductance is **not** dependent on interface pressure
  - Stress output **not** requested
  - Linear thermal properties
    - Conductivity & specific heat are **not** temperature dependent



$T_{surround}$=300 K

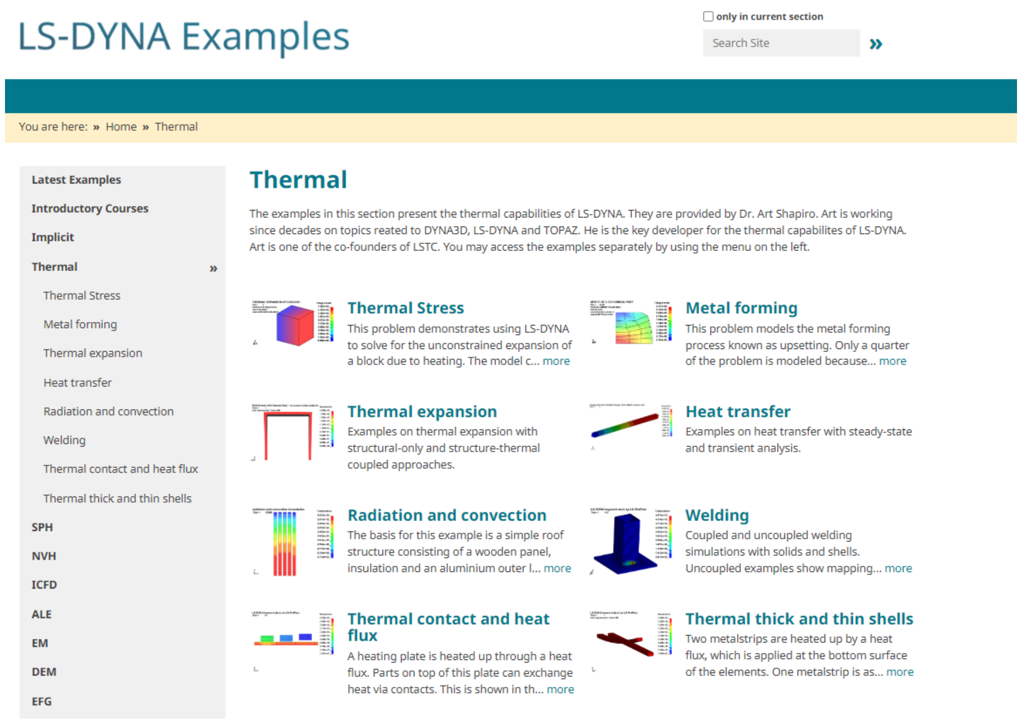
Part of engine block. Heat source is applied to all nodes of this part

Convection defined on all external Faces of the exhaust manifold

May use a linear, thermal only approach

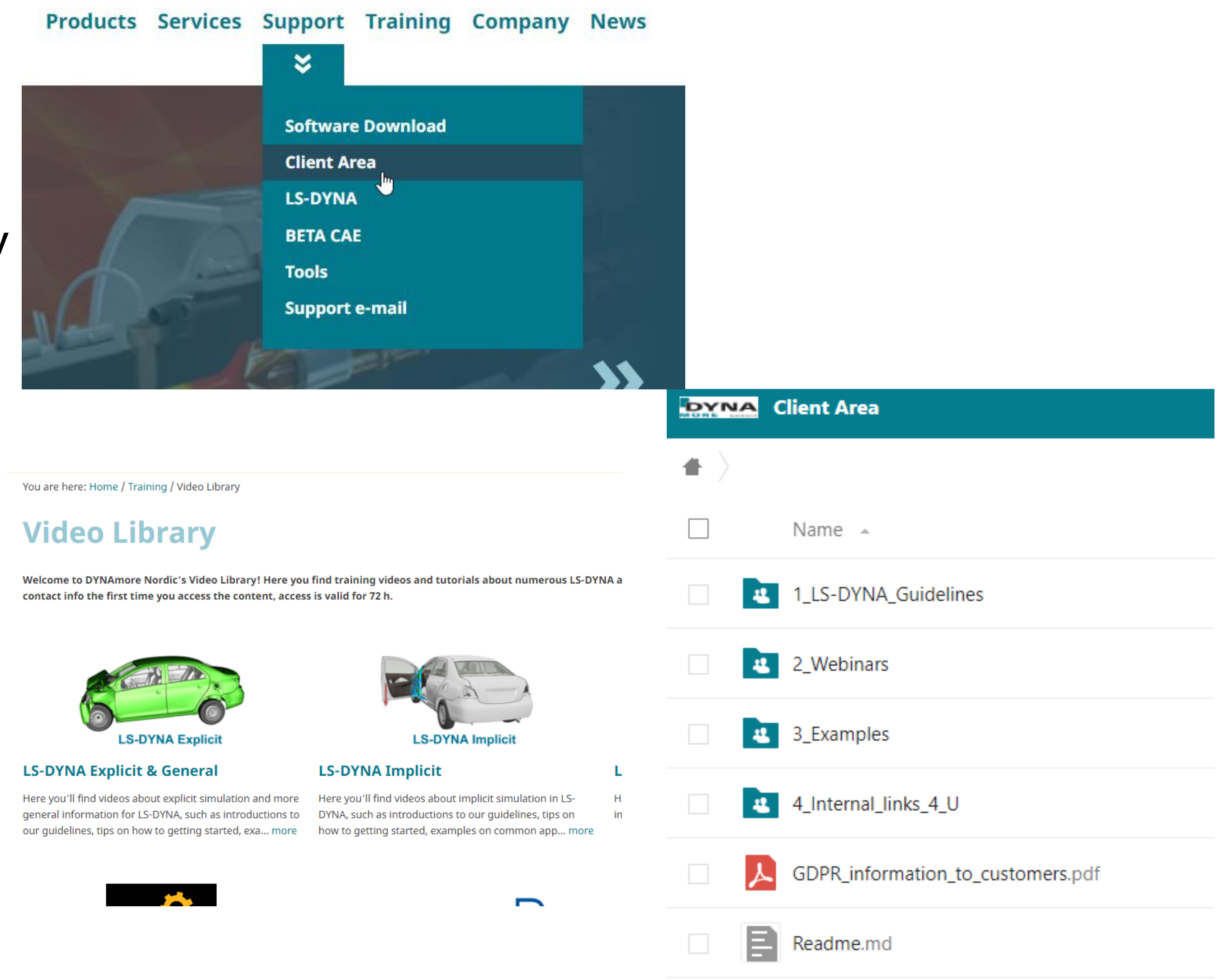
Further information

- Dynalook (www.dynalook.com)
  - Papers from European and International conferences
- YouTube
  - Dynamore GmbH
    - Conjugate Heat Transfer - Tool Cooling
    - …
- LS-DYNA Examples (www.dynaexamples.com)
  - Papers from European and International conferences

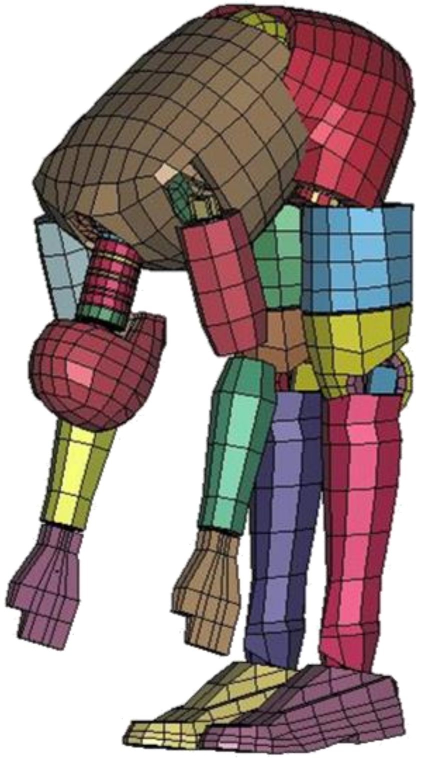


Related webinars and short videos

- Client Area and Short Videos
- Free of charge Webinar every other Thursday



Thank You



DYNAMore Nordic AB
Brigadgatan 5
587 58 Linköping, Sweden

Tel.: +46 - (0)13 23 66 80
info@dynamore.se

www.dynamore.se
www.dynaexamples.com
www.dynasupport.com
www.dynalook.com

Find us on

DYNAmore worldwide
Germany ▪ France ▪ Italy ▪ Sweden ▪ Switzerland ▪ USA