

Coupled Simulation of Forming and Welding with LS-DYNA for the design of Distortion-Compensation

Tobias Loose¹

¹Ingenieurbüro Tobias Loose, Herdweg 13, D-75045 Wössingen (Lkr. Karlsruhe), Germany, loose@tl-ing.de, www.tl-ing.eu

1 Introduction

In the past the simulation methods for forming simulation and welding structural simulation were developed respectively for themselves. This leads to the situation that forming simulation was performed with one code and welding simulation was performed with an other code. The coupling of both simulations is time-consuming and only possible by a loss of accuracy during the mapping of history variables.

In the industry grows the interest in simulations modelling the complete process chain:

Welding simulation considering the state of a previous forming or welding step

Forming simulation considering the state of a previous forming or welding step

A forming process leads to deformation, strain hardening and residual stress, hot forming or welding additionally lead to a change in microstructure and thus to a change of the material properties. These effects have to be considered in the process chain simulation as well. The latest developments for LS-DYNA enable welding structural analysis. Thus the coupled simulation of forming and welding in one finite element code is feasible with LS-DYNA.

If its possible to simulate the manufacturing process in the right way, it is also possible to simulate correction task for the manufacturing process and design them. Correction tasks to compensate welding distortion are necessary to receive the final assembly with the desired geometry.

2 Specific Features for Welding and Heat Treatment Simulation

Welding and heat treatment simulation is based on three piles (fig.1)

Material Simulation

- material properties according microstructure or microstructure evolution
- change of microstructure

Process Simulation

- simulation nearby the process
- heat input generation
- weld pool evolution

Structure simulation

- global temperature field
- global stress strain and distortion field

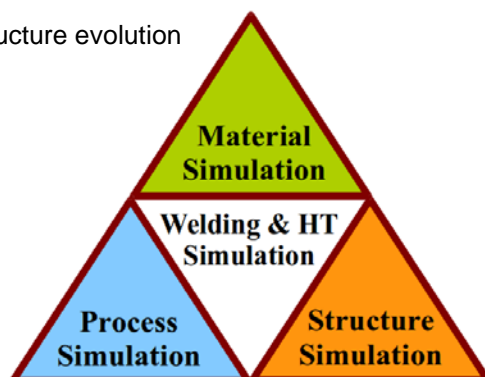


Fig. 1: Three piles of welding and heat treatment simulation

For the structure simulation of welding and heat treatment processes the capability of the simulation has to be expanded to the metallurgical effects of these processes. The simple coupling of thermal and mechanical analysis is not sufficient. Thus additionally has to be taken into account for the modelling of the material:

Material Properties:

- material properties depend on temperature
- material properties change due to thermal loading cycle
- material properties depend on phase

Material Model:

- reset of material history at melting point
- phase transformation
- phase transformation effects
- mixing rule between phases for the properties

History Variables:

- phase proportion
- activation status for filler material

Moreover the model can be expanded to grain size dependency and chemical composition dependency. Carbon dependency is needed for case hardening simulation.

The material model used for welding and heat treatment can be seen as a three dimensional problem:

- nonlinear stress strain relation
- temperature dependency
- microstructure dependency

The material model used for forming or crash can be seen as one dimensional problem:

- nonlinear stress strain relation

Thus a material model for welding and heat treatment can be used for crash and forming as well. Only the sub dimension "room temperature with constant phase proportion" is used.

3 Simulation of manufacturing process

Fig. 2 shows the main simulation components of the manufacturing process arranged around the welding and heat treatment simulation.

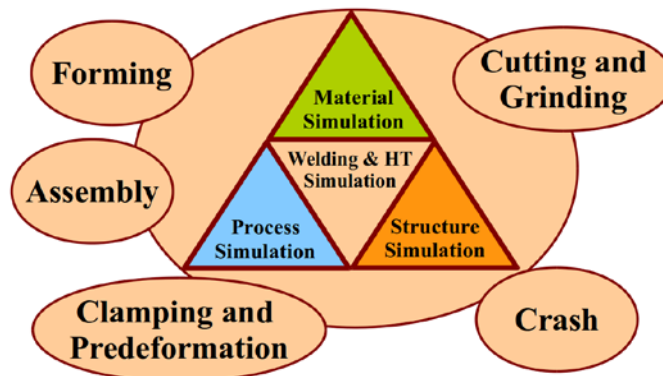


Fig. 2: Welding and heat treatment simulation as a part of the simulation of the process chain

The manufacturing process is not only the chain of single processes. Additionally the assembly of parts has to be taken account. These parts may also have a process history like forming, welding heat treatment.

Two strategies can be applied on the simulation of manufacturing processes:

Method A:

- Each simulation task with a special simulation tool
- Each simulation task with a specific material model
- Transfer of the information between two tasks via special interfaces
- Mapping of results

Consequence Method A:

- Information loss from one step to the next step by mapping
- Problem of interface compatibility
- Multiple license costs

Method B:

- As many simulation task as possible in one simulation tool
 - Each simulation task with the same material model
 - Continuous transfer of information within the same code and the same data structure
- Avoid mapping of results.

Consequence Method B:

- No information loss between single simulation steps
- No trouble with interface compatibility
- Save of license costs

Method B is chosen for the examples in this paper.

Distortion occurs in every process step. The design of final geometry with the demand of no distortion from welding will not remain as state of the art. In future the manufacturing process will be part of the design. The first challenge is the realistic simulation of the whole process. The result of this simulation is the description of the state of the assembly: geometry, stress, strain, microstructure. The next iteration is the design of compensation tasks. These tasks regards the result of the whole process. Thus not each single manufacturing process requires its compensation task. One compensation task can be designed considering distortion effects of multiple steps.

4 Manufacturing of a Box

The manufacturing of a box demonstrates, which single steps has to be taken into account for a welded assembly. The forming of parts is considered. Fig. 3 visualizes the single manufacturing tasks:

1. Forming:

- The roof geometry is made by forming a 3 mm thick sheet made of 1.4301

2. Assembly:

- The sidewalls are added to the roof, also the filler and the clamps

3. Welding:

- The sidewall is welded the roof

4. Clamp and predeformation:

- The distorted sidewall is pressed on measure

5. Assembly:

- The bottom plate is added, also the filler

6. Welding:

- The bottom plate is welded to the sidewall

7. Unclamping

- Final step is the unclamping

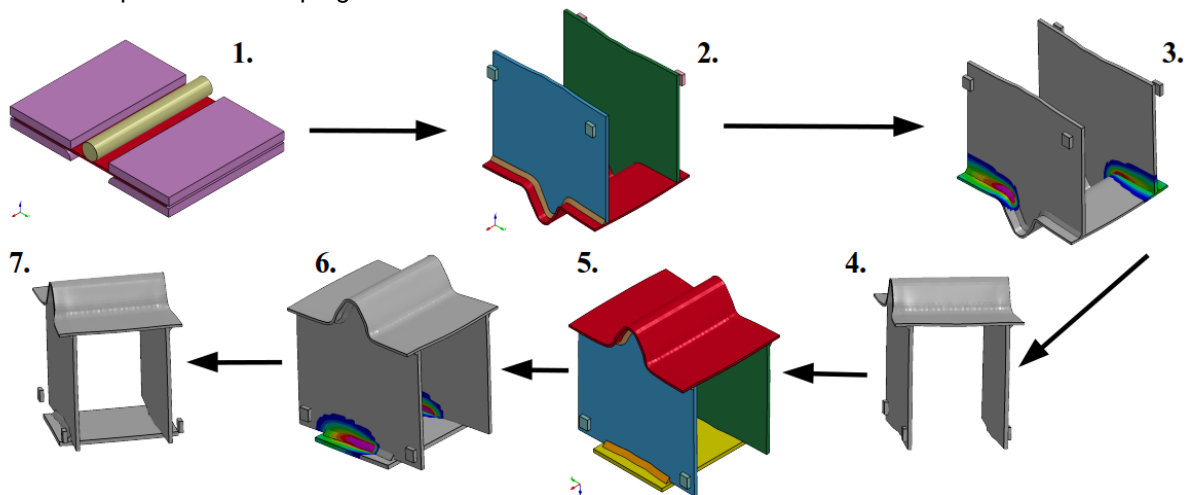


Fig 3. Tasks of the manufacturing of a box

The simulation is performed with a solid element model (Hexaeder, Pentaeder). The stainless steel 1.4301 has no phase transformation during the welding thermal cycle. Thus a single phase model is sufficient: `*MAT_CWM (*MAT_270)` and its counterpart `*MAT_THERMAL_CWM` is used in all steps. This material model considers elastic-plastic flow curves with linear strain hardening. Isotropic hardening is chosen for this model. The activation of the filler material is taken into account. The history variables of stress and plastic strain as well as the activation status of the filler are kept in the dynain file. The transfer of the history variables from one simulation step to the next step is done by dynain file. Contact with friction is defined between the parts and between clamps or the forming tools and parts. The filler of the weld are joined with tied contact to the parts.

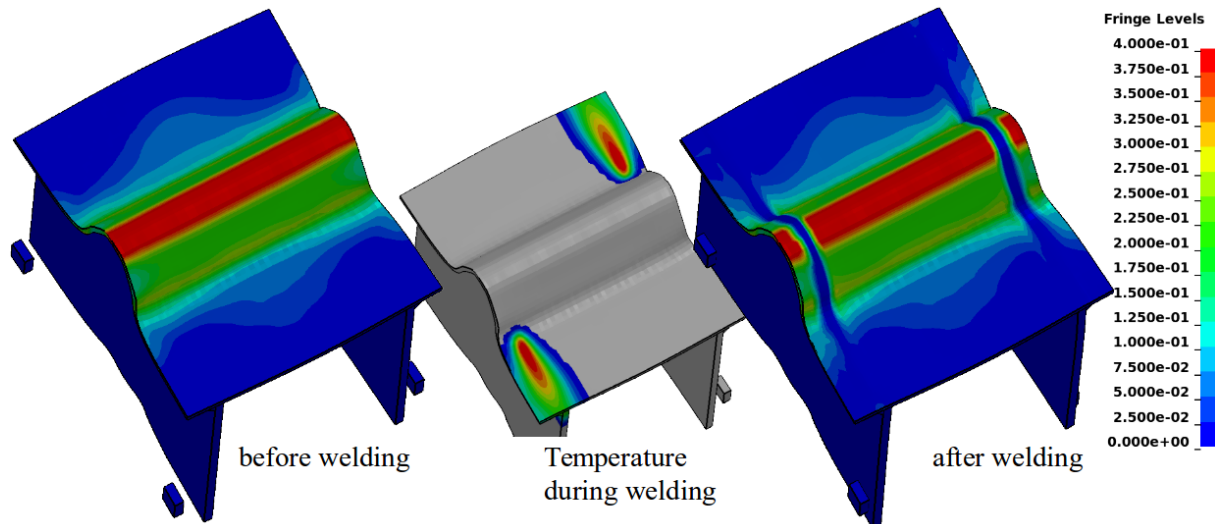


Fig. 4: Plastic strain. Left after forming, before welding. Right: after welding

The plastic strain distribution after forming is shown in Fig. 4 on the left side. The welding leads to a removal of the strain hardening. This effect is represented by the reset of plastic strain. On the right side of Fig. 4 the influence of the weld is recognizable in the plastic strain distribution.

The step by step evolution of equivalent stress (v. Mises) is shown in Fig. 5. Each step gets his own influence on the residual stress. Finally this picture clarifies that the correct state of the stress field requires the simulation of the whole process.

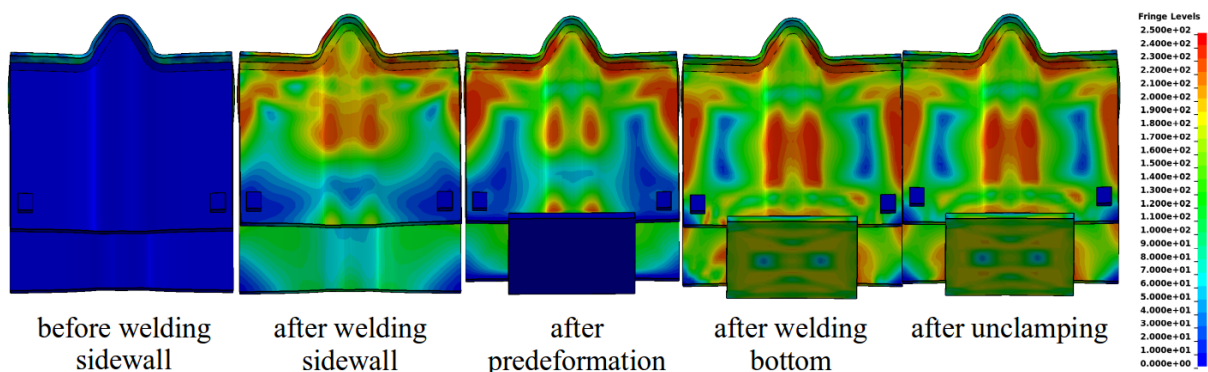


Fig. 5: Step by step evolution of equivalent stress (v. Mises)

5 Laser Welding - Forming - Crash

The next example considers laser welding as the first process and forming, stamping or crash are the secondary processes. This example fits to tailored blanks. For thin metal sheets the modelling by volume elements is not advisable. Recommend is the use of shell models. Welding simulation and forming simulation on shells is performed in this example. Steel S355 requires the consideration of microstructure, microstructure transformation and its transformation effects. This is realised with `*MAT_UHS_STEEL (*MAT_244)`. The geometry of the model is shown in Fig. 6. The plate thickness

is 1 mm. The left side of Fig. 6 illustrates the clamping for forming. The temperature field is drawn in the middle of Fig. 6 and the situation for the impact case (crash) is printed on the right side of Fig 6. Forming is a slow process, the plate needs to be clamped. Impact is a high speed process. The initial velocity of the bullet is 5000 m/s. The plate is not clamped but in free fall when simulation starts. Thus the inertia holds the plate in place.

The welding analysis is performed with the implicit LS-DYNA solver, forming and impact simulation are performed with the explicit LS-DYNA solver.

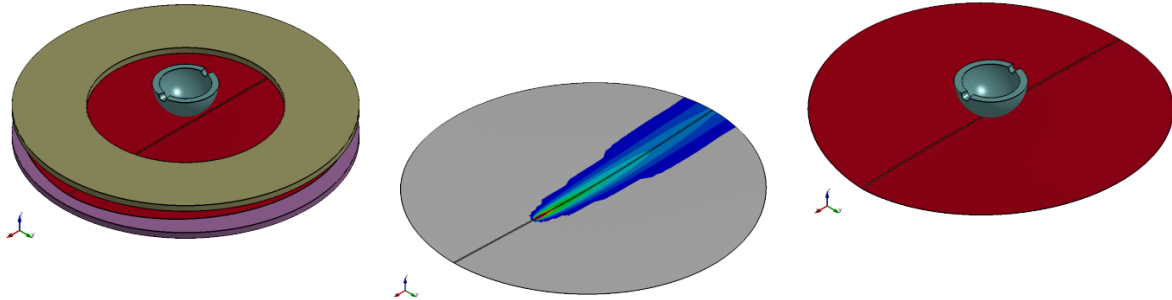


Fig. 6: Model for welding - forming (left) and welding - crash (right) process chain

Fig. 7 illustrates the result of the distortion normal to the plate surface after welding. The simulation results fits the expected distortion shape from the experience of real tests.

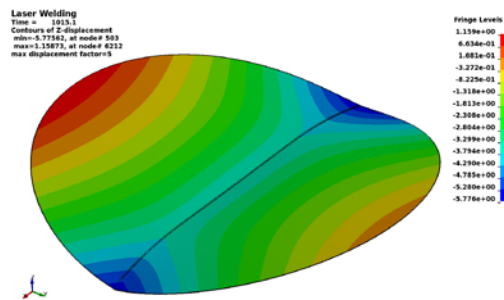


Fig. 7: Distortion normal to the plate surface after welding 5-times scaled

The residual stresses of this numerical experiment are plotted in Fig.8. A good agreement between simulation results and expected values evidence the maximum of longitudinal stress in the heat affected zone.

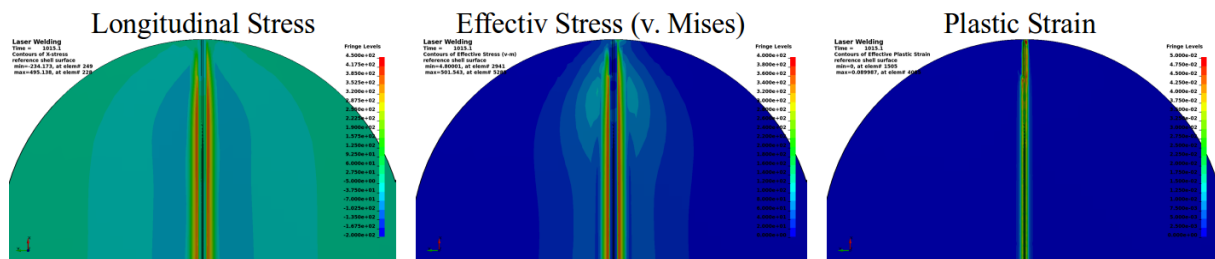


Fig.8: Residual stress and strain after welding

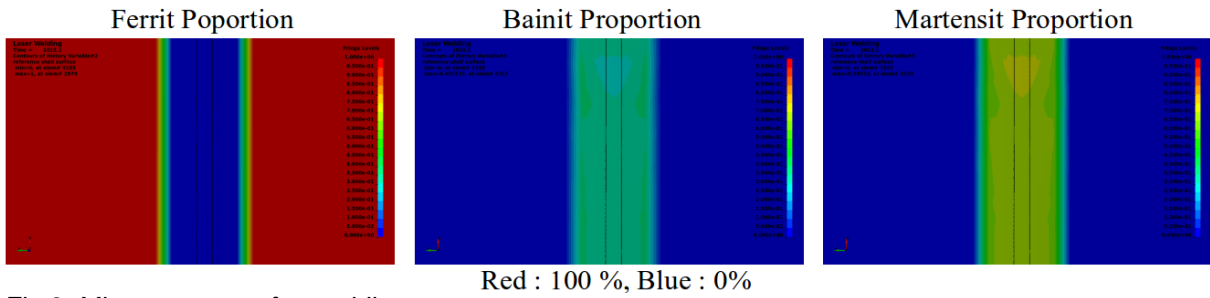


Fig.9: Microstructure after welding

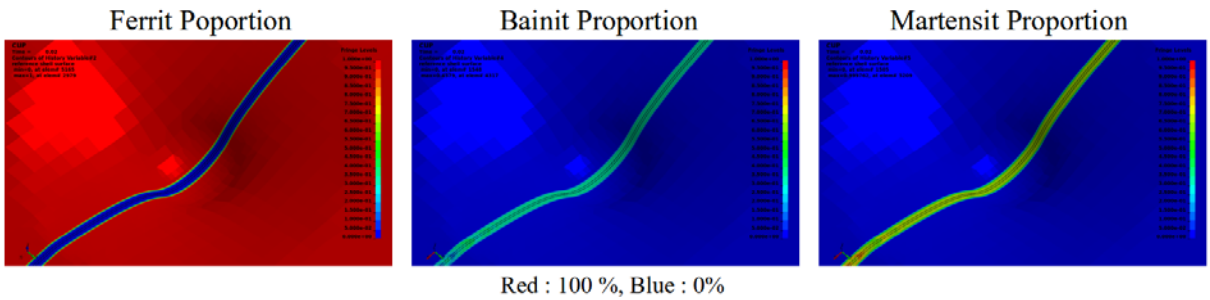


Fig. 10 Microstructure during forming

The change of microstructure in the weld and the heat affected zone is documented in Fig 9. The ferritic structure of the base material changes to an martensitic bainitic structure. The phase proportion from welding is kept in the forming or crash simulation as well (Fig. 10). This means the material inhomogeneity is taken into account. The martensitic bainitic structure of the welded zone has a significant higher yield stress than the ferritic base material.

Effective Stress before Unclamping Effective Stress after Unclamping Plastic Strain after Unclamping

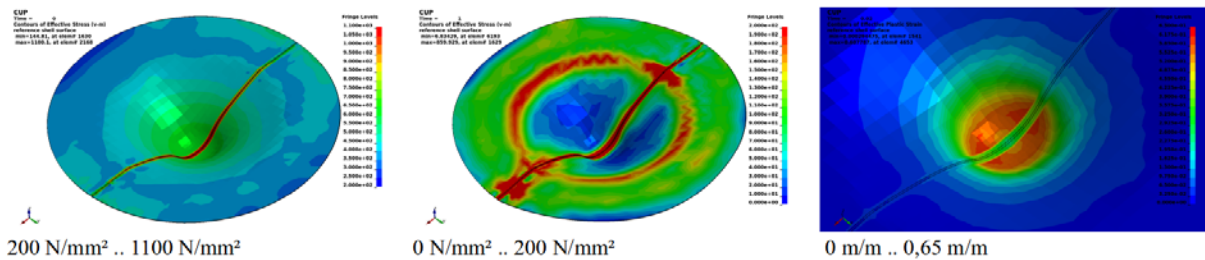


Fig. 11: Residual stress and strain after forming.

In the results of stress and strain (Fig. 11) the effects of the microstructure change are clear to see. The high yield stress of the weld leads to high residual stress. The microstructure change also causes lower plastic strain because of higher yield stress. Additionally the thinning is inhomogeneous too. The welded zone with its higher resistance has a lower thinning than the rest of the sheet (Fig. 11).

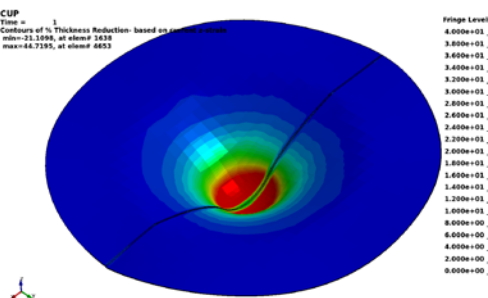


Fig. 12: Thinning after forming.

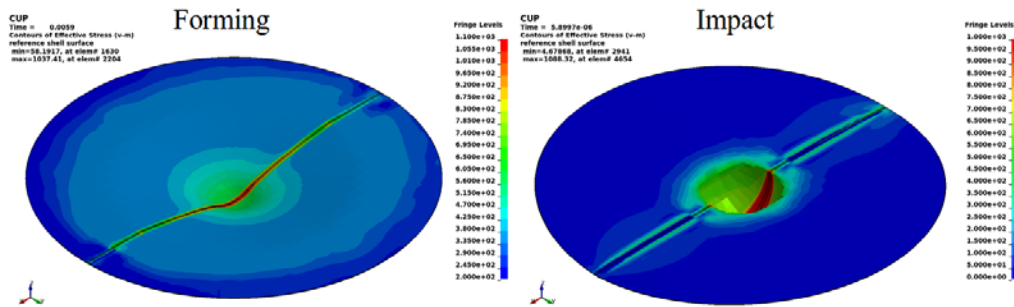


Fig.13: Effective stress (v. Mises) at same penetration depth - comparison between forming and impact.

The influence of the welding process is recognized in the impact analysis as well as in the forming analysis (Fig. 13). The kinetic behaviour and the deformed shape are very different between both types of loading. However the effects - stress distribution, plastic strain and thinning - are similar.

6 Distortion Compensation by Predeformation

Another modelling technique to join shell meshes from forming in a welding analysis is to model the filler material with solid elements and tied them to the shell mesh. In case of tied contact is used LS-DYNA considers the gap between solid surface and shell midsurface (Fig 15).

The tasks of the example are:

Rolling:

- A 1500 mm long sheet (S355) with 1 mm wall thickness is rolled to a groove.
- assembly:
- assemble the groove from forming with the ground plate (1 mm) the filler and the clamps.

Welding without predeformation:

- Weld the ground plate is welded to the groove.

Welding with predeformation:

- Predeform the groove in the middle in vertical direction.
- Weld the ground plate to the groove.
- Unclamp

***MAT_UHS_STEEL (*MAT_244)** is used to consider the phase transformation of S355 in the simulation. In Fig. 14 tree steps of the forming process are illustrated: before during and after rolling.

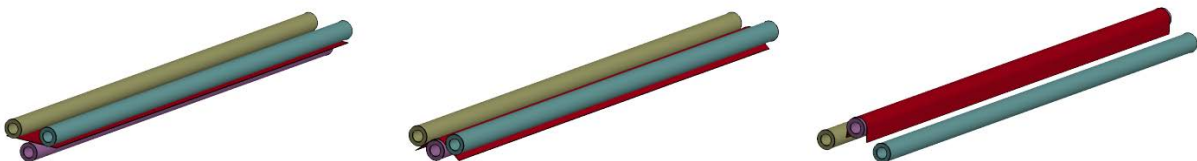


Fig 14: Forming of the groove

The assembled model is shown in Fig15. The groove contains shell elements with its history from the rolling process. The plate is modelled with shell elements as well and the filler consist of solid elements.

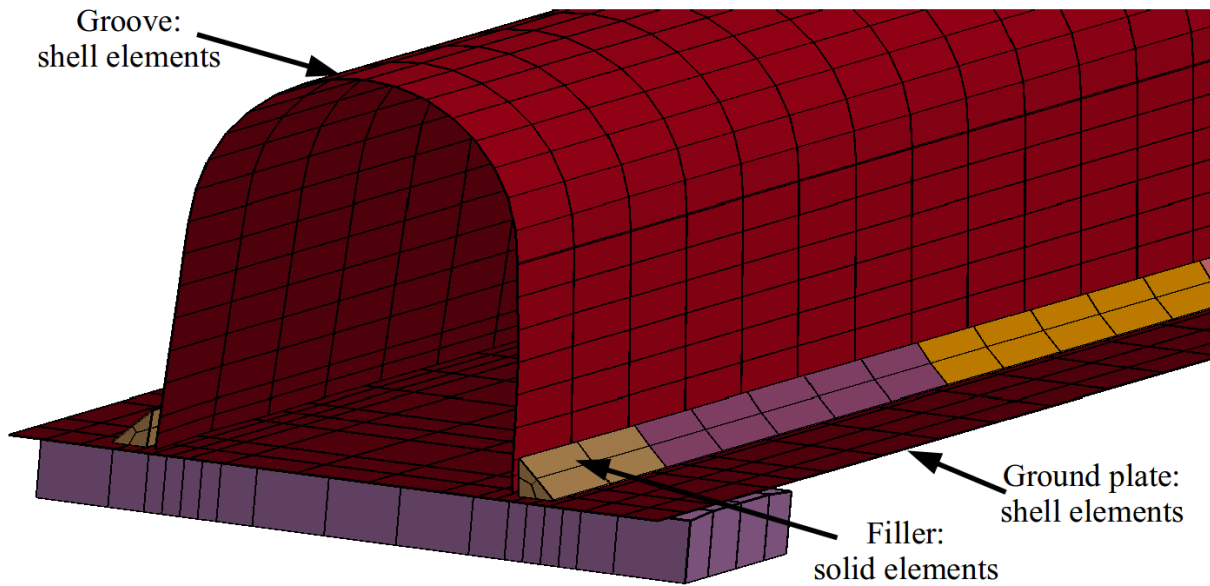


Fig. 15: Preparation for the welding step - mixed shell-solid model

To save computation time the meta-transient method was chosen for welding analysis. Meta-transient means, that not the moving heat source is considered but the heating and cooling of the weld on a larger section. Two tack welds, each on the end of the profile, are welded at the beginning. In Fig. 15 you can see the temperature distribution for the applied meta transient method. The first effect of welding is the distortion downwards caused by heating. During this the material in the molten or mushy zone is compressed. Thus the upward distortion during cooling is larger and a final distortion upward occurs.

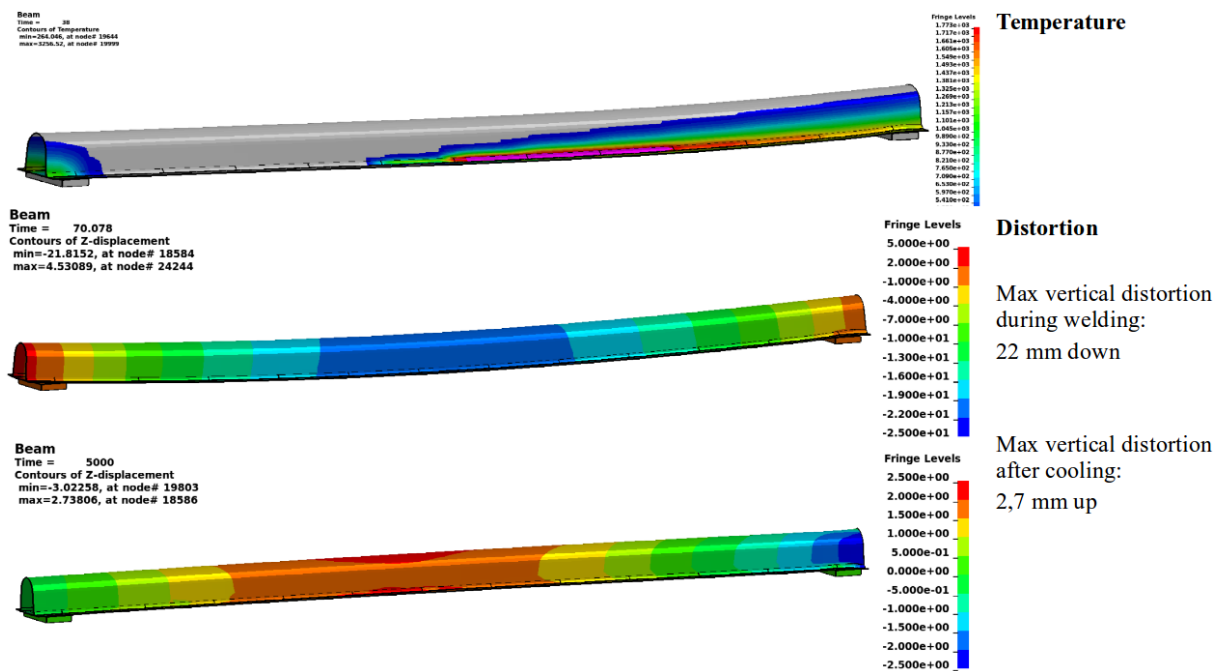


Fig. 16: Temperature distribution by the use of meta-transient approach. Vertical distortion during welding and after cooling.

One possibility to compensate the distortion is the forming of the negative distortion shape. During forming the inverted distortion from the welding process is considered and applied on the final shape. In this example the groove should have an arc rise of 2.7 mm downwards.

The other possibility is to consider a predeformation during welding. As shown in Fig. 17 above before welding the groove is predeformed downwards. With simulation its possible to design the correct measure of the predeformation to achieve zero distortion in the middle of the beam (Fig. 17 down).

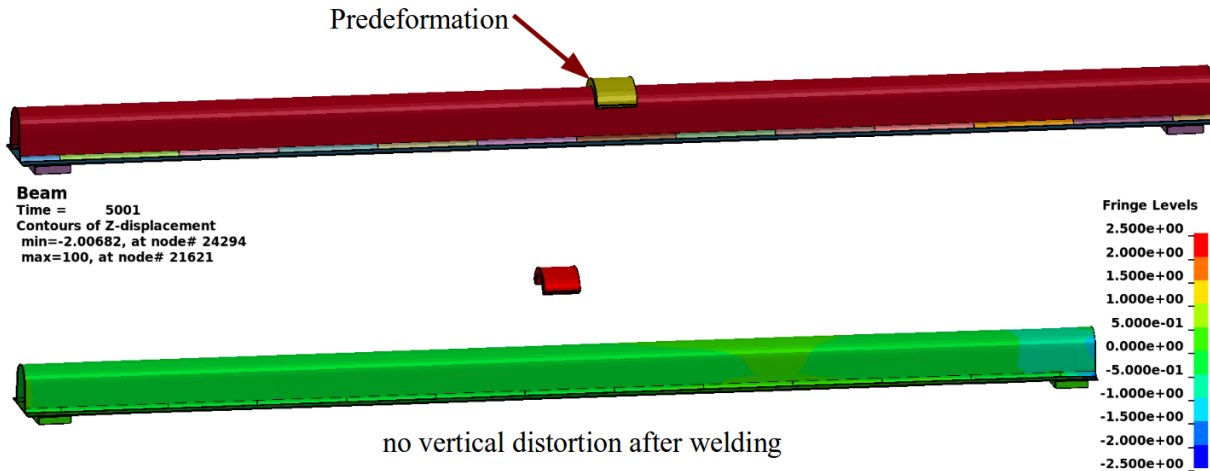


Fig 17: Distortion compensation by predeformation

7 Summary

The manufacturing process comprises several different steps.

These several steps interact and may influence each other.

For a realistic simulation of the manufacturing process results of previous simulation steps has to be taken as initial conditions.

The finite element code LS-DYNA provides the feasibility to simulate the manufacturing steps: forming, assembly, welding, heat treatment, post weld heat treatment, grinding, crash

- in one code
- with continuous data structure
- with continuous material model and continuous history variables
- take into account material property change
- without loss of information by mapping

Shell-, solid- and mixed shell-solid models can be used

The analysis can be performed implicit, explicit or mixed implicit-explicit

The feasibility of a realistic simulation of the manufacturing process, enables a reasonable design of the single processes to get the right final state and right final form of the assembly.

Thus LS-DYNA is a suitable solution for the simulation of the process chain to simulate complex manufacturing processes.

The one-code strategy of LS-DYNA helps to save license costs.