

Efficient Setup and Documentation of Simulations for Welding and Heat Treatment with DynaWeld

Tobias Loose¹

¹ *Ingenieurbüro Tobias Loose, Herdweg 13, D-75045 Wössingen, www.loose.at
E-Mail: loose@tl-ing.de*

Abstract

Simulation models for welding or heat treatment are very complex. It is a special challenge to develop a pre-processor that enables fast setup, automation as far as possible but no limitation in modelling. The aim of DynaWeld [1] is to fulfil these requests. The input data are collected in a spreadsheet. On the one hand this spreadsheet file represents the documentation of the simulation model on the other hand it is the metafile for the DynaWeld keyword generator.

The workflow of DynaWeld allows the user a very efficient model setup even in cases of large models and/ or a huge number of welds.

The data-functions of the spreadsheet program support the input with the sort of data, copy function, auto fill function or the use of cell formulas.

DynaWeld supports the input, import, modification and adjustment of material data for welding and heat treatment. The interface supports several different material simulation software products. The generated data are converted in a spreadsheet file that supports the graphical display of all material parameter for check and documentation.

More than a simple pre-processor DynaWeld provides an environment that links all software packages necessary for welding and heat treatment simulation. It is enhanced by auxiliary tools for model check and post processing. The DynaWeld concept presented in this paper is developed on long term experience in consulting welding and heat treatment simulation.

1. Introduction

In the last decades numerical phenomena for welding and heat treatment processes were investigated by many research projects. The academic basic work is done for the simulation of these manufacturing processes. The challenge now is its application for the industrial cases and use.

The motivation for simulation is various [2][3][4]:

- Design of process
- Design of geometry
- Integrated design of process and geometry
- simulation of process chain for any design reason
- Calculate state of specimen after manufacturing for further numerical analysis
- deeper understanding of process or analysis of failures and damages

Welding and heat treatment processes have some similarities especially from the numerical point of view: They request thermal mechanical coupled analysis, and temperature dependent material properties. Microstructure, phase transformation and phase transformation effects have to be taken into account for heat treatment and some kinds of welding simulation.

The difference between both lies in the phase change of microstructure, difference in some simplification opportunities and the temperature field: Welding requires adjustment for the equivalent heat source, heat treatment requires adjustment for the heat convection function of the quenching medium.

The welding analysis is divided in welding structure analysis which is signed by using an equivalent heat source approach and welding process analysis, which is signed by the calculation of the heating and heat input. In this paper the focus is on the welding structure analysis. The welding process analysis can be used as auxiliary simulation for the determination of the equivalent heat source for the welding structure analysis.

The heat treatment processes:

- Quenching
- Case Hardening
- Inductive Hardening

can be subdivided in the following process steps:

- Heating
- Carburisation (for case hardening)
- Quenching
- Tempering.

The welding processes

- Arc Welding
- Gas Metal Arc Welding (GMAW)
- Submerged Arc Welding (SAW)
- Tungsten Inert Gas Welding (TIG)
- Laser and Electron Beam Welding
- GMAW-Laser-Hybrid
- Resistive Welding (e.g. Resistive Spot Welding RSW)

can be subdivided in the following process steps:

- Clamping, Predeformation
- Heating (Heat Source)
- Cooling
- Reheating (multi pass welding)
- Grinding and Reheating
- Unclamping (welding spring back).

A wide range of weld types and specimen dimension exists.

From the point of modelling and depending of the problem to be analyzed solid element models, shell element models, 2D shells or 2D-axis-symmetric shells or mixed shell-solid element models are recommended. The simulation of the process chain forming - heat treatment - welding - crash becomes more and more of interest [5][6]. Another issue is the assembly of previous simulated process steps within their residual stresses and deformations.

For the design of an pre-processor the processes and process steps mentioned above has to be considered as well as the requests on modelling and process chain feasibility.

LS-DYNA is a well established finite element code including many solvers (thermal, mechanical, electro-magnetic, fluid), supporting coupling between the solvers and explicit as well as implicit analysis. Many features were implemented in the LS-DYNA code to enable welding and heat treatment simulation [7][8][9][10][11] with respect to process chain simulation [12][13]. The one code strategy of LS-DYNA enables the design of a simulation environment within one finite element code for all simulation types and disciplines by use of one continuous data structure. This is the reason why DynaWeld supports the LS-DYNA code.

2. From the model structuring to the GUI structure

Welding and heat treatment simulation touches many engineering and science disciplines:

- Material science
- Welding engineer, heat treatment engineer
- Finite element method engineer
- Meshing engineer
- Software developer

There are not so much engineers in the world who cover all this fields. This might lead to the estimation, welding and heat treatment simulation can only be performed by a small number of experts. But this issue can be solved if the entire simulation task can be subdivided in several subtasks. Each part might be treated by special qualified engineers.

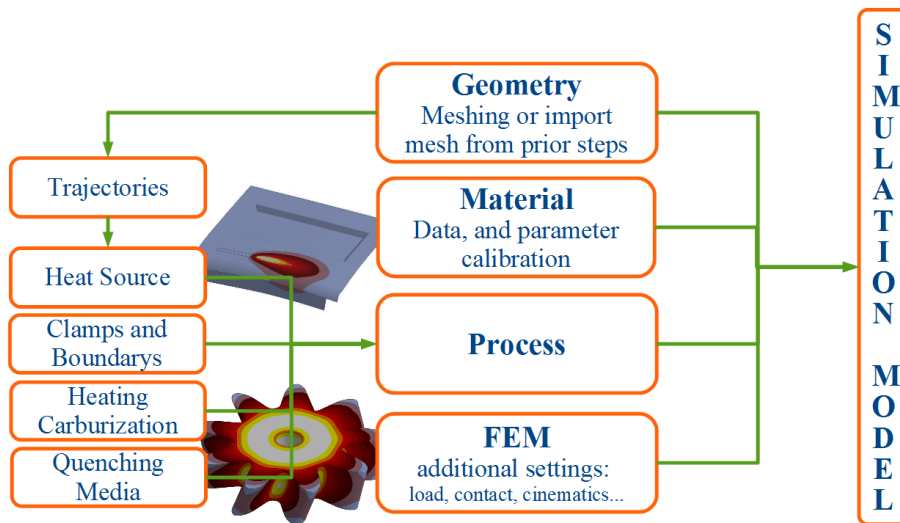


Fig. 1: Sub structuring of the simulation model

Fig. 1 shows the structuring of a welding or heat treatment simulation. The four main tasks are geometry, material, process and additional definitions for Finite Element Method (FEM). The material-card, stored in an includable file, needs to be created once for each material and can be applied to many simulation models. The geometry and mesh has to be created once for each model or for each fabrication step, in case of an assembly simulation. The mesh can be used for many variants of the process. By the process task finally the manufacturing parameter were defined. The process task itself contains subtasks which can be reused several times in the process definition:

- heat source adjustment
- quenching media adjustment

In the last step all other FEM definitions or special model additions are collected. It also might be interpreted as user defined extensions to avoid limitations in modelling

It is possible to outsource the tasks geometry and material. Then the simulation setup become more convenience to those who are familiar with the process but not with the FEM-technology or material science.

The DynaWeld environment follows the concept of model sub structuring. The user has direct access to the main tasks. All software packages which might be in use during modelling can be launched. The user can individually design which software will be launched. With DynaWeld the user can continue with his accustomed software for mesher, FEM pre-processor, FEM post-processor, file manager or spreadsheet program. The environment is related to the projects work directory. All applications are launched related to this directory.

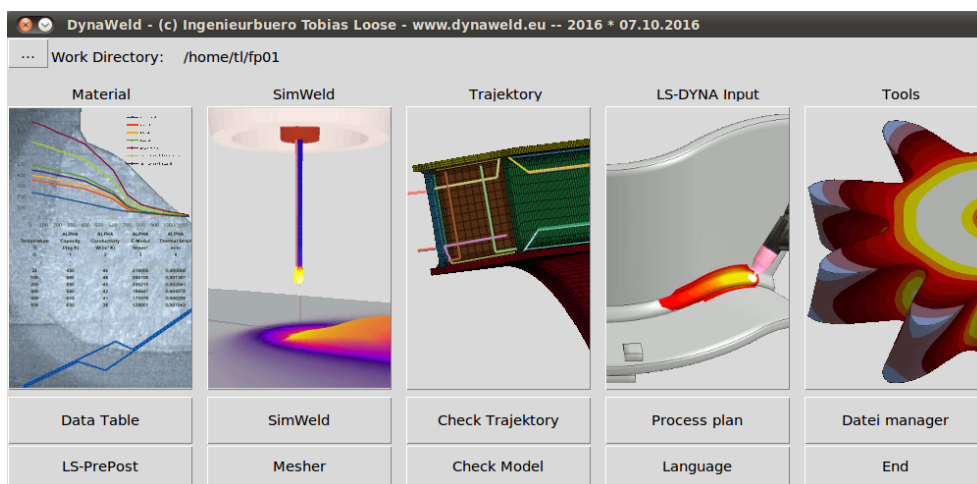


Fig. 2: DynaWeld main frame for welding

3. Geometry, mesh and import from previous simulations

For the used finite element method the geometry needs to be meshed. The mesh represents a grid for the numerical points of calculation in the FEM. For welding as well as for heat treatment a regular mesh is not applicable. The temperature field contains high gradients especially in the weld area or at the surface which requires fine mesh. To get manageable simulation models an adaptive meshing is mandatory with refinement to weld or surface. The simulation performance and the result quality is related to the mesh design. Thus the design of meshing requires experience. In case of industrial application the outline of particular meshing rules, related to the specimen taken into account, are recommend.

To decrease the meshing effort existing meshes from other simulations (crash, forming...) might be used and extended or adapted. For example an existing crash mesh requires mesh refinement in the weld area or the addition of elements representing the weld filler.

In case of process chain simulation or in case of assembly simulation the geometry consists of lots of parts from previous simulations. These parts are implemented in the new simulation by dynain-file with geometry (element, node), stress, strain, history variable and shell thickness information. The new parts are added by mesh only (element, node) [5].

Meshing has to be done by external meshing tools. DynaWeld sources automatically all existing mesh files in the project directory. One single file is not mandatory, the mesh can be stored in different files. Node sets or segment sets are needed to address trajectories for weld, boundary conditions, loadings or contacts. They can also be generated during meshing and stored in the project directory to be auto-sourced from DynaWeld. The model strategy is defined by meshing too: solid, shell-solid, shell or shell 2D which is all supported by DynaWeld.

4. Material

The intention of heat treatment is the design of properties by local change of material properties and microstructure induced by temperature. Microstructure becomes an important impact as well as the change of the phase transformation. This requests the use of multi phase material models with phase kinetic models. Material data is temperature dependent and additionally phase dependent. The strain rate dependency for flow curves is on top. The new LS-DYNA Material *MAT_GENERALIZED_PHASE_CHANGE / *MAT_254 is designed for these applications and features [14]:

- 24 Phases
- Shells, solids, 2D-shells
- Welding features (e.g. reset of plastic strain)
- Phase transformation strain
- Transformation induced plasticity (TRIP)
- Sub cycling for phase transformation calculation
- Tempering
- Hardness computation

Several phase transformation laws are available. Any law can be assigned to each possible phase change relation $i \rightarrow j$:

- Koinstinen-Marburger
- Kirkaldy
- generalized Johnson-Mehl-Avrami-Kolmogorov with Leblond extension
- Oddy
- Time criterion

The list of laws is extendable in case further research provides new laws.

The chemical alloys impact the microstructure. In case of steel carbon is the main alloy. The case hardening process use this effect. The specimen is carburised before quenching. The distributed carbon content leads to distributed material properties, flow curve as well as phase kinetic, according to the carbon content. For this instance the carbon dependency has to be taken into account and a preceding carburisation simulation has to be performed.

The extension for case hardening is actually under development.

While the change of microstructure is wilfully manipulated for heat treatment processes it is an unwanted effect during welding. The effects from phase transformation has to be taken into account for welding as well as for heat treatment. Thus the same material model can be applied. For welding analysis the filler material is assigned initially to the liquid phase and activated when the temperature exceeds the melting point.

In case the materials microstructure does not change (e.g. stainless steels) a single phase model is recommend. In case of distortion analysis for welding a simplified approach for the phase transformation strain, which applies the transformation strain during the temperature

interval of transformation, can be chosen to enable the use of a single phase model. The LS-DYNA material *MAT_CWM / *MAT_270 is such a single phase model for welding analysis:

- Shells, solids, 2D-shells
- Activation of filler
- Reset of plastic strain
- simplified approach of phase transformation strain
- Cooling period (e.g. T_{8-5} -time)

*MAT_270 features bilinear flow curves as well as nonlinear flow curves with strain rate dependency. The simpler the model the faster and more robust runs the simulation. Each model has its preferred application and DynaWeld supports both.

DynaWeld-Material is developed to drive the material data for welding and heat treatment. The user has the choice between input or import of data. The user can adjust the data. If necessary Phase kinetic parameter like JMAK-parameter are adjusted automatically by DynaWeld-Material. Finally the user can choose whether *MAT_254 or *MAT_270 shall be exported. (Fig. 3 left). DynaWeld-Material supports a method to generate multi phase flow curves from a basic flow curve according to yield and ultimate strength of each phase at room temperature. This method is described from Loose in [15].

Actually DynaWeld supports the following input of Data:

- User defined - without data import
- WeldWare import
- Sysweld import
- JMatPro import - single phase data
- JMatPro import - multi phase data

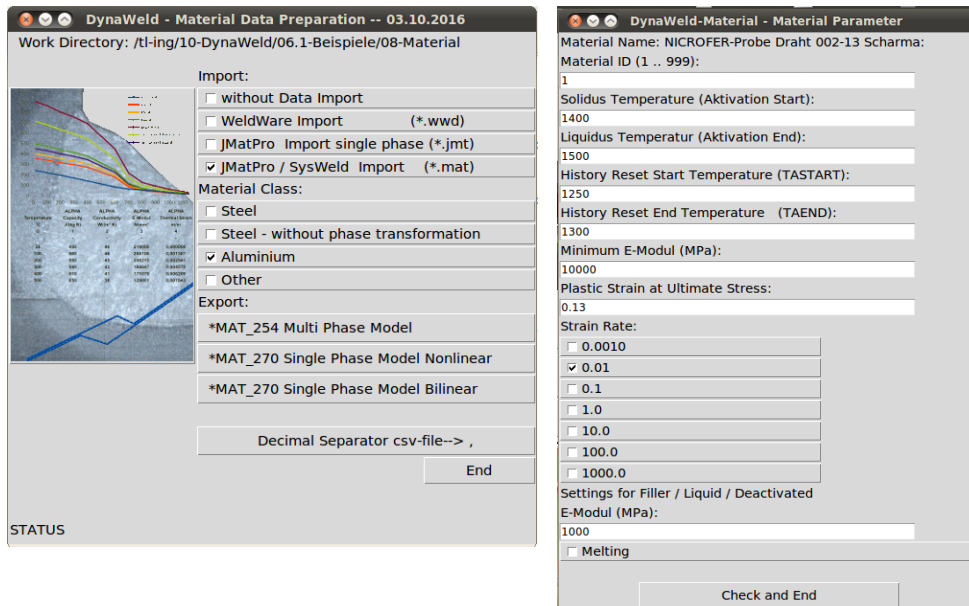


Fig. 3: DynaWeld Material - Import of single phase models

DynaWeld-Material supports several material classes. Each class has its own temperature range phase assignment and predefinitions for selected variables (Table 1).

Table 1: Phase concept multi phase material

Phase ID	Steel	Steel no transformation	Aluminium	other
1	Austenite	Molten	Molten	Phase 1
2	Ferrite	-	HAZ	Phase 2
3	Perlite	-	-	Phase 3
4	Bainite	-	-	Phase 4
5	Martensite	-	-	Phase 5
6	Base Material	Base Material	Base Material	Base Material
7	Liquid/Filler	Liquid/Filler	Liquid/Filler	Liquid/Filler
8	tempered Martensite	-	-	Phase 8
9	tempered Bainite	-	-	Phase 9

DynaWeld-Material - Material Parameter

Material Name: from_JMatPro

Material ID (1 .. 999): 1

Solidus Temperature (Aktivation Start): 1400

Liquidus Temperatur (Aktivation End): 1500

Latent Heat of Melting (kj/kg): 270

History Reset Start Temperature (TASTART): 1445.6904

History Reset End Temperature (TAEND): 1495.6904

Minimum E-Modul (MPa): 10000

Plastic Strain at Ultimate Stress: 0.13

Import electric resistivity from 11-MATERIAL.csv

Settings for Filler / Liquid / Deactivated

Flow Curve as imported

Flow Curve according Austenit

Constant Yield Stress

E-Modul (MPa): 1000

Melting

Basmaterial: Composition of Phase

Phase 1	Phase 2	Phase 3	Phase 4	Phase 5
0.0	0.25	0.75	0.0	0.0

Phase Relation

Target	Source	Yield MPa	Ultimate MPa	Add
DynaWeld	JMatPro / Sysweld			
	P-1 P-2 P-3 P-4 P-5			
Austenit	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input checked="" type="checkbox"/>	182.0	555.48	
Ferrit	<input type="checkbox"/> <input checked="" type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	316.8	586.76	
Perlit	<input type="checkbox"/> <input type="checkbox"/> <input checked="" type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	554.3	922.939999999	
Bainit	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input checked="" type="checkbox"/> <input type="checkbox"/>	768.1	1191.35	
Martensit	<input checked="" type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	842.0	1278.93	
Base Material	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	494.924999999	838.895	<input checked="" type="checkbox"/>
Filler Liquid	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	1000	1100	<input checked="" type="checkbox"/>
Tempered Martensit	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	768.1	1191.35	<input checked="" type="checkbox"/>
Tempered Bainit	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	554.3	922.939999999	<input checked="" type="checkbox"/>

Material Class:

Steel Steel - without phase transformation Aluminium Other

Skip Update Re and Rm according Source Phase Check and End

Fig. 4: DynaWeld Material - import of multi phase models

The base material represents the initial material. It may be a mix of single phases or a rolled / heat treated state. Liquid represents molten material as well as not yet deposited filler material for welding. The user can choose whether melting shall be taken account or not. In case of the import the user can define additional settings. Fig. 3 right shows the import window for single phase material data. Fig. 4 shows the import window for multi phase material models. In this case the user can adjust the phase allocation from import data. He can also adjust the flow curve by yield and ultimate strength according to Loose [15].

Additionally to the material LS-DYNA keyword file DynaWeld-Material prints out a spreadsheet file with the data of all material properties. The material data is displayed in diagrams. This serves the visualisation as well as the documentation.

5. Setup of heat treatment simulations

Reduce the input to the process parameter and let the pre-processor do the job of keyword-writing. To feature this requires a deep understanding of the variable parameters of the process. A spreadsheet file is used as a form of readable metadata to define the parameters. The table HT-Process (Fig. 5) collects the data for quenching media, oven temperature, carbon content and tempering temperature. Load-buttons enable the user to import existing data. Thus a one time defined oven curve can be stored in the user database and may be reloaded for many simulations. A graphic table visualises all input data as seen in Fig. 6:

DynaWeld Heat Treatment - Process							
Load		Load		Load		Load	
Quenching Media		Oven Temperature		Oven Carbon		Tempering	
T in °C	a in W/(mK)	t in s	T in °C	t in s	C in %	t in s	T in °C
1	2	3	4	5	6	7	8
*	*	*	*	*	*	*	*
200	200	0	830	0	0	0	20
300	300,5	200	850	20	0,15	60	250

Fig. 5: Input table HT-process
Oven Parameter

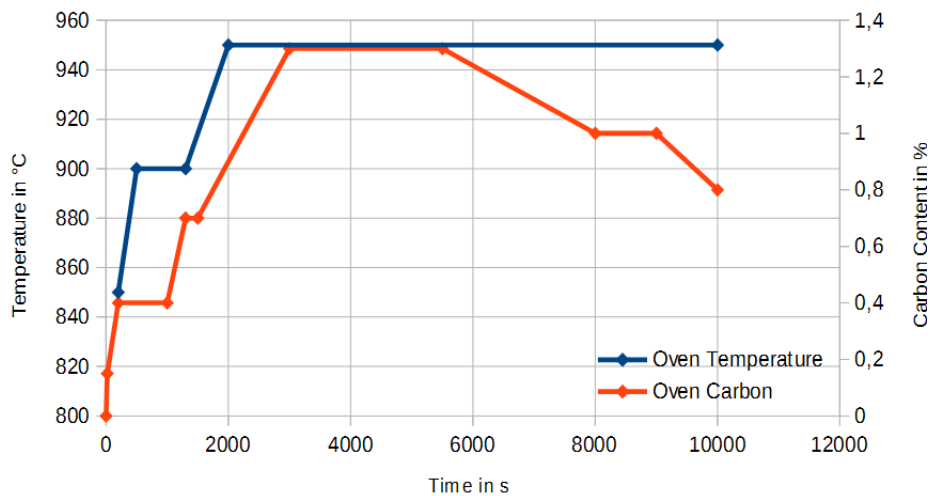


Fig. 6: Visualisation of oven curves

In the Table HT-Parameter (Fig. 7) the remaining process parameters like temperature of quenching media, diving parameter or initial carbon content are defined. Further tables enable the definition of boundary conditions, solver and output settings. To consider the heat treatment of welded structures the definition of contacts is also enabled.

The data access as well as the setup of changes or variants is fast. The spreadsheet-file DynaWeld-Heat-Treat used for the input of data represents also the documentation of the simulation model without additional work.

DynaWeld Heat Treatment - Parameter							Save Table	Save All
Beschreibung	P1	P2	P3	P4	P5	P6		
	-	-	-	-	-	-		
0	1	2	3	4	5	6		
*	*	*	*	*	*	*		
Surface ID	401							
Heat Convection	7,0000	W/(mK)						
Heat Radiation								
Carbon Diffusion Coeff.	1,50E-005							
Initial Carbon Content	0,2200	C in %						
	v in mm/s	x	y	z	Start Node ID	Dir Node ID		
Diving	133,00	0,00	0,00	-1,00				
Medium Temperature in °C	80,0000							

Fig. 7: Input table HT-parameter

The input file generator for DynaWeld-Heat Treatment is now under development as well as the extension to inductive hardening.

6. Heat source for welding simulations

The welding structure analysis uses equivalent heat sources representing an equivalent heat input into the model. DynaWeld supports a wide bibliography of equivalent heat source functions:

- part heating for metatransient method
- surface heat sources
- volumetric heat sources

The heat source can be applied to segment (surface), solids, shells and 2D shells. The new LS-DYNA heat source *BOUNDARY_THERMAL_WELD_TRAJECTORY [9] is supported as well. This heat source applies heating on shell and solids within the same definition. It has an internal energy input control and an sub cycling algorithm over time and space to overcome the mesh dependency and the influence of the overall time stepping of the thermal solver. The heat is applied on the thermal thick shell also with respect to its geometry in thickness direction

The volumetric functions are designed to cover all kinds of fusion welding with the following geometric distributions:

- ellipsoid
- cylindrical
- conical
- double-conical

The intensity function in the geometric shape is constant or Gaussian. The modified ellipsoid heat source for gas metal arc welding according to the recent investigation from Mokrov [16] is supported too.

The heat source calibration can be done by micro section, WPS-approach or by import from process simulation. DynaWeld supports an interface to SimWeld [17][18] for the pre calculation of the heat input for gas metal arc welding.

7. Setup of welding simulations

The moving heat source in the welding simulation model is defined by the trajectory, the reference which defines the direction, start time, velocity and the parameter of the equivalent heat source. The fastest way to define a trajectory is the definition of a set of continuous sorted nodes. This is the method supported by DynaWeld. With the new LS-DYNA heat source only the trajectory is mandatory, the reference is auto detected by the normal of the elements surface. For the other heat sources a second node set for each weld defines the reference.

Rotation around the trajectory, movement in reference direction or movement in lateral direction enable the local fine adjustment of the heat source position. The end time is calculated by the length of trajectory. velocity and start time. The start time of the next weld can be calculated by end time of prior weld and the intermediate time. The DynaWeld-Trajectory procedure sources the mesh files and the node set files and calculates the length of all trajectories as well as its number of elements. The number of elements, trajectory length and velocity is used to define the basic time step for the analysis. The heat source type is addressed by a short code. The codes are explained in a commentary field in the input table. Special parameter can be set to each heat source to include or exclude certain parts from the heat input or to adjust the sub cycling of the new LS-DYNA heat source. A reverse option is available to reverse the start and end point of the trajectory. A check file in LS-DYNA keyword format is written to visualise the trajectories, references and start points by beam elements. This check is important for simulation quality reason.

The input of the heat sources is done in the spreadsheet table DynaWeld Process-plan (Fig. 8). The row Process nr. is used to order the weld sequence. A new weld sequence can be defined and the lines in the table can be reordered by the function "sort". Thus the change of weld order is only a task of few seconds. It is possible to apply more than one robot with simultaneously welding. The heat input is given by the total energy per time (Q in W). The distributed heat input is calculated. Some heat sources allow the input of the surface or volume distributed heat input (q in W/mm² or q W/mm³). The calibration factor k_f is used for the final adjustment of the energy input. The heat input Q is multiplied by k_f.

DynaWeld – Process plan		Save Table	Save All	29.04.16															
Process nr.	Weld ID	Length mm	v mm/s	Duration s	Start s	End s	PAUSE s	Q W	af mm	ar/R2 mm	b/R3 mm	c / v1 mm	br / v2 mm	cr / v3 mm	kf	ay			
0	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16			
1	1001	58,75208426	7,5	7,8336	0,0000	7,8336	0,0000	7838,4482	3	10,72505	5,07272	4,91363			1,33183878	90			
2	1002	64,93651425	7,5	8,6582	7,8336	16,4918	5,0000	7838,4482	3	10,72505	5,07272	4,91363			1,26103721	-90			

Fig. 8: Input table heat sources

The spreadsheet table DynaWeld – Time Schedule Welding contains the information of the time periods with small time steps during welding or the movement of clamps . It uses the start and end times from the process sheet and calculates the basic time step. The user can adjust the time step for mechanical and thermal solver and add a time period of small steps after each

weld. Discrete time periods with small time steps can be added user defined. The time step increase for the cooling time is calculated automatically.

The spreadsheet DynaWeld – Boundary is used for the definition of:

- single point constraints on nodes or node sets
- symmetry planes
- load (force) on node for nodes or node sets
- displacement (movement) on nodes for nodes or node sets

Loads and displacement are applied with user defined start ramp hold time and end ramp. This features the movement of clamps as well as the application of predeformations.

The assignment of materials, shell thickness and number of integration points is given in the table DynaWeld – Part and Segment. Segment sets, node sets or part sets can be defined by selecting the considered parts and can be used for any definition in the model. Predefined materials for rigid body with the block of any combination of degree of translation are addressable. The user can part wise decide between full or reduced integration.

DynaWeld supports a wide range of LS-DYNA contacts.

- Weld contact
- Friction contact
- Mortar contact
- Smooth contact
- Tied contact
- Node to surface contact with interference option

The weld contact is now available for shells and solids. It switches automatically from friction contact to tied contact when the melting temperature is reached in the contact surface. Mortar contact is special for implicit analysis, the smooth contact takes into account the curved surface instead of the sharp shell-element edges from the mesh. This increases the contact robustness especially for cases like automotive bodies. Generally The shell thickness is taken into account automatically. The option ignore shell thickness is also available to enable the modelling of clamps by its contact surface with shells. The contact is defined in the sheet DynaWeld - Contact Table.

The final table DynaWeld - Start and Model Parameter defines general settings, solver settings, post settings dead load and heat transfer by conduction and radiation. The title of the project is given and the LS-DYNA executable to be used for launching the job.

All tables are stored in the file DynaWeld-Processplan. This file contains all information of the simulation model and represents the documentation.

If any keyword requested for modelling is not supported by DynaWeld, the user can define it in a special user-file. This file will be sourced and included automatically. Thus the definition in this file is kept as well if the DynaWeld input is modified.

The DynaWeld Input writer welding (Fig. 9) reads the information from the file DynaWeld-Processplan, sources all mesh and material files and generates the keyword input for LS-DYNA solver. Structured input with subdirectories and include files or one-file input is available. For the structured input the user can choose whether the complete input is written or single tasks. Last feature powers in case of single modifications. Check files are written to prompt the input for control reason. An launch file for LS-PrePost enables the opening of the entire model in LS-PrePost or other pre-processors for checking.

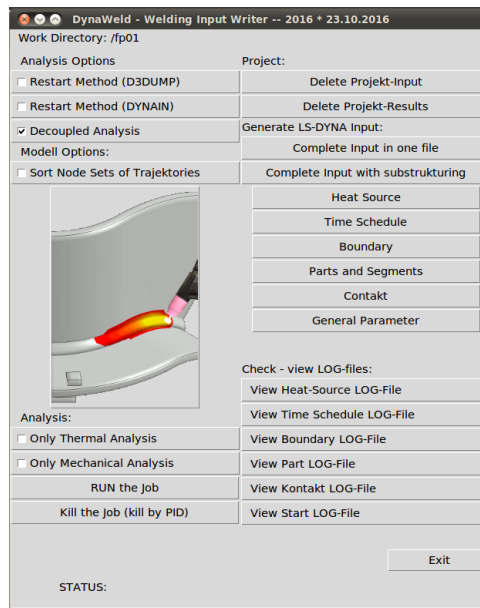


Fig. 9: DynaWeld Input writer welding

The user can erase existing input or results. The input writer supports thermo-mechanical decoupled or coupled analysis and whether the simulation shall be subdivided in restart steps for each weld. Finally the user can launch or kill the computation.

8. Special tools and features

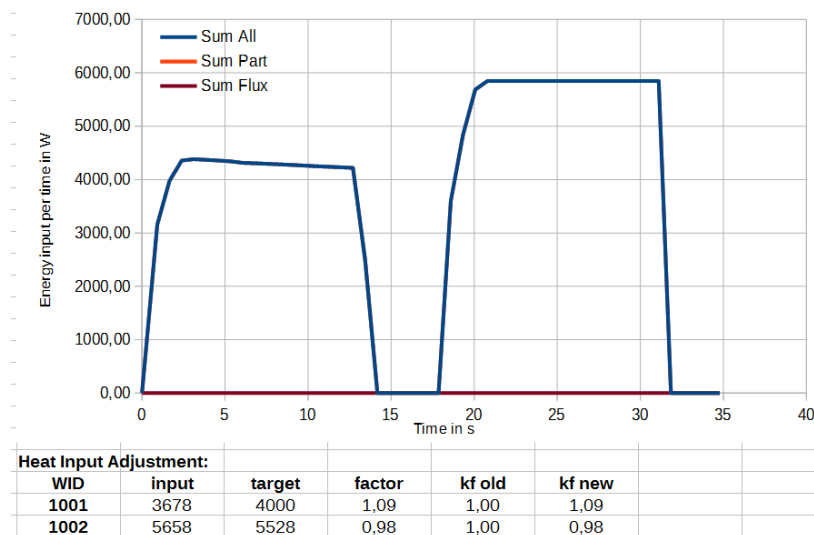


Fig. 10: DynaWeld Heat-Check

The Tool menu contains simulation related auxiliary procedures. Duplicate model generates a clone of the actual model. In case of decoupled analysis the option copy thermal results enables variant runs for mechanic solution only. The post file for thermal or mechanical analysis can be launched. With respect to the material - steel or aluminium - the DynaWeld temperature legend is applied to the temperature range room temperature - melting temperature. Another procedure

writes command files for automatic post processing on path or time history on discrete nodes. Therefore the input are files with the related node lists.

Most important for the result quality is the correct input of energy. DynaWeld Heat Check sources the LS-DYNA prompt file for thermal analysis and evaluates the heat input considered in the simulation run (Fig. 10). Thus the existing heat input can be compared to the desired heat input. If adjustment is needed the calibration factor k_f can be calculated and used for an improved run. This method gives guarantee for the correct heat input which is most important for correct results.

9. Summary

This paper presents the new DynaWeld software as an environment and pre-processor for welding and heat treatment simulations with LS-DYNA. DynaWeld is mentioned to be an add on for existing software environment as well as an tool for the entire setup of models. His intention lies on efficient model setup with no limitations in modelling.

The input uses spreadsheet files as meta-file. These spreadsheet files represent the documentation of the simulation model. The spreadsheet input enables a fast access to the data and enables fast setup and modify of models. DynaWeld use a sub structuring of the model in the tasks geometry, material and process. The auto-source feature of DynaWeld reduces the effort of input.

DynaWeld supports many model and discretisation types. Shell element models are supported as well as solid or 2D-shell element models.

The material tool provides the data management for welding and heat treatment material data. User defined input of data, import from WeldWare, Sysweld or JMatPro with the capability of data adjustment is available. The transformation from single phase input data to an multi phase model with phase transformation is featured. Additional phases for tempered material are able to create. Simple and advanced material models are supported to guarantee efficient simulation run adapted to the simulation task

DynaWeld considers the process chain simulation and the assembly simulation.

Finally the latest LS-DYNA welding and heat treatment features like the new shell-solid heat source with energy input control or the new welding contacts are supported by DynaWeld

10. References

- [1] <http://www.dynaweld.eu>; <http://www.tl-ing.eu>
- [2] Loose, T.: Schweißsimulation - Potentiale und Anwendungen. In: 26. Schweißtechnische Fachtagung 2016 Magdeburg, Verlag Otto-von-Guerike-Universität Magdeburg
- [3] Brand, M. ; Loose, T.: Anwendungsgebiete und Chancen der Schweißsimulation. In: Schweiß- und Prüftechnik (2014) ÖGS Österreichische Gesellschaft für Schweißtechnik (Hrsg.), Nr.05-06, pp 138-142, Wien
- [4] Loose, T. ; Boese, B.: Leistungsfähigkeit der Schweißstruktursimulation im Schienenfahrzeugbau. In: Vortragsband 10. Fachtagung Fügen und Konstruieren im Schienenfahrzeugbau, 2013, pp 61 - 67, Halle
- [5] Loose, T. : Coupled simulation of forming and Welding with LS-DYNA for the design of Distortion-Compensation 13. LS-DYNA-Forum, Bamberg 06.-08.10.2014
- [6] Loose, T.: Einbindung der Schweißsimulation in die Fertigungssimulation mit SimWeld und DynaWeld. In: DVS Congress 2015, DVS-Berichte Band 315, pp 860 - 865

- [7] Shapiro, A. B.: Heat Transfer in LS-DYNA, 4th European LS-DYNA Conference, Ulm, 2003, www.dynalook.com
- [8] Klöppel, T. ; Loose, T.: Recent developments for thermo-mechanically coupled simulation in LS-DYNA with focus on welding processes. 10th European LS-DYNA Conference, Würzburg, 2015, www.dynalook.com
- [9] Klöppel, T. ; Schill, M. ; Loose, T.: Recent Updates for the Heat Transfer Solver in LS-DYNA with focus on computational welding mechanics. 14th International LS-DYNA Users Conference, Detroit, 2016, www.dynalook.com
- [10] Schill, M. ; Jernberg, A. ; Klöppel, T.: Recent Developments for Welding Simulations in LS-DYNA and LS-PrePost, . 14th International LS-DYNA Users Conference, Detroit, 2016, www.dynalook.com
- [11] Klöppel, T.: Recent Updates for the Conjugate Heat Transfer Solver in LS-DYNA. 14. Deutsches LS-DYNA Forum, Bamberg, 2016
- [12] Schill, M. ; Odenberger, E.-L.: Simulation of Residual Deformation from a Forming and Welding Process using LS-DYNA, 13th International LS-DYNA Users Conference, Detroit, 2014, www.dynalook.com
- [13] Klöppel, T. ; Erhart, A. ; Haufe, A. ; Loose, T. : Recent Developments in LS-DYNA to close the virtual process chain for forming, press hardening and welding. 18th International ESAFORM Conference on Material Forming, Graz, 15. - 17.4.2015
- [14] Loose, T.: ; Klöppel, T.: An LS-DYNA Material Model for the consistent simulation of Welding, Forming and Heat Treatment, 11th international seminar on mathematical modelling of weld phenomena, Senggau, 2015
- [15] Loose, T.: Einfluß des transienten Schweißvorganges auf Verzug, Eigenspannungen und Stabilitätsverhalten axial gedrückter Kreiszyinderschalen aus Stahl, Diss., Universität Karlsruhe, 2007
- [16] Mokrov, O.: SimWeld – Neue Entwicklungen und präzisere Modelle der Ersatzwärmequelle für die Struktursimulation, Infotag Schweißen und Wärmebehandlung mit LS-DYNA, Aachen, 27.09.2016
- [17] Loose, T. ; Mokrov, O.: SimWeld and DynaWeld - Software tools to set up simulation models for the analysis of welded structures with LS-DYNA. In: 10th European LS-DYNA Conference 2015, Würzburg, www.dynalook.com
- [18] Loose, T. ; Mokrov, O. ; Reisinger U.: SimWeld and DynaWeld - Software tools to set up simulation models for the analysis of welded structures with LS-DYNA. In: Welding and Cutting 15, pp. 168 - 172, 2016