Ansys Forming® - The New GUI for Forming Simulations with LS-DYNA: An Overview and Outlook

Chris Robinson[1], Pablo Hernandez Becerro[1], Kang Shen[1], Xinhai Zhu[1], Madhu Keshavamurthy[1], Volker Steininger[2]

[1]Ansys
[2]Tiwa Quest

1 Why we do what we do

We are motivated by being the engine behind human innovation that leads to more efficient manufacturing and digital engineering. As we lead out in providing world class simulation solutions, we provide an avenue for individuals and companies to develop products that can improve the quality of life across the globe, by enabling safer more efficient products to be brought to market at a lower overall cost and energy footprint in less time than using traditional methods. In short, digital engineering solutions help improve life, save the environment, and reduce costs.

2 How we do it

Sheet metal stamping is globally one of the most used manufacturing methods in today's world in the automotive and many other industries. Leveraging best-in-class solvers to explore the stamping process digitally before ever cutting any tool steel can drastically help in accelerating time to market and reduces the overall cost of developing a new product. Simulation not only helps understand material thinning, wrinkling, springback and other related process impacts, it also helps to drive die design and can help with overall system integrity validation, such as using forming results in an auto crash simulation to improve the accuracy of safety decisions up front.

3 What is Ansys Forming®

Ansys has created Ansys Forming®, a tool which speaks the die designer language, hides all FEA complexities, and is powered by the highly accurate, scalable, and performant LS-DYNA solver to address sheet metal stamping needs at any stage of the product development cycle. Our aim is to enable users to easily and efficiently set up forming simulations, start the analysis, and analyze the results in an environment that uses the language that process and method engineers speak. It is a multi-stage sheet metal stamping prediction tool that allows users to create full process simulations, including blanking, drawing, trimming, clamping, springback, compensation, and other capabilities.

4 New Features

While Ansys Forming already has basic capabilities that allow users to set up multi-stage forming process validation, manage and use existing material libraries, define the specifics of the process, setup and view tools, define drawbeads and lancing operations, and analyze results from the simulations, there are also many new features being added at a very rapid pace to Ansys Forming. Several of the most important feature additions are listed below and many more features that add value will be added in the years to come.

4.1 Springback Compensation of the Draw Die

If stamped sheet metal was always only plastically deformed and there was no elastic deformation at all, then the overall process would be much easier to deal with, but that is not the reality of most sheet metal stamping scenarios, especially in high production environments. Understanding and overcoming the springback is absolutely critical to capturing key benefits of digital engineering in stamping. Ansys Forming and the LS-DYNA solver have proven to be very effective at accurately predicting the springback. With the next release of Ansys Forming, users will also be able to automatically and iteratively compensate the draw die for the predicted springback and validate, via simulation, that the modification to the die will produce the desired result. The compensation as well as trimming curve

updates are performed automatically for the operator who chooses to use this feature, saving time and effort.
Users will have the ability to determine the max number of iterations, the tolerance of the part and which side of the tool will be compensated. To keep the simulation mesh and digital process under control and to accelerate convergence, users can also set a scale factor that will be used to compensate on each iteration. To help to represent reality, users have the ability to use different coordinate systems for forming and for clamping. Users will be able to view each iteration and track results to help assess the design.

4.2 Trimming Curve Development

Efficiency in production comes through being able to minimize operations in the overall process. Effective trimming is key to minimizing additional operations. The Ansys Forming trim curve development feature allows users to set tolerance limits and number of points that must be within tolerance and then allow the software to automatically define trimming curves. This automatic iterative process allows users to be more efficient and get through more new part developments, enabling them to keep up with the ever increasing need for faster development cycles. The newest version of Ansys Forming will feature adaptations to the trimming curve development feature that will improve convergence and make it even more efficient. The new shift feature will move the trimming curve in space, considering the difference in the center contour of the final panel and the target. The iteration optimization process can then continue to optimize the shape of the trimming curve. Users will also be able to choose to only develop a section of the curve, and will be able to define the beginning and ending points of the trimming curves. There will be additional controls in place to allow users to change the corrections made at the ends of the segments.

4.3 Hot Forming

We have primarily focused efforts in the past strictly on cold-forming, but with the next release of Ansys Forming, users will have the ability to also simulate the Hot Forming process without a cooling system. This entails modifications to the material editor and the ability to visualize thermal results from a thermo-mechanical simulation. Users can choose whether they are setting up a Cold Forming or Hot Forming process and the easy-to-use GUI will guide the user through the process of entering the appropriate information to simulate the given process. Blank, tool, and environment temperatures along with scale factors, convection and heat transfer coefficients, and thermal time steps are all options that users have to effectively represent the process. With the introduction of Hot Forming, we also have a convenient interface to define temperature dependent properties for materials, where users can easily visualize the different property curves.

4.4 Drawbeads

Controlling material flow through the use of drawbeads is effective and commonly used, but it is also very expensive to test and develop drawbeads only through empirical methods. Simulation provides a significant value through the option of digitally determining appropriate drawbeads. However, the fidelity of the prediction of forces due to drawbeads is crucial to minimizing the amount of empirical optimization for getting the “right” drawbeads. Ansys Forming will now have an interface to calculate the uplift and restraining forces with calculations that are more accurate than in past versions. Users will also have access to split their drawbeads into segments to represent the bead with appropriate fidelity. The GUI will provide the option for users to define parameters for three of the most common profile types, trapezoidal, round, and step profiles. Ansys Forming will then automatically run a 2D simulation to calculate the uplift and restraining forces.
One of the really exciting additions to the upcoming version of Ansys Forming is the added accuracy that can be obtained through the use of 3D Drawbeads. These 3D Drawbeads will now be able to be directly used in the tools. Still within the same Ansys Forming interface, users will be able to update their tool geometry to include the drawbeads. Users will be able to control the drawbeads to the tool’s boundary, include open drawbeads with user-controlled fades at the ends of the bead, and control transitions at the beginning and end of the drawbead or when adapting from one segment to another. These additions make it easy to increase the fidelity of the simulation and digital engineering approach that will lead to faster time to market.

4.5 Mesh Check and Repair

Mesh defects can have a significant impact on accuracy, efficiency, and robustness of simulations. It can be very time consuming to evaluate, identify and fix meshing issues. Ansys Forming has introduced a new Mesh Check and Repair feature that alleviates much of the time required to clean up a defective

mesh. Within seconds a user can have meshing defects identified and then they can choose to either automatically repair the mesh, which is also very quick, or they can choose to make manual modifications to the mesh directly within the Ansys Forming GUI. Gaps, holes, folding of elements, inverted normals, and other mesh issues can be quickly identified and easily repaired with this new feature that will certainly save time in getting a high quality mesh for high fidelity simulations.

4.6 Pre-processing Improvements

While all of the amazing new features cannot be called out specifically in this document, there are also several pre-processing improvements that have been made to Ansys Forming that will be a part of the upcoming release. These include features such as Scaling of the tool geometry, an upgrade to the material editor and a beta feature that can support and predict pressure and velocity dependent variable friction coefficients, defined by an algorithm. A significant improvement to the user experience is an enhancement to the CAD import functionality that allows a full CAD model to be imported rather than needing each surface and curve to be separate files. Users will also have the ability to have advanced meshing capabilities with blank meshing from surfaces, and adaptive refinement along boundaries and adaptive remeshing capabilities to support mesh quality and geometric accuracy. Solid element forming simulation will also be introduced in Ansys Forming as an initial beta capability.

4.7 Post-processing Improvements

The addition of enhancements doesn't end with all of the new features, meshing, pre-processing, and material improvements, there will also be improvements in the post-processing capabilities of Ansys Forming. Users will be able visualize the blank thickness and the tool offset in the post-processor. Additional controls and improvements to the Skid Mark analysis will allow users to set a tool radius and contact pressure limits for Skid Mark visualization to really help streamline the part surface quality evaluation and predictions.

5 Summary

Ansys Forming is proving to be an extremely viable tool for accelerating time-to-market for sheet metal stamping applications, which helps us all move our respective industries forward in an efficient manner. It is a very new product, and the pace of new feature development is providing users with powerful and easy-to-use tools to address very complicated forming process problems in a simple to use, but accurate package. The newest features will compliment the multi-stage forming capabilities with new capabilities such as Springback Compensation, Trim Line Development, Hot Forming, Drawbeads, Mesh Check and Repair, as well as other various pre and post processing enhancements. Ansys Forming will continue to evolve and beta features such as variable friction and solid elements will be fully released along with work on other features, such as compensation of line dies, Table Hemming and reporting. The development teams are also looking into high value features such as One-step and Die Face Design for future enhancements. Users today are enabled to innovate more rapidly through the adoption of Ansys Forming, and the future will only hold even more power to innovate for those who leverage Ansys Forming.