Use of virtual process simulations for stamped parts with extremely high geometric complexity

Key components of fuel cells are bipolar plates, which ensure uniform gas distribution and water removal.

In addition to fluid dynamics tasks, uniform cell contact is crucial: contact losses within the stack increase electrical resistance and lead to power losses in the fuel cell. Today's bipolar plates measure up to 500 mm x 350 mm, which is quite small compared with large automotive parts. Their complexity arises from the fine channel structure, embossed into the sheet metal. The FEINTOOL demo plate shown at the bottom right illustrates these geometric details.

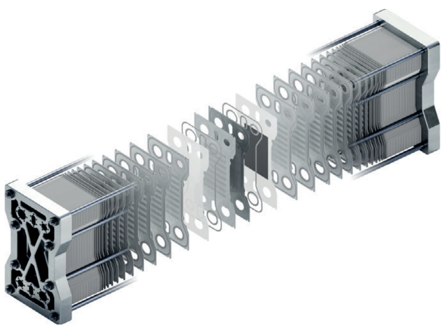
The increasing complexity of channel structures requires extremely fine finite element meshes in the forming process simulation. With sheet thicknesses and channel radii of only 0.1 mm, models comprising up to 25 million elements are created – even when thick shell elements are used. The use of 3D volume elements would further increase the number of elements and make industrial application practically impossible.

Industrial Applications

In collaboration with FEINTOOL, inspire investigated the applicability of Finite Element Method (FEM) for the realistic virtual representation of complex sheet metal forming processes. Among the available tools, the commercial simulation package AutoForm was the only FEM solution suitable for industrial application.

The entire process flow was fully simulated and essentially includes:

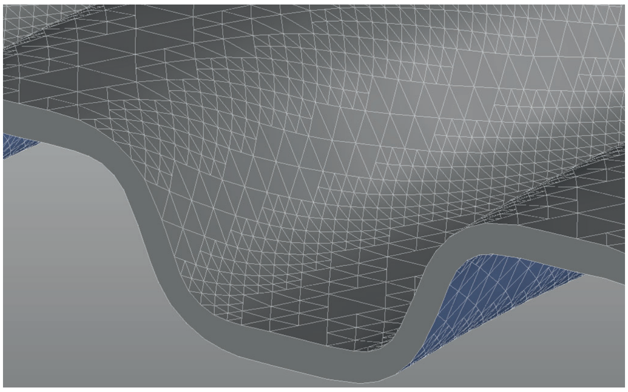
- Tool and blank meshing
- Forming and springback in two stages
- Analysis of part handling between stages
- Trimming operations and prediction of the final geometry



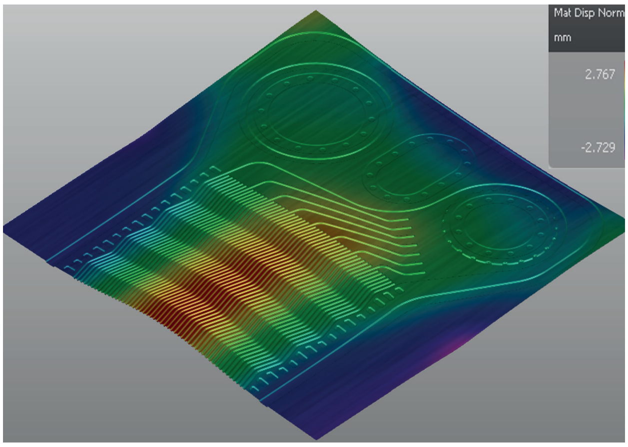
Bipolar Plate Stack
A fuel cell can contain up to 800 bipolar plates.
(Source: FEINTOOL)

Example of a Bipolar Plate
(Source: FEINTOOL)

The thinning distribution and the distance to critical strain limits, Forming Limit Curve (FLC) / Crack Limit Curve (CLC), were evaluated. Local embossing and thermally induced springback reduction were used to optimize the process. A specialized testing tool was developed to accurately determine the actual failure limits at small radii.



Deflection Simulation
Accurate representation of springback allows for the targeted optimization of both tool and process design, reducing geometric deviations in the finished part.



Finite Element Mesh
Local mesh refinement using thick shell elements for more accurate representation of complex geometries and loads.

With the testing tool, the simulation could be checked and optimized. This provides a simulation methodology that allows FEINTOOL to carry out well-founded feasibility studies in the shortest possible time.

Summary

Through the targeted use of AutoForm software, a complex forming process was successfully simulated and designed using FEM. The chosen approach enables robust, efficient, and application-oriented optimization of the complex bipolar plates and leads to a significant reduction in tool manufacturing costs.

Dr. Bekim Berisha
Head of the Material Mechanics and Simulation Group

Prof. Emeritus Pavel Hora

The research presented was conducted in collaboration with Prof. Emeritus P. Hora, ETH Zurich.