Trimmed IGA Solids in LS-DYNA: CADFEM Findings

Dr.-Ing. Yury Novozhilov[1], Dr.-Ing. Lukas Leidinger[2]

[1]CADFEM Germany GmbH
[2]DYNAmore GmbH, an Ansys company

1 Abstract

The finite element method (FEM) remains the standard approach for nonlinear dynamic simulations; however, explicit time integration imposes strict limitations on element size and computational efficiency. Thin-walled structures with complex molded geometries are particularly challenging: shell idealizations are impractical, hexahedral meshes are often unattainable, and linear tetrahedral elements may suffer from severe shear locking. Recent advances in Iso-Geometric Analysis (IGA) provide an alternative framework for addressing these issues.
This paper presents an evaluation of trimmed (immersed) IGA solids in LS-DYNA. Benchmark studies include quasi-static bending of a thin plate, bumper impact against a rigid mast, head impact on a vehicle dashboard, and a drop test of a multi-part assembly. Results demonstrate that IGA is able to reproduce bending behavior and plastic deformation patterns with accuracy comparable to higher-order fully integrated tetrahedral elements. At the same time, IGA significantly reduces preprocessing effort and improves computational performance, with speedups of up to an order of magnitude.
The findings confirm that trimmed IGA solids combine robustness, accuracy, and efficiency, offering a promising tool for industrial applications where conventional FEM discretizations are limited.

2 Motivation

The finite element method (FEM) for nonlinear dynamic problems has been extensively studied and successfully applied for decades. Nevertheless, the explicit time integration scheme imposes strict limitations on its practical use. The integration step is typically very small, often measured in microseconds. The main challenge in explicit simulations is therefore to enlarge the stable time step while ensuring efficient computations at each iteration.
In the classical FEM with a Lagrangian mesh formulation, the computational geometry strictly follows the model geometry (boundary-fitted mesh). For three-dimensional solid formulations, the minimum element size is determined by the smallest geometric features that remain after defeaturing. For example, if a thin stiffening rib plays a significant structural role and cannot be removed, its thickness dictates the maximum allowable element size. Consequently, this thickness also controls the stable time step of the solver.
To maximize efficiency, robust and stable elements are employed. The HEX 1 element (see Fig. 1) – an eight-node linear hexahedron with a single integration point – is widely regarded as nearly ideal for explicit analysis in terms of accuracy-to-performance ratio, absence of shear or volumetric locking, number of supported material models. However, there is still no general algorithm capable of generating high-quality all-hexahedral meshes for models with complex topologies. When tetrahedral elements are used instead, severe volume and shear locking effects arise. These can mainly be mitigated with fully integrated higher-order elements such as TET 16/17 in LS-DYNA (see Fig. 1), which are more computationally expensive, require a significantly smaller time step and less stable under large distortions. Furthermore, curved structures require multiple integration points across their thickness, preferably nonlinear shape functions to capture curved surface. Indeed, with an insufficient number of points along the thickness, it is impossible to correctly restore the stress gradient (and, consequently, the deformations) and capture the correct bending stiffness or plasticization of the cross-section.

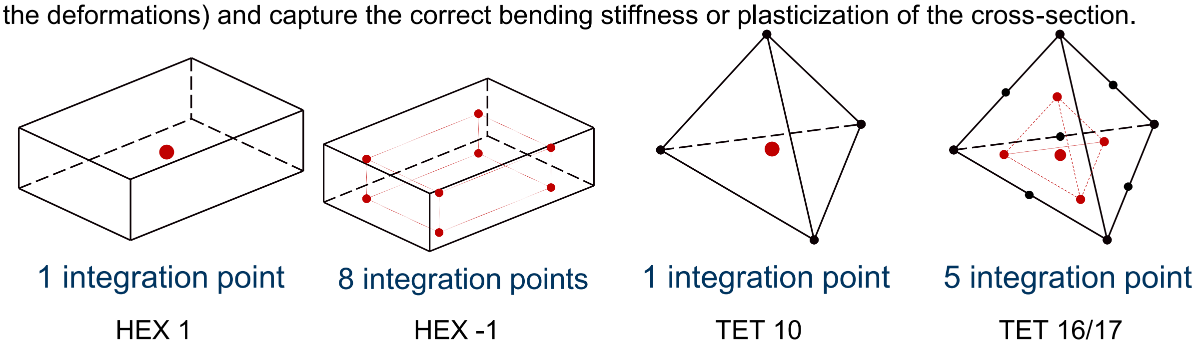


Fig.1: Some commonly used finite elements in LS-DYNA

A particular class of components poses both difficulties simultaneously: thin-walled structures manufactured by casting or molding. Such products are encountered daily, from consumer electronics housings to structural elements in modern vehicles. Their complex shapes make shell idealizations impractical, while generating an adequate hexahedral mesh is often impossible. At the same time, their small thickness makes them highly sensitive to bending, which excludes the use of linear tetrahedral elements due to shear locking. Until recently, the only option was to accept reduced performance and rely on TET16/17 elements. Recent advances in Iso-Geometric Analysis (IGA), however, provide a promising alternative approach to these challenges.

3 Implementations of IGA

Iso-Geometric Analysis (IGA) is a computational approach that enables direct integration of finite element analysis (FEA) into NURBS-based CAD design environments (NURBS = Non-Uniform Rational B-Spline). LS-DYNA has one of the longest histories of IGA development in the CAE market. The first paper on IGA was published in 2005 [1], while the first NURBS-based shell formulations were implemented in LS-DYNA as early as 2011 [2]. Based on promising results obtained from these first implementations in LS-DYNA, the IGA capabilities were continuously extended including solid elements, explicit and implicit analysis, contact, boundary conditions, trimmed NURBS shells, and unstructured boundary-fitted approaches through the Bézier extraction format.
In the last few years, significant progress was especially made with trimmed multi-patch NURBS shells, resulting in the possibility to run full vehicle crash simulations with hundreds of IGA shell components [ref Bauer et. al]. To address to above-mentioned shortcomings of FEA solid elements, the IGA trimming approach was recently extended to tri-variate solid descriptions. With this trimmed (immersed) IGA solid approach, the product (model geometry) is "immersed" into a predefined computational domain (background grid). The solver algorithm automatically determines which cells (elements) of the computational domain contribute to the structural response and to what extent (see Fig. 2, CAD Source: GrabCAD, used with permission of Nishanth Kumar). Based on that, integration points for trimmed and untrimmed elements are generated. For the user, this immersing process may show similarities to the initialization of material distribution in Arbitrary Lagrangian–Eulerian (ALE) analysis. More detailed information about trimmed IGA solid approaches can be found in [3 - 5].

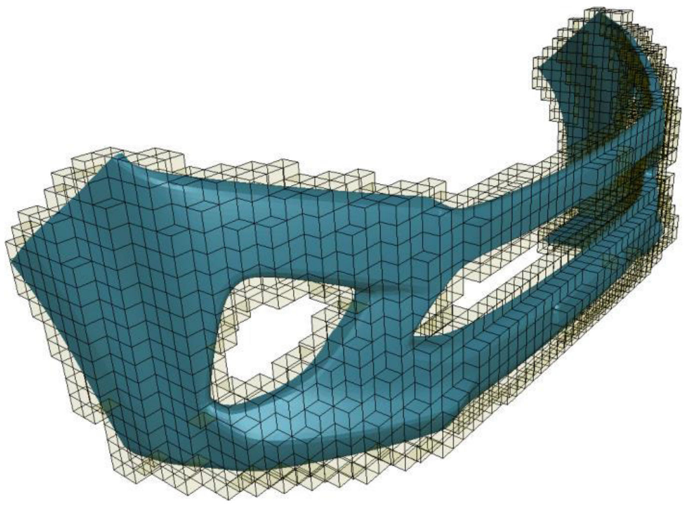


Fig.2: Trimmed (immersed) IGA solid model of a bumper.

The trimmed (immersed) IGA formulation for solid elements is undergoing active development. Use of a R16-beta or DEV build of the solver is recommended, together with special DEV keywords. The syntax is being optimized and may change. A detailed keyword description is therefore omitted. The focus is placed on solving representative test problems and on analyzing the obtained results.
In the current implementation, direct import of CAD geometry into the solver is not yet supported. Some developments in automatic CAD conversion (STEP format) to trimmed IGA solids are available in LS-PrePost 4.13 but studying them is beyond the scope of this research. Model geometry is therefore supplied by an interpolation mesh, that is, a faceted representation composed of linear elements (see Fig. 4). This mesh is used to position material, to define contact interfaces, and to provide visualization. As a result, any post-processor can display LS-DYNA results obtained with an IGA setup.
The background mesh that carries out the computation is generated by the solver based on the model extent and the element size defined by the user. The element size is a model parameter. No manual

remeshing is required, only solver initialization is needed. Failures of model mesh generation become highly unlikely. The same model can therefore be used for a proof-of-concept study with a coarse mesh and for a final high-fidelity analysis with a refined mesh. The two runs differ by a single parameter value.

4 Flat panel quasi-static bending

A classical benchmark of thin plate bending is considered to evaluate the ability of the trimmed IGA solid formulation to resist shear locking and to reproduce bending stiffness (see Fig. 3). The panel dimensions are L x W x T = 50 x 20 x 2 mm. The loading is applied up to the onset of material plasticity.
One of the key features of trimmed IGA solids is revealed in this example: the model thickness no longer limits the element size (see Fig. 4). Elements several times larger than the minimum structural thickness can be used. In this case, element size is up to four times greater than the panel thickness. For explicit time integration, this is particularly important, since the stable time step is governed by the IGA element size.

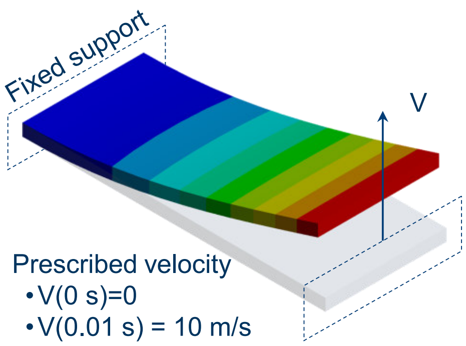


Fig.3: Setup of quasi-static bending test for thin flat panel

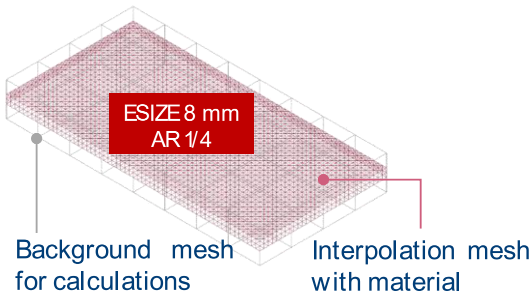


Fig.4: Discretization of the flat panel with IGA elements

The calculations are performed with reduced integration (eight integration points per element) and a second-order polynomial degree. Results are compared against a reference HEX 1 model, discretized with three elements through the thickness (six integration points across the section). The elastic response of the IGA model agrees well with the reference solution, even at aspect ratio (AR) 1/4 (see Fig. 5). Deviations appear after yielding, when only two integration points are available across the IGA thickness, which is insufficient to capture through-thickness plastic gradients. Nevertheless, the results demonstrate that stiffness predictions are obtained efficiently, without evidence of shear locking.

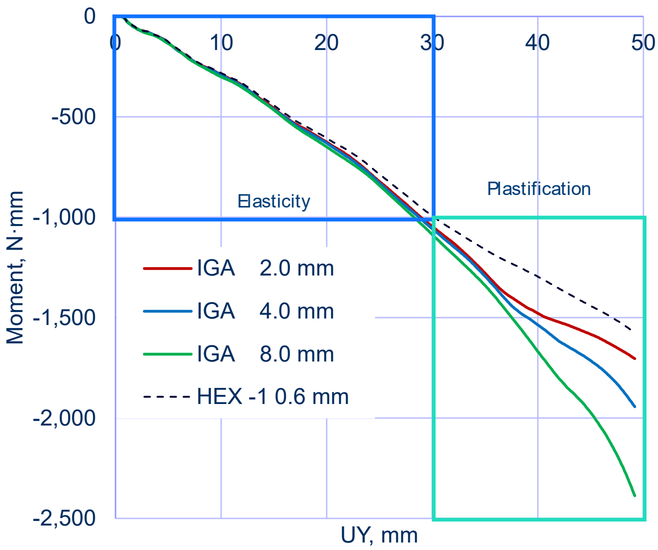


Fig.5: Reaction force response during bending

Visualization of plastic strain further highlights this limitation. For the AR 1/1 case, the effective plastic strain distribution obtained with IGA matches that of the HEX -1 reference (see Fig. 6). With coarser IGA meshes, however, plastic flow through the thickness cannot be fully reproduced.

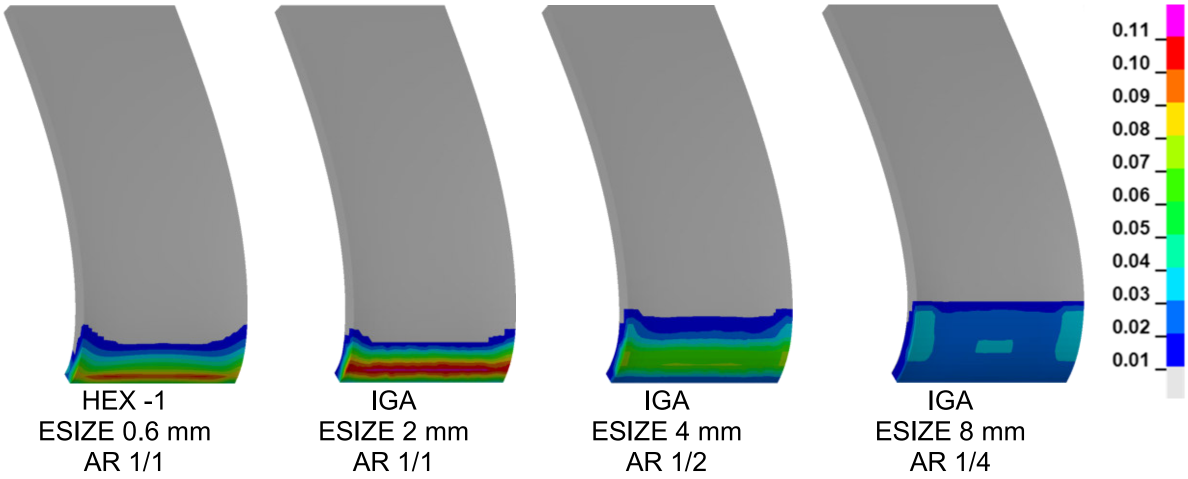


Fig.6: Effective plastic strain distribution, isometric bottom view

5 Bumper impact on mast

The bumper geometry is considered as a representative thin-walled structure with complex molded surfaces (see Fig. 7, CAD Source: GrabCAD, used with permission of Nishanth Kumar). The average thickness of the model is 3 mm. The simulation involves the impact of the bumper against a rigid mast with a diameter of 300 mm. The material is modeled using **`*MAT_024`** with plastic hardening.

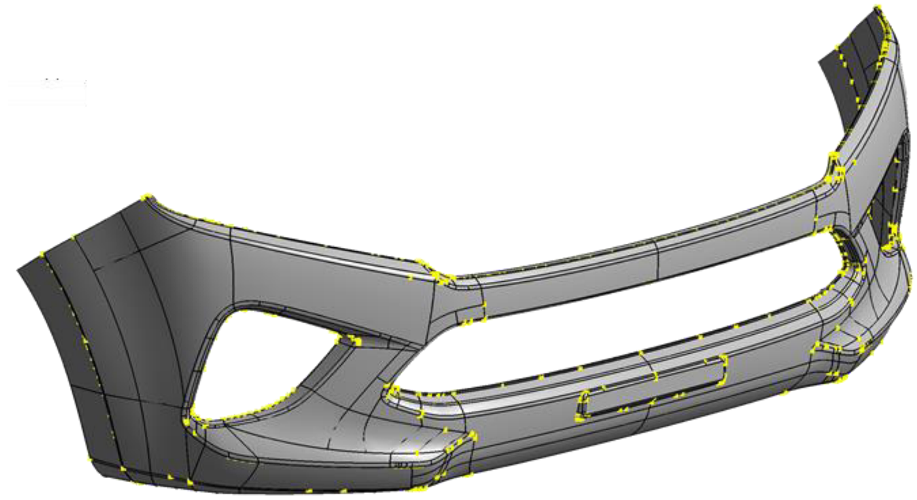


Fig.7: CAD model of bumper with highlighted close vertices (Ansys Mechanical standard tool for highlighting dirty geometry)

The deformed configuration after impact is shown in Fig. 7. The model discretized with linear TET 13 elements is unable to capture the structural deformation due to pronounced shear locking, particularly at poor aspect ratios (see Fig. 8). This once again confirms the inadequacy of linear tetrahedral elements with a single integration point for thin-walled bending-dominated structures. By contrast, the quadratic fully integrated TET 16 formulation is able to reproduce bending stiffness correctly even at an aspect ratio of 1/3 (with an in-plane element size of 9 mm for a wall thickness of 3 mm). The reliability and accuracy of TET 16 results for this model were investigated by CADFEM as part of the training course “Practice-Oriented Meshing in Ansys LS-DYNA” [6].
Both IGA discretization variants also reproduce the deformation response of the bumper (see Fig. 8). The patterns of effective plastic strain observed in IGA are generally consistent with those of the TET16 solution (see Fig. 9).

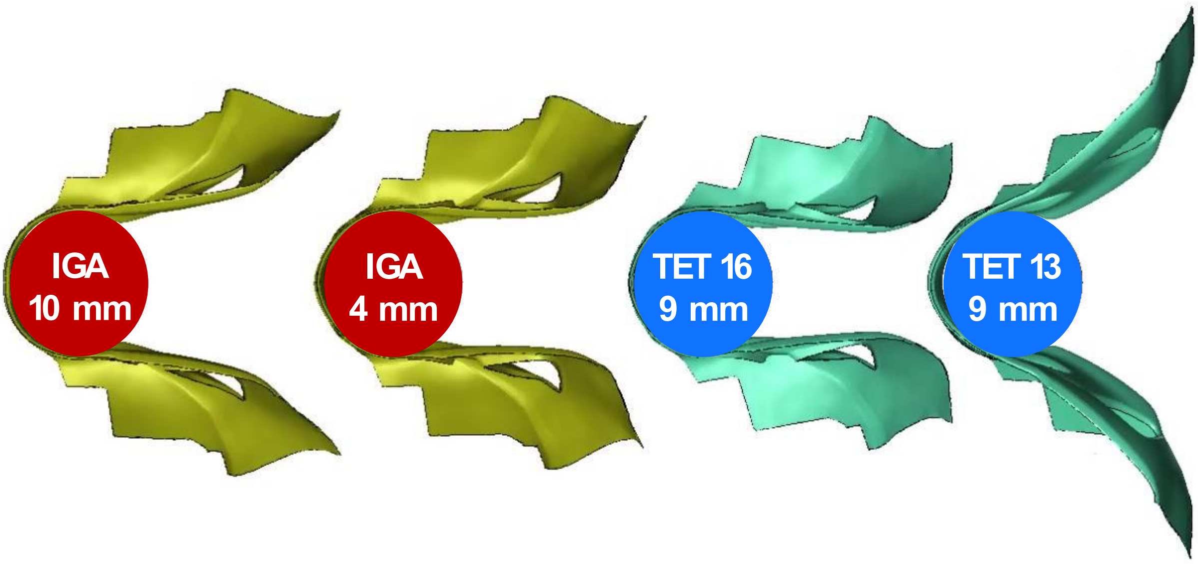


Fig.8: Bumper deformation after mast impact showing shear locking in TET13 mesh

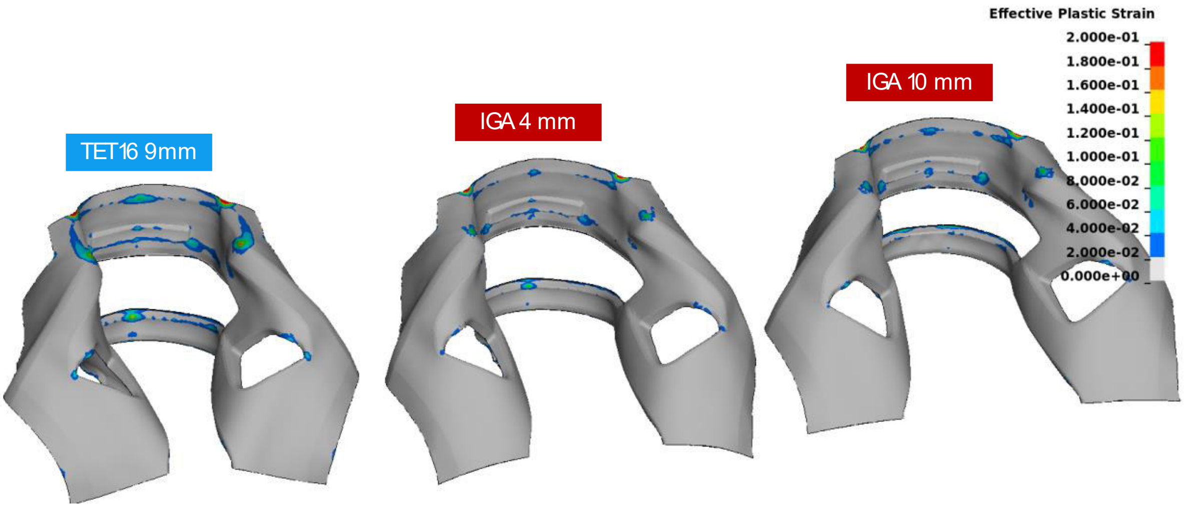


Fig.9: Effective plastic strain distribution in bumper after impact

The performance comparison demonstrates the main advantage of the IGA approach (see Fig. 10). Despite its more complex formulation, IGA permits larger time steps and requires fewer elements. As a result, the proof-of-concept IGA model delivers accuracy comparable to the TET 16 formulation while achieving an x21 (!!!) reduction in computational time.

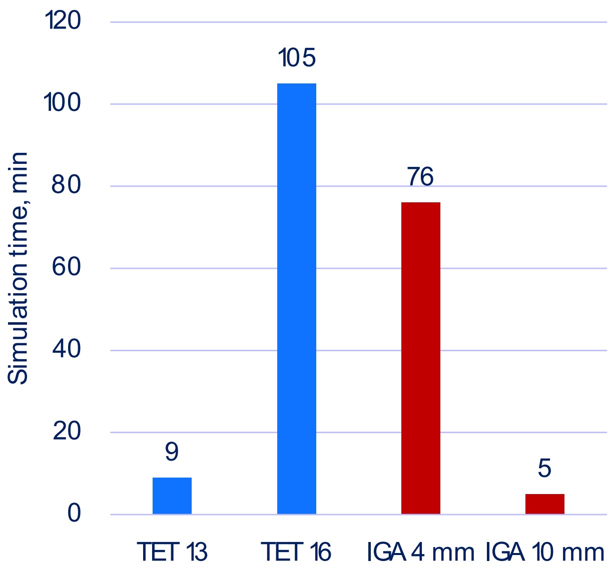


Fig.10: Computational performance comparison of bumper models

6 Dashboard head impact simulation

A dashboard is selected as the next benchmark. In vehicle development, one of the certification requirements is the head impact test, performed with a special dummy that simulates the human head. The dashboard model is representative of a thin-walled molded component, with a thickness of approximately 3 mm in the impact region (see Fig. 11, CAD Source: GrabCAD, used with permission of CQ Xie).

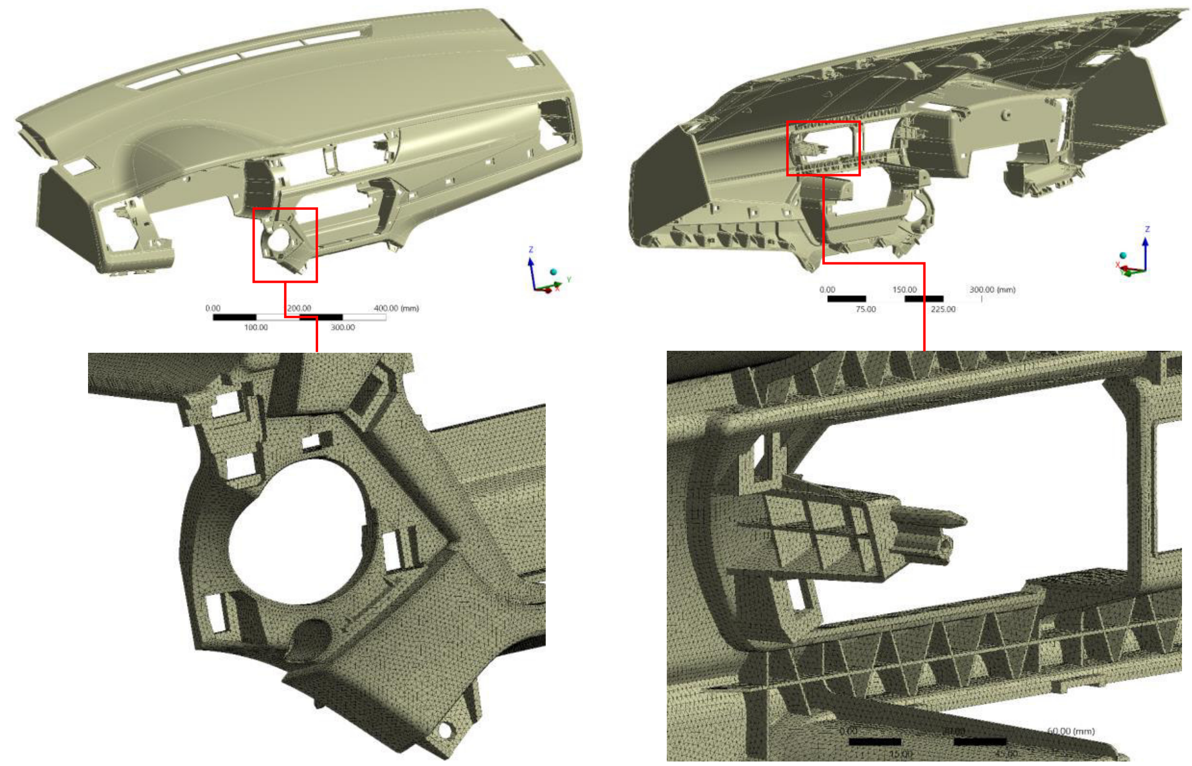


Fig.11: CAD model of car instrument panel and corresponding interpolation mesh

For comparison, two models are analyzed: a finite element model discretized with TET16 elements of 2 mm size, and a trimmed IGA solid model with an element size of 10 mm. Both models reproduce the overall deformation and response with similar accuracy. The TET16 discretization benefits from a larger number of integration points across the thickness, while IGA provides comparable results with significantly coarser discretization (see Figs. 12). In this case, the IGA solution is obtained more than five times faster. Refinement of the IGA model requires modification of only a single parameter in the solver input file.

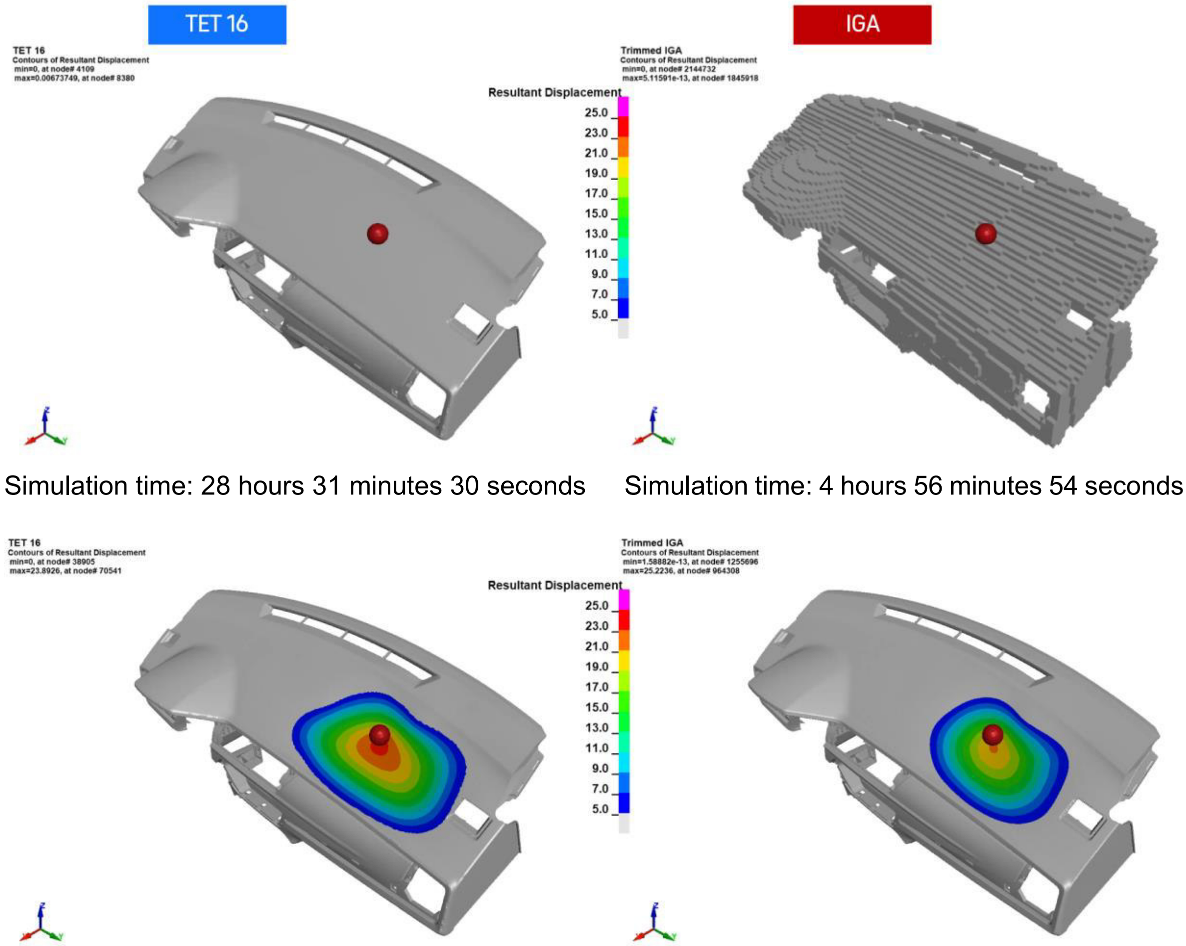


Fig.12: Overview of dashboard model and selected simulation results

A shifted impact location, with the floor acting as the support, highlights the so-called “cross-talk” effect in trimmed IGA solids (see Fig. 13). This effect, also reported for trimmed IGA shells, arises when coarse meshes are used to model relatively small/thin geometric features. In that case, artificial stiffening caused by basis functions with support on disconnected material domains is observed. At present, this limitation prevents the use of trimmed IGA solids for simulations of crushing in models with many closely spaced stiffening ribs. A temporary workaround is to divide the geometry into several parts, model those parts with individual background grids (patches), and subsequently connect these parts with a penalty-based tied contact (e.g. **`*CONTACT_TIED_SURFACE_TO_SURFACE_OFFSET`**), since the cross-talk effect is confined within a single patch.

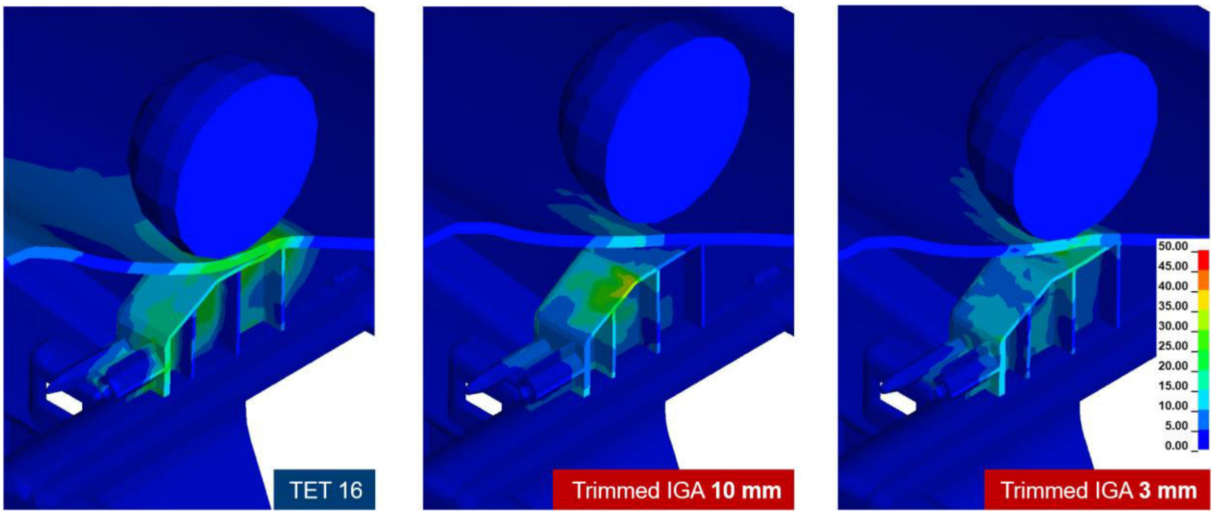


Fig.13: Artificial stiffening in dashboard impact caused by cross-talk effect

7 Cross-talk effect

The cross-talk effect arises from trimming and the fact that disjoint domains are represented through the same basis function. Cross-talk is not specific to IGA, but IGA is more prone to cross-talk because of the higher order and higher continuity of IGA basis functions (B-splines or NURBS) and the correspondingly larger support (see Fig. 14).

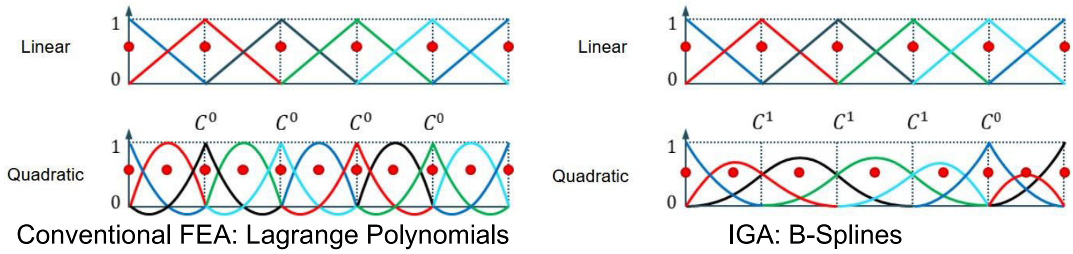


Fig.14: Comparison of IGA to FEA

To illustrate this behavior, a simple compression test of a spring is performed. The spring thickness is 0.5 mm, and self-contact is modeled using a single surface contact definition (see Fig. 15). Several IGA element sizes are tested, ranging from 2.0 mm down to 0.5 mm.

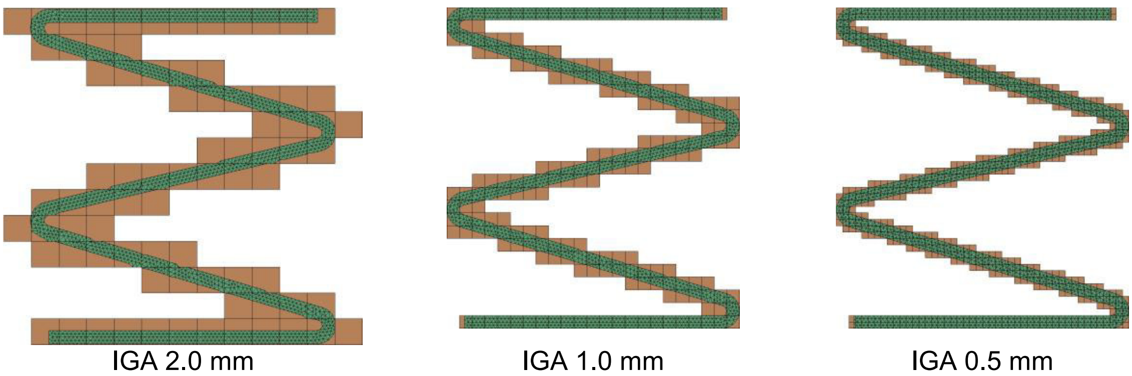
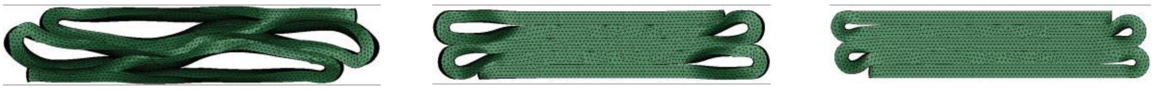


Fig.15: Cross-talk effect in the spring benchmark: model discretization

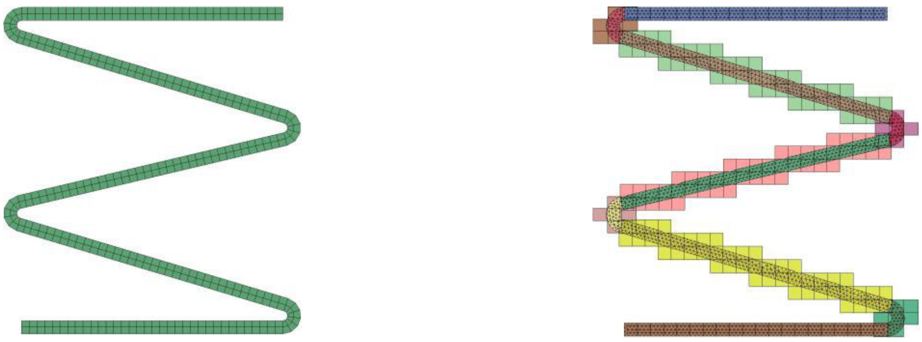
The results reveal the cross-talk phenomenon clearly. Even in the final frame, without following the full animation, the spring corners cannot compress fully (see Fig. 16). This limitation is independent of contact definitions. Removing self-contact does not resolve the issue, as the artificial stiffness originates from the underlying spline-based elements and trimming.



IGA 2.0 mm IGA 1.0 mm IGA 0.5 mm

Fig.16: Cross-talk effect in the spring benchmark: deformed shape

The cross-talk effect is considered important but not critical. It diminishes naturally with mesh refinement. With one element through the thickness, the effect is already minimized (see Fig. 16 for IGA 0.5 mm and Fig. 18 for HEX -1 with 0.6 mm elements). For problems requiring precise description of crushing, such as simulations of cellular fillers, the existing recommendation of using at least one full IGA element through the thickness provides sufficient accuracy. In most industrial impact and drop-test simulations of consumer products, such localized full compression is rare, and coarser meshes remain acceptable.



HEX -1 0.6 mm IGA 1.0 mm split

Fig.17: Cross-talk effect in the spring benchmark workaround: model discretization

Cross-talk can also be mitigated by reducing the continuity of spline basis functions. One practical method is to divide the model into several patches connected by penalty-based contacts (see Figs. 17 and 18 for the IGA 1.0 mm split model). Each interface breaks spline continuity, preventing cross-talk propagation across subdomains. This demonstration highlights potential strategies that could be implemented natively in the solver, eliminating the need for manual splitting. It also confirms that assemblies with small gaps between components can be modeled in IGA without fundamental restrictions.

HEX -1 0.6 mm IGA 1.0 mm split

Fig.18: Cross-talk effect in the spring benchmark workaround deformed shape

8 Cyclone assembly droptest

Since cross-talk does not affect the behavior of the assembly, it becomes possible to perform this calculation. As a final benchmark, a drop test of an assembly consisting of 28 parts is selected (see Fig. 19, CAD Source: GrabCAD, used with permission of AꓘTWOꓘKsSIMO). This case is intended to evaluate the maturity of the trimmed IGA solid formulation, specifically its ability to initialize multiple IGA parts and to robustly handle contact interactions within an assembly.

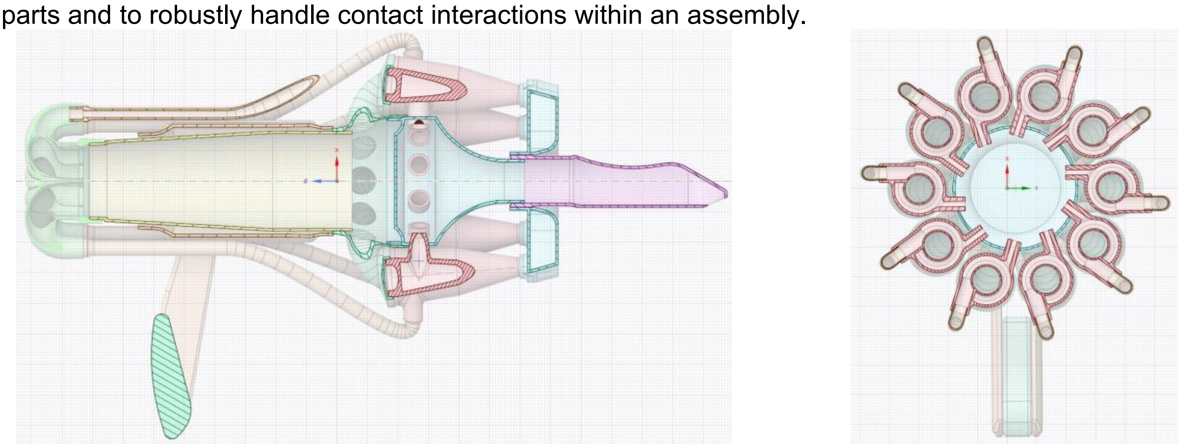


Fig.19: Cyclone assembly in crossection

The creation of an IGA assembly follows a straightforward procedure. Each part is assigned to its own background grid (patch), see Fig. 20. Although it is possible to define a single large parameterization space for the entire assembly, this approach results in excessive memory consumption during initialization. The solver discards completely empty spaces automatically, but RAM usage remains unnecessarily high. For this reason, it is advisable to define the smallest possible background grids, a process that can be easily automated.
For comparison, both IGA and TET 16 models are generated with a target element size of 3 mm. Owing to mesh topology, the TET 16 simulation requires a stable time step of 4.5E-8 s, while the IGA simulation achieves a step of 5.5E-7 s. This corresponds to more than a tenfold increase in the time step.

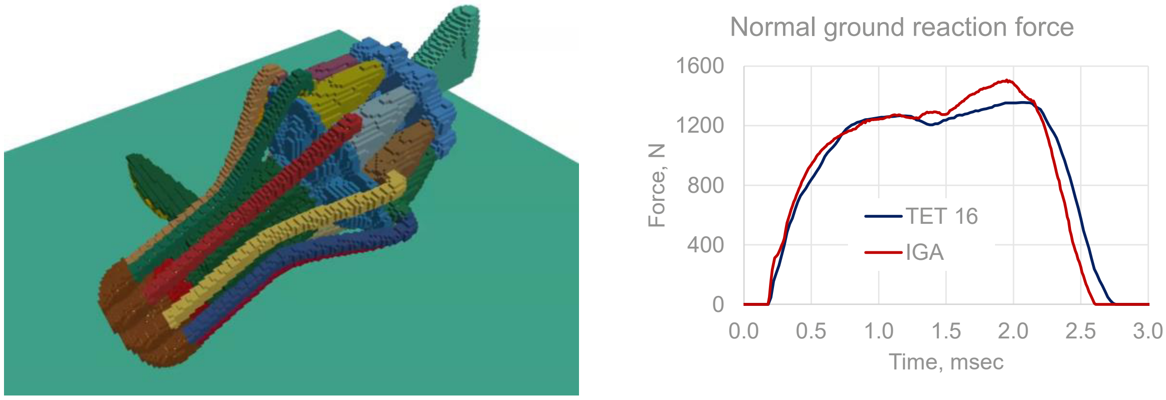


Fig.20: Trimmed IGA solid model of cyclone assembly and drop test results

The trimmed IGA solid calculation completes approximately six times faster than the TET 16 solution. The speedup is not directly proportional to the time-step increase, since IGA computations per step are more resource-intensive and the implementation is not yet fully optimized. Nevertheless, the performance advantage is evident, and it is achieved in addition to the reduction of preprocessing effort due to the absence of mesh generation.

9 Additional tools

During the course of this study, a prototype ACT (Ansys Customization Toolkit) extension for Workbench LS-DYNA is developed to integrate trimmed IGA solids into modern workflows as seamlessly as possible.
The primary function of this extension is to introduce the necessary keywords that allow the conversion of a conventional finite element model with linear elements into a trimmed IGA solid model (see Fig. 21). Due to the specific solver implementation, existing contact algorithms created in Workbench LS-DYNA remain fully operational, and standard post-processing tools can be used without modification.

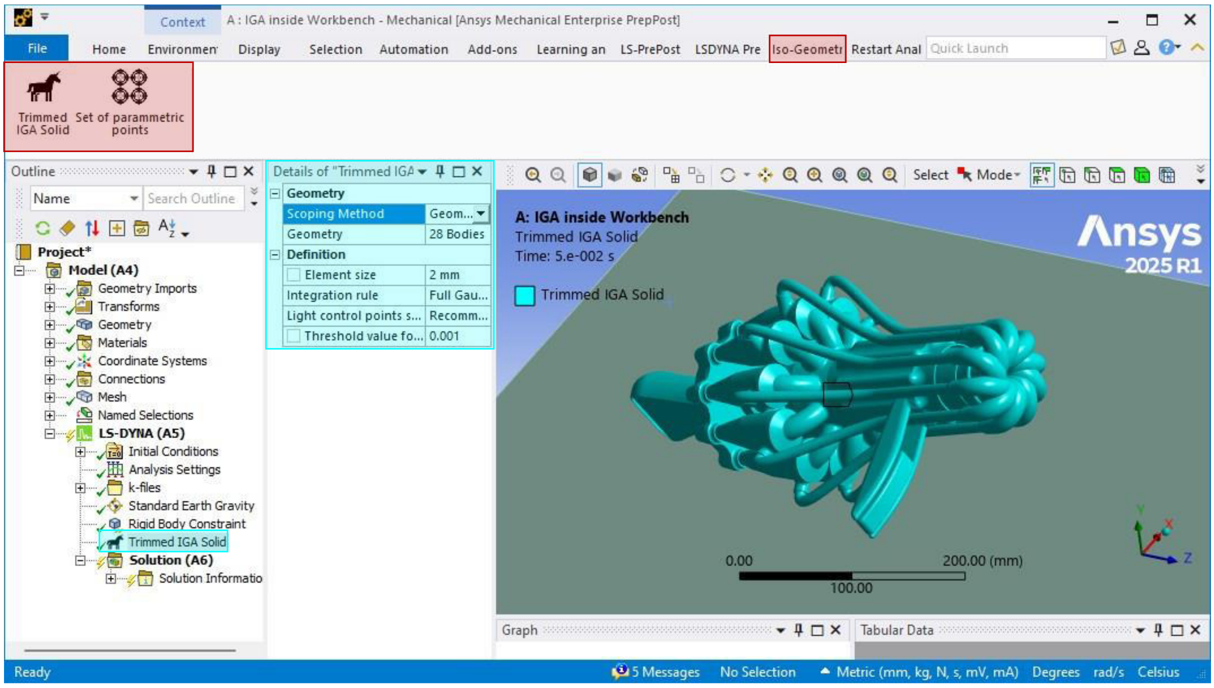


Fig.21: ACT interface for initialization of IGA solid models

A second function of the extension is the ability to create **`*IGA_POINT_UVW`** objects associated with selected geometric entities (see Fig. 22). These points can then be grouped into **`*SET_IGA_POINT_UVW`** collections. With this functionality, standard keywords can be employed to define boundary conditions, prescribe displacements, velocities, and accelerations, as well as to apply external loads.

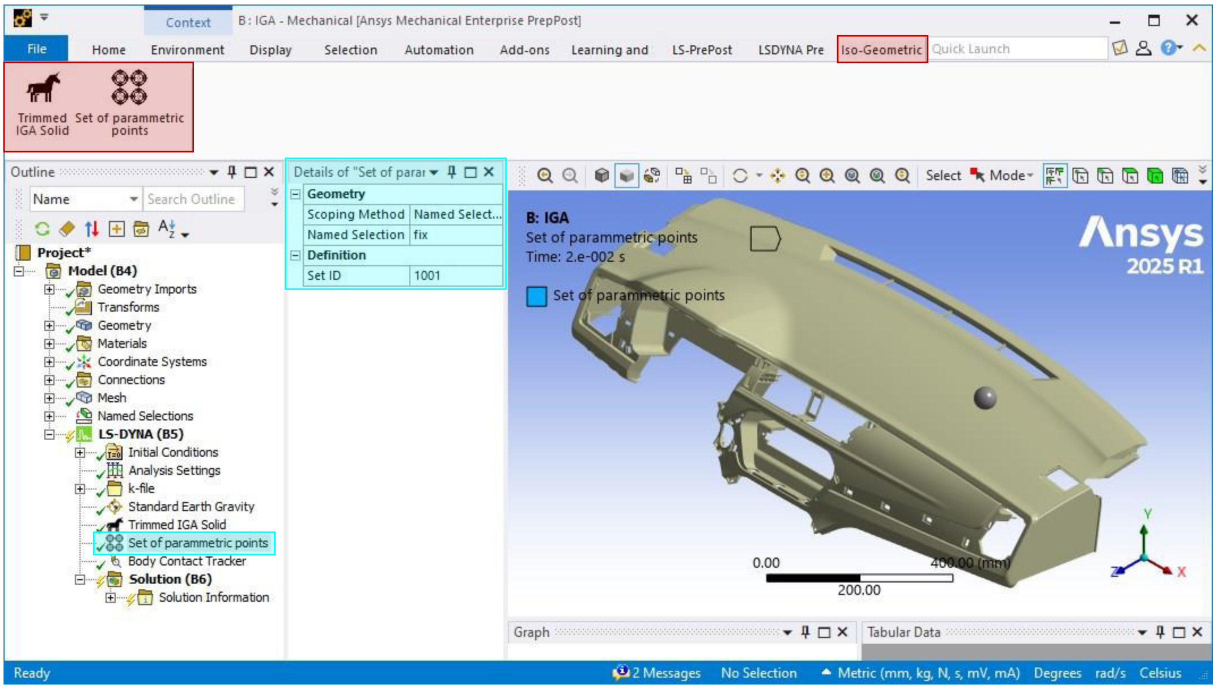


Fig.22: ACT interface for generation of ***`*IGA_POINT_UVW`*** *and* ***`*SET_IGA_POINT_UVW`***

The prototype extension significantly accelerates the testing of IGA capabilities, automates the setup of complex industrial problems, and minimizes the likelihood of human error in keyword syntax. It enables the efficient translation of real-world models into trimmed IGA formulations while maintaining compatibility with established solver features.

10 Conclusion

This study presents the results of CADFEM's evaluation of trimmed IGA solids, carried out with the informational support of Ansys. The investigations are performed from the perspective of design engineers addressing practical industrial problems, rather than developers with direct access to the source code. Such an approach provides an effective stress test for assessing the maturity of the technology.

The results confirm that IGA fulfills the expectations placed upon it. The method enables rapid construction of proof-of-concept models and, by modifying only a single parameter, refinement to production-level analyses. One of the most demanding stages of model preparation – generation and refinement of high-quality meshes – is thus automated.

In terms of accuracy, IGA delivers results comparable to HEX -1 elements. Its performance also resembles that of HEX -1, with the notable advantage that IGA can be applied to geometries for which hex meshing is prohibitively difficult. The element size is decoupled from the smallest geometric features, which allows the method to be used efficiently for complex thin-walled structures.

Certain limitations remain. The use of larger element sizes and larger time steps requires adjustment of penalty function parameters in contact formulations. Guidelines for such tuning already exist, and automated procedures can be expected in future solver versions. In addition, the "cross-talk" effect observed in coarse meshes affects «contact» behavior. While solutions have been developed and implemented for shell formulations [7, 8], corresponding developments for solid elements are anticipated. Until then, excessively coarse meshes are not recommended for problems involving crushing or localized folding. The element size in such cases should remain comparable to the structural thickness to ensure sufficient through-thickness integration.

Overall, trimmed IGA solids provide a robust and efficient alternative to conventional FEM discretization for complex geometries. The method combines accuracy, speed, and simplicity of model preparation, and it is expected to play an increasingly important role in industrial applications.

11 Literature

[1] T.J.R. Hughes, J.A. Cottrell, Y. Bazilevs, „Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement", Computer Methods in Applied Mechanics and Engineering, Vol. 194, 2005, 4135-4195, https://doi.org/10.1016/j.cma.2004.10.008

[2] D.J. Benson, Y. Bazilevs, M. C. Hsu, T.J.R. Hughes, „Isogeometric shell analysis: The Reissner–Mindlin shell", Computer Methods in Applied Mechanics and Engineering, Vol. 199, 2010, 5-8, 276-289, https://doi.org/10.1016/j.cma.2009.05.011

[3] S. Hartmann, L. Leidinger, F. Bauer, D. Benson, A. Nagy, L. Li, M. Pigazzini, L. Nguyen, "Enabling Productive Use of Isogeometric Shells in LS-DYNA", 14th European LS-DYNA Conference 2023, Baden-Baden, German, https://www.dynalook.com/conferences/14th-european-ls-dyna-conference-2023/iga/leidinger_dynamore.pdf/view

[4] S. Hartmann, L. Leidinger, F. Bauer, D. Benson, L. Li, A. Nagy, L. Nguyen, M. Pigazzini, "Updates on trimmed IGA B-Spline Solids", 2024 International LS-DYNA Conference, Metro Detroit, Michigan, USA, https://www.dynalook.com/conferences/17th-international-ls-dyna-conference-2024/iga/hartmann_ansys.pdf/view

[5] Manuel Meßmer, Tobias Teschemacher, Lukas F. Leidinger, Roland Wüchner, Kai-Uwe Bletzinger, „Efficient CAD-integrated isogeometric analysis of trimmed solids", Computer Methods in Applied Mechanics and Engineering, Vol. 400, 2022,115584. https://doi.org/10.1016/j.cma.2022.115584

[6] Y. Novozhilov, "Practice-Oriented Meshing in Ansys LS-DYNA" seminar materials, https://www.cadfem.net/en/training/practice-oriented-meshing-in-ansys-ls-dyna-18696.html

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

[8] Z. Lian, L.F. Leidinger, S. Hartmann, F. Bauer, M. Pabst, C. Krisadawat, R. Wüchner, "Cross-talk effects in trimmed isogeometric shells and the control point duplication approach", Computer Methods in Applied Mechanics and Engineering, Vol. 438, 2025, 117849, https://doi.org/10.1016/j.cma.2025.117849