Recent Developments for Welding Simulations in LS-DYNA® and LS-PrePost®

Mikael Schill[1], Anders Jernberg[1], Thomas Klöppel[2]

[1]DYNAmore Nordic AB, Linköping, Sweden
[2]DYNAmore GmbH, Stuttgart, Germany

1 Abstract

The multi-physics capabilities of LS-DYNA® makes it ideal for simulating the welding process where the mechanical and thermal physical regimes are combined to simulate the resulting part tolerance and residual stresses. Welding capabilities are continuously being added to LS-DYNA®, e.g. the material models ***MAT_CWM** and ***MAT_CWM_THERMAL** which are tailored for CWM simulations. The CWM material models include e.g. ghost element and anneal functionality. However, the challenge when performing this type of simulations is not only related to the solver. The pre-processing quickly becomes a daunting task where the user needs to assign weld paths, weld power and clamping for several weld passes. To accommodate for this, a novel welding GUI has been added to LS-PrePost®. The GUI includes e.g. a welding process planner, welding path selection using the graphical interface and welding heat source definition and visualization. Further, the welding process setup is written to an ASCII input file that can be combined with an optimization software to optimize the process with respect to e.g. welding deformations. This paper will present the recent developments in the mechanical and thermal solvers in LS-DYNA® to accommodate for welding simulations. Also, the novel Welding GUI in LS-PrePost® will be presented together with some application examples.

2 Introduction

LS-DYNA® is a multi-physics solver where several physical regimes can be combined to solve a vast number of real world problems. One of these is welding simulation where the material is heated above the melting temperature and fused together during cooling. This can be done with or without adding filler material. This type of problem is solved using the thermal solver in combination with the mechanical solver that could be either explicit or implicit. The thermal solver is however always implicit. There are basically two ways of solving this today. The uncoupled approach is the simplest where the thermal problem is solved first and then its output is used as input in a subsequent mechanical analysis. The second approach is to couple both solvers by running the thermal and mechanical solvers in a staggered approach where the two solvers exchange results and thus allowing for the two physical fields to affect each other. Both options are available in LS-DYNA® today.

The input for a welding simulation is, apart from thermal and mechanical boundary and initial conditions, material and the heat source description. So far, the development has been focused on the latter subjects and thus accommodating for solving this type of problems in LS-DYNA®. However, the pre-processing of a welding process is quite challenging and its process chain type of simulation requires that simulation results are transferred from one simulation to another while changing e.g. boundary conditions. Also, considering that a user would easily like to change the welding order or the welding direction, it is obvious that a tailored preprocessor is of outmost importance in order to set up this type of problems. This paper will present the recent developments in LS-DYNA® regarding the modelling of the heat source and also the current status of the tailored material models for welding simulations. Further, a description of the recently implemented GUI for welding simulation in LS-PrePost® will be presented. Other papers published in this field is e.g. Schill and Odenberger [1] where a sheet metal part subjected to both deep drawing and welding was simulated and the out of tolerance displacements were compensated in the deep drawing tooling. The welding functionality in LS-DYNA® is under constant development and recent developments regarding e.g. thermally activated contacts and the material models are described in Klöppel and Loose [2].

3 Welding material models

It should be noticed that basically all thermo-mechanical material models in LS-DYNA® can be used to model the welding process, but the specific welding material models have two important

characteristics that makes them suitable for simulation of welding processes. Firstly, the materials have ghost functionality. This means that the material has negligible thermal and mechanical properties until it is activated at a specified user defined temperature. By this, it is convenient to distinguish between three different material states.

- Solid material: Material that is always solid or a weld pass that has been previously activated
- Liquid material: Material that is in a ghost state but will be activated during the current weld pass.
- Ghost material: Material that will not be activated during this weld pass. But it will be activated in a subsequent pass.

The procedure for using this modeling technique has been described by Lindström [2] and verified against benchmark experiments of a multi pass girth weld in Lindström et al. [4]. The material is state is given by the activation temperature where a solid state material is given a very low activation temperature, a liquid material has an activation temperature of typically the melting temperature, and the activation temperature for the ghost material is set to a very high value. It should be noticed that both the thermal and the mechanical material models have activation temperatures and can thus be used independently of each other.

The second characteristic of a suitable welding material model is the anneal functionality. This is implemented to simulate a limiting temperature where history variables such as effective plastic strain or back stress are zeroed out. Thus, above this user specified temperature, the material will behave as ideal plastic.

The current material models in LS-DYNA® with the welding functionality is **`*MAT_CWM/#270`** and **`*MAT_UHS/#244`**. Starting with the **`*MAT_CWM`** material model, it is a thermo-elasto-plastic material model based on the von-Mises yield surface with isotropic or kinematic hardening. Apart from the previously mentioned welding capabilities it also includes a maximum temperature dependent thermal expansion coefficient. By this, the user can model thermal dilatation due to changes in the material phase content. In practice this means that the thermal expansion is input as a table as a function of temperature and maximum temperature. The material model will follow a certain thermal expansion load curve when heating and choose another one when cooling depending on the maximum temperature, see **Fehler! Verweisquelle konnte nicht gefunden werden.**.
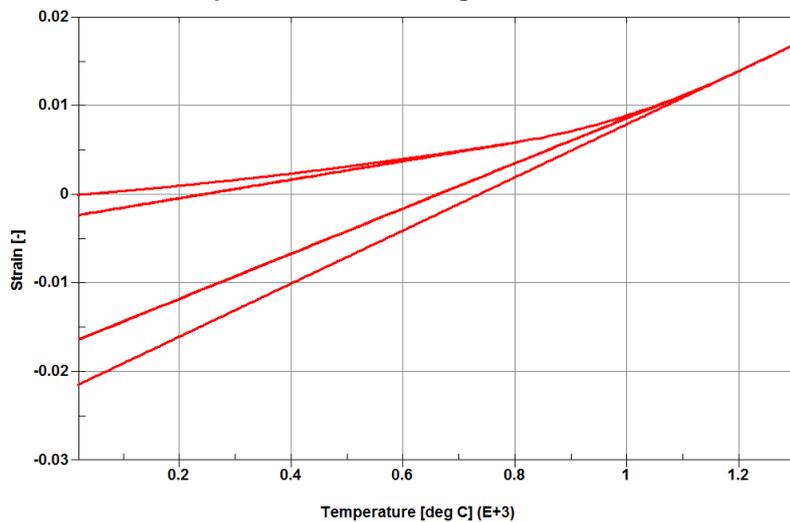

Fig.1:   Thermal dilatation in *MAT_CWM

The history variables for the **`*MAT_CWM`** material model are shown in **Fehler! Verweisquelle konnte nicht gefunden werden.**. The user has the possibility to specify a temperature range where the cooling rate is calculated and then stored as history variable 11. This output can be used to estimate the final phase content of the material assuming a constant cooling rate. Other important history variables are number 9 that indicates if the material is solid or ghost, and number 10 that stores the maximum temperature in the integration point that can be used to visualize a heat affected zone. Other recent implementations in the **`*MAT_CWM`** material model is a tabulated input of the hardening using stress-strain curves as a function of temperature and support for shell elements.

History variable number	Description
1-6	Back stress
7	Temperature at last time step
8	1 if yielding, else 0
9	Indicator of material state, 0 for ghost, 1 for solid
10	Maximum temperature
11	Temperature rate from T2PHASE to T1PHASE

*Table 1:   *MAT_CWM history variables*

The thermal counterpart ***MAT_CWM_THERMAL/#T07** has also been updated to allow for maximum temperature dependent input. In this case, the heat capacity can be input as a table depending on temperature and maximum temperature.

The phase kinetics model ***MAT_UHS/#244** was initially implemented for hot stamping simulation, see Åkerström et al. [5] and [6]. The model has evolved through time and now it includes not only cooling from a fully austenite state but also heating and the decomposition of hard phases into austenite. This makes it suitable for predicting the resulting microstructure from a welding process where the material has been heated and cooled in sequence. The input for the material model is the chemical composition and the plastic hardening of the respective phases. The resulting hardening properties are determined through a mixture rule depending on the phase content. For the solid material, the initial phase content also has to be included. To accommodate for welding simulations, the ghost material functionality is added using the activation temperatures similar to ***MAT_CWM**.

4 Heat source modeling

The Goldak double ellipsoid heat source is by many considered to be the standard choice for modeling the weld torch, see Goldak et al. [7]. A volumetric heat load is applied to a volume defined by an elliptic shape, see **Fehler! Verweisquelle konnte nicht gefunden werden.**. Further, it is possible to adjust the flux intensity towards the rear or the front of the region. The heat source origin is defined by a moving node and its direction is defined by a second node. Thus, in order to move the heat source, it is necessary to prescribe the motion for both nodes. The Goldak heat source has been available in LS-DYNA® for quite some time using the keyword ***BOUNDARY_THERMAL_WELD**.
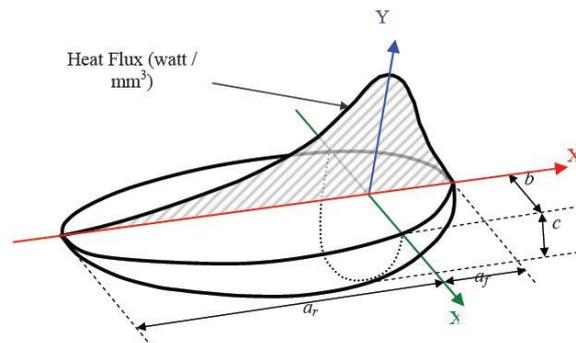

Fig.2:   Goldak double ellipsoid heat source

A novel keyword has been implemented for the heat source modeling denoted ***BOUNDARY_THERMAL_WELD_TRAJECTORY**. This has been done in order to meet some of the drawbacks of the old heat source model. Firstly, the old heat source needed a mechanical solver present in order to move the heat source. The mechanical materials could all be rigid which would limit the simulation time, but managing both solvers was found to be a bit awkward by the users. The new heat source only requires the thermal solver. Thus, to in order to switch between a thermal only and a coupled simulation, the user only need to change the parameter SOLN from 1 to 2 under ***CONTROL_SOLUTION**.

When running a coupled simulation, the material will deform due to the extreme temperature increase. This will of course affect the path of the weld torch and making it virtually impossible to predict in advance. In the ***BOUNDARY_THERMAL_WELD_TRAJECTORY** keyword, the user has new possibilities of prescribing the movement of the heat source. The weld torch origin is now defined by a node set.

The nodes defining the trajectory can thus be part of the deforming structure and will then be adaptively updated with the deformation. The orientation of the heat source can be described in three ways:

- Node set ID
- Aiming direction vector
- Normal to Segment set

The new heat source keyword will also present a number of different heat source shapes. Apart from the well-known double ellipsoid shape it is now possible to define i) a double ellipsoid heat source but with constant heat flux density ii) double conical heat source with constant density and iii) a conical heat source, see **Fehler! Verweisquelle konnte nicht gefunden werden.**.

Fig.3: Ellipsoid heat source (left), double conical heat source (middle), conical heat source (right)

Due to the incremental nature of the heat source, the user cannot arbitrarily choose the time step size in a welding simulation. In fact, there is a relation between the time step, heat source velocity and the element size. If a too large time step is used, the pulsating heating will cause severely distorted elements, see **Fehler! Verweisquelle konnte nicht gefunden werden.**.
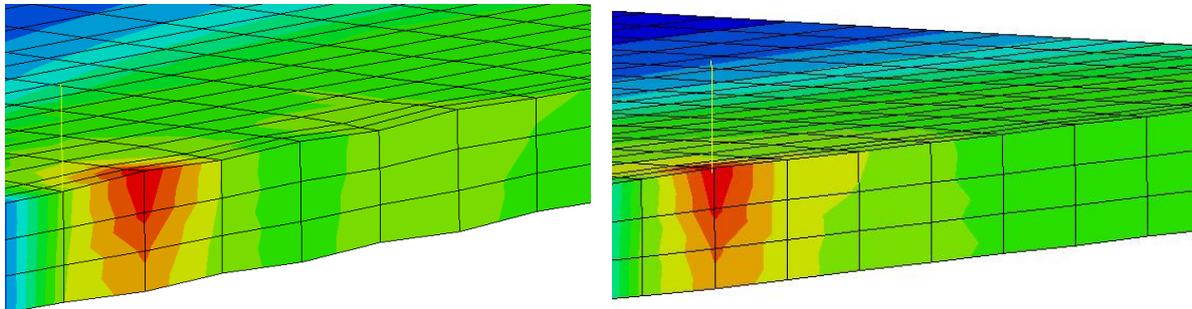

Fig.4: Simulation with a too large timestep (left) and a correctly defined timestep (right)

The new heat source implementation includes an option to sub-cycle the heat source. Thus, the user can choose how many sub-cycle time steps the solver should use for the heat source within one thermal time step. This will limit the previously described element size and time step constraint. If a large time step and sub-cycling number is chosen, the heat will be applied over several elements, see **Fehler! Verweisquelle konnte nicht gefunden werden.**. The idea of this implementation is to allow for the user to choose the thermal and mechanical time step more freely depending on the necessary level of accuracy.
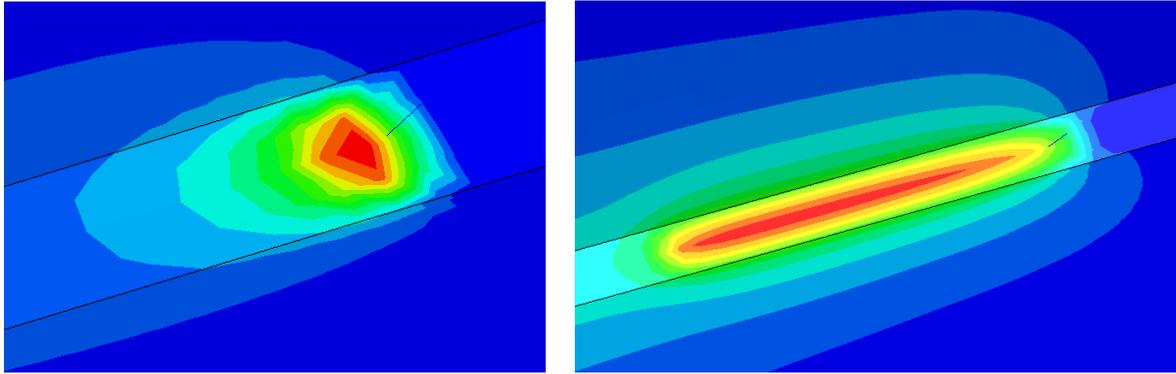
Fig.5: Low time and sub-cycling steps (left), high time and sub-cycling steps (right)

When defining a negative weld power in the input, LS-DYNA® will use an accurate integration of the applied heat. The higher numerical effort increased the simulation time and the user had no way of affecting the level of accuracy. The new heat source also has an option of accurately integrating the heat source but with the option to control the edge length of the integration cells.

5 LS-PrePost® Welding GUI

The tasks when setting up a welding simulation are:

- Path of the weld source and orientation
- Speed, type and power of the heat source
- Mechanical boundary conditions, thus clamping
- Thermal boundary conditions, thus convection and radiation
- Assign the material state, thus solid, liquid and ghost

This is done for a certain process order, and if the process is split up in several simulations the results from a previous state has to be transferred to the sub-sequent step. This quickly becomes complicated for a multistage analysis, especially if the process order or directions are varied. The LS-PrePost® Welding GUI aims at solving the following pre-processing issues:

- Choose the weld path and weld direction
- Vary the process order
- Automatically transfer the results from one process stage to the next
- Automatically set the correct material state
- Choose the thermal and mechanical boundary conditions
- The setup file should be text based for easy connection to e.g. an optimization software

The GUI is to be considered as a standalone module. Thus, it does not add anything to the model until the user outputs the keyword file. The setup data for the welding GUI is instead stored in a separate process file. The input for the GUI is thus e.g. the mesh, contacts, materials and sections, node and segments sets for boundary conditions etc. without the welding specifics. Please note that the weld pass parts should have the word "weld" in the part name, and the material should be defined in a liquid state. The parts will then be automatically identified as weld passes by the GUI. Further, the weld and orientation paths are identified by beam elements. These have to be created before entering the GUI.

When starting the GUI (Application/Tools/Welding simulation) the sequence folder is visible, see **Fehler! Verweisquelle konnte nicht gefunden werden.**. Here, the user can set the welding order and corresponding mechanical and thermal boundary conditions by clicking or un-clicking boxes. If e.g. a weld pass is undefined, the box will be marked as red and if it is defined it will be colored green. The same holds for the mechanical and thermal boundary conditions. The air cooling segments are automatically identified by the GUI.

Fig.6: Process sequence folder

When double clicking on a weld definition, the user is transferred to the welds folder, see **Fehler! Verweisquelle konnte nicht gefunden werden.**. Here it is possible to select the weld and orientation paths as well as the start of the weld path by clicking. Also, it is possible to animate the weld pass by dragging the scroll bar. The weld pool geometry, power and efficiency is also defined here and the weld pool geometry is visible in the graphics window, see **Fehler! Verweisquelle konnte nicht gefunden werden.**. A cooling stage is associated to each welding stage and the time is set in this folder. It is also possible to set the number of time steps/element. The GUI will thus identify the element length along the weld and adjust the time step of the respective solvers depending on the weld velocity.

In the structural boundary conditions folder, the constraints are set for each defined node set and in the thermal boundary condition folder, a convection and/or radiation boundary condition can be defined for each segment set. When the user is finished setting up the process, the process information can be saved in a text file that can be imported for later use. Due to the vast difference in solver setup between the welding and the cooling stage as well as allowing for the user to run separate weld passes, each weld pass and cooling stage will be a separate simulation. Thus, when the GUI outputs the keyword files, each weld pass will be found in a folder denoted X.1 where X is the weld pass order and the corresponding cooling stage will be found in folder X.2. The simulations can then be run in sequence by a simple script. The GUI comes with pre-installed control files for welding and cooling. These can be easily modified by the user.
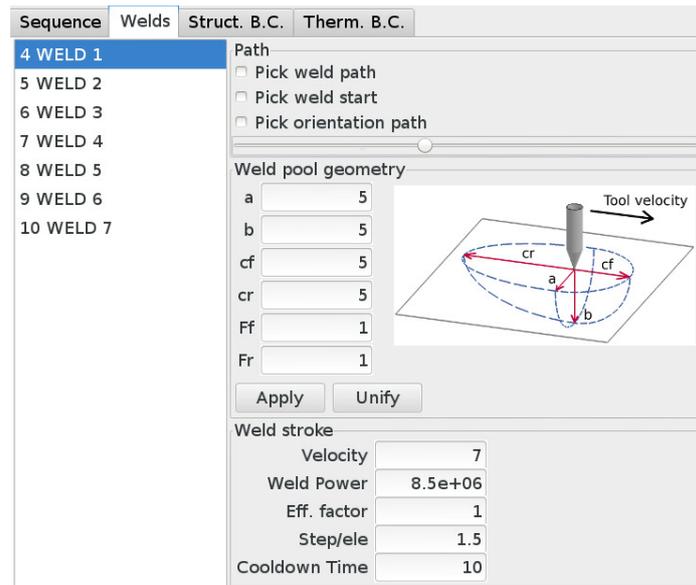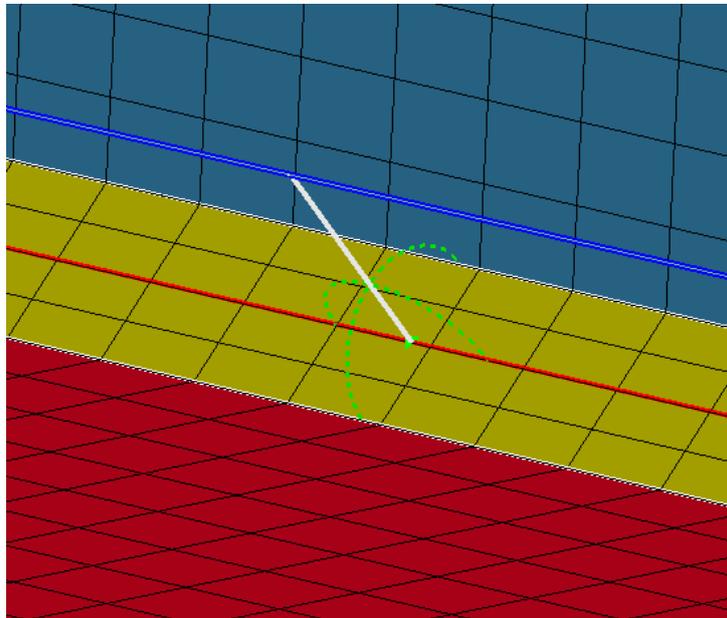
Fig.7:   Weld pool definition folder

Fig.8:   Weld pool, pass and orientation is visible in the graphics window

6 Application example

The novel heat source implementations and the GUI have been used to model a T-joint weld simulation. The main purpose of the example is to evaluate the sub-cycling of the heat source and the influence from the heat source integration parameter on the accuracy and the simulation time. The T-joint example consists of 2 plates (500 x 500 x 9 mm and 300 x 500 x 9 mm) joined together by 2 fillet welds, see **Fehler! Verweisquelle konnte nicht gefunden werden.**
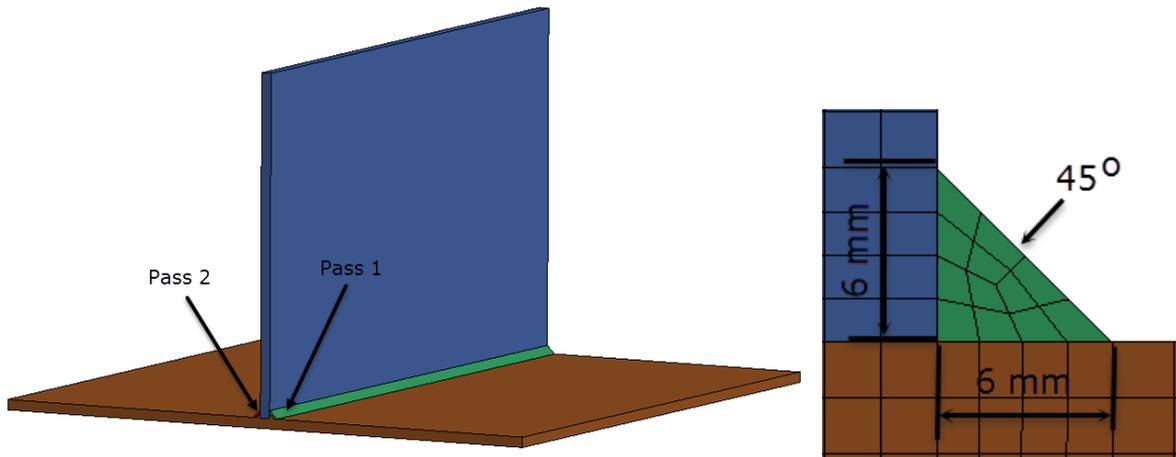
Fig.9: T-joint geometry and fillet weld pass geometry

The material of the T-joint is S355 with material data from [7]. The FE-model consists of 43000 solid elements and the welding process specifics are presented in **Fehler! Verweisquelle konnte nicht gefunden werden.**. The base plate is constrained according to **Fehler! Verweisquelle konnte nicht gefunden werden.**, the attaching plate however is free to move.

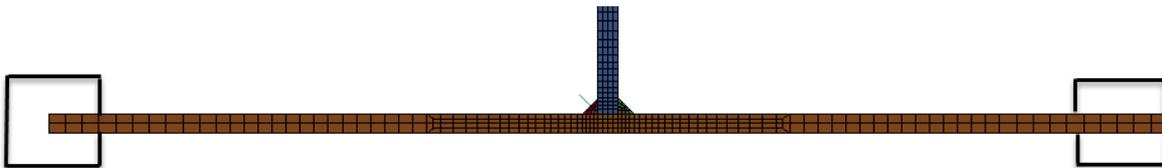| Power [W]	Width [mm]	Depth [mm]	Forward [mm]	Backward [mm]	Distribution Forward/Back	Velocity [mm/s]	Cooling time [s]
3000	4.35	4.35	4.35	4.35	1./1.	3.33	1000

Table 2: Weld pass data

Fig.10: Base plate constraints

The first part of the study aims at choosing a proper setup of the time step and sub-cycling. This was done for the first weld pass. As a basis for the comparison 1.5 time steps/element were chosen for the mechanical and thermal time steps and using 1 sub-cycle step of the heat source. Considering the weld velocity, this yields a time step of 0.834 seconds and a total of 181 time steps. The time step was then subsequently doubled as well as the heat source sub-cycling thus keeping the same number of heat source time steps, see **Fehler! Verweisquelle konnte nicht gefunden werden.**.

Time step [s]	# Sub-cycle steps	#Time steps
0.834	1	181
1.668	2	91
3.336	4	46
6.672	8	23
13.34	16	12
26.7	32	6

Table 3: Combinations of time step and sub cycle steps used in the study

The temperature, stresses and attached plate displacement was evaluated at the center of the model according to **Fehler! Verweisquelle konnte nicht gefunden werden.**. Figures 12 through 14 show a comparison of for the different settings of time step and sub-cycling. It was decided that a time step of 6.67 s and 8 sub-cycling steps yielded reasonable accuracy.
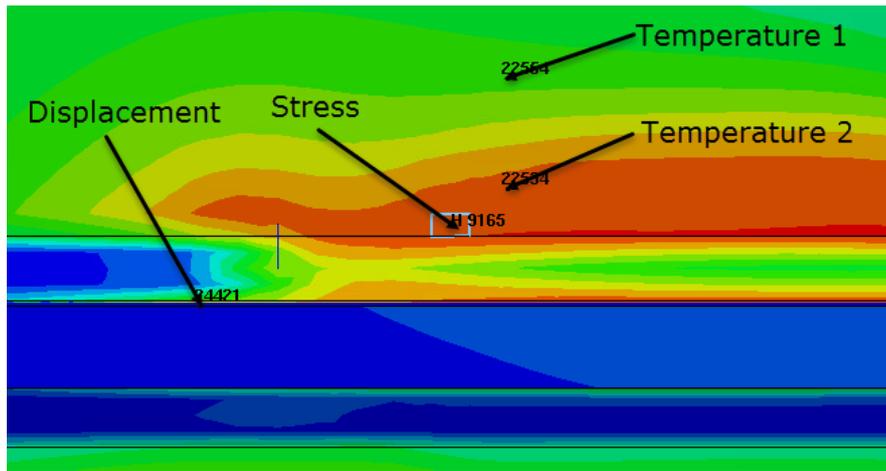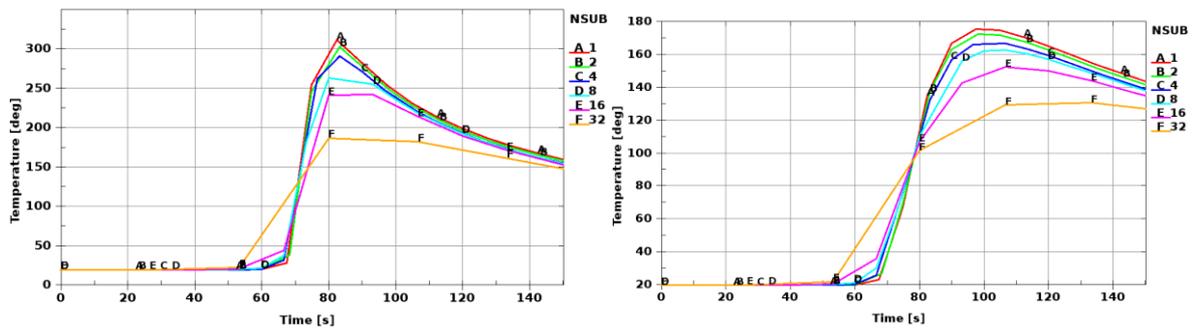
Fig.11: Evaluation points

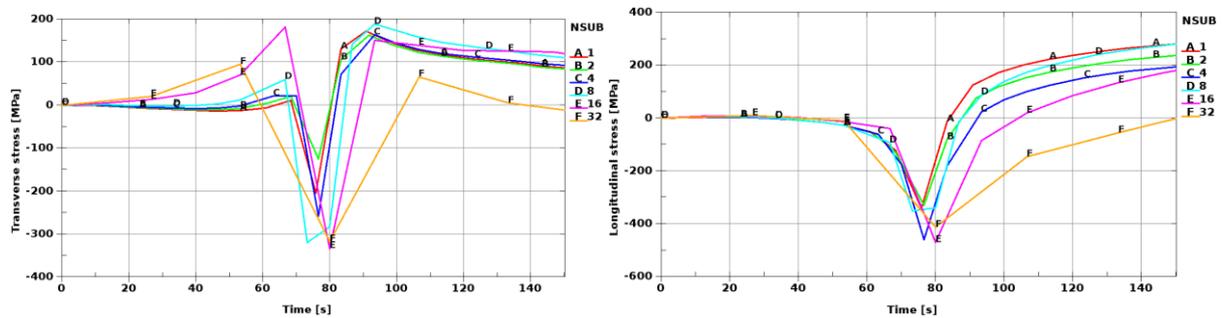*Fig.12: Temperature during weld pass 1 at evaluation point 1 (left) and 2 (right)*

Fig.13: Transverse stress (left) and longitudinal stress (right) during weld pass 1
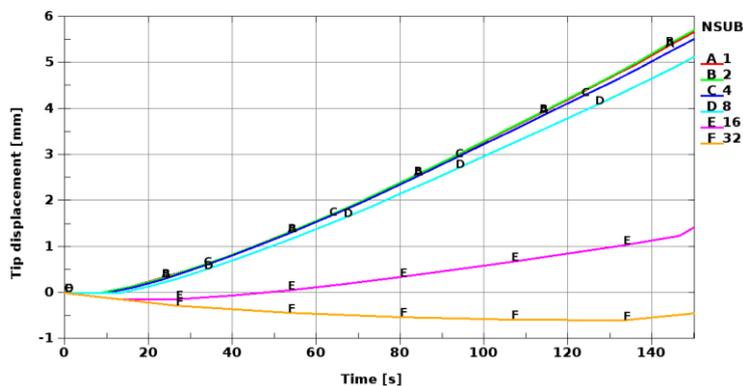

Fig.14: Attached plate tip displacement

To further reduce the simulation time, the edge length of cells of the heat source integration was doubled without significant difference in result. The complete welding process was simulated including 2 weld passes, 2 cooling stages and a finishing un-clamping with springback. The longitudinal and transversal stress across the baseplate is shown in **Fehler! Verweisquelle konnte nicht gefunden**

werden., and the out of plane deformation of the base and the time history displacement of the attach plate top edge are shown in **Fehler! Verweisquelle konnte nicht gefunden werden.**. The simulation time was reduced by 60 % with the current choice of the heat source parameters. There is a quantitative difference in the stress distribution but the final shape of the plate assembly is within a reasonable tolerance.
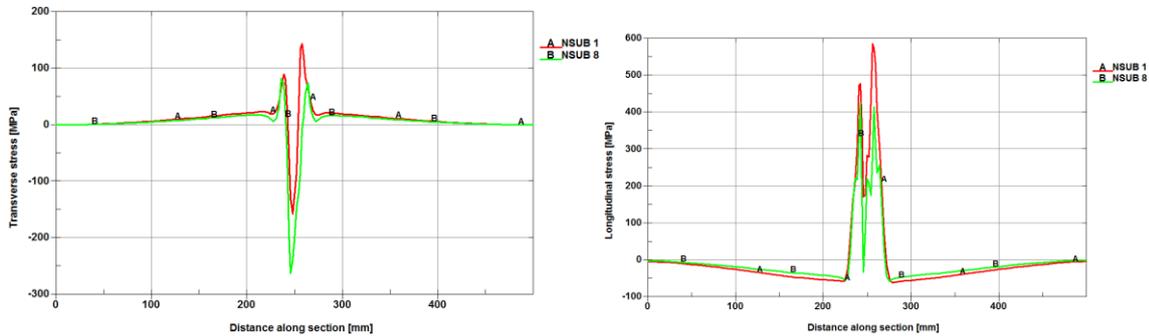

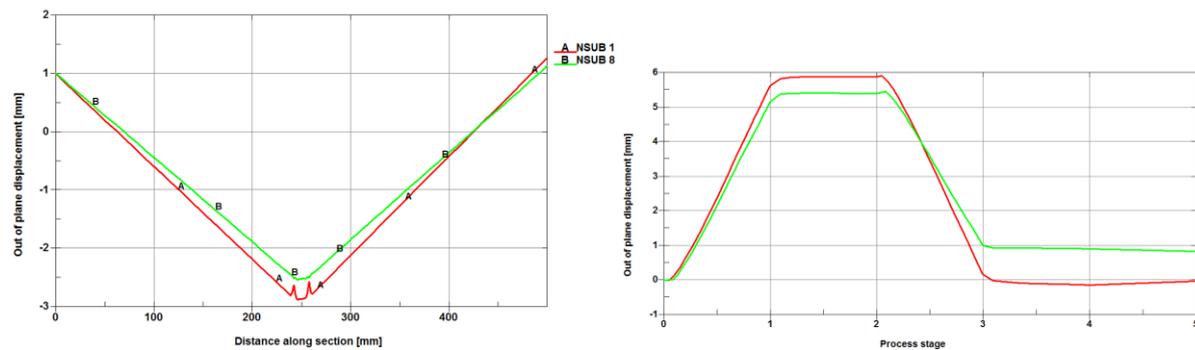Fig.15: Transversal stress (left) and Longitudinal stress (right) after springback

Fig.16: Baseplate (left) and attach plate top edge (right) out of plane deformation after springback

7 Conclusion

Welding simulation capabilities are continuously improving in LS-DYNA®. These capabilities include material models that are tailored for CWM analysis including ghost element and anneal functionality. There is now a possibility to use the common ***MAT_CWM** but also the much more advanced ***MAT_UHS** that include the phase kinetics of the material. The heat source model has also been subject for major improvements. The heat source is now a thermal solver keyword that allows for thermal only simulations. Further, the problem with deforming weld paths are also addressed by allowing the weld path to be defined by a node set rather than a fixed path. One of the biggest issues with using an incremental heat source is that it will pointwise heat up the structure causing a weld velocity, element size and time step dependency. The heat source can now be sub-cycled and thus integrated within a thermal time step. This will give the user more freedom to choose the time step depending on accuracy and simulation speed. Setting up a welding simulation can be quite demanding, especially for a multi pass weld and if the weld order is to be varied. LS-PrePost® now includes a GUI that reduces the amount of work by allowing the user to graphically choose the weld and orientation paths, welding direction and choosing the welding and boundary condition order in a process chart.

8 Literature

[1] Schill, M, Odenberger, E-L, "Simulation of residual deformation from a forming and welding process using LS-DYNA", Proc. of 13th International LS-DYNA conference, 2014.

[2] Klöppel, T, Loose, T, "Recent developments for thermo-mechanically coupled simulations in LS-DYNA with focus on welding processes", Proc. of 10th European LS-DYNA Conference, 2015.

[3] Lindström, P.R.M, "DNV Platform of Computational Welding Mechanics", Proc. Of Int. Inst. Welding 66th Annual Assembly 6, 2013

[4] Lindström, P.R.M, Josefson, L.B., Schill, M., Borrvall, T.," Constitutive Modelling and Finite Element Simulation of Multi Pass Girth Welds", Proc. Of NAFEMS NORDIC Conference, Gothenburg May 22-23, 2012

[5] Åkerström, P., Oldenburg, M., "Austenite decomposition during press hardening of a boron steel – Computer simulation and test", Journal of Materials Processing Technology, 174, 399-406, 2006.

[6] Åkerström, P., Bergman, G., Oldenburg, M.,"Numerical implementation of a constitutive model for simulation of hot forming", Modeling and Simulation in Materials and Engineering, 15,105-119, 2007.

[7] Goldak, J., Chakravarti, A., Bibby, M., "A Double Ellipsoid Finite Element Model for Welding Heat Sources", IIW Doc.No. 212-603-85, (1985)

[8] Eurocode 3:" Design of steel structures – Part 1-2: General rules – Structural fire design", EN 1993-1-2, (2005)