Enabling Productive Use of Isogeometric Shells in LS-DYNA

Lukas Leidinger[1], Stefan Hartmann[1], Frank Bauer[2],
Dave Benson[3], Attila Nagy[3], Liping Li[3], Marco Pigazzini[3], Lam Nguyen[3]

[1]DYNAmore GmbH, an ANSYS Company
Industriestr. 2, 70565 Stuttgart, Germany

[2]BMW Group
Knorrstraße 147, 80788 Munich, Germany

[3]ANSYS
7374 Las Positas Road, Livermore, CA 95411, USA

1 Introduction

Isogeometric Analysis (IGA) [1] is a novel Finite Element Analysis (FEA) technology based on splines known from Computer Aided Design (CAD). In an isoparametric sense, IGA uses the higher-order and higher-continuity spline basis functions, e.g., Non-Uniform Rational B-Splines (NURBS), not only to describe the model geometry, but also the solution field. This may yield a more accurate geometry description, higher solution accuracy and a larger time step in explicit analysis compared to conventional FEA.

With the goal to exploit these potential benefits in productive industrial use, the IGA capabilities in LS-DYNA have been successively extended and improved over the last few years with a focus on vehicle crash simulations and trimmed (multi-patch) NURBS shells [2]. A simple and convenient way to apply IGA in large-scale crash simulations is to replace certain components of existing FEA vehicle models with their IGA counterparts, resulting in hybrid IGA/FEA models. To enable such a 1:1 component exchange, LS-DYNA was enhanced to support the state-of-the-art FEA connection technology for spotwelds, seam welds, bonds and rigid bodies also for isogeometric shells. This includes a wide range of penalty- and constraint-based tied contacts and constraint definitions (Section 2). As will be shown by Bauer et al. [3] at this conference, LS-DYNA is now able to run crash simulations of full vehicle models with hundreds of isogeometric shell components.

This contribution shall provide more information about the underlying developments and capabilities that were necessary to achieve this. Another crucial aspect to be discussed is the possibility to use the existing elasto-plastic material cards including damage and failure for isogeometric shells, ideally without modifications (Section 3). Furthermore, recent developments such as initialization of isogeometric shells with mapped material history data, and the definition of multi-thickness and multi-material patches for tailor-rolled/-welded blanks are presented in Section 4 and 5, respectively. In Section 6, an outlook is provided to a more sophisticated, feature-based IGA modeling approach for a tight connection between the CAD and the simulation model.

2 Connection Technology for Hybrid IGA/FEA Modeling in LS-DYNA

An essential requirement for a simple implementation of IGA parts in existing FEA LS-DYNA models, is to support the state-of-the-art connection technology for spotwelds, seam welds, bonds and rigid bodies also for isogeometric shells. This includes penalty-based tied contacts, constraint-based tied contacts, **`*CONSTRAINED_INTERPOLATION_SPOTWELD`** and the attachment of **`*NODE`**s through **`*IGA_POINT_UVW`**. To understand the implementations for IGA, it is important to know that LS-DYNA automatically generates a linear interpolation mesh on top of the spline-based isogeometric shell surfaces that can be used for contact and visualization. The elements of this interpolation mesh are fully constrained to the IGA surface. Thus, they are not evaluated, they do not affect the critical time step size, but only map nodal forces to the IGA control points. This allows the application of existing penalty-based contact algorithms to IGA. In recent years, also penalty-based tied contacts were enhanced to work for IGA shells via interpolation meshes, for example **`*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET`**, see [4]. Similarly, for modeling of spotwelds **`*CONSTRAINED_INTERPOLATION_SPOTWELD`** was extended to act on the interpolation mesh of IGA shells [4,5]. Another common use case is the attachment of conventional nodes to IGA shells, for example from existing nodal rigid bodies or from existing node sets used in tied contacts. This can be done through the **`*IGA_POINT_UVW`** keyword, which associates a **`*NODE`** with a parametric point (in the UVW space) of the IGA shell [4,5]. Depending on its usage, the associated **`*NODE`** is either attached in a weak or a strong sense. The latter option is new and may improve accuracy in case the **`*NODE`** is used in a penalty-based tied contact.

What was missing so far, are constraint-based tied contacts because they cannot be applied to the already constrained interpolation mesh. Therefore, constraint formulations are now implemented such that nodes defined by SURFA are directly tied to the isogeometric shell surface defined by SURFB. Fig. 1 shows how **`*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET`** is used in combination with **`*IGA_POINT_UVW`** to model seam welds between two IGA shells and between an IGA and an FEA shell. Fig. 2 shows how **`*CONTACT_SPOTWELD`**, which is equivalent to **`*CONTACT_TIED_SHELL_EDGE_TO_SURFACE`**, is used to attach cohesive hexa solid finite elements to IGA shells. Here it should be noted that these constraint-based tied contacts for IGA are only available for the MPP version of LS-DYNA and that the contact keyword option **`_MPP`** in combination with GRPABLE=1 must be set.

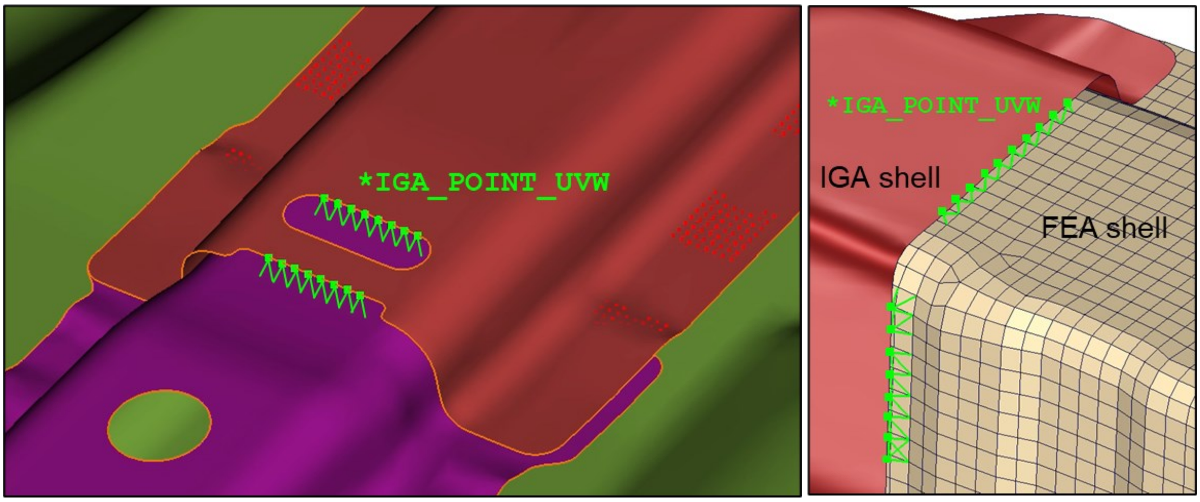


*Fig.1: Modeling of weld seams between two IGA shells (left) and between an IGA and an FEA shell (right) using **`*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET`** together with nodes attached to IGA shells via **`*IGA_POINT_UVW`** (schematic visualization, Courtesy of BMW Group).*

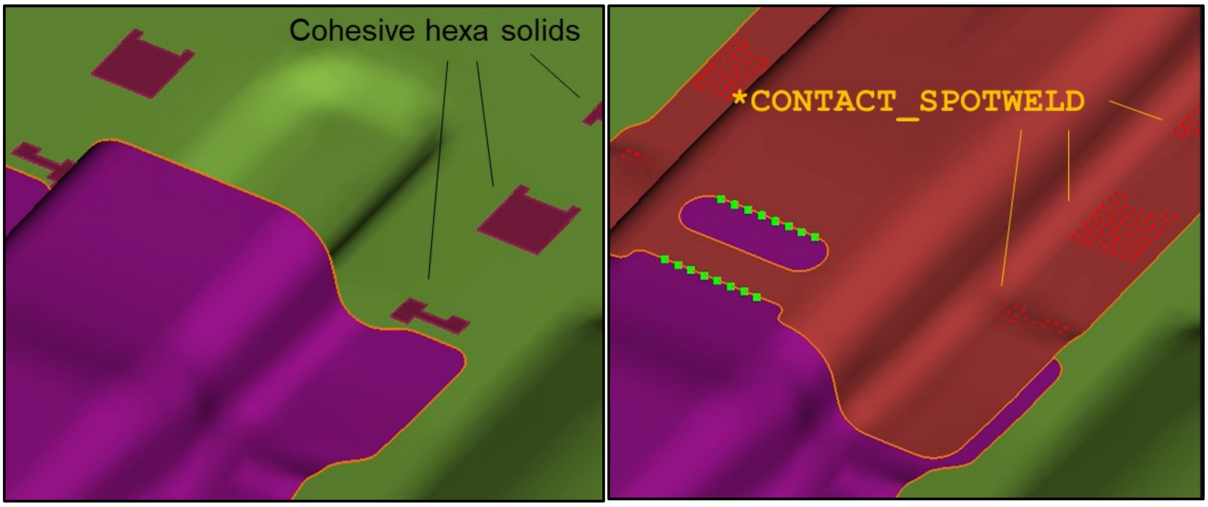


*Fig.2: Modeling of bonds between two IGA shells using cohesive hexa finite elements and **`*CONTACT_SPOTWELD`** (Courtesy of BMW Group).*

3 Improved Damage and Failure Modeling with DIEM

An important goal in the development of the IGA solver is to enable seamless portability of material cards, as well as the damage and erosion control settings, between FEA and IGA models. To achieve this, things like an element erosion scheme or characteristic element lengths for damage regularization also need to be implemented in LS-DYNA for IGA. Due to the inherent complexities introduced by higher-order and higher-continuity IGA shape functions, this is still considered an active area of research in the framework of softening damage models.

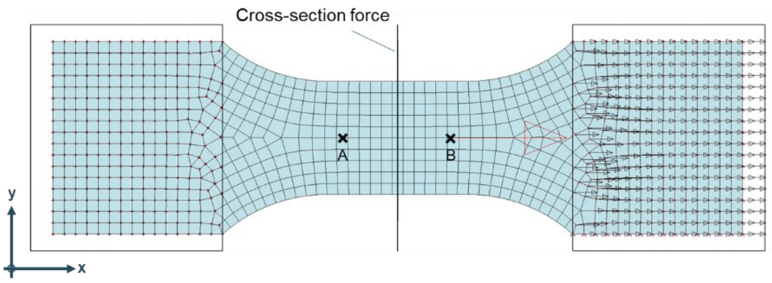
Here, we consider a standard tensile test example to show that an existing elasto-plastic material card (**`*MAT_024`**) including damage and failure (**`*MAT_ADD_DAMAGE_DIEM`**) can be directly used for IGA. As a reference, the results of a validated FEA model are used.
The FEA and IGA tensile test specimens with an average element length of around 2mm are depicted in Fig. 3. The FEA specimen is modelled with a quad-dominant mesh of linear Belytschko-Tsay shell elements (ELFORM=2). The IGA specimen is modelled with bi-quadratic trimmed NURBS shells using a Reissner-Mindlin shell formulation (ELFORM=3) with reduced Gauss integration (IRL=0). Both models use five integration points in thickness direction. To enable precise comparability, the boundary conditions (**`*BOUNDARY_SPC`** and **`*BOUNDARY_PRESCRIBED_MOTION`**) are applied to nodes and control points, respectively, collected via identical boxes for both models, as shown in Fig. 3. Damage and failure are modelled through DIEM (Damage Initiation and Evolution Model) [6] with a distinct damage evolution phase. For both models, the cross-section force is measured in the middle of the specimens, while the displacement difference is measured between the two points A and B, see Fig. 3. The simulations are performed with an MPP single precision version of LS-DYNA.

Fig. 4 compares the force-displacement responses of the FEA and the IGA model, both using the exact same material card. As can be seen, both curves are not only almost identical in the elasto-plastic phase until damage initiation, but also match well in the damage evolution phase.
In the following, two important aspects that enabled such a good accordance are discussed in more detail.

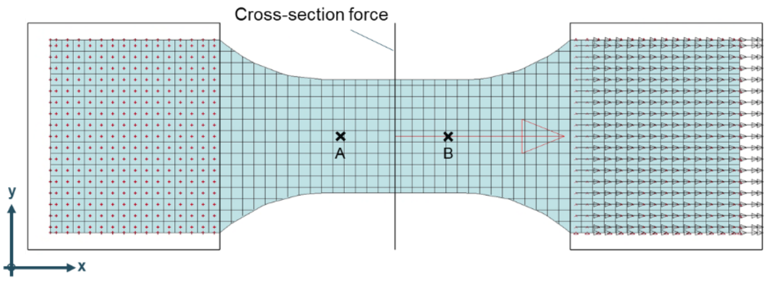
1. Element erosion scheme: Higher-order isogeometric shell elements usually have multiple in-plane integration points, i.e., multiple integration points per shell layer. In that case, the number of in-plane integrations to be failed until a whole layer is considered as failed needs to be defined. For DIEM with conventional FEA, a shell element layer (NIP defines the number of layers) is considered failed if any of the integration points in this layer is failed [7]. Adopting this definition as a default also for IGA elements when using NUMFIP.LT.0.0 in **`*MAT_ADD_DAMAGE_DIEM`**, turned out to give good agreement with FEA reference solutions as also visible in Fig. 4. Another setting, the percentage of layers which must fail to consider an element as failed (NUMFIP) was also kept identical for FEA and IGA. However, the user has the freedom to adjust these settings through **`*DEFINE_ELEMENT_EROSION_IGA`** [7].
2. Characteristic element length for damage evolution: Within DIEM, the evolution of the damage parameter D depends, among other quantities, on a characteristic element length h to suppress mesh dependence [6]: $\dot{D} = \dot{D}\left(\dot{\varepsilon}^p, \eta, D, h\right)$. Thus, a suitable definition of this characteristic element length for IGA, a non-trivial task for higher-order and higher-continuity IGA elements, is crucial to obtain the desired damage evolution behavior. A definition solely based on geometric properties yielded good agreement with the FEA reference solution for many test cases.

A (small) difference that still exists in the element erosion behavior can be explained by the higher continuity of isogeometric elements. That is, deleting one row of elements perpendicular to the loading direction does not result in a complete discontinuity, because some element basis functions still have support on both sides of the “crack”. This can be imagined as a small amount of remaining stiffness between the visually disconnected domains. However, the elements directly adjacent to the crack are already weakened and fail soon in the following time steps. This effect is subject to current research and more information can be found in the contribution by Lian et al. [8].

In summary, this tensile test example showed that good agreement between IGA and FEA results is achieved with the same material card in both the elasto-plastic and the damage phase. As next steps, test cases with different model geometries and loading scenarios are planned along with a comparison to hardware tests.



(a) FEA model.



(b) IGA model.

Fig.3: Standard tensile test: Specimens, discretization and boundary conditions.

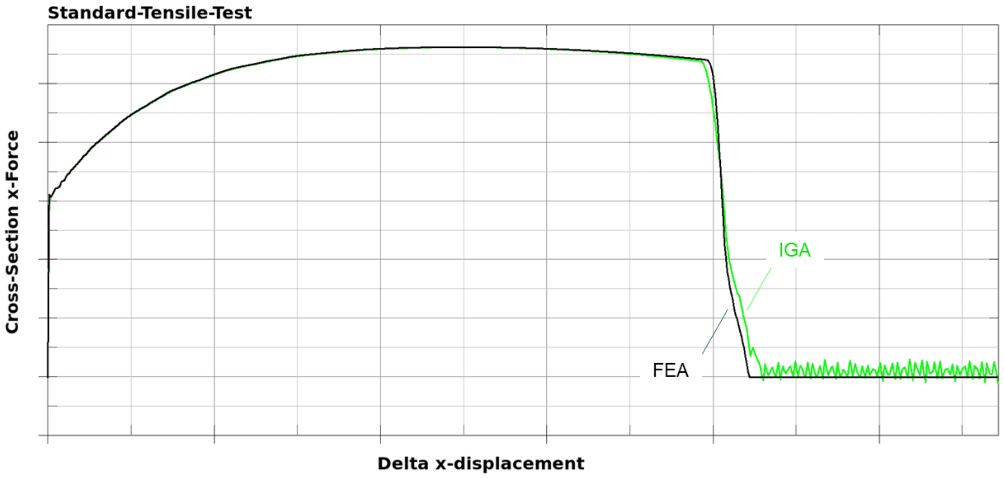


Fig.4: Standard tensile test: Force-displacement curves for FEA and IGA.

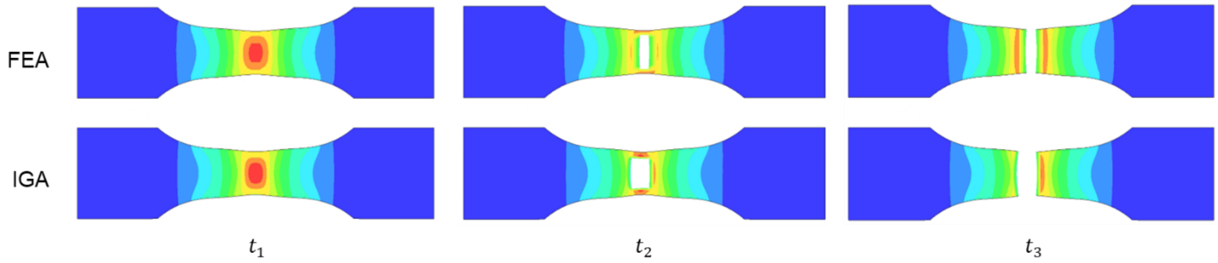


Fig.5: Standard tensile test: Effective plastic strain and element erosion at three different simulation states for FEA and IGA.

4 Initialization with Material History Variables

An important factor that not only allows state-of-the-art vehicle crash simulations, but also meaningful comparisons with hardware tests, is the consideration of the material history from manufacturing processes in IGA shell models. In analogy to the **`*INITIAL_STRAIN/STRESS_SHELL`** keywords for FEA, the keywords **`*INITIAL_STRAIN/STRESS_IGA_SHELL`** are implemented for IGA. With these two keywords, isogeometric shells can be initialized with material history variables like effective plastic strain, stress components or shell thickness.

Figure 6 shows a longitudinal beam consisting of two IGA shells with initialized shell thickness and effective plastic strain. The data for this initialization stems from a FE sheet metal forming simulation and the mapping is performed through a prototypical workflow using the DYNAmore tool Envyo [9]. These IGA shell components with mapping data are implemented into a hybrid IGA/FEA component model and simulated in a crash load case. The deformed IGA shell structure depicted in Fig. 6 showed good agreement with the according pure FEA model and corresponding hardware tests.

This example demonstrates that LS-DYNA has the capability to consider material history data in IGA shell models. To enable broad industrial usage of it, practical workflows including preprocessing or mapping tools need to be developed around LS-DYNA. This is currently work in progress.

More detailed information about the **`*INITIAL_STRAIN/STRESS_IGA_SHELL`** keywords can be found in the LS-DYNA keyword user's manual volume I [10].

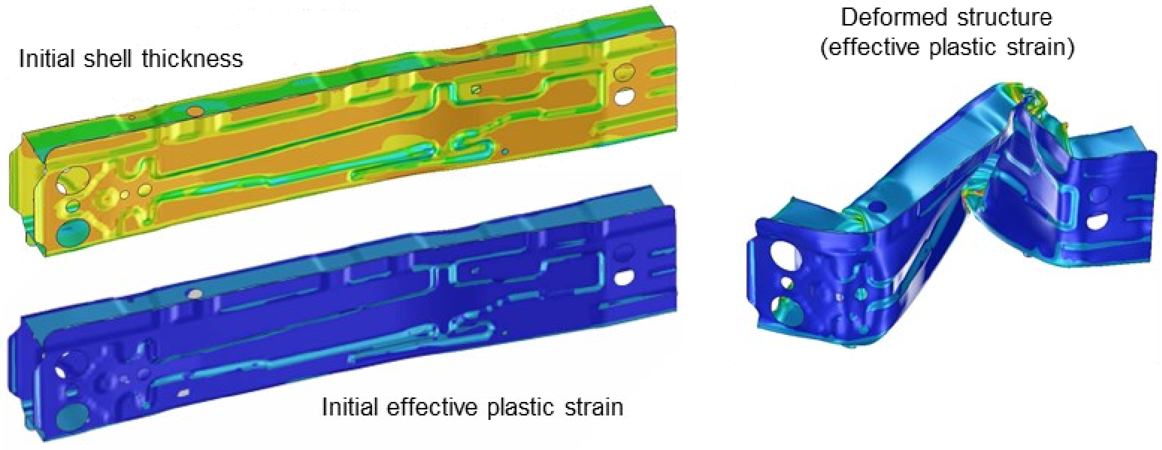


Fig.6: Longitudinal beam with initialized shell thickness and effective plastic strain obtained and mapped from forming simulations. The deformed structure is shown on the right.

5 Modeling of Multi-PID Patches (Tailor-welded/-rolled blanks)

Tailor-rolled and tailor-welded blanks with varying thickness and varying material properties are commonly used in automotive components. In LS-DYNA, such components may be modelled with dozens of finely graduated part, section, and material definitions. To realize such a component with isogeometric shells, so far, the whole component had to be split into multiple patches (multiple **`*IGA_2D_NURBS_XYZ`**). That is, one patch was required for each part, as shown in Fig. 7 for a simplified IGA shell model with only three different parts, sections and materials. These patches would then be coupled with a weak penalty-based approach along the topologically connected edges during the analysis (**`*IGA_TIED_EDGE_TO_EDGE`**).

To avoid an extensive number of patches and coupling edges for such tailor-rolled and tailor-welded components, the possibility to define multiple parts, sections and materials on one single patch is implemented in LS-DYNA. As Fig. 8 shows, the three different parts, isogeometric shells (**`*IGA_SHELL`**) and faces (**`*IGA_FACE_XYZ`**) can now refer to the same underlying patch (**`*IGA_2D_NURBS_XYZ`**). Thus, the maximum continuity within the IGA component can be retained and penalty coupling is no longer required. As always in modeling, the user must decide whether a high continuity is desired across thickness or material jumps.

Here it should be noted that the model for the B-pillar example in Fig. 8 was generated manually, i.e., the curves dividing the patch into multiple domains were created "by hand". The corresponding preprocessing capabilities to generate such models are expected to follow soon. This would allow the integration of complex tailor-welded and tailor-rolled isogeometric shell components into vehicle models more easily.

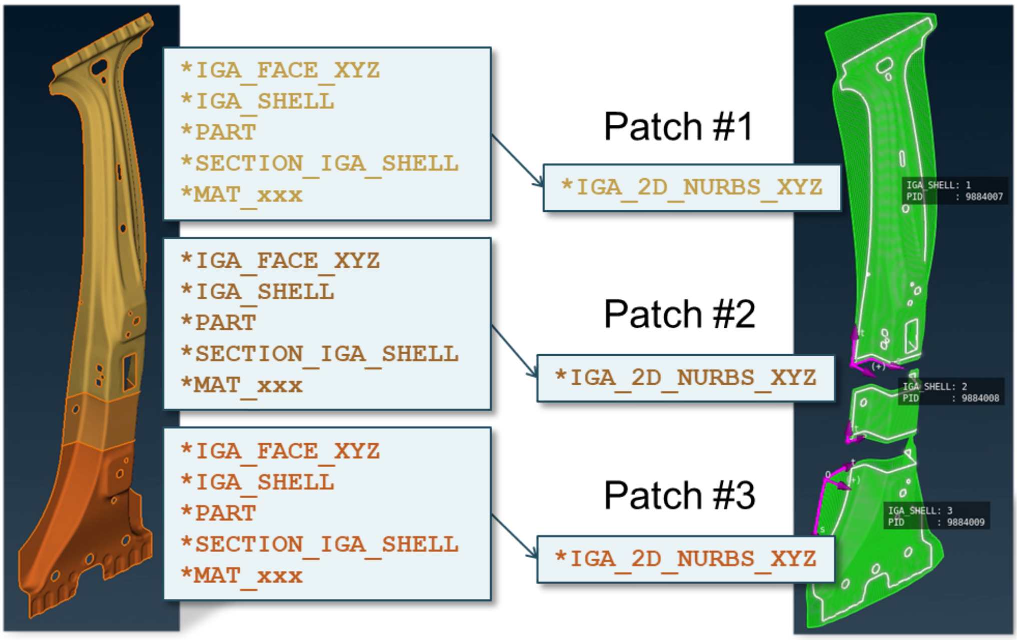


Fig.7: Old approach: Multiple patches for multiple part definitions (Courtesy of BMW Group).

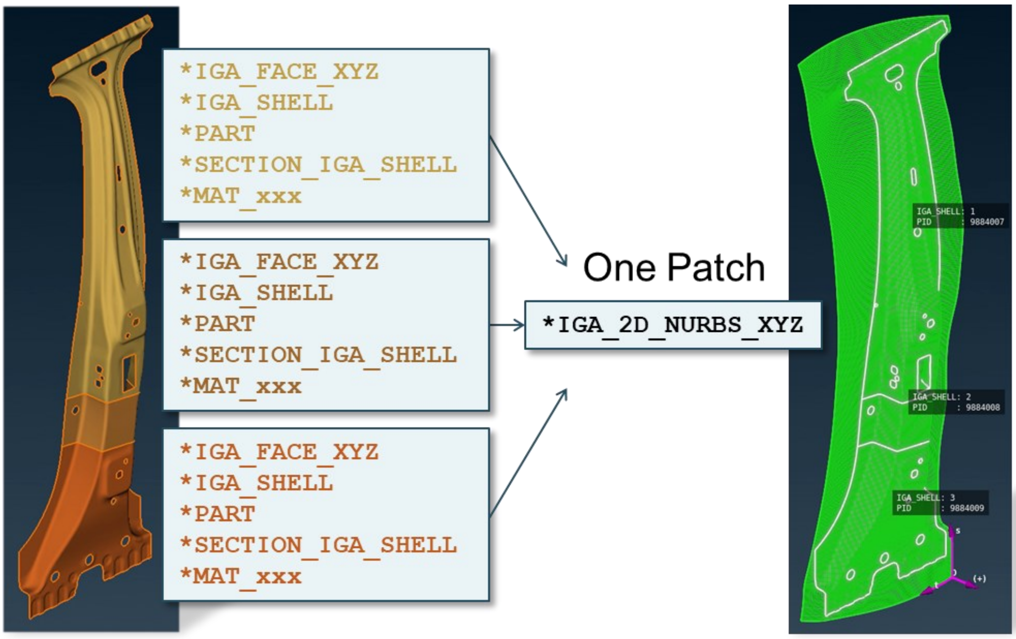


Fig.8: New approach: One patch for multiple part definitions (Courtesy of BMW Group).

6 Feature-based Modeling with Isogeometric Analysis

The connection modeling approaches presented in Section 2 were extended in a straightforward manner for IGA to allow compatibility and a simple 1:1 component exchange between FEA and IGA. However, many of these connections still rely on discrete `*NODE`s and therefore cannot exploit the potential of IGA for feature-based modeling, independent of nodes and meshes. The advantages of such a feature-based approach are clear: same modeling technique and data structure as in CAD and therefore a fast and simple transition, see Fig. 9. It furthermore provides an intuitive way of applying connections and boundary conditions to entities like faces and edges instead of individual nodes or node-sets.

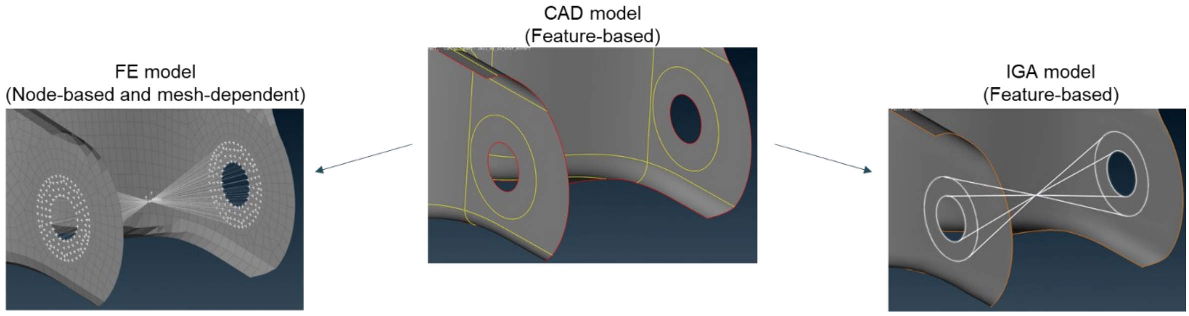


Fig.9: Comparison of modeling techniques: Node-based and mesh-dependent modeling with FEA versus feature-based modeling as in CAD with IGA.

Several keywords for a feature-based application of boundary conditions and loads to points, edges, and faces are already available for IGA in LS-DYNA, for example:

```
*BOUNDARY_PRESCRIBED_MOTION_POINT_UVW
*BOUNDARY_PRESCRIBED_MOTION_EDGE_UVW
*BOUNDARY_PRESCRIBED_MOTION_FACE_XYZ
*LOAD_EDGE_UVW
*LOAD_FACE_XYZ
```

A logical extension of these IGA capabilities in LS-DYNA are feature-based rigid body definitions. Rigid bodies are commonly used in component simulations to represent connected parts like shafts and bolts. The IGA suspension component depicted in Fig. 10 is such an example. With the current capabilities, the rigid bodies are modeled with the conventional FE keywords **`*CONSTRAINED_NODAL_RIGID_BODY`** and **`*CONSTRAINED_INTERPOLATION`** and the associated **`*NODE`**s are then attached to the IGA shells through **`*IGA_POINT_UVW`**.

A more attractive approach is sketched in Fig.11. This approach uses feature-based alternatives to **`*CONSTRAINED_NODAL_RIGID_BODY`** and **`*CONSTRAINED_INTERPOLATION`**, which are not defined between individual **`*NODE`**s but between an isogeometric face (**`*IGA_FACE_XYZ`**) and a **`*NODE`** or an isogeometric edge (**`*IGA_EDGE_UVW`**) and a **`*NODE`**. Such feature-based rigid bodies could be already defined on faces and edges in the CAD environment and then easily transferred to the IGA model. The development of such feature-based rigid bodies is currently ongoing.

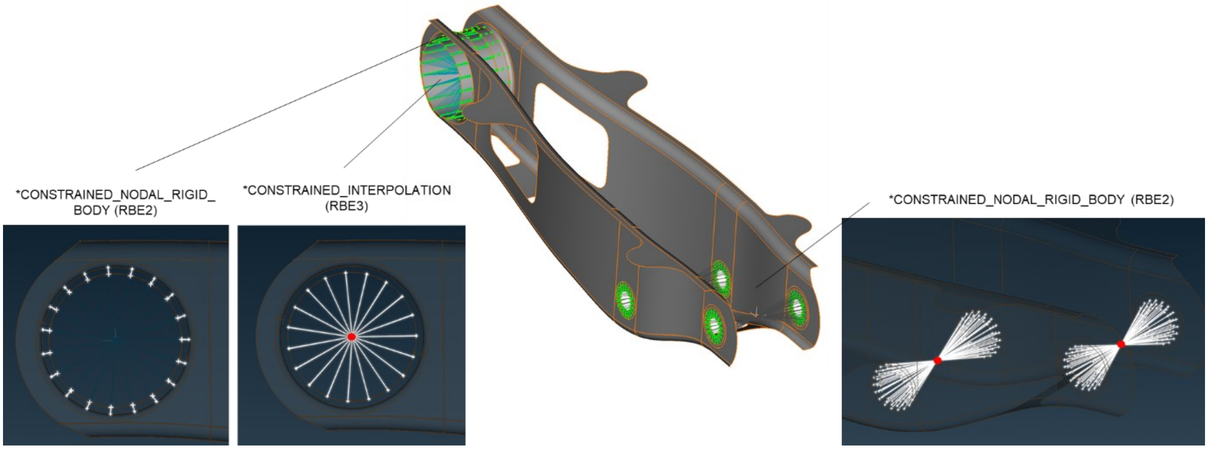


Fig.10: Currently: Node-based modeling of rigid bodies with IGA to enable a 1:1 substitution of FEA components.

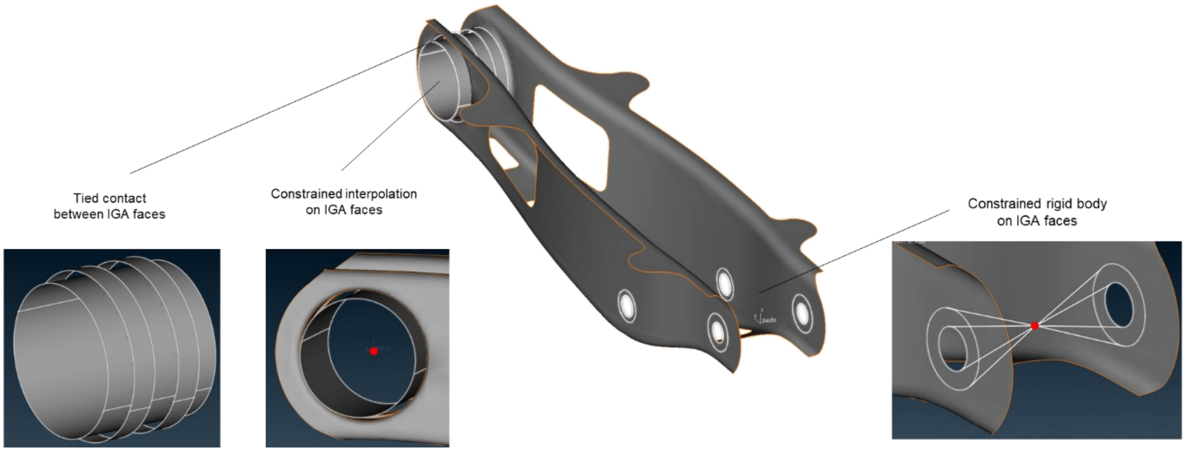


Fig.11: Future: Feature-based modeling of rigid bodies with IGA.

7 Summary and Conclusion

This paper presented several novel LS-DYNA capabilities crucial for state-of-the-art vehicle crash simulations with isogeometric shell components:

1. Constraint-based tied contacts for IGA shells acting directly on the isogeometric faces (spline-based), in addition to the already available penalty-based tied contacts acting on the interpolation mesh of IGA shells.
2. Improved damage and failure modeling with DIEM due to improved element erosion schemes and an appropriate characteristic element length for the damage evolution phase. Good agreement with validated FE results was achieved.
3. The possibility to consider the material history from preceding manufacturing simulations on IGA shells through the keywords **`*INITIAL_STRAIN_IGA_SHELL`** and **`*INITIAL_STRESS_IGA_SHELL`**.
4. A convenient way to model tailor-welded and tailor-rolled blanks with varying material properties and wall thicknesses on a single patch instead of multiple coupled patches.
5. The future way of IGA modeling inspired by CAD with feature-based, node- and mesh-independent boundary conditions and rigid bodies to enable faster processes through a tight connection between design and analysis.

The latter three capabilities are supported in LS-DYNA, but for practical industrial usage also the corresponding preprocessing capabilities and workflows are required. As in the past, this will be achieved through a close collaboration between solver developers, preprocessor developers and the users. The discussions in this paper were limited to isogeometric shells, but similar capabilities are also available for isogeometric solids or will be implemented in the future.

8 Literature

[1] Hughes, T.J.R., Cottrell, J.A., Bazilevs, Y.: "Isogeometric Analysis: CAD, finite elements, NURBS, exact geometry, and mesh refinement", Computer Methods in Applied Mechanics and Engineering, Vol. 194, 2005, 4135-4195.

[2] Leidinger, L., Hartmann, S., Benson, D., Nagy, A., Li, L., Pigazzini, M., Nguyen, L., Bauer, F.: "Isogeometric Shell Components in Full Vehicle Crash Simulations: Hybrid Modeling", 16th LS-DYNA Forum 2022, Bamberg, Germany.

[3] Bauer, F., Yugeng, T., Leidinger, L., Hartmann, S.: "Experience with Crash Simulations using an IGA Body in White", 14th European LS-DYNA Conference 2023, Baden-Baden, Germany.

[4] Leidinger, L., Hartmann, S., Benson, D., Nagy, A., Rorris, L., Chalkidis, I., Bauer, F.: "Hybrid IGA/FEA Vehicle Crash Simulations with Trimmed NURBS-based Shells in LS-DYNA", 13th European LS-DYNA Conference 2021, Ulm, Germany.

[5] Hartmann, S., Leidinger, L., Benson, D., Nagy, A., Pigazzini, M., Li, L., Nguyen, L.: "Isogeometric Analysis in LS-DYNA R13 - key steps towards industrial applications", 13th European LS-DYNA Conference 2021, Ulm, Germany.

[6] Borrvall, T., Johansson, T., Schill, M., Jergéus, J., Mattiasson, K., DuBois, P.: “A General Damage Initiation and Evolution Model (DIEM) in LS-DYNA”, 9th European LS-DYNA Conference 2013, Manchester, UK.

[7] LS-DYNA dev Keyword Manual Vol II – Material Models, Latest DRAFT Versions of LS-DYNA Manuals: https://lsdyna.ansys.com/manuals/

[8] Lian, Z., Leidinger, L., Hartmann, S., Bauer, F., Wüchner, R.: "Towards the Solution of Cross-Talk in Explicit Isogeometric B-Rep Analysis", 14th European LS-DYNA Conference 2023, Baden-Baden, Germany.

[9] Envyo, https://www.dynamore.de/en/products/process-chain/envyo, 2023.

[10] LS-DYNA dev Keyword Manual Vol I, Latest DRAFT Versions of LS-DYNA Manuals: https://lsdyna.ansys.com/manuals/