

Springback Calculation of Automotive Sheet Metal Sub-assemblies

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Abstract

In recent years the spring back calculation of a sheet metal part after forming has achieved a high accuracy. Today we are able to calculate the spring back amount after all forming operations of the part including trimming, piercing and reforming. But the spring back of a single sheet metal part is only the first step to solve the problem. What matters at the end is the spring back of the assembly or the sub-assembly of multiple sheet metal parts.

This paper describes a GUI for an efficient setup of a spring back calculation of multiple sheet metal parts, taken into account the complete forming history of the parts. It will be shown how to position the sheet metal parts, how to define an assembly sequence and joining method and the hemming process of outer and inner panels and how to launch the LS-DYNA[®] simulation.

Motivation

Springback simulation and compensation of springback for automotive panels can be achieved today. Commercial software is able to calculate springback with a sufficient accuracy and it is just a question of how much effort has to be spent to get the correct result. But a hood outer and a hood inner both within dimensional tolerance does not automatically lead to a hood which fulfills dimensional accuracy. Unfortunately the flanging of the parts, the different stiffness and the restraining parts influence the dimensional accuracy of the component. The goal is to calculate the springback behavior of the component to get an idea of the dimensional accuracy requirements of the individual parts.

There are several strategies to do a springback simulation of an assembly:

Strategy 1:

Complete springback simulation of individual sheet metal parts. Define assembly with clamping and joining of parts, hemming simulation and finally springback simulation of the assembled component

Advantage: Close to reality

Disadvantage: Huge effort, needs time to setup and to calculate

Strategy 2:

Drawing simulation of the main individual parts, no springback simulation of individual parts, define assembly, join individual parts, hemming simulation and finally calculate springback.

Advantage: No clamping necessary, simple to build assembly

Disadvantage: Doesn't work with compensated tools

Usage: Early estimation of springback of component, useful for influence and parameter studies

Strategy 3:

Use scanned data of parts, define assembly, clamp and join parts including hemming and finally calculate springback

Advantage: No simulation of individual sheet metal parts necessary

Disadvantage: Problems to get a good mesh of scanned parts, constant thickness and constant strain distribution is assumed

Basically each of these strategies has its advantages and its disadvantages. Depending on the time of the calculation with respect of the development time, only one of the methods can be used, e.g. in an early development stage a full springback simulation with compensated tools of each of the parts does not exist. Therefore one possibility is, to start with the most simple strategy 1: simulation of the drawing, cutting and reforming process of the main parts without springback simulation, definition of assembly, joining of parts, hemming and springback simulation of assembly.

The GUI should of course be able to handle each of the strategies. What are the requirements for such a GUI?

- Read in geometry and results of different parts
- Transform and position these parts
- Define assembly sequence including clamping and joining method
- Define clamps and joints
- Define hemming of inner and outer parts
- Calculate springback of component

This paper presents a first implementation of such a GUI.

The GUI

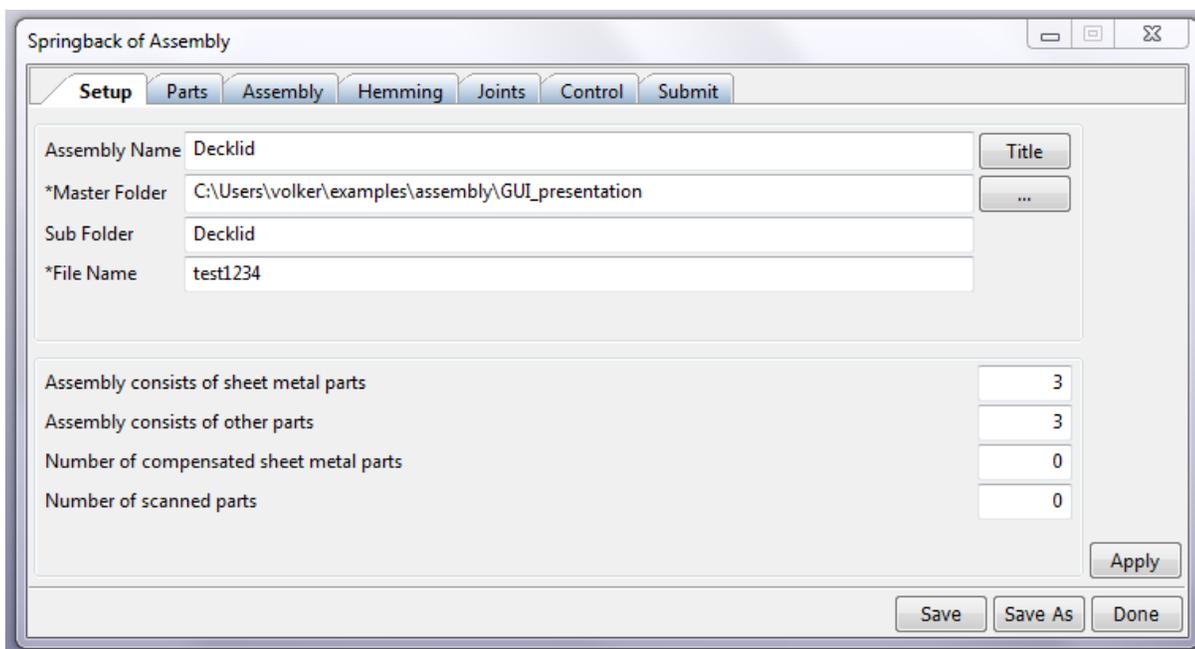


Fig. 1: First page of GUI for assembly and springback calculation of assembly

The Setup page of the GUI allows one to input assembly name, file and path names. Furthermore the GUI asks for number of parts:

- Sheet metal parts are parts for which a simulation file exists (dynain format)
- Other parts are parts for which only a mesh exists.
- Compensated sheet metal parts are parts which deviate from nominal geometry
- Scanned parts are scanned after springback

After input the relevant number and pressing Apply, GUI will automatically create the respective number of input fields.

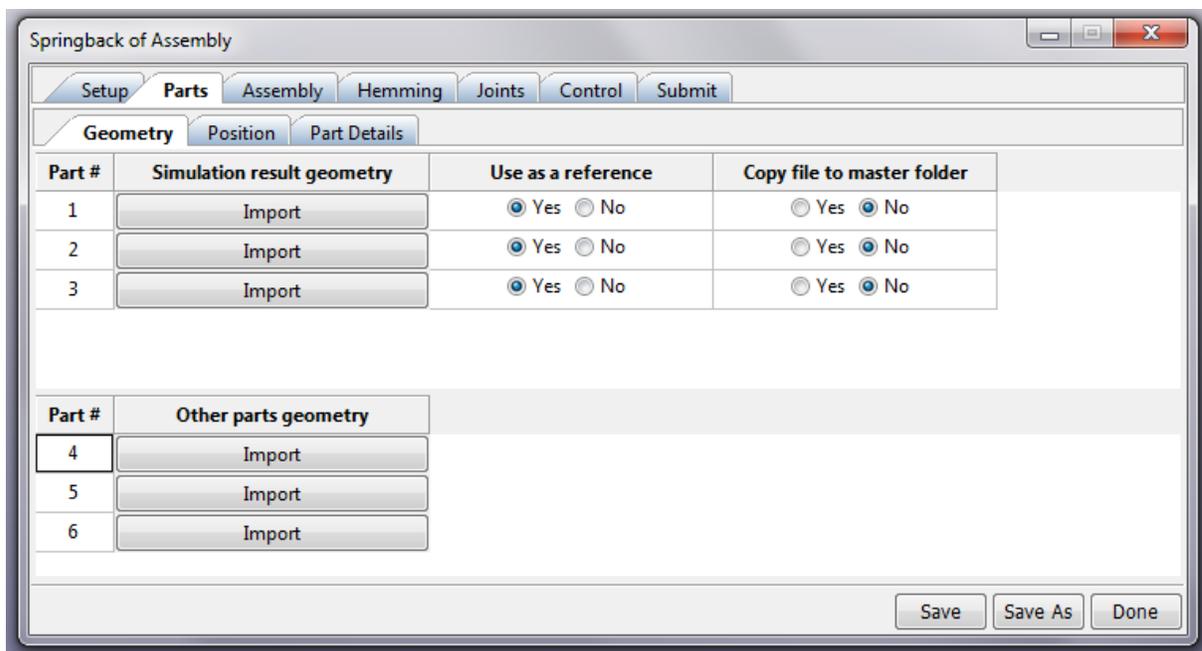


Fig. 2: Parts > Geometry page

Fig. 2 shows e.g. Parts > Geometry page with 3 import fields for sheet metal parts and 3 import fields for other parts, according to the input on Setup page.

Simulation result geometry is read in in dynain-format. GUI automatically gives a PID and automatically rennumbers element and nodes. User doesn't have to bother with meshes anymore. The PIDs are numbered in structured way so that the parts can easily be identified, e.g. first imported part gets PID 1100, second PID 1200 and third PID 1300.

After import of parts, input field contains file name of imported geometry.

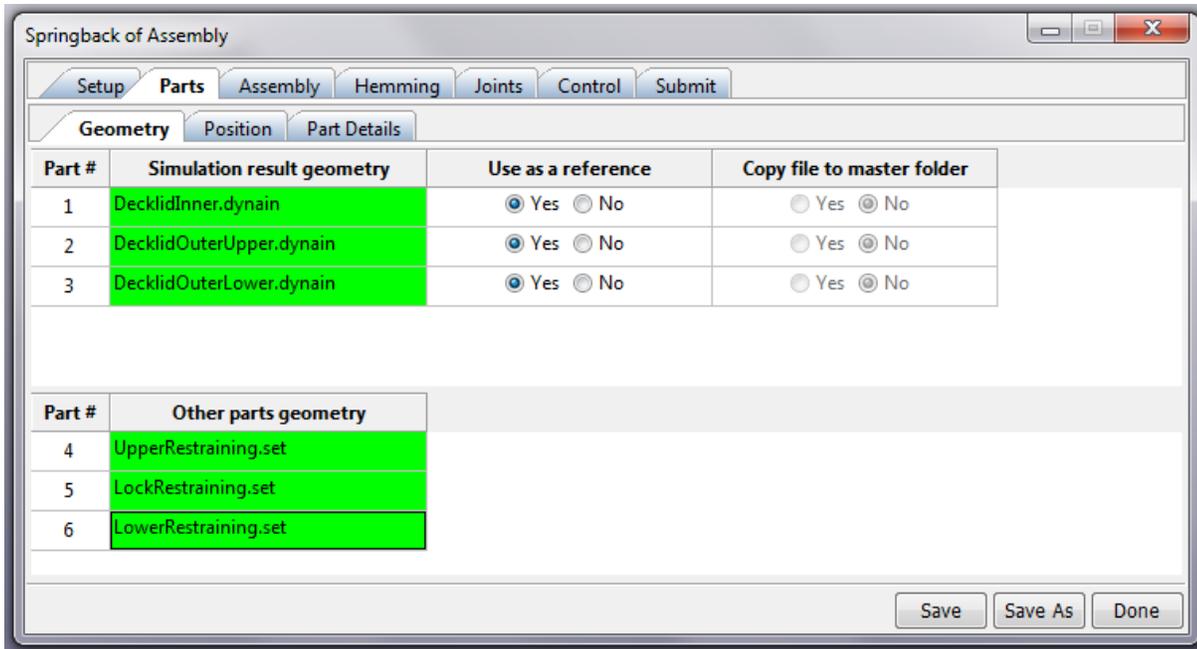


Fig 3: Parts > Geometry page after import of files

Use as reference can be used if imported geometry is nominal geometry.

Copy file to master folder can be used to copy the geometry file into the master directory. This might be important if user wants to have all files which are relevant for his project stored in one directory.

If result of simulation is read in, these geometries are normally in press position whereas the assembly is done in either part position or in a special position for assembly. The transformation of parts is done on Parts > Position page.

GUI allows one to define the transformation of each part to position it to car or to assembly position. Transformation info is rotation angles for each axis and translation into each direction. Values can be input manually or can be imported from CAD system. The supported format is a dyna keyword format for transformation. Fig 4. shows the Parts > Position page.

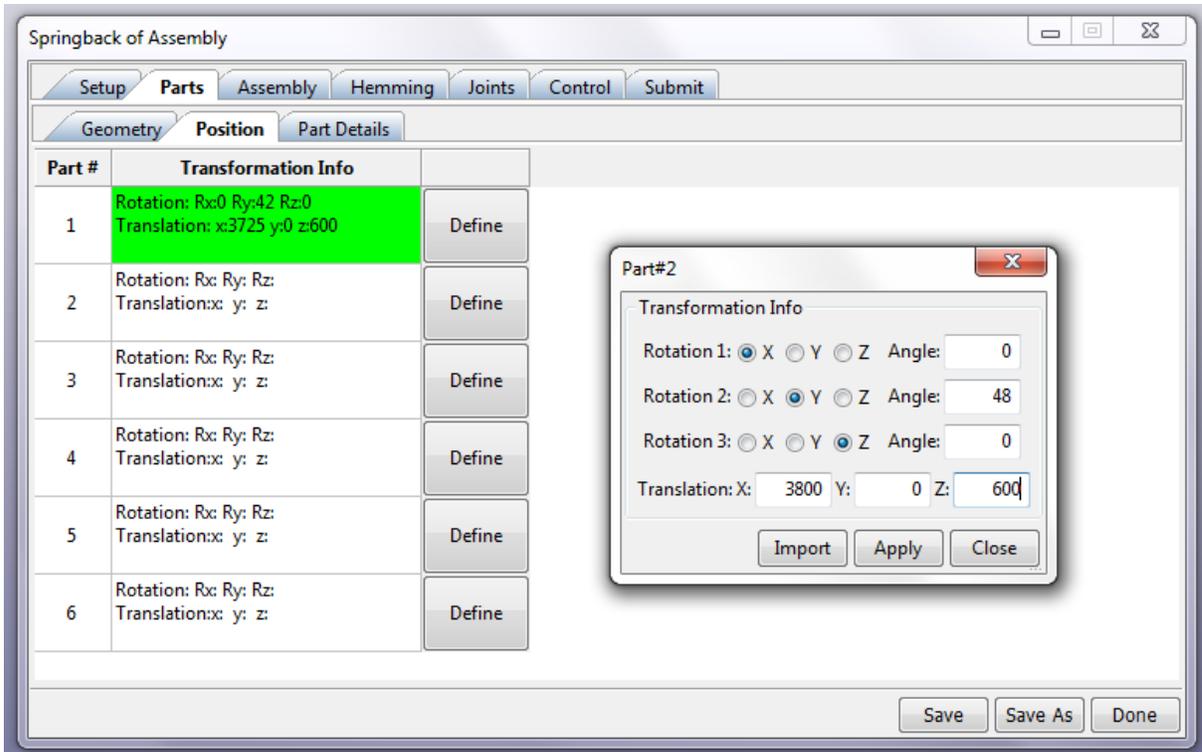


Fig. 4: Parts > Position page

Parts > Details page allows to input part names and material IDs.

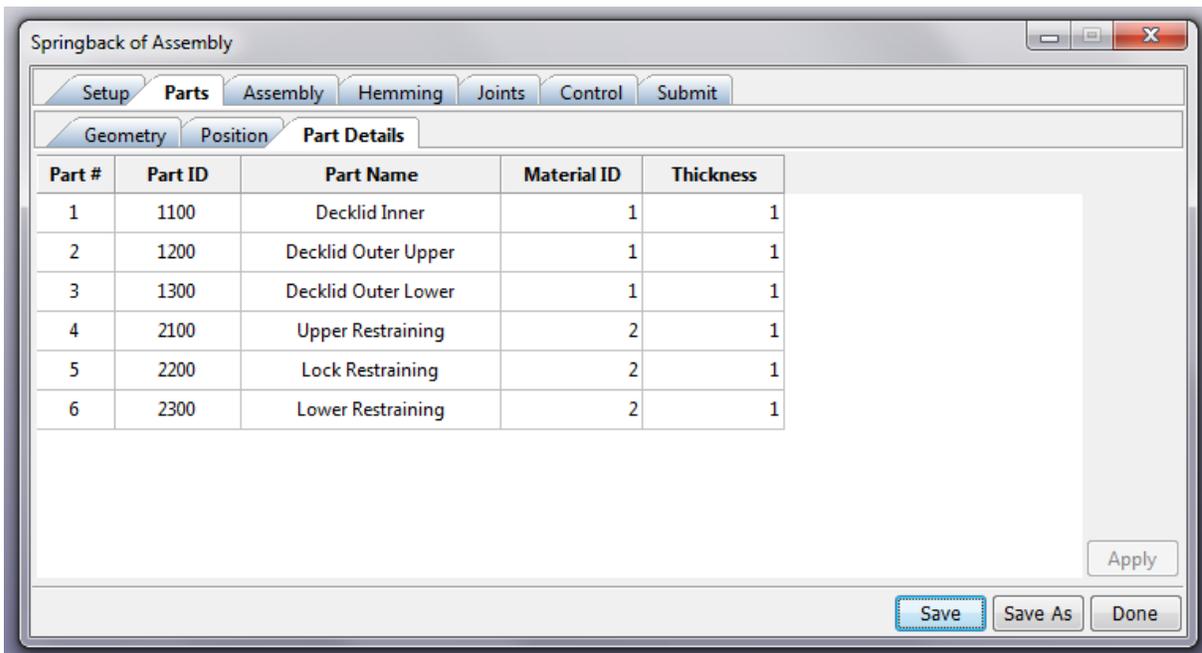


Fig. 5: Parts > Details page

If all files are imported, named and transformed to correct position, an assembly has to be defined. Defining an assembly means to specify the master part to which all other parts are joined, to specify the assembly sequence, joining method and position of joints and, if necessary, the number and position of clamps. This definition is done on Assembly page.

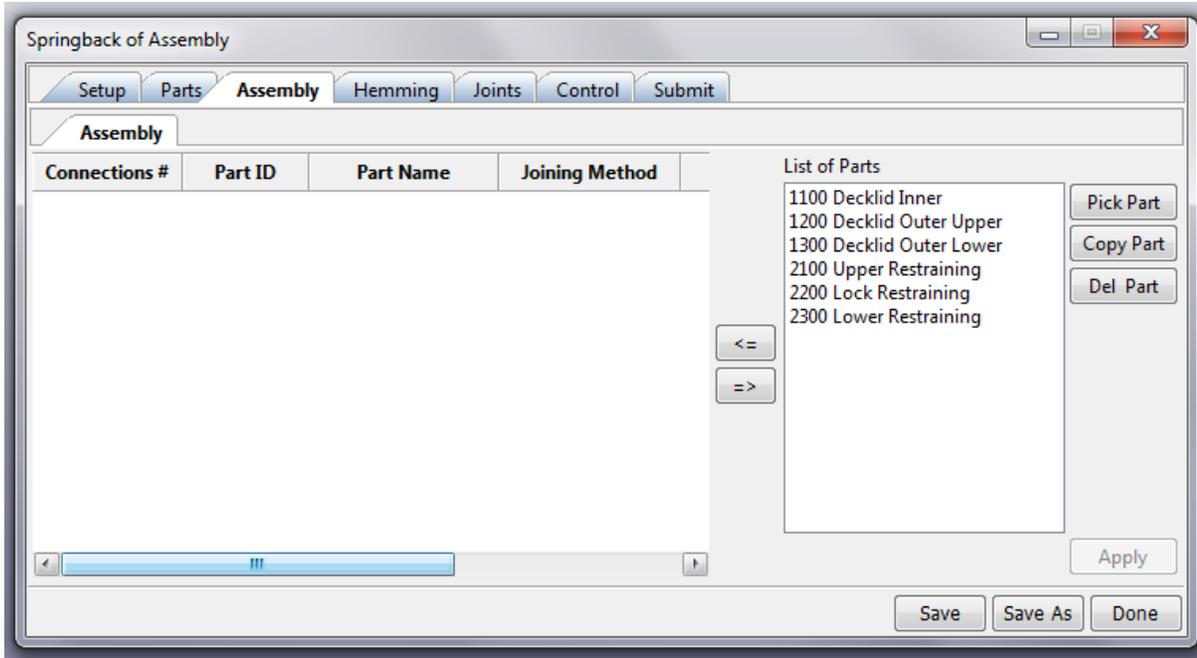


Fig 6: Assembly page

On Assembly page all imported geometries are listed in the List of Parts window. To build an assembly the first part is highlighted and moved to the Assembly section. The first part is the master part.

For the second part that is moved to Assembly section, the part to which it is assembled (Part ID), Joining Method, Number of Clamps and Number of Joints have to be specified.

Joining Method can be:

- Spot Welding
- Fusion Welding
- Clinching
- Laser Welding
- Hemming
- Bonding

Depending on joining method, the input on joints page will be automatically defined.

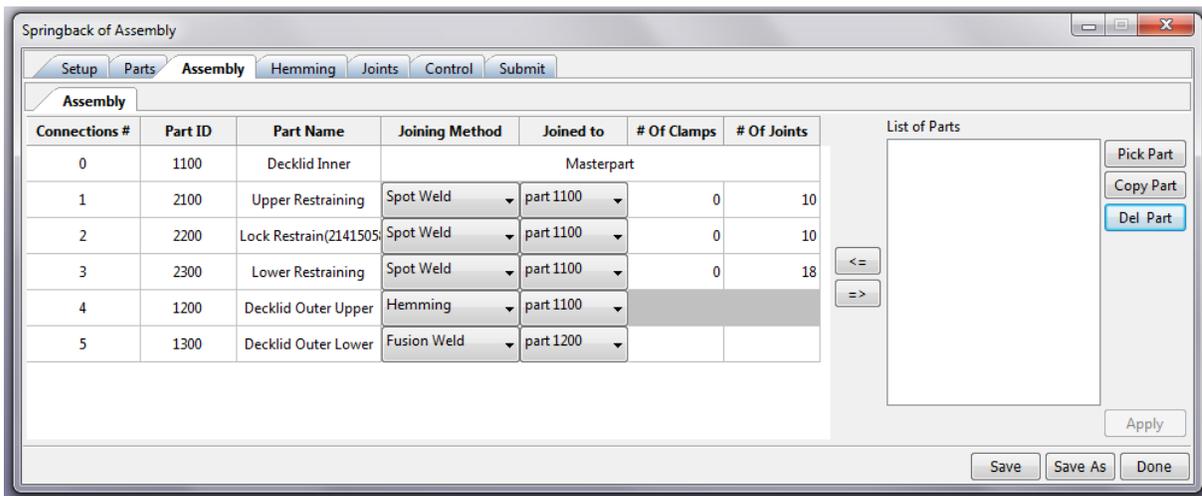
Fig 7 shows the Assembly page after the definition of the assembly.

No clamps are needed for this component because geometry of the sheet metal parts corresponds to nominal geometry. No springback was calculated for these parts.

In this example, decklid inner is the master part. The 3 restraining parts, Upper restraining, lock restraining and lower restraining are joined to decklid inner (PID 1100) with spot welds. Number of spot welds is 10 for upper and lock restraining and 18 for lower restraining.

The decklid outer consists of two parts, decklid outer upper (PID 1200) and decklid outer lower (PID 1300). These parts are joined with a fusion weld before the decklid outer is hemmed to decklid inner.

Assembly page allows a very simple and efficient way to define the assembly sequence of the component.



g. 7: Assembly page after definition of assembly

After assembly is defined, the position of the joints has to be defined on Joints page.

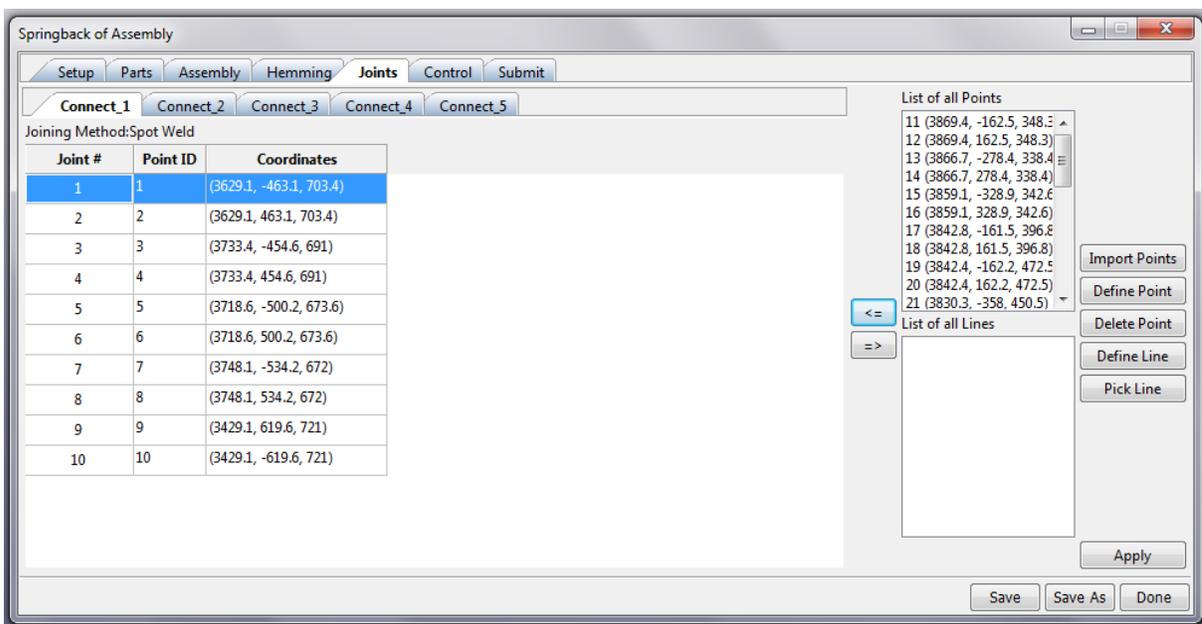


Fig. 8: Joints page

Joints page allow one to import coordinates of all joints and these points are displayed in List of Points. GUI has created automatically for each Connection the relevant number of input rows for joints. Joints can now be highlighted and moved to the right connection.

If a point is highlighted it is displayed in the graphic area on the part geometry. This makes it very simple to identify the right spot weld points for each connection.

For joining methods fusion weld, laser weld and bonding a line has to be defined.

For the specification of the hemming process of decklid inner and decklid outer Hemming page is used.

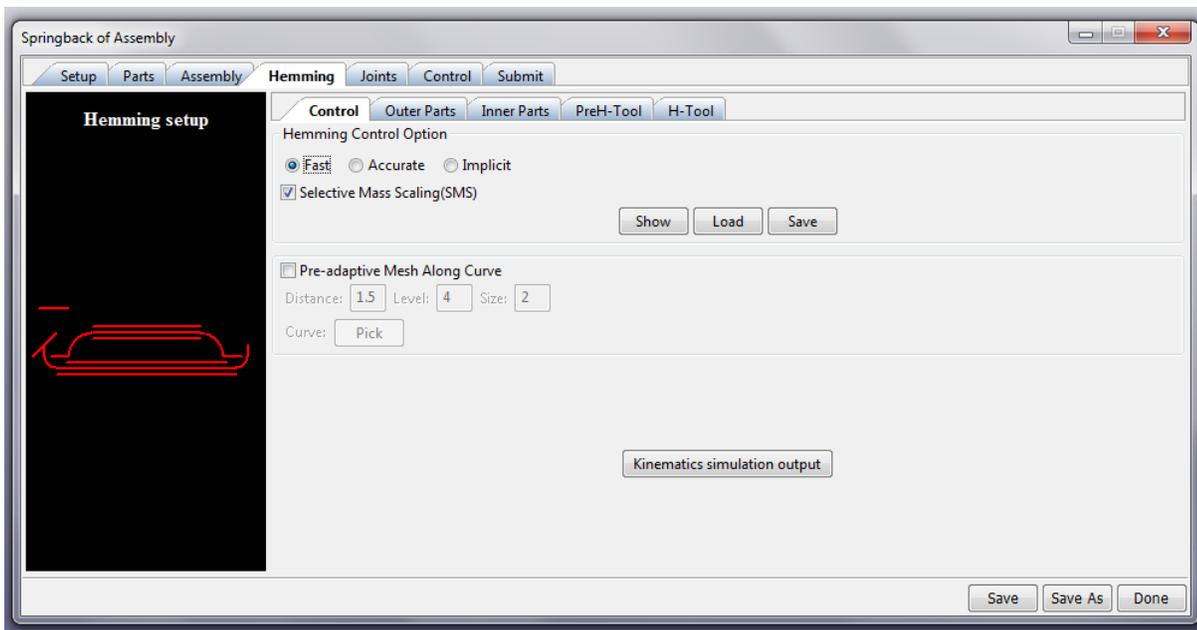


Fig. 9: Hemming page

This page allows an efficient setup of a table hemming process.

Control tab allows one to specify accuracy and adaptive mesh parameters, which is very useful to speed up simulation.

The other tabs allow the input of the parts and the tools that are necessary for the hemming process. On Outer part page the outer sheet metal part is specified as well as the supporting tool for the outer part.

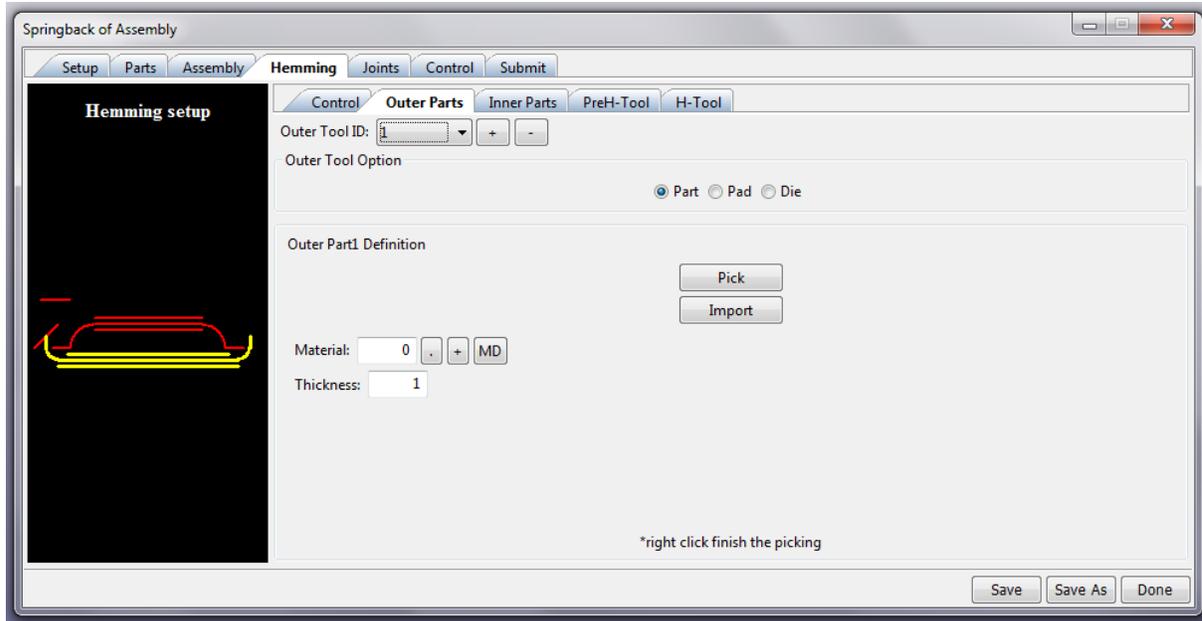


Fig 10: Hemming > Outer part page

The expected input is highlighted on the left side and part, die or pad can be clicked and defined.

Same input tab exists for inner part.

PreH-Tool tab allows the definition of the pre hemming steels and H-Tool tab allows definition of hemming tools. GUI assumes a two step hemming process with one pre hemming and a final hemming step.

After definition of hemming process all input is done, a Dyna input file can be created and the simulation can be started.

Expected Results are:

- Clamping forces (if clamps are used)
- Final dimensional accuracy of component (springback)
- Parameters that influence the final dimensional accuracy

Solver must be able to handle different parts with different joining methods, e.g. spot welding, clinching, welding and laser welding, hemming with bonding. And solver must also be able to simulate different materials for the parts incl. aluminum, roll formed parts, hot formed parts, plastic parts and adhesives.

Summary

This paper shortly describes a first approach for a GUI for definition of an assembly and springback calculation of assembly. The goal of the GUI is to allow users to specify a very complex process with different inputs in an easy and efficient way.

This GUI is still under development. Meanwhile, a lot of requirements for additional functionality exist.

Springback calculation of components just starts and this GUI might help to implement this method in a productive environment.