Isogeometric Analysis in LS-DYNA R13 - key steps towards industrial applications

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

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

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

1 Introduction

Hughes et al. [1] introduced the term i*sogeometric analysis* (IGA) in the framework of f*inite element analysis* (FEA). Its main idea is to use the same mathematical description for the geometry as well during the design process in a c*omputer aided design* (CAD) environment as in the later analysis phase using FEA. Numerous research papers devoted to IGA have demonstrated beneficial and superior analysis properties, using higher order and higher continuity basis functions compared to standard, low order finite elements. As B-splines and non-uniform rational B-splines (NURBS) are the most widely used geometry descriptions in CAD, NURBS-based finite elements have been developed and implemented into LS-DYNA over the last few years.
This paper presents an overview about the analysis possibilities using IGA elements in LS-DYNA. The focus will be on crucial developments added to the latest R13 version, which allows the numerical analysis of large-scale industrial models of interest using IGA finite elements.
The paper will be organized as follows. Basic concepts used in CAD are presented in Section 2. The newly developed and implemented **`*IGA`** keyword family is introduced in Section 3. Various additional important new IGA features added to LS-DYNA R13 are presented in Section 4. The paper closes with a summary and an outlook in Section 5.

2 Geometry fundamentals used in CAD

For a better understanding of the IGA method it is helpful to get familiar with a few key methods used and established in the CAD environment. This section explains the idea of boundary representation (B-Rep), trimming, topology and some other fundamentals.

2.1 NURBS-based B-Rep model

A geometrical 3D object is shown in Figure 1 (left), together with its NURBS-based boundary representation structure in Figure 1 (right). The B-Rep structure defines the 3D object solely using the boundaries, in this case the surfaces while the interior or the 3D object remains hollow. The boundary representation of a geometrical object can be divided into two parts:

- shape (geometry), which defines the spatial position and curvature
- structure (topology), which allows to make links between geometrical entities

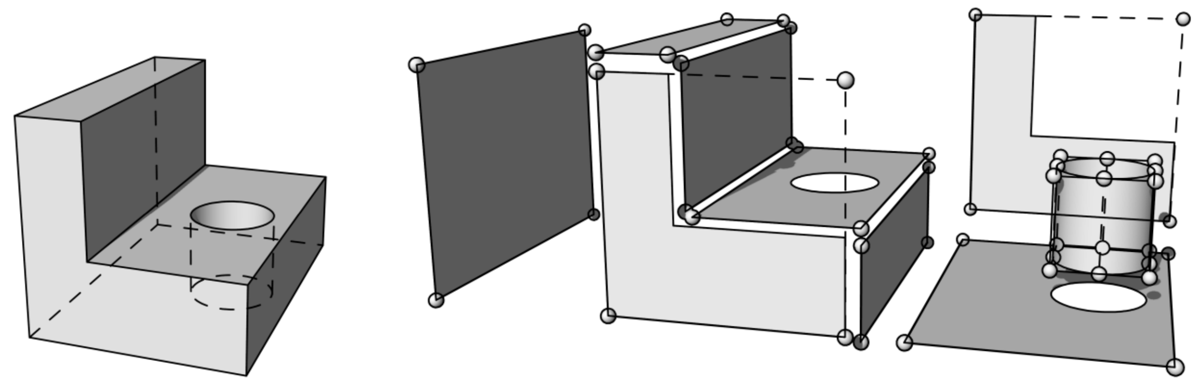

Fig.1: 3D object (left) and NURBS-based B-Rep model (right) [2]

2.1.1 B-splines, NURBS and trimming

For the geometry (shape) representation, usually non-uniform rational B-spines (NURBS) are used in commercial CAD systems. Its core ingredient, the B-spline basis functions (see Fig. 2) are constructed recursively until the desired polynomial degree of the functions is reached. The definition of a knot vector, which is a set of parametric coordinates arranged in an ascending order is necessary to define the B-spline basis functions. In contrast to the standard Lagrange polynomials, which are widely used for finite element analysis, B-spline basis functions are always positive regardless of their polynomial order. Furthermore, B-spline basis functions fulfill important properties necessary for FEA, like partition of unity and they can preserve higher continuity across element boundaries.

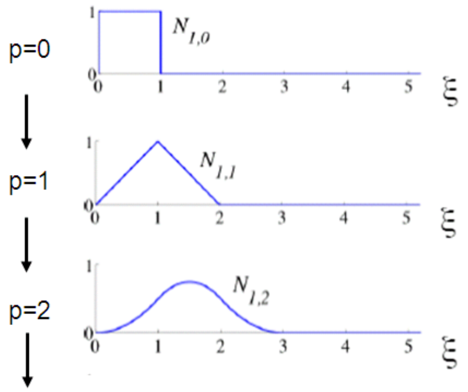

Fig.2: B-spline basis functions of order 0,1 and 2 for uniform knot vector [1]

The representation of curves, surfaces and volumes is done in a similar manner to standard FEA. Instead of interpolatory finite element nodes, control points $\mathbf{P}_i$ are used as coefficients together with the B-spline or NURBS basis functions. NURBS is an extension to B-splines, where additional weights are associated with the control points which leads to rational basis functions that, for example, allow for an exact representation of conical sections.

A particularly important method used in CAD is trimming. Additional closed loops of curve segments allow the partition of the underlying geometry in visible and void domains. A simple trimming procedure is shown in Fig. 3. The designer would like to generate a plate with a hole (Fig. 3a). To do so, the first step would be to generate a NURBS surface without a hole (Fig. 3b). In a second step, a clockwise (cw) trimming loop will be defined (Fig. 3b). The orientation of a trimming loop is important as it defines which side of the trimming curve will be recognized as material domain and which one as void. Traveling along the curve, the right side of the trimming curve will be assigned the void domain. In a last step, the CAD software will perform the trimming operation and only display the material domain.

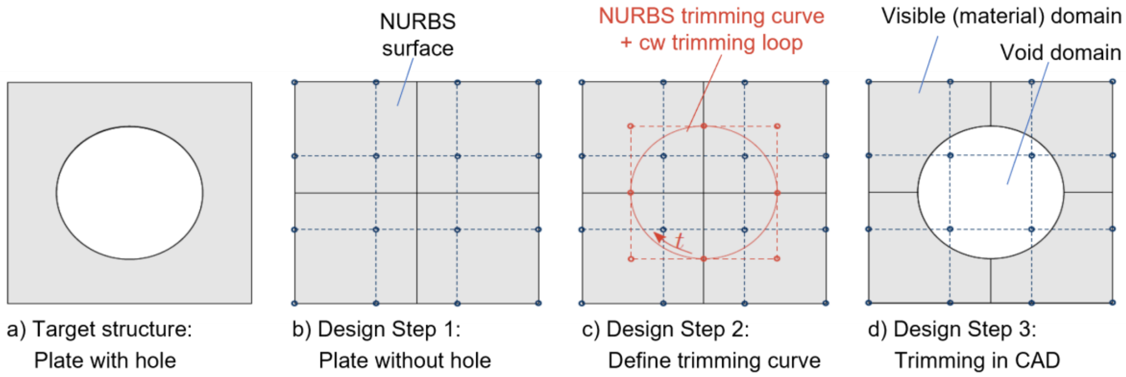

Fig.3: The logic of a trimming operation in CAD

2.1.2 Topology

In addition to the shape of individual parts of a geometric object, topological information is necessary to define links between geometric entities. The main topological entities are vertices (V), edges (E) and faces (F). A simple trimmed B-Rep model is shown in the real geometry space in parameter spaces and as an abstract topology in Fig. 4. Geometrical entities, like curves and surfaces have a representation in the common physical geometry space as well as in individual parameter spaces that are local to each underlying (untrimmed) NURBS surface. The most important topological information for later finite element analysis is the link between individual trimming curve segments of different surfaces, which define a common interface edge. In the example shown in Fig.4a two trimmed surfaces have a common interface represented by the trimming curves $\boldsymbol{C}_2$ and $\boldsymbol{C}_9$ on the first and second patch, respectively. Since the trimming curves are usually defined in the parametric space of their respective NURBS surface, their images in the physical space only approximately match in general. Therefore, the two trimming curves $\boldsymbol{C}_2$ and $\boldsymbol{C}_9$ are linked to the edge $\boldsymbol{E}_2$ (Fig.4b), which in its essence, establishes the topological connection between the two adjacent patches. It is worth noting that for most B-Rep CAD models, the interface between topologically connected trimmed patches is not watertight and may have gaps and overlaps.

The pure topological relation between adjacent surface patches is sufficient to correctly display multiple connected trimmed patches as one single part in a CAD system. In an analysis software package, however, it is important to apply suitable interface boundary conditions along common interface edges to perform a proper mechanical coupling between the individual trimmed surface patches.

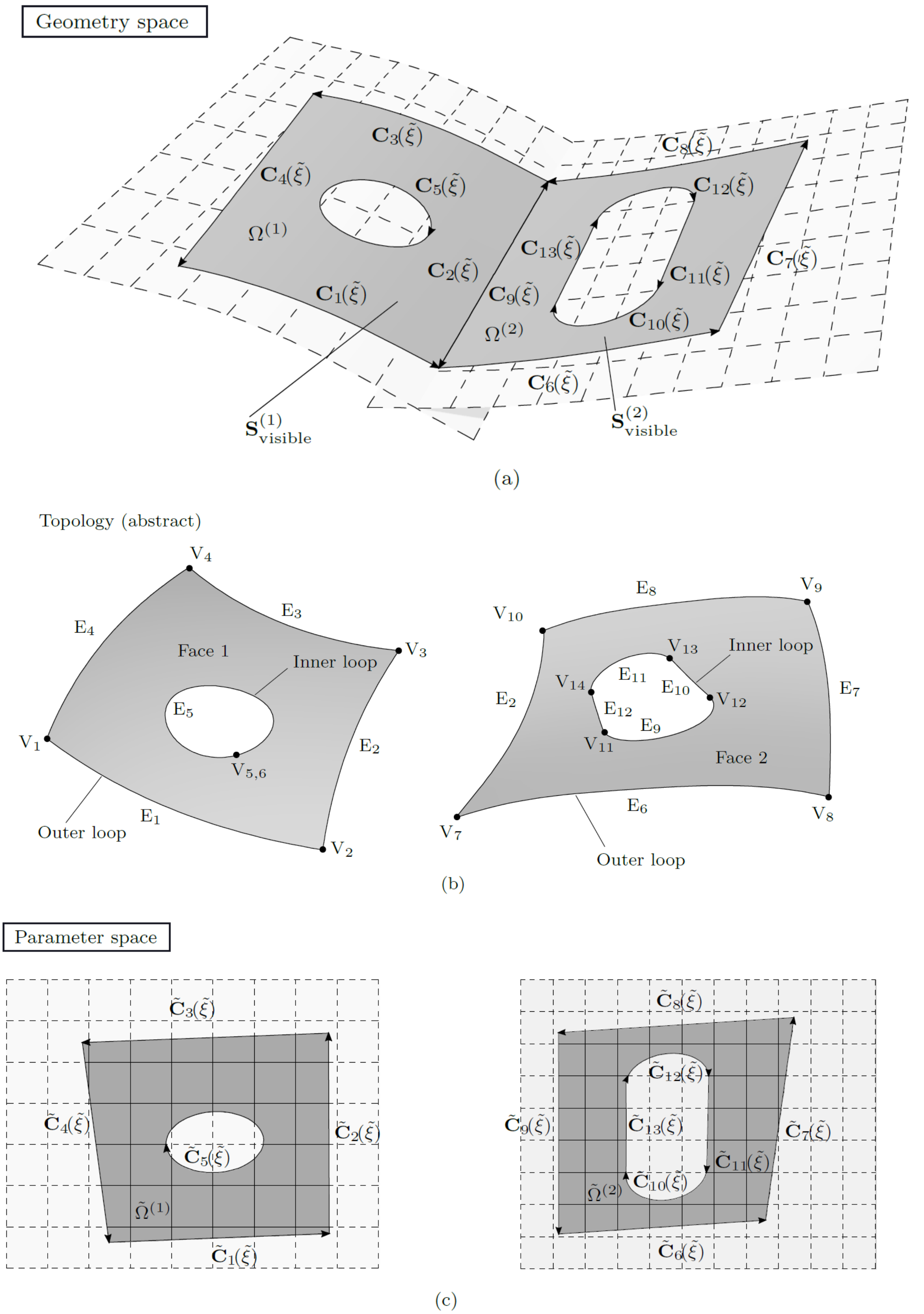

Fig.4: Trimmed surface B-Rep model represented in geometry space, as abstract topology and in parameter space, taken from Leidinger et al. [3]

3 The *IGA keyword family

Over the recent years significant IGA capabilities have been implemented into LS-DYNA using the keywords ***ELEMENT_SHELL/SOLID_NURBS_PATCH**. However, it became obvious that previous keywords will not be able to support all the necessary flexibility for a more seamless transition from CAD to analysis in the future, due to the lack of necessary topology information. Therefore, starting with R13, a new set of ***IGA** keywords has been implemented into LS-DYNA that reflect the general ideas of the above introduced typical CAD data structures, including the important topology information. The main idea of these new keyword family is to have a clear differentiation between the geometry definition and the actual discretization with appropriate IGA finite elements in LS-DYNA. This generalization offers a lot more freedom and flexibility and allows future developments and modeling strategies that are increasingly feature-based and independent of the actual discretization for the FEA.
The ***ELEMENT_SHELL/SOLID_NURBS_PATCH** keywords will be maintained in future LS-DYNA versions, but new IGA developments will be made exclusively for the ***IGA** keywords and it is expected that LS-DYNA users will switch to the new format in the foreseeable future.

3.1 Geometry- and topology-related *IGA keywords

As shown in the previous section, there are geometric entities that can be defined with respect to the global physical space as well as to some local parametric space of another geometric object. To distinguish this, those keywords will get the ending **_XYZ**, once it is related to the global physical coordinate system or the ending **_UVW**, once it is defined with respect to some parametric coordinate system, embedded in a higher-level geometric entity. Furthermore, curve, surface, and volume entities will get the addition **1D**, **2D** and **3D**. Some of the currently available keywords that are related to geometry and topology definition include, e.g.:

- ***IGA_1D/2D/3D_NURBS_UVW/XYZ**: define NURBS curves, surfaces and volumes
- ***IGA_EDGE_UVW/XYZ**: define edges (trimmed NURBS curves) and define topology
- ***IGA_FACE_UVW/XYZ**: define faces (trimmed NURBS surfaces)
- ***IGA_VOLUME_XYZ**: define volumes (trimmed NURBS solids)
- ***IGA_1D_BREP**: definition of a trimming loop
- ***IGA_POINT_UVW**: define a point on any parametric location of a NURBS surface

Usually, it is not necessary to know and understand all these keywords, as a proper preprocessor should automatically translate the CAD geometry information. However, it will be helpful to understand the general concept. With the above keywords, any geometric object can be described, and it is also possible to define various faces (trimmed NURBS surfaces) using the same underlying (untrimmed) NURBS surface definition. This might be interesting once a particular boundary condition, like a pressure load, shall be applied on a subsection of a shell structure. For this a second face could be defined as a subset of the whole shell by defining additional trimming loops.

3.2 IGA discretization

The actual discretization of the geometry defined above is done with the two keywords ***IGA_SHELL** and ***IGA_SOLID**. This means there could be various faces or volumes defined in terms of geometry, but not all of them are actually discretized as IGA elements, as some of them may be used to define boundary conditions of other connection properties.

3.3 Additional IGA-related keywords

Using IGA with the new set of keywords requires an IGA-specific section card, i.e. ***IGA_SECTION_SHELL/SOLID**. Furthermore, it is possible to define various sets for IGA related entities, like ***SET_IGA_EDGE**, ***SET_IGA_FACE**, etc.

3.4 Mechanical coupling of topologically connected surfaces

For a suitable multi-patch analysis, a mechanical coupling along the topologically connected interfaces is necessary. To achieve this, the user needs to define the keyword ***IGA_TIED_EDGE_TO_EDGE**.

Once this keyword is defined, LS-DYNA will collect all the given topology information between parametric (***IGA_EDGE_UVW**) and physical edges (***IGA_EDGE_XYZ**) and apply a suitable mechanical coupling there. In the rare case that the connected interfaces are untrimmed surface boundaries and the discretization in terms of knot vectors, order and control-point spacing fits along the common interface, LS-DYNA will merge the corresponding control points. If this is not the case, LS-DYNA will use a penalty-based tied contact method based on the IBRA approach developed by Breitenberger et al. [4] and presented at the 12th European LS-DYNA Conference by Hartmann et al. [5]. This approach is general enough to allow a proper coupling along T-joints, where one edge of a surface is topologically connected to a face of another surface as shown in Fig. 5. Generally, the number of surface boundaries coupled at a common interface is not limited.

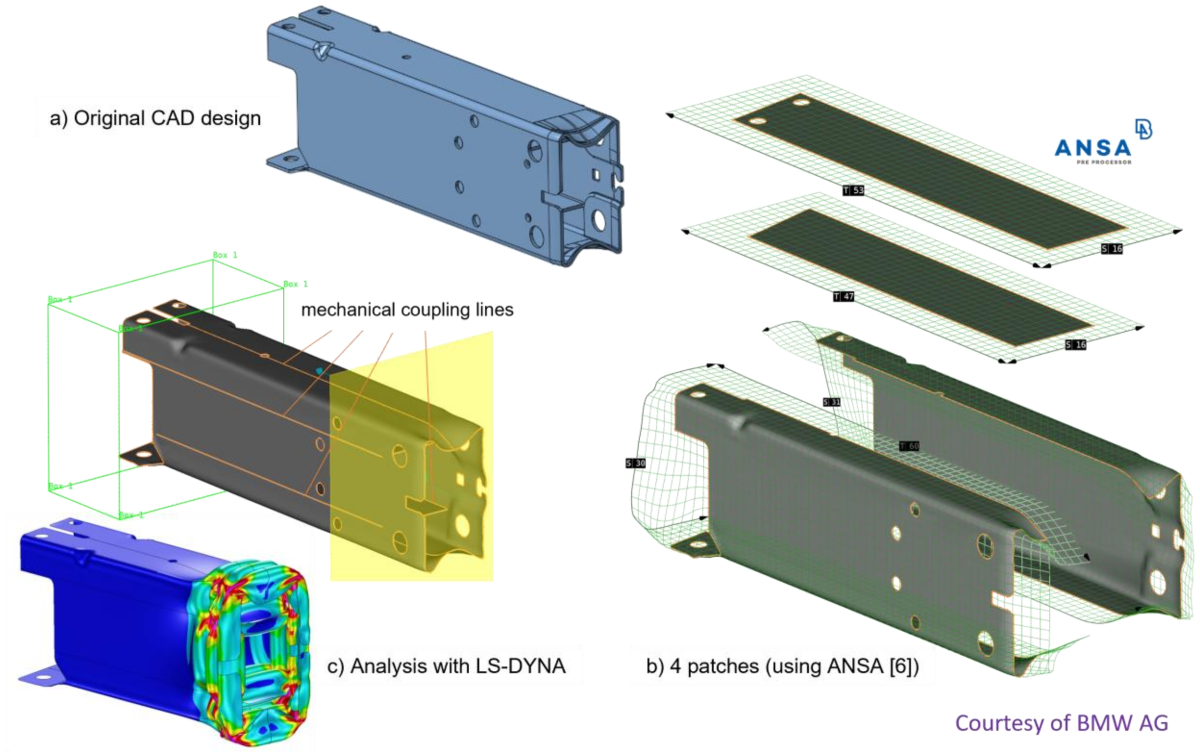

Fig.5: Crashbox model: From CAD design to LS-DYNA analysis

4 Additional new IGA features

4.1 Stabilization of small trimmed elements – light control points

It has been shown [7], that trimming of NURBS surfaces yields basis functions of reduced support. Consequently, the associated control points only get a tiny portion of the elemental mass, hence the naming *light control points*. These light control points can lead to instabilities due to out-of-range velocities in an explicit finite element analysis that yield premature termination of the solver. To overcome this problem, a suitable stabilization method has been added to LS-DYNA R13. The method ensures that nodal velocities of these light control points stay within a reasonable range and guarantee a stable solution. This is an important addition to the implementation as trimming in CAD is a fundamental and heavily used method and thus it is not possible to avoid the occurrence of light control points in realistic industrial models.

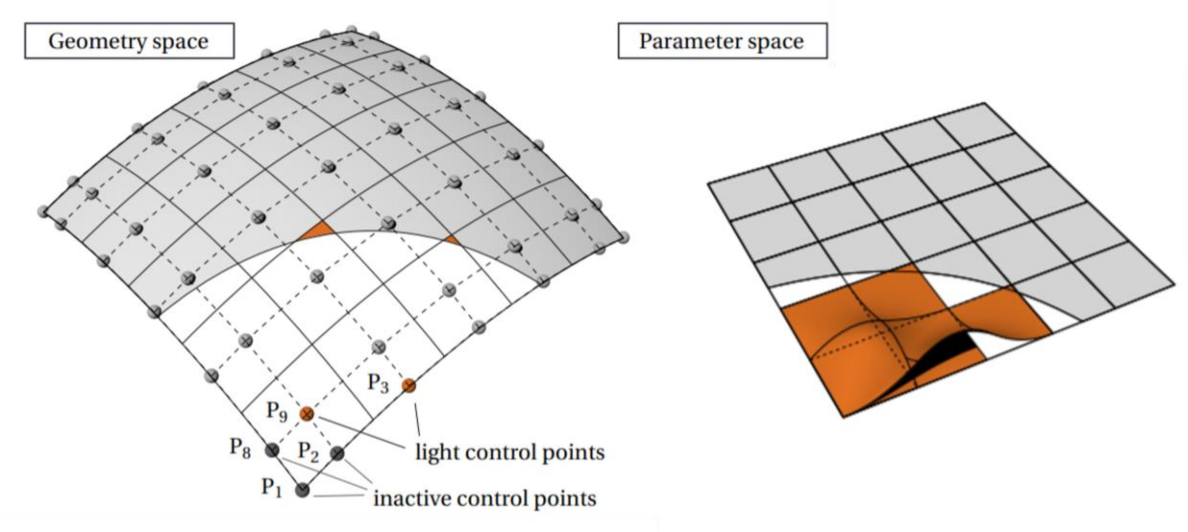

Fig.6: Small trimmed elements with light control points (left) and the corresponding NURBS basis functions (right) [7]

4.2 Spotweld modelling

In addition to ***CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET**, which was already available for IGA, spotwelds for isogeometric shells can now be also modelled through the keyword ***CONSTRAINED_INTERPOLATION_SPOTWELD** (prior notation ***CONSTRAINED_SPR3** still works) using a plasticity-damage model including failure.
With this modelling approach, the spotweld definition comprises (i) the part ID of the first and the second sheet, (ii) the node set ID of the spotweld location nodes, (iii) the total thickness of both sheets, (iv) the spotweld radius and (v) elasto-plastic and damage material parameters.
This spotweld modelling approach acts upon the interpolation shell finite element mesh, which is automatically generated for the isogeometric shell elements and which is also used for contact and visualization purposes, see [8]. In this way, all nodes of the interpolation shell finite element mesh located within the defined spotweld radius are automatically involved in the spotweld formulation during analysis. The main benefit of this spotweld modelling approach for IGA is its mesh-independency i.e., no modifications are required in case the underlying shell components or the positions of the spotwelds change during the development cycles.

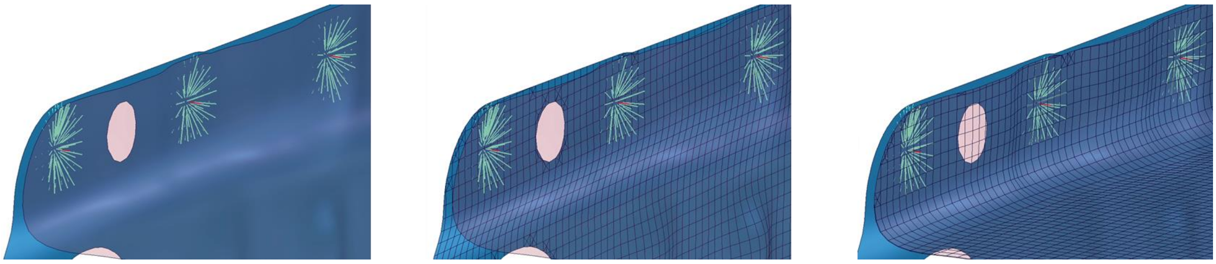

Fig.7: Spotwelds between IGA shells modeled via ****CONSTRAINED_INTERPOLATION_SPOTWELD*** *with visualized interpolation elements of sheet 1 (middle) and sheet 2 (right)*

4.3 Constrained nodal rigid bodies

Together with the keyword ***IGA_POINT_UVW**, nodal rigid bodies (***CONSTRAINED_NODAL_RIGID_BODY** or also ***CONSTRAINED_INTERPOLATION**) can now be attached to IGA shells as shown in Fig. 8. Herein, ***IGA_POINT_UVW** enables constraining a conventional node (of a nodal rigid body) to a parametric point of a 2D NURBS surface via a penalty approach, while the definition of the underlying rigid body remains unaltered. The required input for ***IGA_POINT_UVW**, namely a parametric point ID, the ID of the constrained node and the parametric coordinates of the point, can be

easily obtained through the preprocessor ANSA (from version 22) by projecting the desired node onto the corresponding IGA shell.
It should be noted that this is a preliminary approach that allows using the nodal rigid body definition from conventional FEA without modifications. A real feature-based IGA-type approach would directly introduce constraints between a certain reference point (A in Fig. 8) and NURBS surfaces (B in Fig. 8) and is currently under development.

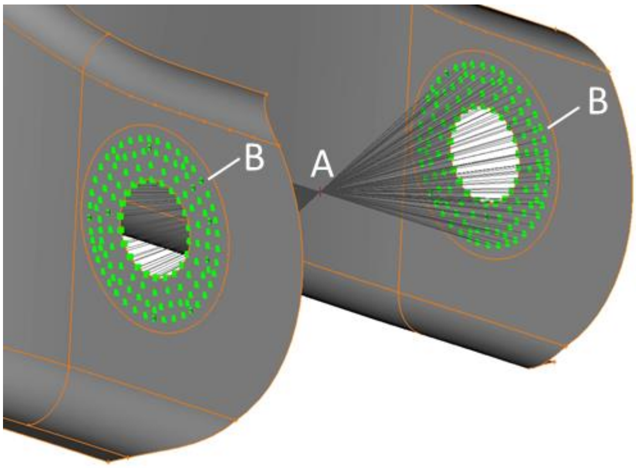

Fig.8: A ****CONSTRAINED_NODAL_RIGID_BODY*** *attached to IGA shells via* ****IGA_POINT_UVW*** *(highlighted in green)*

4.4 Hybrid finite element analysis

An aspect that will strongly facilitate the introduction of IGA in industry, is the capability of LS-DYNA to combine components represented by either conventional or isogeometric elements within the same simulation. Detailed descriptions of hybrid IGA/FEA vehicle crash simulations are provided in the paper “Hybrid IGA/FEA vehicle crash simulations with trimmed NURBS-based shells in LS-DYNA” by Leidinger et al. [9] also presented at this conference.

5 Summary and outlook

Starting with the presentation of some fundamental methods used in the CAD environment, this paper introduces the new ***IGA** keyword family which is now available in LS-DYNA R13. Although the previous keywords used for isogeometric analysis in LS-DYNA are still available and maintained, the future development of IGA in LS-DYNA will be mostly focused on the new keyword structure. Due to the rigorous separation between geometry and discretization, it is now possible to further develop the IGA capabilities in LS-DYNA to a more feature- and geometry-driven modelling strategy.
In addition, crucial new IGA developments added to R13 are presented, like:

- mechanical coupling of topologically connected trimmed surface patches, including T-joints
- stabilization of heavily trimmed IGA shell elements
- spotweld modelling
- definition of nodal rigid bodies
- hybrid analysis, using IGA together with standard FEA

The generality of the new ***IGA** keyword family allows a straightforward extension to IGA solid elements, whose implementation into LS-DYNA is currently taking place. Instead of faces (trimmed NURBS surfaces) that are discretized with IGA shells, volumes (trimmed NURBS solids) will be discretized using IGA solids. For the definition a 3D geometry, NURBS surfaces and faces are embedded into the parametric space of the 3D NURBS object, which allows for the representation of any 3D volumetric object. A suitable numerical integration scheme for the resulting trimmed solid elements is currently developed and should be available in future LS-DYNA releases. This will open the possibility to easily analyse complex 3D volumetric parts without the need of using low-order and low-quality tetrahedron finite elements.

With the described enhancements for IGA in LS-DYNA R13, the hybrid numerical analysis of large-scale industrial models is now possible. This leads to additional enhancements requests especially in terms of computational costs. To address this, enhancements in the time step estimates as well as in the numerical integration scheme for trimmed IGA elements are currently implemented and tested. Further work includes more sophisticated parallelization methods, leading to a better load balancing of all the processors involved in a massively parallel computing environment. It is expected that these enhancements will speed up the numerical cost for IGA significantly, such that IGA analysis will be cost competitive in the future.

Finally, as always, the future IGA development in LS-DYNA will be highly driven by customer requests. The more LS-DYNA customers starting to use and deploy the technology, providing feedback and suggestions, the better for the future development process.

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

[3] Leidinger, L.F., Breitenberger, M., Bauer, A.M., Hartmann, S., Wüchner, R., Bletzinger, K.-U., Duddeck, F., Song, L.: “Explicit Dynamic Isogeometric B-Rep Analysis of Penalty-Coupled Trimmed NURBS Shells”, Submitted to Computer Methods in Applied Mechanics and Engineering, 2019.

[4] Breitenberger, M., Apostolatos, A., Philipp, B., Wüchner, R., Bletzinger, K.-U.: “Analysis in computer aided design: Nonlinear isogeometric B-Rep analysis of shell structures”, Computer Methods in Applied Mechanics and Engineering, Vol. 284, 2015, 401-457.

[5] 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.

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

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

[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] Leidinger, L.F., 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.