Recent Updates to the Structural Conjugate Heat Transfer Solver

Thomas Kloeppel, Peter Vogel
DYNAmore GmbH, Stuttgart, Germany

Abstract

In this contribution recent developments in the structural conjugate heat transfer solver in LS-DYNA® are presented and discussed. The motivation for new implementations results from the specific needs in complex manufacturing processes such as for example hot forming, heat treatment and welding or multi-physics problems such as battery modelling.
The paper first addresses two new options for the thermal contact. First of all, heat transfer between a shell edge and a surface (either shell or solid face) can now be considered. Second, a special welding contact formulation has been implemented. Above a certain temperature, the formulation switches from a sliding to a tied formulation and uses different parameters for the heat transfer. Although both modifications have been motivated by line welding simulations, they have also proven to be helpful for other applications.
As second topic of the contribution, the recently implemented thermal composite thick shell element is discussed. Here additional temperature degrees of freedom are introduced for each layer of the composite structure. As main application a coupled model for batteries is presented that accounts for the mechanical, electro-magnetical and thermal properties of battery cells.
Third, recent extensions to the thermal Dirichlet boundary conditions are shown. The definition of birth and death times allows activating or removing the constraints at certain times. Furthermore, as a simplified modelling of resistance spot welding, temperatures can directly be prescribed on an ellipsoidal region of the model that represents the weld nugget.

Shell edge thermal contact

This section introduces the first of two newly developed contact algorithms in LS-DYNA [1]. The implementations have been motivated by computational welding mechanics, but are general enough to be helpful in other thermo-mechanical coupled applications as well.
Process simulations in manufacturing industries usually use shell elements or discretizations with both shell and solid elements. The standard thermal contact algorithms in LS-DYNA only consider surface-to-surface type contacts. Consequently, simulations of heat transfer between shell edges to solid or shell surfaces are out of the scope of these algorithms. Since the simulation of T-joints modelled with shell elements and the connection of a shell edge to a solid face are of particular interest in for example welding simulations, a new contact option has been developed.
The new implementation makes use of the thermal thick shell formulation in LS-DYNA that features bi-linear in-plane shape functions and a quadratic temperature shape function across the thickness of the shell. Virtual nodes are created at bottom and top of the shell element and those can be used to reconstruct the three-dimensional geometry at the shell edges.
From a user's perspective the new implementation is not considered as a new contact type, but rather a variation of existing contact definitions. In the keyword file the parameter **ALGO** in the thermal contact card is to be set set to 2 (two-way) or 3 (one-way) to activate the reconstructed geometry in the contact algorithm.

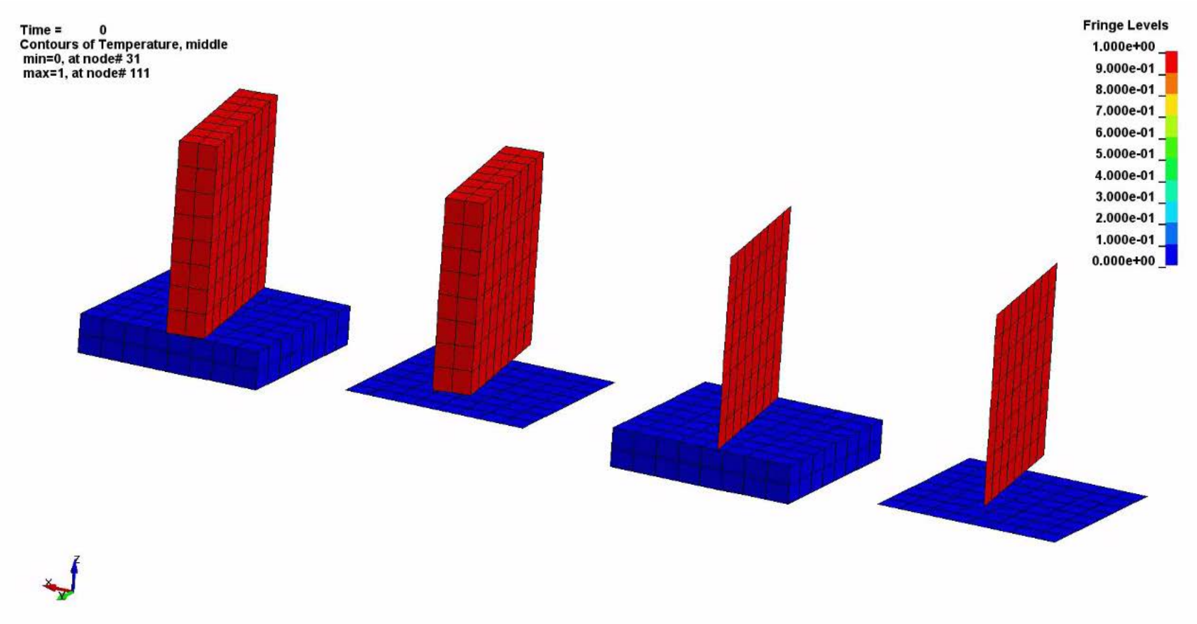

Fig.1: Validation test cases for shell edge thermal contact, initial temperature distribution.

The formulation has been validated using the heat transfer in a T-joint. Four different models are investigated and shown in Fig. 1. In all cases a hot upper part is brought in contact with a cold surface. The resulting temperature fields in Fig. 2 demonstrate that the heat transfer across the edge of the upper part does not change significantly if these parts are modelled with solid or shell elements.

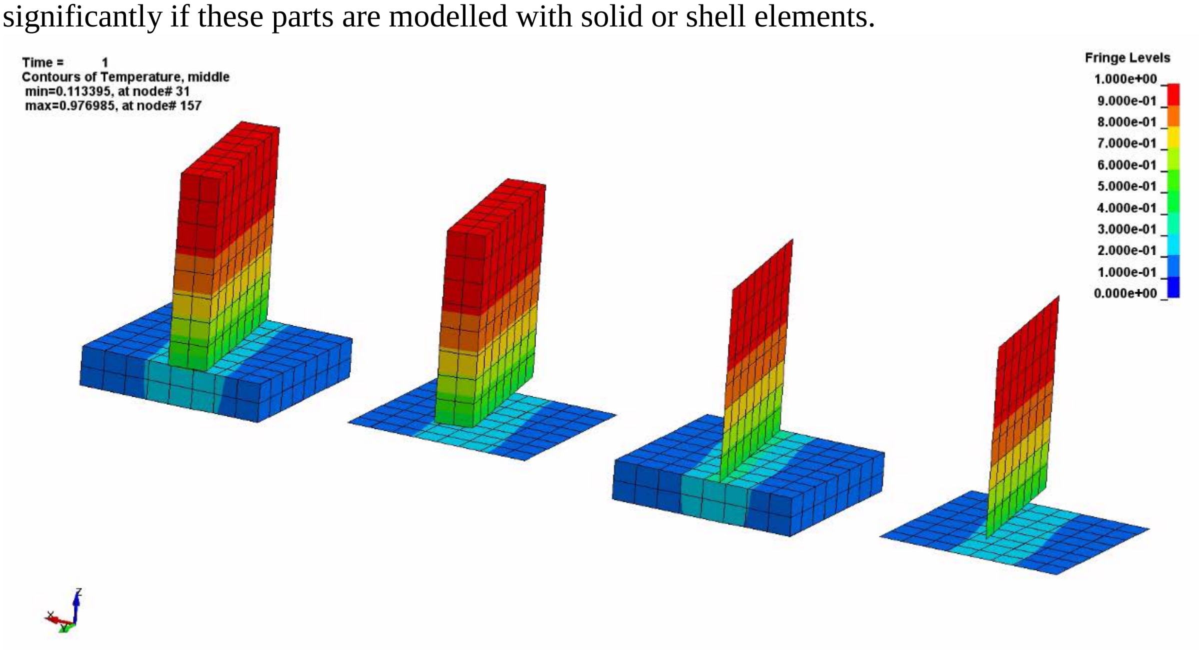

Fig.2: Validation test cases for shell edge thermal contact, resulting temperature distribution.

Welding Contact

The second new development has also been motivated by welding applications. Many of the processes used today do make use of a filler material, but the materials of the parts to be combined are locally heated up, e.g. by a laser beam. If the melting temperature is exceeded in the contact surface, a strong connection is established and the parts can no longer be separated.
This behavior is represented by a new thermo-mechanical contact formulation. It can be addressed in the LS-DYNA input as *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIED_WELD_THERMAL. Mechanically, this contact can switch locally from a sliding contact to a welded, i.e. tied, contact condition. The switch is triggered if the temperature on both surfaces in the contact zone reaches a certain user- defined value. Of course, the contact formulation for the rest of the contact surface is not affected by this switch. Once the welded state of the contact has been reached the contact cannot return to a sliding formulation.
On the thermal side a similar behavior is modelled. The contact keyword introduces an additional parameter to define the thermal conductivity in the welded zone of the contact. In contrast to the mechanical counterpart, the change of the thermal properties is initiated if one side has reached the critical temperature and, thus, has started to melt.

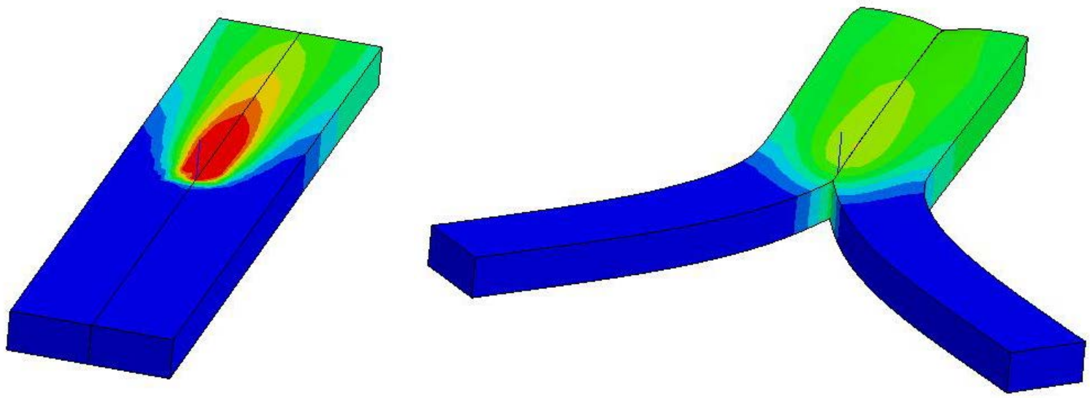

Fig.3: Welding and subsequent loading of a butt weld of two bars. Temperature field at the end of the welding step is shown (left). Loading only results in a separation of the contact areas that have not been affected by the heat.

An example application for this novel feature is the butt weld shown in Fig. 3. As depicted on the left, the weld torch only moves through one half of the contact area. After welding, forces are applied onto two end points of the structure. The deformation shows that the parts can only separate in areas for which the contact has not been affected by the heat source. The tied contact in the welded regions does not allow for separation of the parts.
The new welding functionality has been extended to shell-to-shell contact formulations to enable for example the simulation of laser welding of two or more stacked blanks. A small example of such an application is shown in Fig. 4. Here, a heat source partially welds together two thin blanks. After a short cooling phase forces applied to the non-welded edges pull apart the blanks, showing that the local heating prevents the blanks from separating.

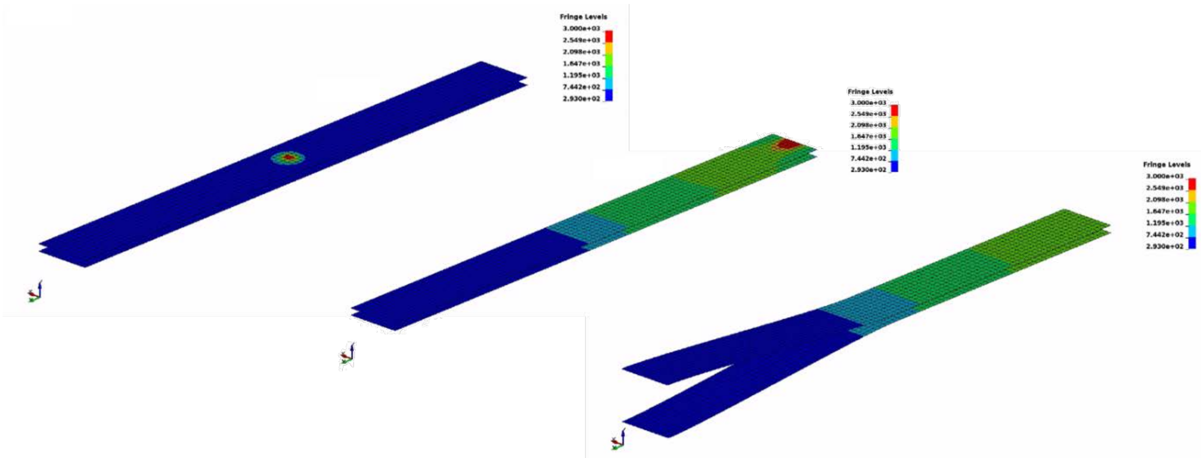

Fig.4: Welding and subsequent loading of two sheet metals. From left to right: Temperature fields at beginning of the welding step, at the end of the welding step and after mechanical loading. Separation prohibited in heat affected contact zones.

Thermal Composite Thick Shell Elements

Lithium-ion batteries have gained importance in various applications. Particularly their usage in electric vehicles requires a reliable prediction of the multiphysics response of the battery packs to abusive conditions. As it is state of the art in automotive industries computer aided engineering tools are expected to support this analysis during design phase and, thus, to reduce the amount of necessary physical testing. A reliable numerical analysis necessarily has to at least consider the structure mechanics, electromagnetics and the thermal behavior of the battery system.

The mechanical response can well be modelled with rather well-established modelling techniques in LS-DYNA. A numerical representation of the complex electromagnetics (EM) and electrochemistry in the battery has been added to LS-DYNA more recently using the so-called “distributed randles circuit” model [2]. It is coupled to the structure solver to account for the deformation of the cell. Furthermore, joule heating is transferred to the thermal solver as an extra heat source and the resulting temperature distribution affects the EM materials and parameters.

In general, the simulation of battery cells faces somewhat contradicting requirements on the model set up of EM and thermal solvers on one hand and the structure mechanical solver on the other hand: A very fine resolution of the different layers of the battery cell is required by the EM and thermal solvers, but such a fine resolution of layers with very different mechanical properties is challenging for the structure solver and drastically reduces the maximum time step of explicit time integration schemes. Moreover, the construction of very finely resolved models is rather cumbersome for the user.

Therefore, an extension for the distributed randles circuit model to composite Tshell elements has been implemented in LS-DYNA, cf. [3]. From a structure mechanical point of view, multiple possibly different layers can be defined within the thickness direction of one hexahedral or pentahedral element. The number of layers corresponds with number of through the thickness integration points but has no effect on the number of degrees of freedom in the system or on the maximum time step for explicit calculation. The EM solver, however, uses the composite definition of the element to internally reconstruct the highly resolved model. Consequently, additional degrees of freedom are generated and included into the system of equations.

A similar handling for composite shell elements has been part of the thermal solver in LS-DYNA for some time. A thermal composite Tshell element however has been implemented only recently. As it is done in the EM solver, the thermal solver internally and automatically reconstructs a resolved mesh based on the user definition of the lay-up within a Tshell element, cf. Fig 5.

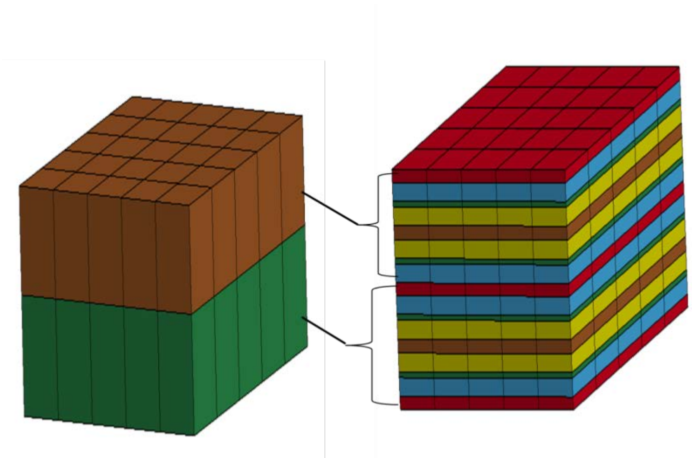

Fig.5: Tshell mesh provided by the user on the left and reconstructed EM/thermal mesh based on the composite lay-up in the Tshell elements on the right.

Naturally, the reconstruction introduces additional virtual elements and virtual nodes and, thus, requires the handling of additional degrees of freedom in the implicit system. The thermal solver keeps track of lists that link the virtual nodes of the reconstructed mesh and the "physical" nodes of the user mesh. This allows providing temperature information for the individual integration points of the thick shell elements as seen by the structure solver and also sharing information between the virtual nodes of the EM solver and of the thermal solver, which not necessarily feature the same numbering.
To test the implementation, various coupled simulations have been set up. Of particular interest is a comparison between a battery cell modelled with a finely resolved solid mesh and the same cell lay-up represented by composite thick shell elements. As expected, the results of the EM and thermal simulations are in perfect agreement.
The presented development allows for a much easier set-up of the models. But it is important to note that the postprocessing of data for the reconstructed meshes is not yet available. So far, only through the thickness averaged information is provided by the EM solver and the temperature data is only available for the nodes of the original mesh. If needed, an output for the reconstructed entities can be implemented in the future.

Recent and Future Extensions for Temperature Boundary Conditions

In the thermal solver of LS-DYNA the keyword *BOUNDARY_TEMPERATURE_NODE/SET defines Dirichlet conditions for nodal temperature degrees of freedom. The prescribed temperature values can vary with time and even be given as a function of spatial position and velocity using *DEFINE_FUNCTION.
Recently, death and birth times have been added to the keyword making it possible to activate or deactivate the boundary condition during the simulation. Before activation and after deactivation, the nodal temperature is a standard degree of freedom for the thermal solver.
One possible application is the process simulation of resistive spot welds (RSW) for automotive industries with possibly hundreds of different spot welds. Here it is almost impossible to include all connection points within a detailed coupled simulation. Instead, for each of the spot welds the temperature distribution in the nugget during the process is more or less known from the calibration phase. Now this information can be used to prescribe the temperature during the welding of an individual spot weld during the process and deactivate the condition once the welding tools are removed from the welded parts.
To simplify the input process, particularly for large assemblies with many spot weld, a new keyword *BOUNDARY_TEMPERATURE_RSW is under development. With this keyword the region of the weld can be defined and as well as the temperature values for the center and the boundary of the region. In between a

quadratic distribution is assumed. Furthermore, a load curve can be provided to scale the temperature during the weld process, i.e. between birth and death time of the individual boundary condition.

References

[1] Klöppel, T, "The Structural Conjugate Heat Transfer Solver - Recent Developments", Proc. of 11th European LS-DYNA Conference, Salzburg, 2017.

[2] L'Eplattenier, P. et al., "A distributed Randles circuit model for battery abuse simulations using LS-DYNA", Proc. of 14th International LS-DYNA Users Conference, Dearborn, 2016

[3] L'Eplattenier, P. et al, "Battery abuse simulations using LS-DYNA", Proc. of 11th European LS-DYNA Conference, Salzburg, 2017.