

New Implementation of a Weakly Thermal-Mechanical Coupling Scheme in LS-DYNA[®]

Thomas Kloeppel

DYNAmore GmbH, Stuttgart, Germany

Abstract

*In this paper a novel implementation of a weakly one-way coupled approach for thermal-mechanical coupled problems is introduced. The core of the implementation is the new keyword *LOAD_THERMAL_BINOUT. The main advantage is a very flexible input structure, which allows defining the results of several thermal simulations, for example the temperature evolution in different weld stages, as time-dependent boundary conditions for a structural simulation. The temporal order, in which the temperature boundary conditions are processed, can easily be modified. Hence, the effect of, for example, a modified weld order can be considered without recalculation of the thermal results.*

Several examples will demonstrate the application and advantages of the new approach in the context of the manufacturing process chain. Limitations of the approach, further possible simplifications for coupled simulations and future implementations are also discussed.

1 Introduction

Over the last years, efforts have been made to improve the forecast quality of numerical simulations for (cold) forming processes. These simulations have reached a high level of accuracy at least for standard materials, such that it has become state-of-the-art to take their results into account for crashworthiness analyses in automotive industries. The effect of the assembly processes that in most cases follow the forming stages is often neglected, although it is well-known, that those have a significant influence on the properties and the final distortions of the processed part.

Particularly thermal assembly stages tend to alter the mechanical properties of the parts and to result in relatively large deflections in the structure. In this context, the different welding processes are of great importance due to their frequent usage during a typical manufacturing process. Consequently, welding simulations have to be added to the virtual process chain to allow for a realistic and accurate prediction of the product properties and part deformation developing during the complete production cycle and, thus, to evaluate the quality of the manufacturing process.

The multi-physics nature of these problems makes the numerical handling relatively complex. Their solution necessitates a powerful solver with strong multi-physics capabilities, such as the Finite Element (FE) software package LS-DYNA. In general, it integrates a large variety of different solution algorithms for different physical fields and a large variety of different coupling strategies between the fields. LS-DYNA has been successfully used to simulate thermal-mechanical coupled problems.

In this paper, focus is put on the different coupling strategies available in LS-DYNA, which must be considered in order to find the best compromise between efficiency and accuracy requirements for the problem of interest. In the next section, the standard two-way coupled approach in LS-DYNA is briefly presented, in which effects of the temperature field on the structural mechanics and vice-versa are considered. Exemplarily its application to hot forming and laser-cutting processes is discussed.

In contrast to those processes, it has shown for the realistic simulation of most welding processes one-way coupled approaches are sufficiently accurate. Those approaches rely on the fact that the effect of the structure deformation on the evolution of the temperature field is negligible. Technically, this can be realized by using the results of a thermal-only simulation as thermal loading conditions within a subsequent structure-only simulation. It is only possible, if the heat input including motion of the heat source can be accounted for in the corresponding boundary conditions of the thermal solvers. Those conditions are briefly presented in the third section. But the focus of this section is put on the new LS-DYNA keyword *LOAD_THERMAL_BINOUT. It defines the data transfer between thermal results and the loading condition of the structure solver, provides the possibility to account for multiple thermal runs, and only requires partial overlap of the meshes.

2 The Standard Two-Way Coupled Approach

2.1 Algorithmic Details

As mentioned above, one of the key features of the FE package LS-DYNA is the seamless integration of different solvers for different physical fields. For example, a conjugate heat transfer solver can be coupled with a structural mechanical solver. Per default this coupling follows a two-way approach, which means that information is transferred in both directions. In a coupled analysis the thermal solver takes additional input from the structure into account such as heat from plastic work, current contact status, and interface friction in order to evaluate the temperature field. This temperature distribution serves as input for the mechanical problem to evaluate for example thermal expansion, temperature dependent material properties or phase kinetics.

Reflecting the nature of the heat transfer equation, an implicit time integration scheme is implemented in the thermal solver. For the mechanical counterpart, the user can choose between explicit and implicit integration depending on the problem. To provide this flexibility in temporal discretizations and algorithms, a loosely staggered coupling approach is realized in LS-DYNA, which is sketched in Figure 1.

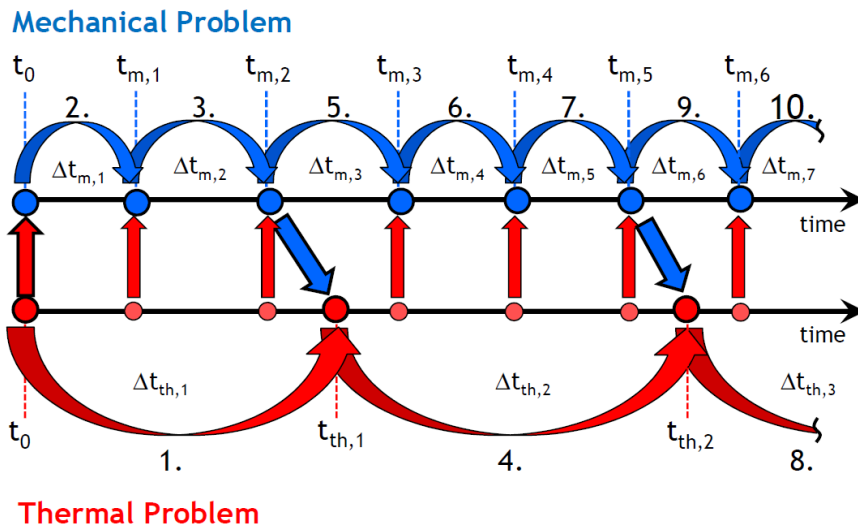


Figure 1: Schematic representation of the loosely staggered approach for thermal-mechanical coupled simulation in LS-DYNA

2.2 Example Applications

There is a large variety of problems for which this two-way coupled approach is best feasible. For example, hot forming problems require the symmetric information transfer. During the forming process the contact status constantly changes and, thus, the heat transfer between tools and blank must be frequently reevaluated based on the updated geometry. The resulting nonhomogeneous temperature distribution for a small generic hot stamping example can be found in Figure 2. Conversely, to develop the desired properties of the hot formed materials usually certain cooling rates must be ensured.

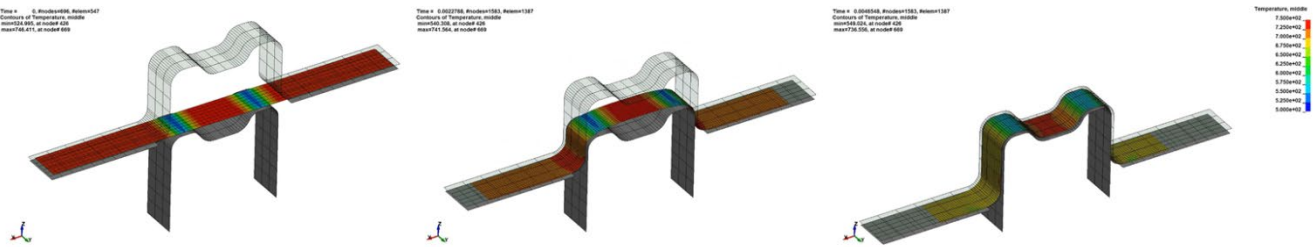


Figure 2: Non-homogeneous temperature evolution in the blank during a generic hot stamping process. The process starts after a gravity step.

As a second example that naturally demands a two-way coupled approach a laser cutting application is considered. Basis of the simulation is a relatively new keyword `*BOUNDARY_FLUX_TRAJECTORY` that has been presented in detail in [1] and therefore only the basic features are briefly discussed here. This new feature defines a moving flux boundary condition applied on a segment set. To define the characteristic surface heat density of the heat source the user can either choose from a list of predefined densities or provide an analytic expression for it with the `*DEFINE_FUNCTION` functionality. If activated, this base heat density is automatically modified to account for changes in the laser intensity due to tilting.

The heat source follows a nodal path, defined by a set of nodes, with a prescribed velocity profile, which implies that the motion is completely controlled by this thermal boundary condition. Consequently, the temperature distribution resulting from a circular motion of a laser on a rectangular plate as depicted in Figure 3 can be evaluated in a thermal-only simulation.

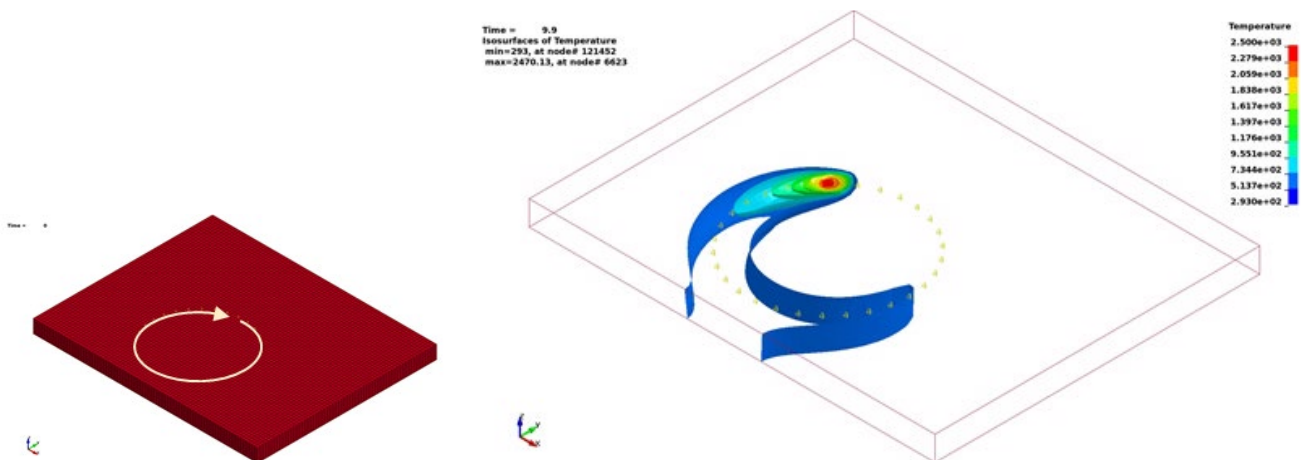


Figure 3: Circular laser motion on a plate: visualization of the path on the left, resulting temperature iso-surfaces on the right together with the node set from the trajectory definition.

It is one of the core features of the boundary condition, that after element erosion e.g. due to *MAT_ADD_EROSION it is propagated to the newly exposed surfaces and becomes applicable to laser cutting problems. Naturally, such a simulation requires the presence of the mechanical solver, as it provides information about topology changes whenever elements are eroded. As a proof of concept, the previous example is extended by element erosion based on a maximum temperature criterion and a mechanical loading applied on the plate in the center of the circle. This set-up is sketched in the left of Figure 4 and the results are shown on the right. The laser power in the example is sufficiently large to penetrate eight layers of solid elements and to cut a circular disc from the block.

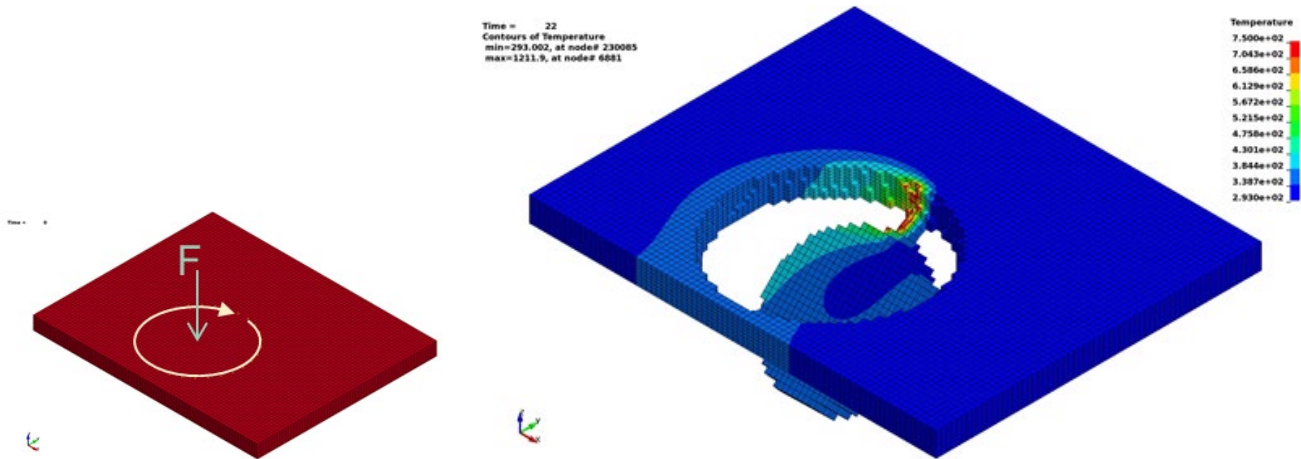


Figure 4: Circular laser cut through eight element layers. Visualization of the path and mechanical loading conditions on the left, resulting temperature distribution on the modified topology at the end of the process on the right.

3 A New One-Way Coupled Simulation Approach

3.1 Motivation for a new Implementation

Whereas the applications presented in the previous section are definitely not the only ones that require a two-way coupled approach, some of the most commonly used temperature assisted assembly steps in the automotive industries, namely line welding and resistance spot welds, can be represented by one-way coupled simulations. During the process, the topology of the parts does not change and in most cases the effect of deformations on the temperature evolution can be neglected. In contrast, the effects of the temperature field on mechanical material properties and particularly on the part deformation are significant.

Therefore, it has become state-of-the-art for this kind of process to first calculate the temperature distribution throughout the process and use the results as thermal loading condition for the mechanical solver. It must be noted that this approach is only feasible for line welding applications if the thermal solver can accurately predict the temperature distribution also for moving heat source. As the thermal solver exclusively considers temperature degrees of freedom, this has to be incorporated in tailored boundary conditions. Two of those will briefly be presented here.

First, *BOUNDARY_THERMAL_WELD_TRAJECTORY has been developed for line welding processes. It is closely related to the already discussed keyword *BOUNDARY_FLUX_TRAJECTORY but defines a volumetric heat source moving along a nodal path. Again, the motion is completely controlled by the node set and the given velocity profile in the boundary condition. A variety of different input possibilities are available for the

beam aiming direction as well as for the shape of the volumetric heat density. For a comprehensive introduction of the keyword, its input structure, the main features, and example applications the reader is referred to [2, 3]. Some of the capabilities of the keyword are shown in Figure 5 with the rather simple example of a Goldak equivalent heat source [4] moving along a curved T-Joint. The two plates that are joined are discretized with shell elements. The beam aiming direction is assumed to be always orthogonal to the surface of the filler material in the weld seam, which is meshed with solid elements.

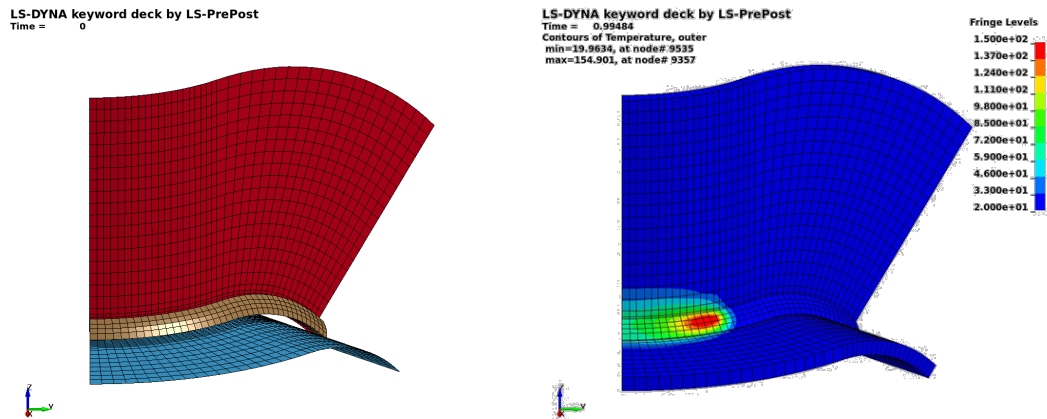


Figure 5: Welding of a curved T-joint discretized with shell elements. Geometry (left) and temperature field (right). Only to visualize the temperature gradients in shell normal direction, the shells are shown as volume elements in post-processing.

The keyword `*BOUNDARY_THERMAL_WELD_TRAJECTORY` has not only been used for line welding but also for a simplified process simulation of resistance spot welds. Instead of trying to predict the weld nugget (zone of molten material during the process) by a complex coupled simulation using the electro-magnetic, thermal and structure solver of LS-DYNA as for example proposed in [5, 6], an equivalent heat source can be used. Even though it is a simplified approach, it has to be calibrated to reproduce the weld nugget in a non-trivial and elaborate procedure. A successful calibration procedure requires knowledge of the resulting weld nugget in the process.

This information is usually available from preliminary hardware tests. With the novel keyword `*BOUNDARY_TEMPERATURE_RSW` the knowledge of the process and, particularly, of the nugget's shape and size can directly be used. In contrast to the other approaches, the boundary condition does not define the heat entry to the structure but directly describes the temperature field in the weld nugget and optionally also of a surrounding heat affected zone (HAZ). Consequently, only an easily manageable calibration effort, if necessary at all, is required.

The shapes of both the nugget and HAZ are assumed to be combinations of two half-ellipsoids with a common axis of rotation, which is defined by two nodes of the model, and a common center. The temperature distribution in the affected area is calculated based on user input for temperatures in the center, at the boundary of the nugget and, optionally, for the boundary of the HAZ. Within the nugget a quadratic approximation is used, whereas temperature is linearly interpolated in the HAZ. A small example with only one RSW joining two solid parts is shown in Figure 6. Details on this keyword and on the input structure are given in [1].

These previously discussed features of LS-DYNA enable the user to accurately predict the temperature field evolving during line and spot-welding processes. In the next step, the numerical results serve a thermal loading condition for the mechanical problem. Up until recently, the only way to realize this transfer was to use

LS-DYNA's binary d3plot data base for the results. The corresponding keyword for the structure simulation is *LOAD_THERMAL_D3PLOT. For practical applications, this approach shows certain shortcomings.

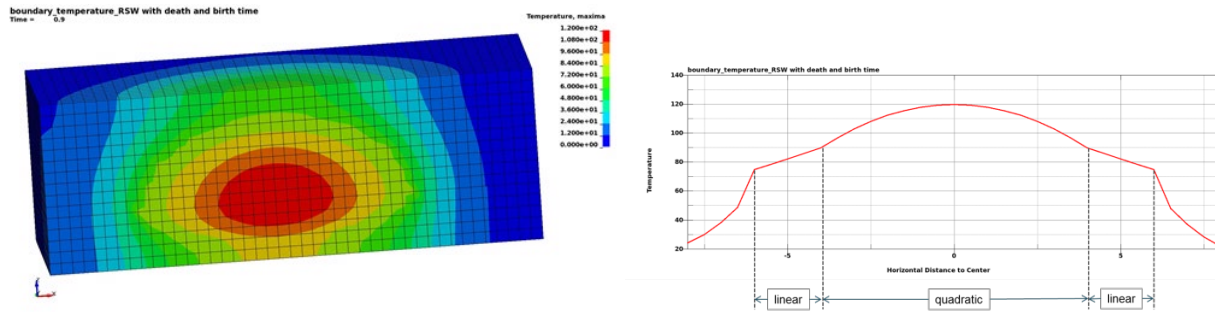


Figure 6: Visualization of *BOUNDARY_TEMPERATURE_RSW. Resulting temperature distribution in a solid block assembled from two parts on the left. Graph on the right shows course of temperature along a horizontal line through the center.

For one, the models for the thermal and the structure problem have to coincide. It is not possible to only consider certain parts of interest either in the thermal or the mechanical run, which could increase the efficiency of the computation. This is especially true for welding applications since heat source usually has only a local effect on the temperature field. Theoretically this problem could be reduced by the employing an external mapping software, which maps the temperature results around the welds onto the complete mesh to provide a new d3plot database for the structure simulation. But this demonstrates the second shortcoming of the approach: the complexity of the d3plot format. It is almost impossible to manipulate existing d3plot databases or create new d3plot databases by external software.

3.2 Discussion of *LOAD_THERMAL_BINOUT

Based on user feedback for *LOAD_THERMAL_D3PLOT and in order to meet current user requests, it has been decided to use the open and very flexible lsda database for a new data transfer approach. This database is also binary, but in contrast to the d3plot format, there are open tools available to read, manipulate and create such databases. In LS-DYNA the format is widely used, particularly for the binout database, which among others contains a TPRINT section. In this section, results from the thermal solver such as for example nodal temperatures for certain discrete points in time as well as the nodal ids corresponding to those results are provided.

Table 1: Keyword input for *LOAD_THERMAL_BINOUT

Card 1	TDEF							
Card 2	FILENAME							
Card 3	STARTT	TSF						

The nodal values of the TPRINT section in the binout database are enough for the thermal load of a structure. Therefore, these are considered by the new LS-DYNA loading condition *LOAD_THERMAL_BINOUT, the keyword structure of which is given in Table 1. The name of the file containing the lsda database is to be given in the second card. The mapping of the temperature data from the lsda file to the current structure simulation is done based on the user-defined nodal ids. A geometrical mapping is not performed.

This simple mapping approach on the one hand requires to use the same meshing for those parts of the model that are subjected to a thermal loading. On the other hand, it means that there is no need for a total agreement of the two models, as it has been with the approach followed by `*LOAD_THERMAL_D3PLOT`. If there are nodes in the thermal model that are not considered in the structure simulations, then their temperature data is just not used for the thermal loading. The inverse case requires more considerations. For nodes in the structure model without counterpart in the thermal simulation a default temperature must be applied. It is defined by parameter `TDEF` in the first card of the input, cf. Table 1.

As already mentioned above, the binout database contains temperature data for discrete points in time. The time stepping of the structure solver is completely independent from the thermal run and a linear interpolation of the temperature data is applied. There is often the need for an artificially accelerated mechanical analysis, particularly for explicit time stepping schemes. In contrast, thermal problems can usually be solved in the physical time scale. Therefore, an acceleration factor α_t can be defined by parameter `TSF` in the third card of the input.

Furthermore, there are applications in which the thermal load does not start at the beginning of the mechanical analysis. For example, parts must be clamped together before they can be welded. In order to allow considering only the thermally active phase of the problem in the thermal simulations and, thus, reducing the amount of data to be applied, a time offset t_{start} has been included into the keyword by means of parameter `STARTT`. Consequently, the relation between a time t_{mech} in the structure simulation and time t_{th} in the thermal problem is given by $t_{\text{mech}} = t_{\text{start}} + t_{\text{th}}/\alpha_t$.

Since we could realize a relatively flexible data transfer in `*LOAD_THERMAL_BINOUT` with respect to the requirements on the congruence between the meshes of thermal and structure simulation, it is also possible to consider multiple thermal result files, each corresponding to a certain region of the model. The second and third input cards can be repeated as often as necessary, i.e. each thermal run has an individual starting time and time scale factor. Naturally, the different starting times are of particular interest when it comes to the simulation of complex welding processes, since they allow to easily analyze the effect of the weld ordering in structure-only simulations. A corresponding example will be discussed in the section 3.3.

While a binout data base written by a thermal simulation can be used in the keyword without any further modification, it usually contains more data than is actually required by the thermal loading condition. The mapping tool `envyo`®, presented in [7, 8], contains the necessary routine to read and write `lsda` data bases. Hence it was relatively easily possible to create a reduced `lsda` database to be used in the new keyword from a given binout file. The minimum design of that database contains only data for nodal temperatures and nodal ids for the different time steps. While significantly reduced in size there was of course no change in the resulting thermal load on the structure. A further reduction in file sizes could also be obtained by reducing the temporal resolution in the database for phases with moderate temperature rates in the model.

With the help of `envyo` or other mapping tools it has also been possible to use thermal results from third-party software tools, such as for example tailored tools for the temperature calculation in the paint shop for automotive industries, for thermal loading in a mechanical analysis with LS-DYNA. Another interesting aspect for the application of `envyo` is that geometrical mapping becomes possible, such that individually optimized meshes could be used for thermal and structural analysis.

3.3 Example Application for `*LOAD_THERMAL_BINOUT`

The features of the keyword are demonstrated with a line welding example that is shown in Figure 7. Four curved stiffeners are welded onto one slit curved base plate and all the welded parts are modeled with shell elements and clamped into position during the whole process. The simulation of the clamping follows a very simplistic approach. For the thermal analysis it is assumed that the heat transfer to the environment is higher than in areas

surrounded by air. In the mechanical analysis the clamped areas are subjected to a single point constraint, i.e. any motion of the respective nodes is suppressed.

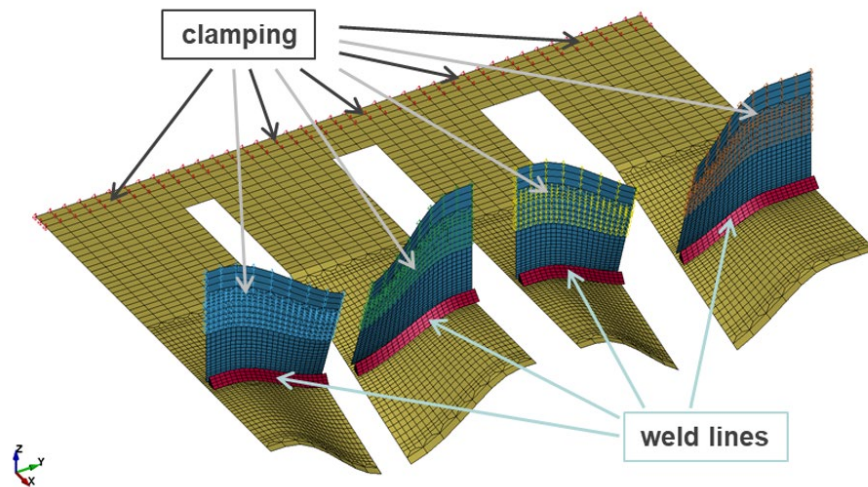


Figure 7: Welding of four curved stiffeners onto slit curved base plate.
Sketch of thermal and mechanical boundary conditions.

The slits in the base plate ensure a thermal decoupling of the four weld lines and it seems, thus, reasonable to divide the model in four disjunct models and run separate thermal analyses for those. Figure 8 shows the four models and the resulting temperature distributions at one point during the welding process. It is important to note that the clamped regions are contained in all the thermal models. They are necessary to control the heat transfer to the surrounding and, consequently, significantly influence the temperature distribution.

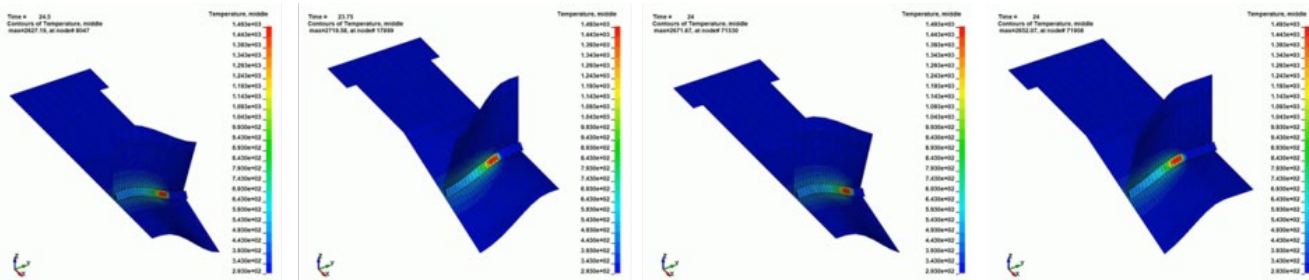


Figure 8: Temperature distribution during the welding process in the four separate thermal models.

To start with, only one thermal run, namely the welding of the leftmost weld line in Figure 8, is considered and transferred to the structural analysis. As there cannot be any motion within the clamped regions based on the assumptions made, it is possible to omit most of the nodes of those regions in the structural simulation and subject only one row of nodes to single point constraints instead. This approach is not motivated by efficiency considerations but made to demonstrate that a partial overlap of thermal and structural model can be dealt with by the new keyword.

The temperature distribution as output by the structure simulation run is shown in Figure 9 on the left. Comparison with the result in Figure 8 demonstrates that the temperature data has correctly been transferred to the thermal loading condition.

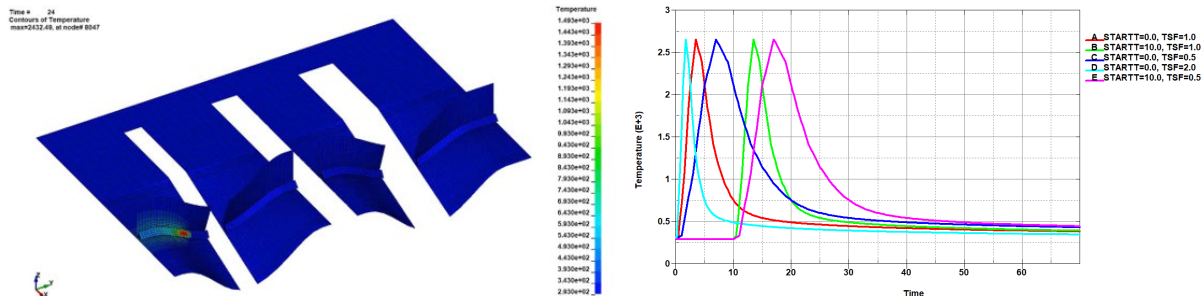


Figure 9: Resulting temperature distribution as output of the mechanical analysis on the left. Temperature profiles for a node in the first weld line for certain combinations of start time and time scale factor.

In the right the temperature evolution in a node that is located on the surface of the weld seam close to the starting edge is depicted for certain combinations of start time and time scale factor. The base curve is the one obtained with STARTT=0.0 and TSF=1.0, i.e. congruent time scales for thermal and mechanical simulation. As can be expected a non-zero starting time results in a shift of the temperature curve to the right. Increasing the scaling factor above narrows the curve, whereas defining values below one widens the temperature profile.

The temperature distributions shown in Figure 10 have been obtained from different ordering of the four weld lines. In both cases, the same temperature results from Figure 8 have been used. It demonstrates that the loading condition can indeed account for multiple thermal results and allows for an easy reordering of the weld procedures. On the left, all weld lines have been started simultaneously. On the right, a sequential welding has been simulated. From left to right, the starting times have been 0, 100, 50, and 125 and figure shows the temperature distribution of time $t=130$.

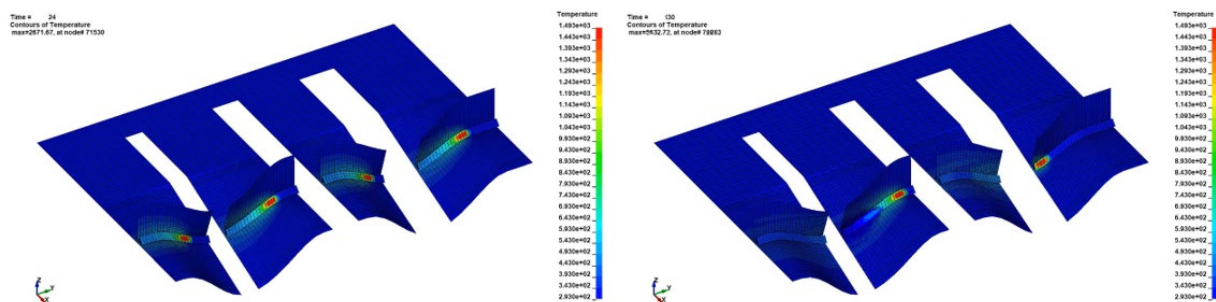


Figure 10: Resulting temperature distribution for simultaneous (left) and a sequential (right) welding.

4 Summary

In this paper, different strategies for the simulation of thermal-mechanical coupled assembly processes have been discussed. First, the standard two-way coupled approach in LS-DYNA has been briefly summarized. Processes such like hot stamping and laser-cutting require a symmetric data transfer between the physical fields as has been shown with simple examples.

For welding applications, the effect of the structure deformation on the heat transfer analysis can often be neglected and the result of a thermal simulation can be used as thermal loading for a subsequent structure-only analysis. The new keyword `*LOAD_THERMAL_BINOUT` has been devised for this purpose. Data transfer is established based on the very flexible and yet efficient lsda data format and only requires partial overlap between the thermal and the structural model.

In the new implementation multiple result files from thermal simulations can be considered. For each of those, a start time and time scale factor can be defined. As a simple welding example demonstrated this allows for a very simple and efficient evaluation of different weld orders.

References

- [1] T. Klöppel, „Recent LS-DYNA Developments in the Structural Conjugate Heat Transfer Solver,“ in *proc. of 12th European LS-DYNA Conference*, Koblenz, 2019.
- [2] T. Klöppel, M. Schill und T. Loose, „Recent Updates for the Heat Transfer Solver in LS-DYNA with focus on computational welding mechanics,“ in *Proc. of 14th International LS-DYNA Conference*, Detroit, 2016.
- [3] T. Klöppel, „The Structural Conjugate Heat Transfer Solver – Recent Developments,“ in *proc. of 11th European LS-DYNA Conference*, Salzburg, 2017.
- [4] J. Goldak, A. Chakravarti und M. Bibby, „A Double Ellipsoid Finite Element Model for Welding Heat Sources,“ *IIW Doc.No.212-603-85*, 1985.
- [5] P. L'Eplattenier, I. Caldichoury, S. Bateau-Meyer, T. Loose und U. Reisgen, „Resistive Spot Welding Simulations Using LS-DYNA,“ in *proc in 11th European LS-DYNA Conference*, Salzburg, 2017.
- [6] I. Caldichoury, P. L'Eplattenier, S. Bateau-Meyer, T. Loose und U. Reisgen, „Update on Resistive Spot Welding Capabilities in LS-DYNA,“ in *proc of 15th International LS-DYNA Users Conference*, Detroit, 2018.
- [7] C. Liebold und A. Haufe, „process2product simulation: Closing Incompatibilities in Constitutive Modeling and Spatial Discretization with envyo®,“ in *proc. of 15th International LS-DYNA Users Conference*, Detroit, 2018.
- [8] Dynamore GmbH, „envyo® User's Manual,“ 2020.