Hybrid IGA/FEA Vehicle Crash Simulations with Trimmed NURBS-based Shells in LS-DYNA

Lukas Leidinger[1], Stefan Hartmann[1], Dave Benson[2], Attila Nagy[2], Lambros Rorris[3], Ioannis Chalkidis[3], Frank Bauer[4]

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

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

[3]BETA CAE Systems International AG
Platz 4, 6039 Root D4, Switzerland

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

1 Introduction

Isogeometric Analysis (IGA) [1] is a rather new approach to Finite Element Analysis (FEA), using spline basis functions known from Computer Aided Design (CAD) for describing both the geometry and the solution field. The main motivation for IGA is the integration of design and analysis. Achieving such a full integration requires a holistic approach with a fundamentally different modeling strategy and development process to exploit the full potential of IGA. Such changes certainly take time and cannot be achieved overnight. Fortunately, IGA with its higher-order and higher-continuity elements also offers several additional advantages such as an accurate geometry description, superior analysis qualities, a larger explicit time step size or smart modeling techniques. Thus, users may benefit from IGA immediately, even without a full paradigm shift.

Several recent developments made the application of IGA in an industrial context more attractive for users. First, the introduction of the new **`*IGA`** keywords in LS-DYNA R13, which are able to capture the data structure of common CAD models, so-called B-Rep (boundary representation) models, including geometry and topology information. This comes along with the capability to couple and stabilize the multiple trimmed NURBS (Non-Uniform Rational B-Splines) shell patches appearing in industrial models during analysis. Second, the increasing availability of preprocessing capabilities for IGA, especially in ANSA [2]. From version 22, ANSA enables an efficient generation of industrial NURBS shell models for LS-DYNA, based on the new **`*IGA`** keywords. A third reason for the attractiveness of IGA in LS-DYNA is the possibility to simulate hybrid models, that is, models consisting of both, IGA and conventional FEA components.

The focus of this paper is on hybrid IGA/FEA models with isogeometric shell components. In particular, Section 2 describes the generation of isogeometric NURBS shell models from current (not analysis-suitable) CAD models. Section 3 demonstrates the possibility to insert IGA components into an existing FEA (vehicle) model via a simple one-by-one component exchange. It furthermore shows that connection technology like spotwelds, bolts and rigid bodies can now be directly applied to isogeometric shells in LS-DYNA, without further model modifications. Section 4 provides two industrial crash examples of BMW models, dynamic buckling of a crashbox-type component and impact of a hybrid IGA/FEA vehicle front end structure against a rigid wall. Section 5 gives a brief summary and an outlook to future steps and developments.

2 IGA model generation workflow based on the new `*IGA` keywords

A tight integration of CAD and analysis requires a common communication basis. Therefore, a new **`*IGA`** keyword family, representing the B-Rep data structure of industrial CAD models including geometry and topology information, has been recently implemented into LS-DYNA. These new keywords are developed with a particular focus on trimmed multi-patch NURBS models, most common in industrial CAD, for both shell and solid analysis. This paper, however, is restricted to isogeometric shell components. Detailed descriptions of the underlying theories and concepts is beyond the scope of this paper. For more information on the **`*IGA`** keywords, the reader is referred to the latest LS-DYNA keyword manual [3] and the paper “Isogeometric Analysis in LS-DYNA R13 – key steps towards

industrial applications" by Hartmann et al. [4] also presented at this conference. Details on trimmed multi-patch NURBS shell models can be found in references [1,4,5,6].
Although direct analysis on CAD models is possible with the new `*IGA` keywords, this task turned out to be not straightforward for current industrial CAD models, because these models were not designed with analysis-suitability in mind. For conventional FEA, a completely new model discretization is generated anyway. Current industrial CAD models may exhibit many small faces with small elements. This leads to a small stable time step size and therefore inefficient explicit dynamic crash analysis. Moreover, current CAD algorithms may also generate surfaces with relatively high polynomial degrees (e.g. up to p=13), which lead to a large number of integration points per element with standard Gauss integration, i.e. $(p+1) \times (p+1)$ for full and $p \times p$ for reduced integration.
There are basically two ways to overcome this issue. The first approach, currently required for existing CAD models, is to make the model suitable for isogeometric analysis in a preprocessing step. This includes merging of small faces and reducing the polynomial degree, for example with the preprocessor ANSA [2]. Although one may not exploit the full potential of IGA regarding fast model generation with this approach, this preprocessing step was still found to be simpler and faster than for conventional FEA. Furthermore, one may still benefit from better results due to the higher-order and higher-continuity basis functions, or an increased time step size compared to standard FEA. The second, more visionary approach is to consider isogeometric analysis as much as possible already during CAD model construction. For this purpose, the design engineer could be given a set of modeling guidelines to consider, e.g. a minimum patch (surface) size, a minimum element size or already providing a midsurface description in association with the B-Rep model for shell analysis. Similarly, CAD algorithms and systems could be accordingly adapted, e.g. using the lowest possible degree for a surface by default or additional capabilities for the application of analysis-related properties such as material definitions and boundary conditions. With these measures, isogeometric analysis could be performed on CAD models without or with a minimum number of modifications in the future, yielding a significantly faster development process.
In the following, a model generation workflow employing the first approach for current CAD models with an additional preprocessing step is presented. The key ingredients of the workflow depicted in Fig.1 are the preprocessor ANSA for IGA model generation and LS-DYNA as the FEA/IGA solver. For the CAD and the postprocessing step, standard commercial software can be used (LS-DYNA provides a finite element interpolation mesh with projected results to enable standard FEA postprocessing). Fig.2 describes the shell model generation steps in ANSA in more detail. The first step from a) to b) is the import of the B-Rep CAD model (hollow volume model) into ANSA. In the next step, the midsurface is generated, which is an automated process for most thin-walled components. Based on this midsurface description in c), ANSA is able to automatically generate an analysis-suitable model consisting of only one trimmed NURBS patch based on user parameters such as min. and max. element size, polynomial degree and max. deviation from the original model. In Fig.2 d), the trimmed IGA model is depicted together with the underlying untrimmed patch. Finally, the IGA model in e) is output in the new `*IGA` keyword format for analysis in LS-DYNA, see f). Please note that no manual geometry cleaning steps were required for the BMW vehicle components studied in this paper.
Depending on the model complexity, generating an IGA model consisting of a single trimmed NURBS patch as shown in Fig.2 may not always be possible, e.g. for models with closed cross sections or T-joints. In such cases, the model is divided into multiple domains, which are then reparametrized as single trimmed NURBS patches in ANSA as shown in Fig.3. During analysis in LS-DYNA, these multiple trimmed NURBS shells are coupled through a weak penalty-based approach, see [5,6,7].
The modeling techniques to connect such IGA components to other (FEA) components within a hybrid IGA/FEA (vehicle) model, are described in the following section.

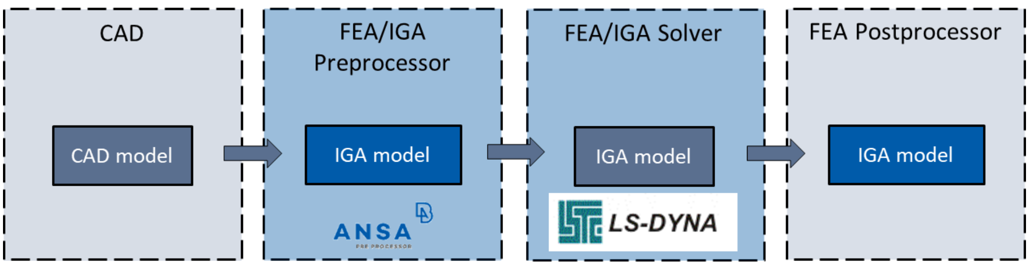


Fig.1: Isogeometric analysis (of hybrid IGA/FEA models) within a conventional FEA workflow.

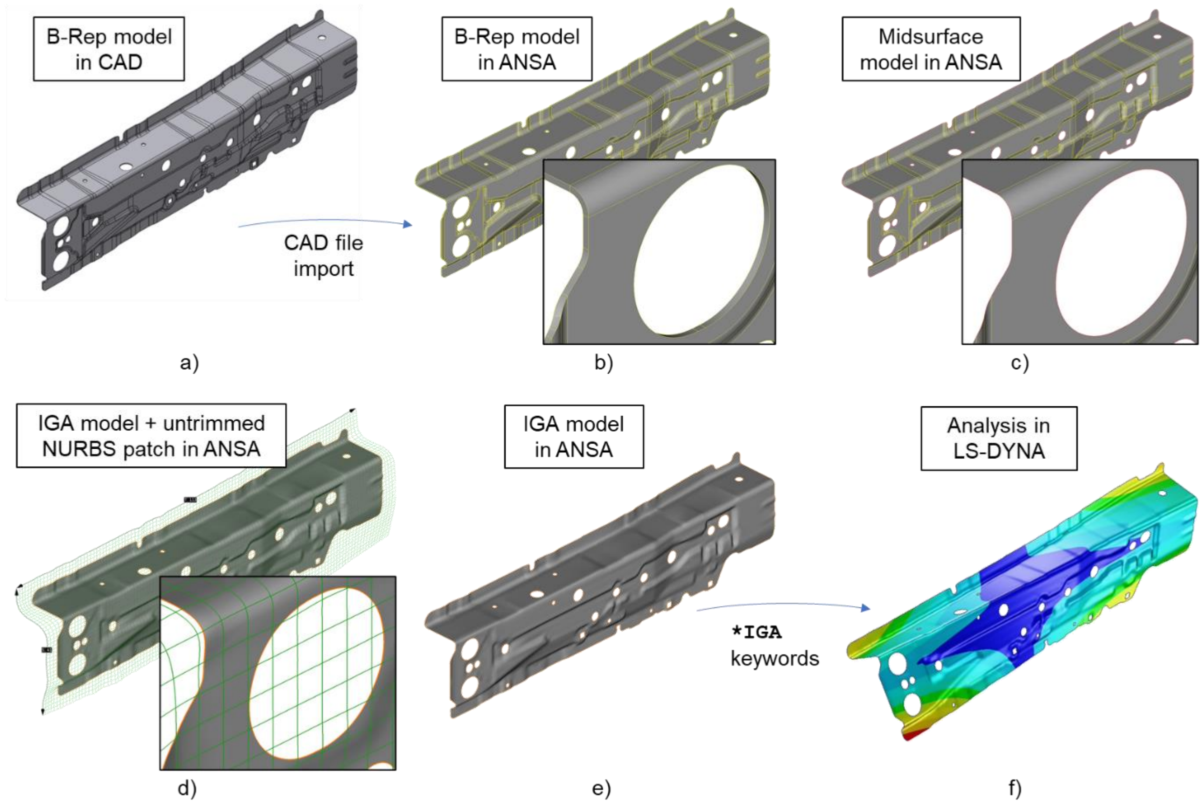


Fig.2: IGA model generation: From a CAD model to an LS-DYNA IGA shell model with ANSA.

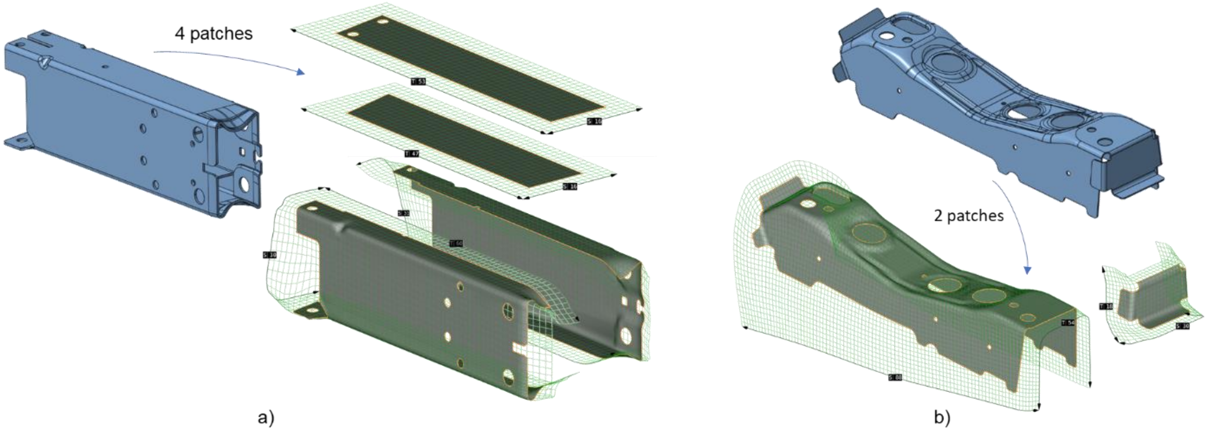


Fig.3: IGA model generation: Trimmed multi-patch shell models generated with ANSA.

3 Hybrid IGA/FEA modeling for a one-by-one component exchange

IGA has a large potential to speed up current virtual development processes by integrating design and analysis. With a trimming approach, IGA models employ the same feature-based modeling paradigm used in CAD. This not only means that holes and cut-outs are modeled independently of the underlying mesh, avoiding remeshing in case the position of a feature changes. The fact that model entities and features like faces, edges, vertices, beadings, holes, or cut-outs can be selected and referred to, even enables an associative model assembly as in CAD. Imagine, for example, a seat mounted on a seat cross-member which in turn is mounted on the floor panel of a vehicle. If the position of the seat had to change during the development process, the position of the cross-member would change accordingly together with the shape and the attachment points of the floor panel.
Although such a visionary approach is certainly desirable, it requires fundamental changes in the modeling strategy and the overall development process. IGA technology, however, has not yet reached the point at which such a transformation would be done by OEMs, although isogeometric simulations of a full body-in-white (BIW) have already been performed recently. The intermediate step to be taken now, is to remain the current processes mainly unaffected and to insert certain IGA components into existing

FEA (vehicle) models without additional modifications. Replacing a standard FEA component with an IGA component should be as simple as changing the element formulation. In this way, more and more LS-DYNA users are expected to test IGA, to build trust in the technology and to contribute to the development, elevating IGA in LS-DYNA to a mature technology for industrial applications. Once this is achieved, the focus can be moved to a real IGA-type modeling approach for the integration of design and analysis.

With the one-by-one component exchange approach, only a part of IGA's large potential can be exploited for now, similar to the IGA model generation approach for current, not analysis-suitable CAD models described in the previous section. Nevertheless, the accurate geometry description, the higher-order and higher-continuity elements, as well as the larger time step size may still provide enough benefits to apply IGA in this manner.

The remainder of this section provides examples of IGA solver capabilities, mainly regarding connection technology, required to achieve such a simple one-by-one component exchange in LS-DYNA.

3.1 Spotwelds

The probably most common connection technology in a BIW are spotwelds. Within a hybrid IGA/FEA model, spotwelds between FEA components, between IGA components, and between IGA and FEA components need to be modeled. Here, the keyword **`*CONSTRAINED_INTERPOLATION_SPOTWELD`** is applied, see [4] for more details. Another option available for IGA would be **`*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET`**.

For conventional FEA, a specific (finer) mesh around spotwelds is often generated, be it to ensure a sufficiently high number of nodes involved in the spotweld formulation or to assign different material parameters to shell elements in the vicinity of spotwelds, see Fig.4. For IGA with **`*CONSTRAINED_INTERPOLATION_SPOTWELD`**, this is no longer required. A sufficiently fine mesh can be guaranteed by a relatively fine interpolation finite element mesh (no element evaluation, no effect on time step, see [8]), while the heat-affected zone with varying material parameters around spotwelds (and line welds) may be modelled using **`*DEFINE_HAZ_TAILOR_WELDED_BLANK`**. The shell parts involved in the spotweld definition are simply identified via their IDs (**`MID`** and **`SID`**), while the spotweld positions are defined via a node set and the corresponding ID (**`NSID`**), see Fig.5. In this way, the spotweld definition for IGA is completely mesh-independent and thus, manual adaptions can be mostly avoided, regardless of whether the underlying shell model or the position of the spotweld changes.

3.2 Bolts

Various bolt modeling approaches with different levels of detailedness exist for standard FEA. The goal here is not to remodel bolts with IGA, but to just use these existing bolt models without further modifications for the connection of isogeometric shell components. Without going into detail about bolt modeling strategies, in most cases, the interaction between a bolt and a shell component is established via some type of contact definition, in our case a **`*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_xxx`**. Therefore, contact between the bolt model and the isogeometric shells needs to be modeled. By default, contact for isogeometric shells is handled via a finite element shell interpolation mesh, automatically generated on top of the isogeometric shell elements, see [8]. In this way, the standard FEA contact algorithms are also available for IGA.

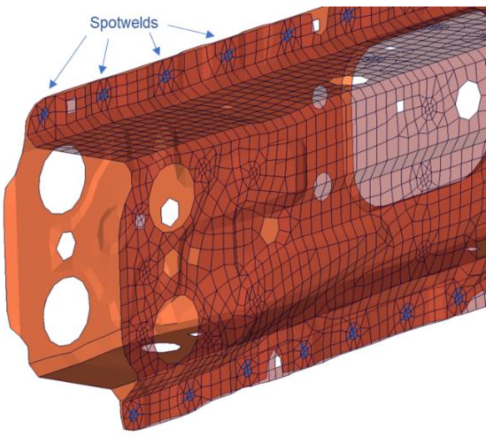


Fig.4: Spotweld modeling for conventional FEA with specific mesh around spotwelds.

To conclude, as long as the interaction between bolts and shells only includes a kind of penalty-type contact, the existing bolt modeling approaches can be directly applied to isogeometric shells without further modifications, as shown in Fig.6.

3.3 Rigid bodies

Also rigid body definitions such as ***CONSTRAINED_NODAL_RIGID_BODY** or similar keywords like ***CONSTRAINED_INTERPOLATION** are commonly used within BIWs. The conventional approach for attaching rigid bodies to FEA shells, is to incorporate certain nodes of the finite element shell mesh in the rigid body definitions. The goal is to enable this also for IGA without modifications. For this purpose, the original finite element shell nodes incorporated in the rigid body definition can be reused and "glued" onto the IGA shells. This is achieved via the keyword ***IGA_POINT_UVW** in the preprocessor ANSA, see [4] for more details. Fig.7 shows a rigid pin connected to a finite element mesh in a) and how the very same pin is attached to isogeometric shells via four nodes and ***IGA_POINT_UVW** (highlighted in green).
Here it should be noted that this is again a modeling approach for a one-by-one component exchange in a hybrid IGA/FEA model. In a real IGA-type approach, one would not attach the rigid body to four discrete points on the IGA shell, but rather to the hole's edge or to a part of the IGA face directly, following a feature-based modeling philosophy.

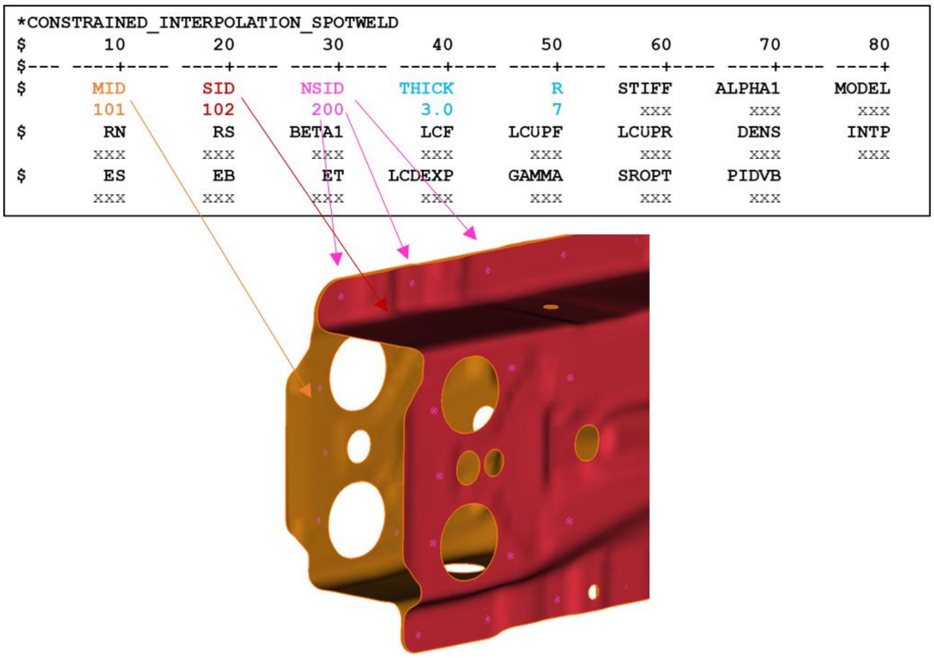


*Fig.5: Mesh-independent spotweld modeling for NURBS-based shells and ***CONSTRAINED_ INTERPOLATION_SPOTWELD**.*

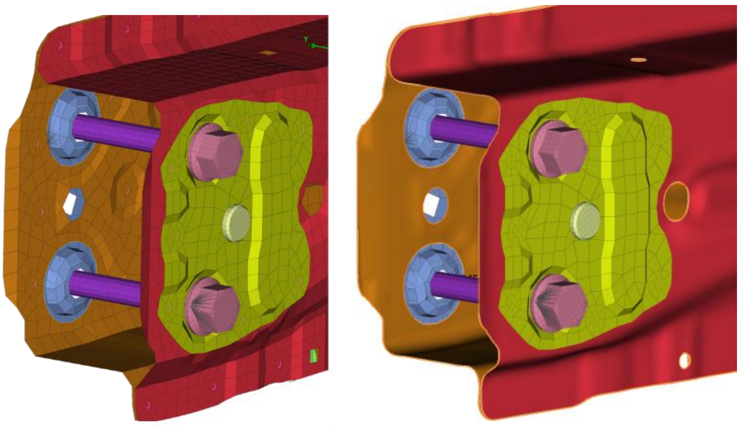
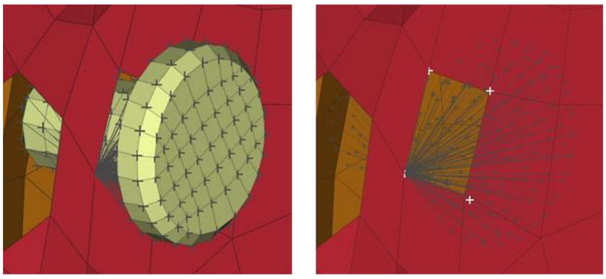
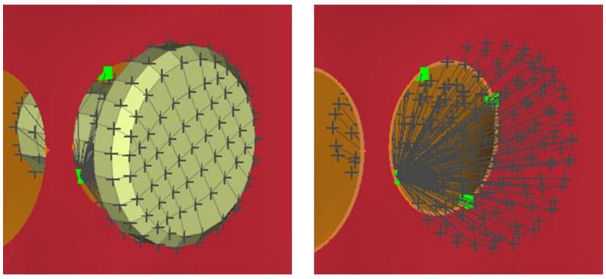


Fig.6: Bolt modeling: The pure FEA model (left) and the hybrid IGA/FEA model with isogeometric shell elements (right).



a)



b)

Fig.7: Rigid body definitions for conventional FEA shells in a) and isogeometric shells with ****IGA_POINT_UVW*** *in b).*

3.4 Additional solver capabilities

This section provides some additionally required or beneficial solver capabilities for explicit dynamic (crash) simulations of hybrid IGA/FEA models.
Explicit dynamic analyses of large models are commonly performed on multiple processors using the MPP (massively parallel processing) version of LS-DYNA. To enable this for hybrid IGA/FEA models, also isogeometric entities including shells, interpolation shells, attached points via ***IGA_POINT_UVW** and related spotweld definitions need to be decomposed and accordingly distributed over the available processors. This is properly supported in LS-DYNA, starting from R13.
Established development processes for FEA usually have predefined analysis settings and a set of proven control keywords. When using hybrid IGA/FEA models, the IGA implementations must either be compatible with these settings or the IGA components need to be excluded from certain control cards.
Furthermore, time step size and mass scaling are important aspects to consider in explicit dynamic simulations of hybrid IGA/FEA models. First of all, a single IGA component should not diminish the time step of the whole model. It is therefore advisable to choose the element size of the IGA model in a similar range as for the FEA models. By excluding the $(p-1)$ elements at the boundary of so-called open knot vector patches, which can be done with the 'extend' option during IGA model generation in ANSA [2], one can even achieve a larger time step for the same element size compared to FEA, see [6]. An enhanced time step estimate for NURBS-based elements accounting for this is currently implemented in LS-DYNA. With this estimate, one can immediately benefit from this larger time step. In case a predefined time step is used for the analysis, the larger time step for IGA components may result in a lower amount of mass scaling and therefore better accuracy compared to standard FEA.
Another crucial aspect is the stabilization of so-called light control points, caused by small trimmed elements, see [4,6]. In LS-DYNA R13, a suitable stabilization method has been developed and implemented, which appears to be very effective for explicit dynamic (crash-type) problems.

4 Examples

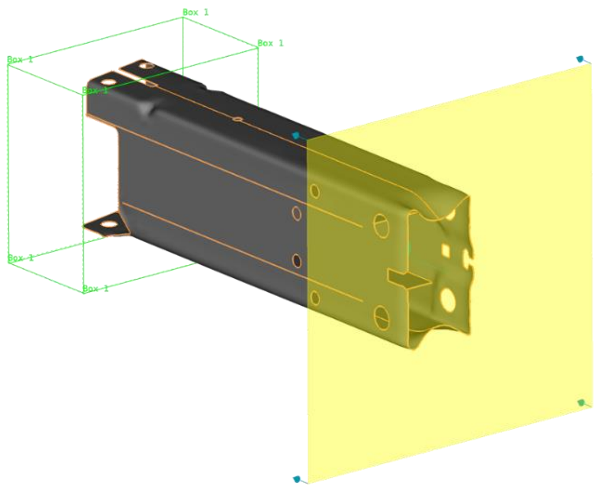
In this section, two industrial examples are provided to demonstrate the practical applicability of the LS-DYNA IGA capabilities for trimmed NURBS shells based on the new ***IGA** keywords. Both examples were set up using the model generation workflow described in Section 2. For the hybrid IGA/FEA model in the second example, the modeling approach for a simple one-by-one component exchange including existing connection technology from Section 3 is employed.

4.1 Dynamic buckling of an energy-absorbing component

This first example shows the dynamic buckling of an energy-absorbing BMW vehicle component with large deformations, plasticity, failure, and contact. It furthermore provides a detailed comparison between pure IGA and pure FEA results in terms of simulation accuracy and numerical effort. The problem definition and analysis settings are given in Fig. 8, showing a crashbox-type component clamped on one end and impacted by a rigid wall on the opposite end. The multiple trimmed NURBS patches of the model, see also Fig.3, are coupled via a weak penalty-based approach, invoked by **`*IGA_TIED_EDGE_TO_EDGE`**.

A close-up comparison of the conventional FEA model geometry and the superior IGA model geometry is provided in Fig.9. As can be seen, smoothly curved areas can be captured more accurately with the NURBS-based IGA model. Fig.10 depicts the dynamic buckling process through a sequence of deformed shapes for an FEA model with approximately 4mm element length (top), and for two IGA models with an approximate element size of 6mm and 3×3 resp. 2×2 integration points per element (middle resp. bottom). The final deformed shapes at $t = 30$ms are provided in Fig.11. Although the individual buckling patterns differ slightly, a good overall agreement between the FEA and the IGA results is observed, despite the coarser IGA discretization. Furthermore, no significant difference between the two IGA configurations with one-time (3×3) and two-times reduced (2×2) integration is observed for this example (full integration for cubic shell elements would be 4×4). The good agreement is also confirmed with the force over time plot provided in Fig.12, in which the IGA models exhibit a slightly higher average buckling force level and thus a slightly higher energy absorption.

A detailed comparison of numerical costs in terms of average CPU time (explicit analysis, 6 CPUs, MPP) is provided in Fig.13. As can be seen, the cost for IGA with 3×3 integration points and the original time step estimation ($\Delta t_0 = 4.44 \times 10^{-7}$s) is with 1962.7 CPU seconds around 3.6 times higher than for standard FEA with an initial time step size of $\Delta t_0 = 3.27 \times 10^{-7}$s. Reducing the numerical integration to 2×2 points per element, significantly reduces the CPU time by around 38% to 1223.3s, without a noticeable loss in accuracy, see Fig.10-12. The highest gain, however, is achieved through the



- MAT_024 (piecewise linear plasticity)
- Fixed DOFs within Box
- Rigid Wall $m = 150\text{kg}, v = 50\text{km/h}$

IGA settings

- Cubic NURBS-based RM shell ELFORM=3
- 5 out-of-plane integration points
- 3×3 or 2×2 in-plane integration points / element

FEA settings

- Linear Belytschko-Tsay elements ELFORM=2
- 5 out-of-plane integration points

Fig.8: Dynamic buckling of an energy-absorbing component: Problem definition and analysis settings.

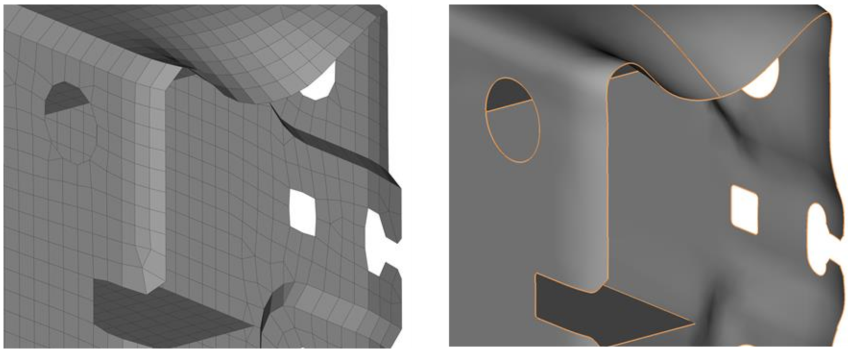


Fig.9: Dynamic buckling of an energy-absorbing component: Comparison between FEA model (left) and IGA model (right).

enhanced time step estimates, which exploit the full time step potential of IGA shells with trimmed-off boundary elements ('extend' option in ANSA). With this new estimation, a substantial time step increase to $\Delta t_0 = 1.31 \times 10^{-6}$s is achieved. This results in a total CPU time of only 415.6s, which is lower than the CPU time of the reference FEA model.
To conclude, for this example, the combination of a slightly larger element size, reduced integration, and enhanced time step estimation enables accurate isogeometric crash-type simulations of trimmed multi-patch shell models with a computational effort comparable to (or lower than) conventional FEA.

4.2 Hybrid IGA/FEA vehicle front end structure

This example shows the crash simulation of a hybrid IGA/FEA vehicle front end structure impacting a rigid wall. In this hybrid model, the FEA longitudinal members are replaced by their IGA counterparts as shown in Fig.14. Each of the two longitudinal members consists of two spot-welded sheet metal parts, each of which is modeled via a single trimmed NURBS shell, see also Fig.2. Due to IGA capabilities enabling a direct application of FEA connection technology to IGA shells, see Section 3, inserting the IGA components is achieved via a simple one-by-one include exchange without further model modifications. These connections comprise spotwelds between IGA components and between IGA and FEA components, bolts including tied contact formulations, and rigid bodies attached to IGA shells. In combination with trimmed isogeometric shell models, these connections become independent of the underlying shell mesh. This avoids manual adaptions in case the model geometry or the position of connections change within the various design cycles.

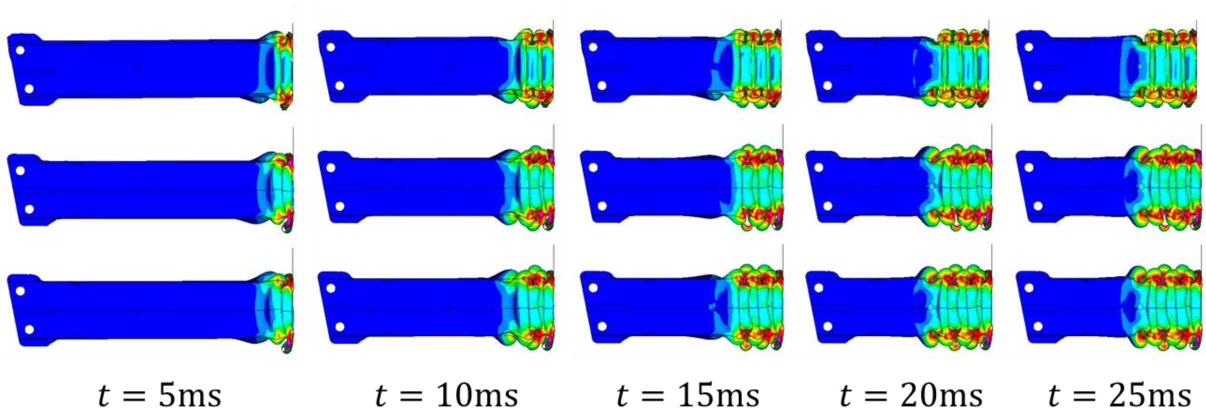


Fig.10: Dynamic buckling of an energy-absorbing component: A sequence of deformed shapes for FEA 4mm (top), IGA 6mm with 3×3 integration points (middle) and IGA 6mm with 2×2 integration points (bottom). The color plot indicates maximum plastic strain.

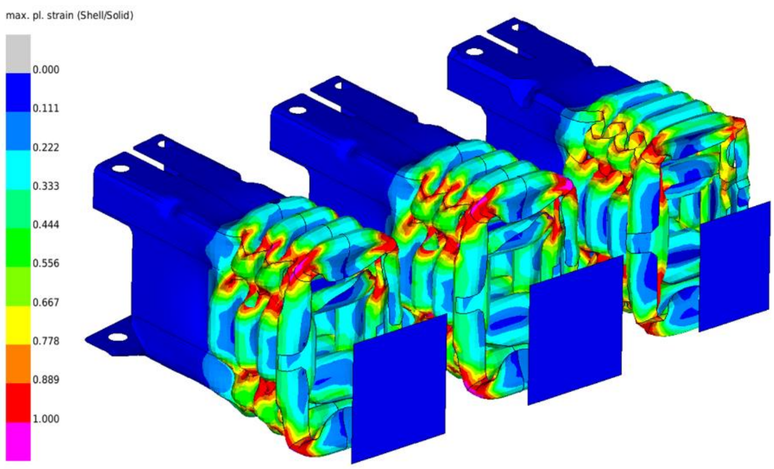


Fig.11: Dynamic buckling of an energy-absorbing component: Final deformed shapes at $t = 30ms$ for FEA 4mm (right), IGA 6mm with 3×3 integration points (middle) and IGA 4mm with 2×2 integration points (left).

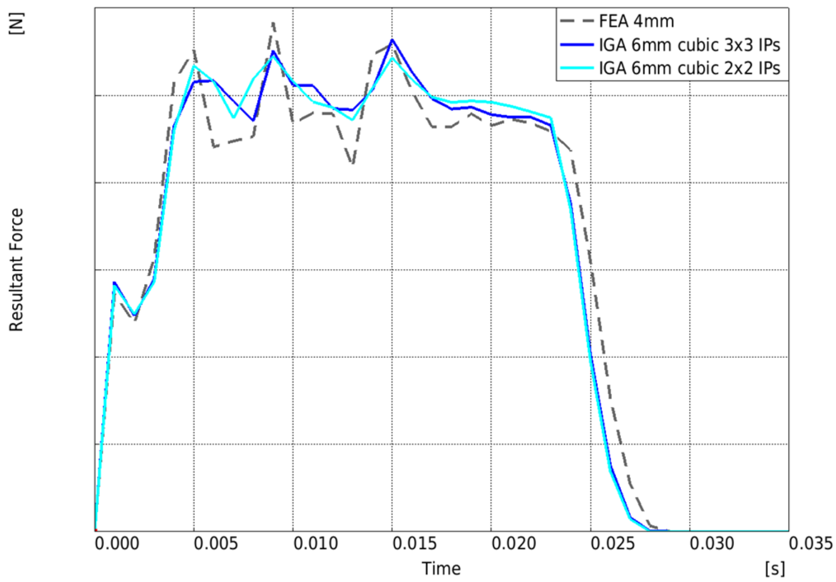
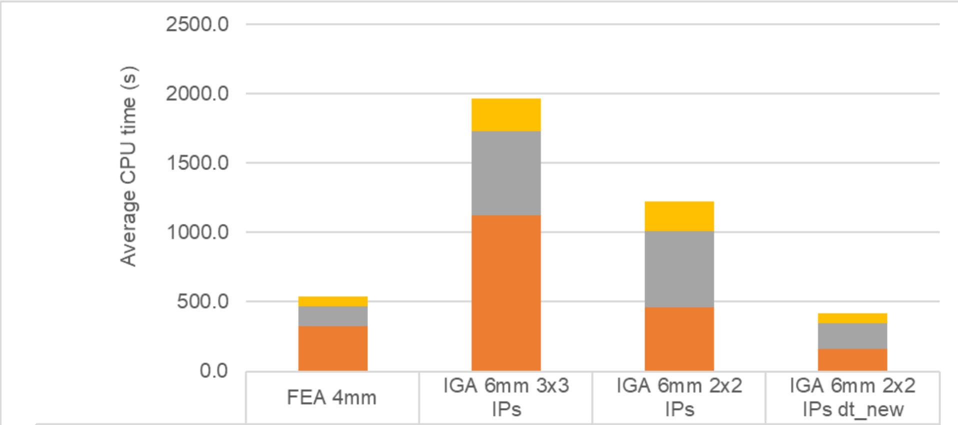


Fig.12: Dynamic buckling of an energy-absorbing component: Resultant force over time plot for the FEA reference solution and the IGA configurations.



	FEA 4mm	IGA 6mm 3x3 IPs	IGA 6mm 2x2 IPs	IGA 6mm 2x2 IPs dt_new
Misc	72.5	236.0	211.4	72.6
Contact Algorithm	143.3	602.6	550.6	184.3
Element Processing	322.0	1122.3	460.1	157.1
Initialization	0.1	1.8	1.2	1.7
Total	537.9	1962.7	1223.3	415.6

Fig.13: Dynamic buckling of an energy-absorbing component: Comparison of average CPU time per processor with a time step size determined by LS-DYNA (no predefined time step).

During the analysis, the rear part of the vehicle is represented via a correspondingly attached point mass. The vehicle structure impacts the rigid wall with 15m/s. For this model a fixed time step size is used together with an appropriate mass scaling strategy. Since the IGA shell models would allow for an initial time step larger than the standard time step size used here, less mass scaling than for the FEA components is required during the analysis. The overall increase in computational time compared to the pure FEA model is insignificant, because the IGA components only take up a small proportion of the whole model.

The vehicle structure after impact at $t = 100$ms is depicted in Fig.15 with the significantly deformed IGA components highlighted in red and gold. As can be seen, the connections between the IGA longitudinal members and the adjacent FEA components work as intended and therefore transmit forces along the upper load path of the structure. The general deformation behavior of the hybrid model agrees well with the pure FEA model, although this vehicle substructure was found to be sensitive to parameter variations. Therefore, detailed comparisons are omitted. Here it should be noted that this vehicle substructure with crucial components for a front crash missing, was artificially generated only for the purpose of this study and does not represent the behavior of the actual vehicle. Nevertheless, this example still shows the possibility to perform practical crash simulations of hybrid IGA/FEA models, generated on the basis of existing FEA models via a simple one-by-one component exchange without further model modifications. Due to these promising results, the next step is to run crash simulations of a hybrid full vehicle model.

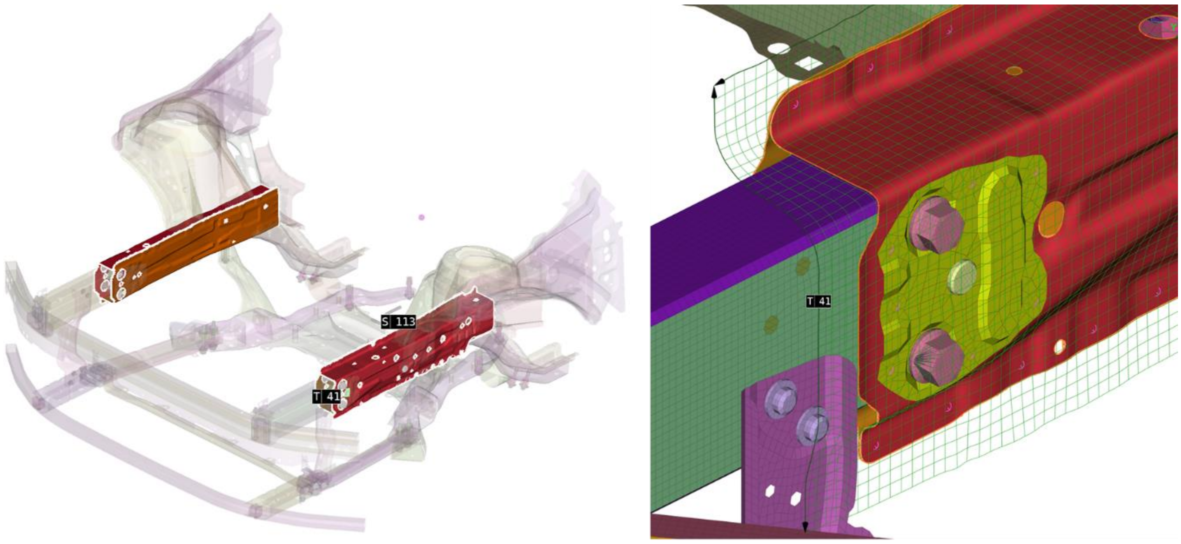


Fig.14: Hybrid IGA/FEA vehicle front end structure: IGA components highlighted in red and gold, FEA components faded (left), IGA longitudinal member with indicated untrimmed patch (right).

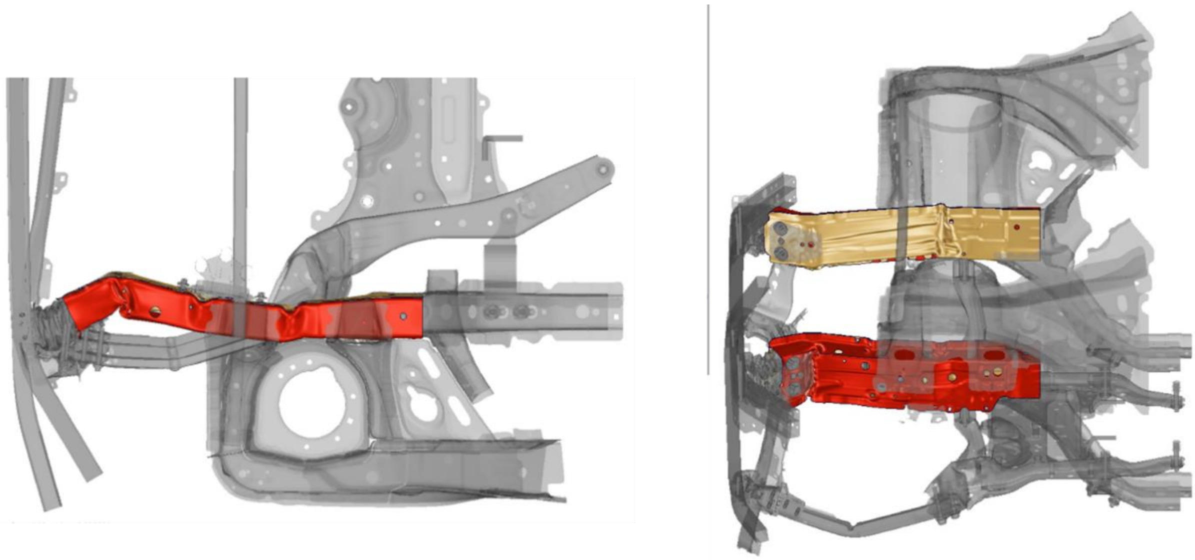


Fig.15: Hybrid IGA/FEA vehicle front end structure: Deformed shape after impact on rigid wall with IGA longitudinal members highlighted in red and gold, FEA components faded.

5 Summary and outlook

This paper demonstrated the possibility to use isogeometric analysis with existing CAD models and within existing virtual development processes by means of hybrid IGA/FEA models generated with minimum effort. Although this approach does not yet exploit the full potential of IGA regarding integration of design and analysis, users may still benefit from a more accurate geometry description, higher-order and higher-continuity elements, a larger explicit time step size, faster model generation with ANSA or smarter modeling techniques.
The basis for the industrial application of isogeometric analysis, are the recently developed `*IGA` keywords, able to capture the data structure of B-Rep CAD models including geometry and topology information. Several new IGA capabilities in LS-DYNA allow applying existing connection technology from FEA also to isogeometric shells. In this way, hybrid IGA/FEA models can be generated from existing FEA models through a simple one-by-one exchange of individual components. Two crash-type

simulations of BMW models demonstrated the possibility to perform accurate explicit dynamic analysis of trimmed NURBS-based shell models with computational costs comparable to standard FEA.

Due to the promising results obtained with the hybrid IGA/FEA substructure model, the next step is to insert IGA components into a full vehicle model and to conduct more detailed comparisons with pure FEA models. Also further studies regarding plasticity, damage evolution and failure of isogeometric elements are planned. Despite the already decent performance of isogeometric shells, the potential for further improvements in the future is still high, for example more efficient implementations or optimized integration rules to mention but a few.
The IGA models in this paper were restricted to trimmed NURBS shells. However, the basic concept of trimming and the **`*IGA`** keywords can also be extended to NURBS solids, see the paper “Isogeometric Analysis on Trimmed Solids: A B-Spline-Based Approach Focusing on Explicit Dynamics” by Meßmer et al. [9] presented at this conference. This trimmed solid approach is not only expected to speed up the model generation process (compared to hexahedron FEA solids), but to also provide superior analysis capabilities (compared to linear tetrahedron FEA meshes). With IGA shell and solid models able to represent complex industrial geometries, most of a vehicle could be modeled with isogeometric components. The increasing proportion of isogeometric components within a vehicle model, could then slowly lead towards the visionary goal of a real IGA-type modeling strategy and development process for a tight integration of design and analysis.

6 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] ANSA – Pre Processor: www.beta-cae.com/ansa.htm

[3] LS-DYNA dev Keyword Manual Vol I – Latest DRAFT Versions of LS-DYNA Manuals: www.lstc.com/download/manuals

[4] 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.

[5] Breitenberger, M.: "CAD-integrated design and analysis of shell structures", PhD Thesis, Technical University of Munich, Germany, 2016.

[6] Leidinger L.F., “Explicit Isogeometric B-Rep Analysis for Nonlinear Dynamic Crash Simulations”, PhD Thesis, Technical University of Munich, Germany, 2020.

[7] Hartmann, S., Leidinger, L.F., Benson, D., Li, L., Nagy, A., Pigazzini, M.: “Enabling the Analysis of Topologically Connected Multi-Patch Trimmed NURBS Shells in LS-DYNA”, 12th European LS-DYNA Conference 2019, Koblenz, Germany.

[8] Hartmann, S., Benson, D., Lorenz, D.: “About Isogeometric Analysis and the new NURBS-based Finite Elements in LS-DYNA”, 8th European LS-DYNA Conference 2011, Strasbourg, France.

[9] Meßmer, M., Leidinger, L., Hartmann, S., Bauer, F., Duddeck, F., Wüchner, R., Bletzinger, K.-U.: “Isogeometric Analysis on Trimmed Solids: A B-Spline-Based Approach Focusing on Explicit Dynamics”, 13th European LS-DYNA Conference 2021, Ulm, Germany.