Updates on trimmed IGA B-Spline Solids

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

[1]DYNAmore GmbH, an ANSYS company
Industriestr. 2, 70562 Stuttgart, Germany

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

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

1 Introduction

Recently, trimmed Isogeometric B-Spline solid elements have been introduced in Ansys LS-DYNA ® nonlinear dynamics structural simulation software [1]. The numerical analysis methodology Isogeometric Analysis (IGA) dates to the paper by Hughes et al. [2] in 2005. While in standard Finite Element Analysis (FEA) polynomial basis functions are typically used for the discretization of the geometry and the unknown fields, IGA aims to use the same shape functions employed in the Computer Aided Design (CAD) environment for numerical analysis.
This paper reviews the main ideas and concepts of the trimmed IGA B-spline solid finite elements in LS-DYNA and provides an update on the available capabilities. The current modeling workflow for these new element types will be first introduced. Modeling strategies for boundary conditions, contact and connections, and in-core refinement possibilities will be presented, and some recommendations and best practices will be discussed. The potential benefits of this new type of solid finite elements will be illustrated by two numerical examples.
The paper closes with a summary and an outlook on future development activities.

2 Trimmed IGA B-spline solid finite elements

The idea of developing trimmed IGA B-spline solid finite elements in LS-DYNA is a direct consequence of the work done on trimmed Non-Uniform Rational B-Splines (NURBS) shell finite elements. While the development builds on the same framework designed for shells, and the extension to 3D solid elements appears to be straightforward, some major differences exist. These differences will be further discussed in the paper. Additionally, the current approach has many similarities with other embedded methods, such as the Finite Cell Method (FCM) in its B-spline version, described by Schillinger [3] in 2012. In this section the B-spline solid elements implemented in LS-DYNA will be presented, while a direct comparison with similar methods described in literature will be omitted because it is outside of the scope of this work.

2.1 Embedding approach

In contrast to other standard finite element methods, in the embedded approach the geometry is not discretized with a boundary conforming mesh, like tetrahedral or hexahedral finite elements. Indeed, the geometry of the 3D body is embedded into the parametric space of a trivariate B-spline patch. As the parametric space of the trivariate B-spline is defined by three knot vectors, the parametric space can always be represented as a regular grid of knot spans, which is commonly referred to as the background grid, see Fig 1. Depending on how the 3D body is embedded into this background grid, the knot spans are categorized as:

- Fully embedded (untrimmed): The whole knot span is filled with material
- Partly embedded (trimmed): The knot span is partly filled with material
- Inactive: Knot spans without any material in it, i.e., fully outside of the 3D body

Inactive knot spans are ignored, whereas the untrimmed and trimmed knot spans are considered as finite elements for the purposes of analysis. Untrimmed B-spline elements are treated similarly to standard finite elements and a predefined numerical integration rule can be used, like the well-known

© 2024 Copyright by Ansys Inc.

Gauss integration. However, trimmed B-spline elements need more attention, and a specific numerical integration rule needs to be defined depending on the geometry of each individual trimmed knot span. The requirements of these numerical integration rules, as well as the basic concepts on the design of individual quadrature rules for trimmed B-spline solids, are described in [1] and some references therein.

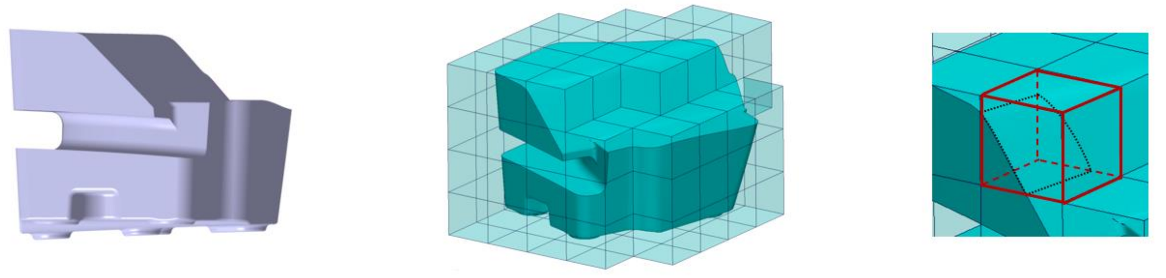

Fig.1: Embedding the geometry into a hexahedron B-spline background grid [4].

The embedding approach allows to independently modify the accuracy of the geometric representation of the embedded 3D body, and the parametrization of the mechanical solution field defined through the trivariate B-spline. The accuracy of the solution depends on the parameterization of the volume, i.e., the knot vector and the polynomial degree in each parametric direction. For example, Fig. 2 shows a crash pad model with a relatively coarse background discretization and a very accurate geometric representation.

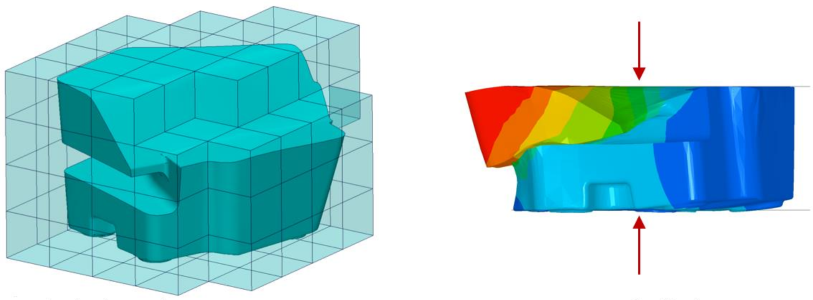

Fig.2: Decoupling of geometry and mechanics.

2.2 Why B-Splines?

While other embedding approaches oftentimes use higher-order standard finite elements as a background grid, at least C^1 continuous, trivariate B-splines are selected for this work. In Leidinger [5] it has been shown, that trimming of standard C^0 finite elements has a significant negative impact on the critical time step size for explicit dynamics simulations. On the other hand, Leidinger [5] has also demonstrated that the trimming of higher order NURBS shell elements with at least C^1 inter-element continuity do not suffer from a significant negative impact on the critical time step size when using a lumped mass matrix. Similar studies were conducted by Meßmer et al. [6] in 2021 for trimmed B-spline solids, and the results were consistent. Although the current implementation for the trimmed B-spline solids in LS-DYNA can also be used for implicit analysis, the efforts have been primarily focused on explicit dynamics simulations, e.g., crash analysis.

2.3 Interpolation mesh and boundary conditions

Similarly to the approach followed for IGA shell elements in LS-DYNA, an interpolation mesh is automatically generated that matches the geometry of the embedded body. This interpolation mesh is comprised of standard linear finite elements, such as tetrahedrons or hexahedrons. Its main purpose is to enforce contact boundary conditions and for post processing, i.e., visualization. The interpolation elements are not integrated during the analysis cycle and thus they do not influence the critical time step size. Furthermore, the interpolation nodes are fully constrained to the background B-spline grid and do not carry additional degrees of freedom. Therefore, there are no requirements regarding the quality of the interpolation mesh.

2.3.1 Contact

Contact forces are evaluated on the interpolation mesh. Therefore, all penalty-based contact methods available in LS-DYNA are already supported. After the contact evaluation, the contact forces acting on the interpolation nodes are mapped back to the actual degrees of freedom of the B-spline elements, i.e., at the control points of the trivariate B-spline background grid. Once the updated deformation of the B-spline elements is computed, the location of the interpolation nodes is updated accordingly, and the analysis proceeds to the next cycle.

2.3.2 Constrained nodal rigid body

Constrained nodal rigid bodies are frequently used to connect various parts in larger LS-DYNA models. In a typical workflow, a subset of finite element nodes on the surface of a 3D part are collected within a node set. A constrained nodal rigid body is then defined for this set of nodes. However, if the embedded approach with trimmed B-spline solids is used, it is not possible to directly define a geometrical constraint to an embedded node in the background grid. To establish this type of constraint, an **`*IGA_POINT_UVW`** ***P*** can be defined with respect to the parametric space of the B-spline solid. Furthermore, in the **`*IGA_POINT_UVW`** definition, a link to a **`*NODE`** ***N*** needs to be established to indicate that the parametric location of the point coincides with the location of the node in the real physical space. LS-DYNA will then automatically generate a new **`*NODE`** ***M*** that is constrained within the B-spline solid at the parametric location of the **`*IGA_POINT_UVW`**. It will also generate a potential zero-length discrete beam between the new **`*NODE`** ***M*** and the given **`*NODE`** ***N***. The original **`*NODE`** ***N*** may remain in the constrained nodal rigid body definition and its effect on the B-spline solid is transferred in a force-based manner via the discrete beam connection to the constrained **`*NODE`** ***M***.
An alternative approach would be to define a penalty-type tied contact between the set of nodes in the constrained rigid body and the trimmed B-spline solid. With this strategy, no **`*IGA_POINT_UVW`** need to be defined. However, the initial kinematic constraint is again translated into a force-based constraint enforcement via the appropriate tied contact definition.

2.3.3 Constrained interpolation spotweld (SPR3)

A frequently used method to model spotweld connections in LS-DYNA is the use of **`*CONSTRAINED_INTERPOLATION_SPOTWELD`** (SPR3). This particular type of connection is directly applicable with the trimmed B-spline solids. It works similarly to the treatment of contact boundary conditions. Instead of evaluating the SPR3 directly on the B-spline solids, it is evaluated on the embedded interpolation elements. The resulting forces computed at the interpolation nodes are then mapped back to the actual degrees of freedom at the control points of the B-spline background grid.

2.4 Trimmed IGA shells vs. trimmed IGA solids

The current implementation of the trimmed B-spline solids is similar to the implementation of trimmed NURBS-based shells. However, there are a few differences between the two approaches that shall be discussed here.

2.4.1 B-splines vs. NURBS

Non-uniform rational B-splines (NURBS) are a generalization of B-splines. They are created by adding weights to the control points. NURBS have been developed to provide an exact description of conic sections like circles. Depending on the specific surface structure of shell-like geometries, the use of NURBS may be necessary to correctly represent the desired surface geometry. Trimming may be added to define the actual boundary of the underlying surface or to cut out holes within the domain. Conversely, a trivariate background B-spline grid usually does not need to align with the 3D geometry and could be placed rather arbitrarily. Therefore, it is typically not necessary to add weights to the trivariate B-Spline solids. However, for certain 3D structures, like very thick-walled cylinders, 3D NURBS may be used in the future as a background grid. The fundamental concept of the trimmed IGA solids remains unchanged by this and the current implementation in LS-DYNA is already general enough to support this if needed.

2.4.2 Analysis-suitability of CAD models

In the computer aided design (CAD) environment, the geometry is usually described by a boundary representation (BREP). A common modeling approach is to represent a part using several individual trimmed surface patches with rather high polynomial orders, that are topologically connected to each other. If the underlying geometry is a shell-like structure, and if it shall be analyzed with IGA shell elements, it is usually necessary, or at least preferred, to:

© 2024 Copyright by Ansys Inc.

a) Combine different surface patches to reduce their number.
b) Reduce their polynomial order.

In the best case, it is possible to describe the whole part with one trimmed surface patch. However, if the part to be analyzed is a 3D volumetric solid, it is not necessary to make any changes to the individual surface descriptions, as the whole BREP will be embedded into the B-spline solid background grid. This facilitates the preprocessing effort for the IGA B-spline solids compared to its shell counterpart.

2.5 Current capabilities

Many necessary capabilities for the trimmed B-spline solids are already available in the latest development version of LS-DYNA. This includes implicit and explicit analysis, the support of many material models, including material damage, failure and element erosion, standard mass scaling approaches, various connection technologies, MPP support for all implemented features and stabilization techniques for light control points. Light control points may exist due to the presence of trimmed knot spans that are only filled with very little material. In such cases, some active control points have support in elements whose material volume fraction is low, and thus their nodal mass becomes very small, i.e., they are light. As previously discovered investigating trimmed NURBS shell elements, such light nodes may become unstable in highly dynamic explicit analysis, and they need to be stabilized. Similar approaches that have been developed for the IGA shell elements were extended to the trimmed B-spline solid elements.

3 Model setup

This section describes two methods for setting up a LS-DYNA model with trimmed IGA B-spline solids. The first approach described in this section is considered to be the future, long-term, and standard modelling solution and it is based on the extension of the **`*IGA`** keywords from shells to solids. The second approach is a temporary intermediate workflow. This is a prototypical approach that has been developed to evaluate, test, and improve the B-spline solid element implementation and capabilities within the LS-DYNA solver.

3.1 From CAD to `*IGA` keywords

The **`*IGA`** keywords known from the IGA shell elements in LS-DYNA have been systematically extended to 3D in order to define a closed boundary representation (BREP) for volumetric parts. Some of these keywords are shown in Fig. 3. The background grid, in which the solid part shall be embedded, is defined via **`*IGA_3D_NURBS_XYZ`**. As described above, usually it is sufficient to have a B-spline instead of a NURBS definition. Furthermore, a new keyword **`*IGA_REFINE_SOLID`** allows for an automatic refinement of the background grid. When using the refinement option, it is sufficient to simply define a rectangular box, big enough encompass the whole body that shall be embedded. More information about the refinement option will be given in section 3.3. The boundary representation of the solid part is defined via **`*IGA_VOLUME_XYZ`** which collects the various surfaces (**`*IGA_FACE_XYZ`**) of the **`*IGA_2D_BREP`**. Once the set of surfaces is topologically connected, such that it describes a closed boundary, the geometric definition of the 3D part is complete. The **`*IGA`** keyword cards reflect the geometric description as defined in CAD. Therefore, a conversion from CAD should be straightforward for any preprocessor. However, it is worth mentioning that LS-DYNA is not yet able to fully support all the necessary geometric entities. This is the reason why an intermediate, albeit temporary, workflow has been introduced to allow for further developments within the LS-DYNA solver.

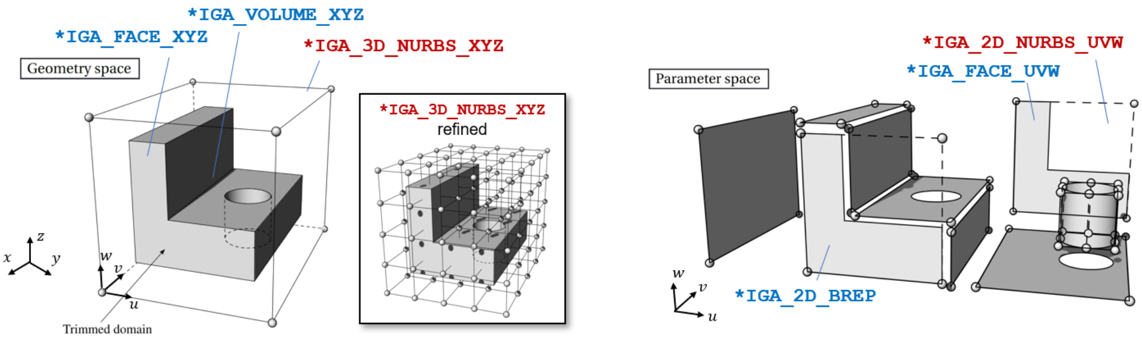

*Fig.3: **`*IGA`** keywords for trimmed B-spline solids (pictures taken from Breitenberger, 2016 [7]).*

3.2 Temporary workflow

An intermediate approach for defining B-spline solids has been temporarily implemented. It does not take into account the CAD geometric entities. Instead, it uses a tetrahedral mesh approximation of the 3D body as input for the representation of its geometry. This is, of course, a crude approach because it makes use of a discretized version of the actual geometry as a starting point. However, it facilitates the conversion from existing solid finite element models to trimmed B-spline solid element models. Still, the 3D background grid needs to be defined via a ***IGA_3D_NURBS_XYZ** large enough to fully encompass the full 3D body. Instead of the official ***IGA_VOLUME_XYZ** keyword, this workflow is enabled by a temporary ***IGA_*DEV*_VOLUME_XYZ** keyword. This **DEV** volume card does not collect a set of surfaces, but points to a solid part that is represented via a tetrahedral mesh, see Fig. 4.

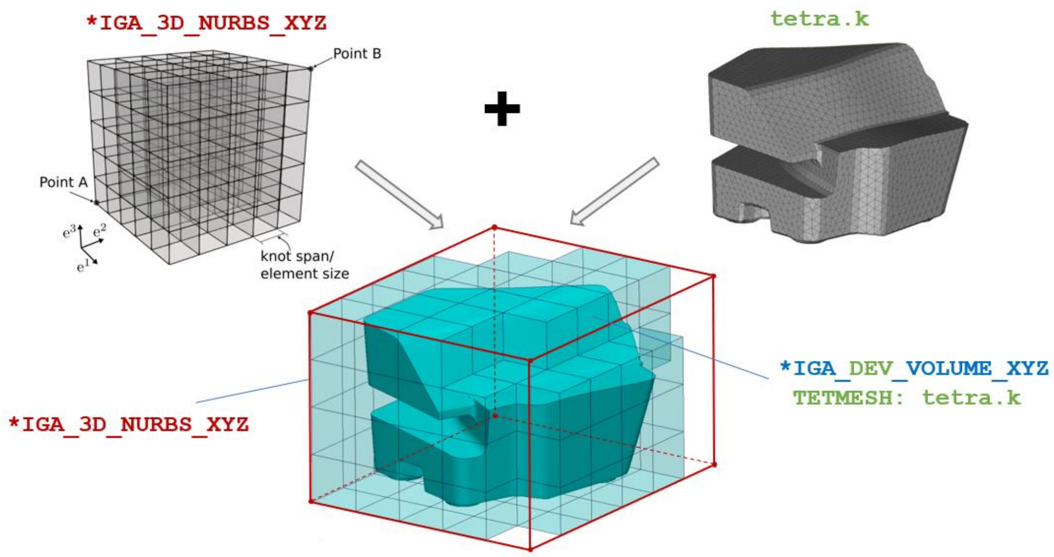

Fig.4: Embed a 3D solid part, represented by tetrahedral mesh into a B-spline background grid.

3.2.1 Discussion and recommendations

This temporary workflow is well suited for users who already have existing tetrahedral LS-DYNA solid models. In such a scenario, it is easy to convert the existing model into one that uses the trimmed B-spline approach. First, a rectangular box, large enough to embed the solid part of interest, must be defined. In the simplest case, a tri-linear B-spline patch with just one element can be defined. This may be automatically refined to the desired knot span size and polynomial order by including the ***IGA_REFINE_SOLID** keyword (see Section 3.3). As the given tetrahedral mesh describes the 3D body in the real physical domain, it is helpful to define the parametric space of the background grid equal to the physical space. As long as the background box is aligned with the global coordinate system, this can easily be achieved by using a uniform knot vector (UNIR=UNIS=UNIT=1 in ***IGA_3D_NURBS_XYZ**) and setting the knot vector boundaries (RFIRST, RLAST, …) equal to the minimum and maximum boundaries of the B-spline background grid definition. In this case, no internal transformation from the physical space back to the parametric space of the B-spline background grid needs to be performed within LS-DYNA. Furthermore, it may be helpful to define the background box at least one knot span size larger than necessary to embed the 3D structure. This ensures that the outermost knot spans in each parametric direction will be trimmed away and thus become inactive. This is especially beneficial in explicit dynamics simulations, as the elements with the smallest critical timesteps are associated with knot spans at the parametric boundaries of open knot vectors. If these knot spans are inactive, i.e., if they are trimmed away, a higher explicit time step size can be used.

3.3 Refinement of the background grid

As already mentioned before, a new keyword ***IGA_REFINE_SOLID** has been implemented into LS-DYNA which eases the definition and adjustment of the B-spline background grid significantly. A description of this keyword is shown in Fig. 5 below. Three different types of refinement (h-, k- and p-refinement) are supported. Furthermore, two h-refinement options are available. Either the number of subdivisions or the desired physical element length in each parametric direction can be defined. These refinement options simplify the definition of the background grid significantly and keep the input data size small, independently of the size of the B-spline background grid. The desired element length of the

B-spline elements and the polynomial order can be changed by varying a few parameters and, notably, without the assistance of any preprocessor. This simplifies, for example, performing convergence studies based on different mesh sizes.

```
*IGA_REFINE_SOLID
$      rid      rtyp
         1         2
$    hrtyp        rr        rs        rt
         2       5.0       5.0       5.0
$      itr       its       itt
         2         2         2
```

```
RID  - solid refinement ID
RTYP - solid refinement type
       = 1 : h-refinement
       = 2 : k-refinement, i.e. p-refinement followed by h-refinement
       = 3 : p-refinement
HRTYP - h-refinement type
        = 1 : subdivision-based refinement
              Each non-zero knot span is subdivided to NINT(R*) equal parts along
              the parametric *-direction.
        = 2 : physical length-based refinement
              Each non-zero knot span is split up such that the resulting spatial
              dimension of the resulting knot vectors is about RR x RS x RT. Currently
              limited to trilinear parametrizations with a single knot
              span only, i.e. P*=1 and N*=2 on *IGA_3D_NURBS_XYZ.
R*     - see HRTYP above
IT*    - target polynomial degree in the *-parametric direction
```

Fig.5: `*IGA_REFINE_SOLID` keyword with description.

4 Numerical examples

The numerical examples recently presented in [1] focused on the performance of the trimmed B-spline solids compared to standard tetrahedral finite elements. In terms of material behavior, these examples were limited to nonlinear plasticity without damage and failure. In this paper, two numerical examples are presented that include damage evolution, material failure, and element erosion. These modeling capabilities are paramount for crash analysis.

4.1 Car door lock

The first example is a model representing a test performed on a car door lock. The test setup as well as the results of three physical experiments are shown in Fig. 6. A test device is pulled in vertical direction until the shackle fails.

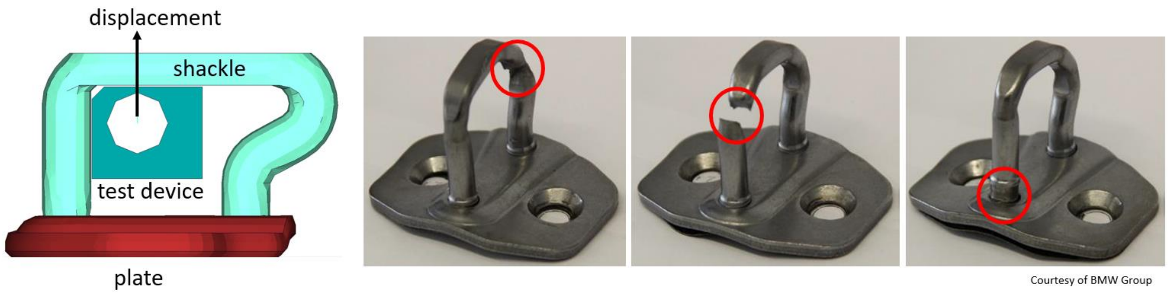

Fig.6: Test setup and experimental results with different material failure locations.

Although the three physical experiments lead to material failure at different locations, the resulting force-displacement curves (see Fig. 7) correlate very well. The differences between the pre-failure peak forces are smaller than 5%.

Fig.7: Force-displacement curve of the three test specimens.

For the numerical test, a standard finite element model using linear tetrahedral solid elements (ELFORM=13) is compared with the trimmed solid approach, using a tri-quadratic B-spline background grid, see Fig. 8. In both numerical models, a segment-to-segment (SOFT=2) eroding single surface contact is used. The base plate is fixed using screws, which are modeled via prestressed beams, while the test device is modeled as a rigid body.

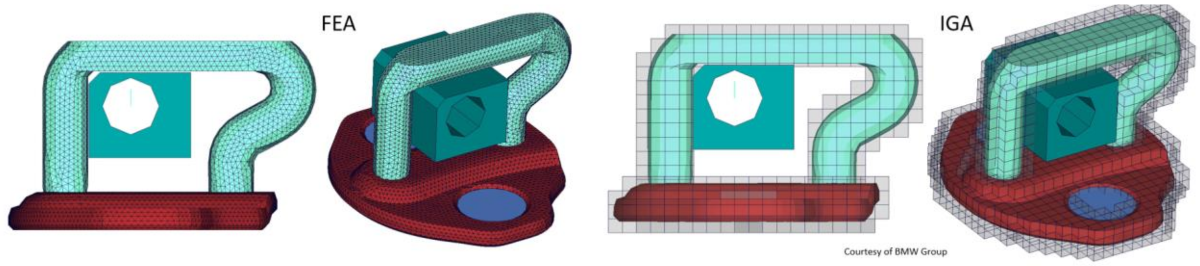

Fig.8: Finite element model with linear tetrahedral elements (left) and trimmed B-spline solids (right).

The force-displacement curves as well as the deformed configuration, right before and after failure, are shown in Fig. 9. Both numerical models can predict the global behavior observed in the experiments. The failure is initiated at the same locations and the deformed configurations after failure are comparable. By looking at the force-displacement curves it can be observed that failure occurs a little earlier in the IGA model. More research is required to investigate the differences. Also, the element erosion algorithm for trimmed B-spline solids may be revised. Furthermore, it is worth noting that with the trimmed B-spline solid approach a critical time step size increase of more than 40% (1.21E-07s → 1.74E-07s) can be achieved without any additional mass scaling, which then leads to a nearly 30% decrease (22min18s → 15min54s) in the overall computational time.

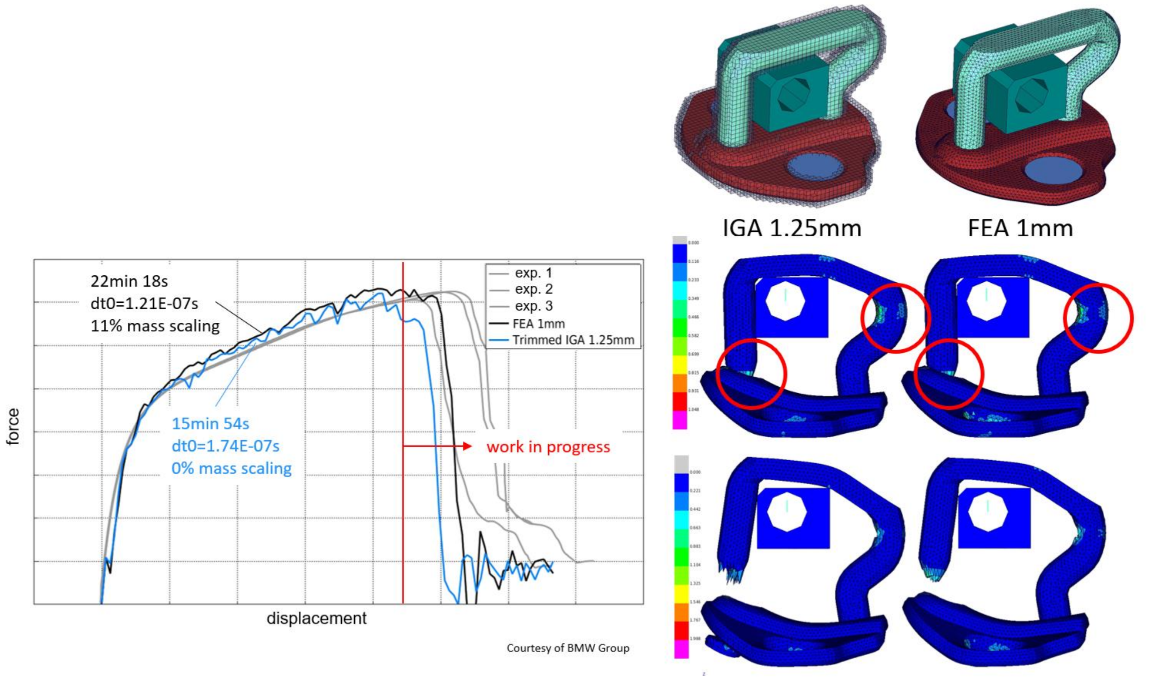

Fig.9: Force-displacement curves and deformed configurations at different times.

© 2024 Copyright by Ansys Inc.

4.2 Bolt model

The second example investigates the behavior of a bolt under shear loads. Two plates are connected via a bolt and a prescribed displacement is applied to the top plate. No prestress is applied to the bolt in this analysis. The model setup and the finite element discretization are shown in Fig. 10.

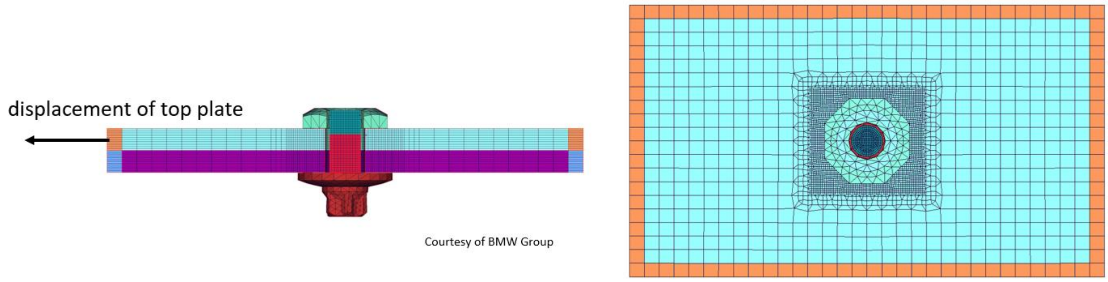

Fig.10: Bolt model: side view with mesh and boundary conditions (left), top view of mesh (right).

Two numerical models are compared. The only difference is the discretization of the cylindrical shaft of the bolt. In the pure finite element model, the cylindrical shaft is discretized with linear standard finite hexahedral elements whose reference length is, approximately, 0.3mm. In the mixed FEA-IGA model, the cylindrical shaft is discretized with 1mm trimmed B-spline solid elements using a regular background grid (see Fig. 11).

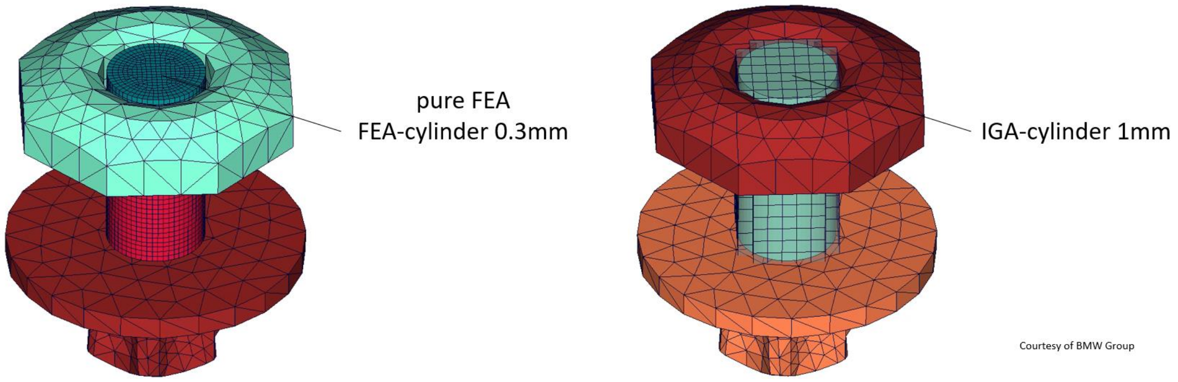

Fig.11: Different discretizations: The cylindrical shaft of the bolt discretized with standard finite elements (left) and with trimmed B-spline solids (right).

A comparison of the deformed configuration of the two models, at four different time states during the analysis, is shown in Fig. 12. Although the elements used in the trimmed B-spline model are roughly three times larger, it can be observed that the predictions of both models are consistent in terms of deformation and failure pattern.

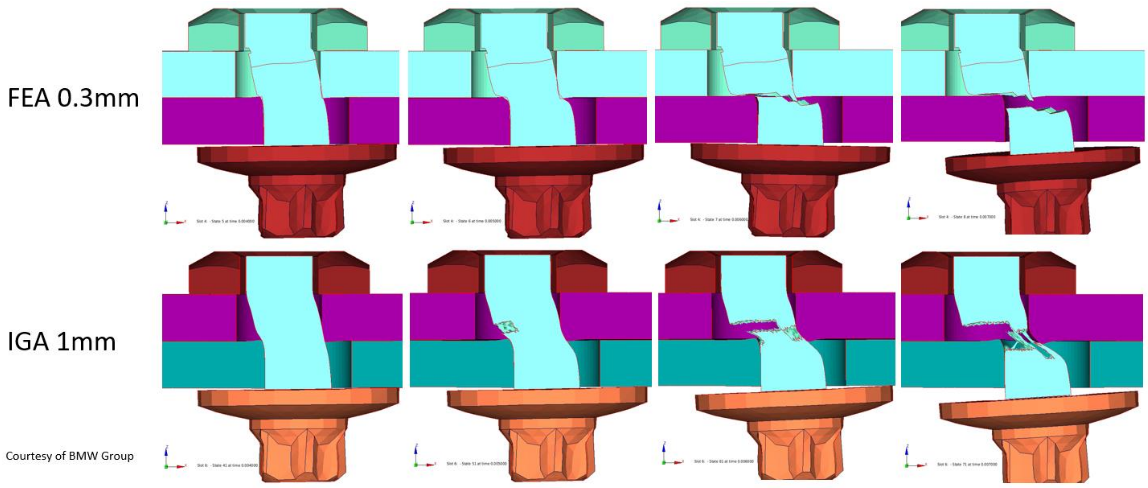

Fig.12: Comparison of the deformed configuration at four different time states.

The force vs. time curve is shown in Fig. 13. It can be observed that both models predict a very similar behavior until the maximum force is reached. Also, the peak force predicted by both models is similar. The only obvious difference is in the post peak behavior. The pure finite element model softens faster after reaching the peak force, while the IGA model exhibits a more ductile behavior. More work is required to investigate the differences.

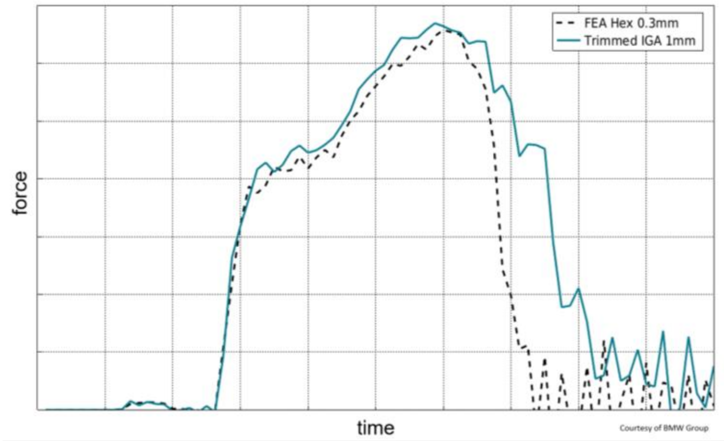

Fig.13: Force vs. time curve.

The potential advantage in computational time cannot be inferred by this example because:

a) Both simulations are run with the same prescribed time step size, dictated by the discretization of the remaining FEA parts
b) Only a small part of the whole model is discretized with trimmed B-spline solids

However, the example showed that with trimmed B-spline solid elements it is possible to capture the cylindrical geometry as accurately as needed, while using comparatively large B-spline elements. Therefore, the advantage of decoupling the geometry representation from the mechanical behavior becomes clear. With standard, conforming linear tetrahedral or hexahedral elements, a reasonably accurate geometry representation of the cylindrical shaft requires a relatively small mesh size, which results in a reduction of the critical time step size and an increase in the number of elements and degrees of freedom.

5 Summary and outlook

5.1 Summary

This paper provides an overview of the recently introduced trimmed isogeometric B-spline solid finite elements in LS-DYNA. The concept of embedding the geometry of a 3D part into a trivariate B-spline background grid is first illustrated. Several modeling strategies for contact boundary conditions, nodal rigid bodies or spotweld connections like SPR3 are shown. The differences between trimmed IGA NURBS shell elements and the trimmed B-spline solid elements are discussed.
Two different approaches for modeling embedded B-spline solids in LS-DYNA are then introduced. The first makes use of the **`*IGA`** keywords. This approach is intended to become the standard IGA workflow, where the LS-DYNA model will use CAD-like geometric entities that could be directly imported from the original CAD data. Since this workflow is not yet fully supported within LS-DYNA, a second temporary workflow is presented. Instead of using CAD data for the geometric representation, a lower-order linear tetrahedral element mesh is used as a starting point.
This paper also provides some recommendations regarding the definition of the B-spline background grid and the choice of the parametric space with respect to the physical space.
The new **`*IGA_REFINE_SOLID`** keyword is then introduced. This keyword allows effortlessly refining a B-spline background grid in terms of polynomial degree as well as element size. Refinement studies are also greatly simplified as they can be performed by varying a few parameters in the input deck without the assistance of any preprocessor. Furthermore, the size of the input file is not affected by any refinement performed via **`*IGA_REFINE_SOLID`**.
Finally, two numerical examples are presented. These examples, in contrast to the previous ones studied in [1], focus on material damage, failure, and element erosion. It is shown that trimmed B-spline solids are capable of accurately representing damage evolution and predicting failure while maintaining larger element sizes compared to standard finite elements. These examples have shown the great

© 2024 Copyright by Ansys Inc.

potential of the trimmed B-spline solids. Larger elements may be used for the mechanical representation of the physical problem, i.e., the analysis, without affecting the accuracy of the 3D solid geometrical representation. This leads to larger critical time step sizes and less mass scaling in explicit dynamics simulations, which ultimately results in a significant decrease of the overall computational time.

5.2 Outlook

The paper shows the great potential of the trimmed IGA B-spline solid elements in LS-DYNA. However, additional work is required:

- Support of the missing geometric entities in LS-DYNA. These are required to enable a seamless workflow from CAD geometries to the LS-DYNA solver.
- Investigate and enhance the damage and failure behavior of these type of elements.
- Adjust the contact stiffness estimation when using interpolation elements for contact boundary conditions of trimmed B-spline solids need to be added to LS-DYNA.

While these are considered priorities for the development pipeline, novel technologies and modeling features will also be considered based on the feedback of IGA users and customer requests.

6 Acknowledgement

Part of the work presented in this paper was developed within the project “DigiTain – Digitalization for Sustainability”. The project “DigiTain” is funded by the European Union and the German Federal Ministry for Economy and Climate Protection within the framework of the economic stimulus package no. 35c in module b under grant #19S22006c on the basis of a decision by the German Bundestag.

7 Literature

[1] Hartmann, S., Leidinger, L., Bauer, F., Benson, D., Nagy, A., Li, L., Pigazzini, M., Nguyen, L.: "Trimmed IGA B-Spline Solids vs. Standard Tetrahedra Finite Elements", 14th European LS-DYNA Conference 2023, Baden-Baden, Germany.
[2] 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.
[3] Schillinger, D.: "The p- and B-spline versions of the geometrically nonlinear finite cell method and hierarchical refinement strategies for adaptive isogeometric and embedded domain analysis", PhD Thesis, Technical University of Munich, 2012.
[4] Leidinger, L., Hartmann, S., Meßmer, M., Benson, D., Nagy, A., Li, L., Pigazzini, M., Nguyen, L.: "Analysis of 3D structures using IGA solids: Current status and future directions", 16th LS-DYNA Forum 2022, Bamberg, Germany.
[5] Leidinger, L.: "Explicit Isogeometric B-Rep Analysis for Nonlinear Dynamic Crash Simulations", PhD Thesis, Technical University of Munich, 2020.
[6] 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.
[7] Breitenberger, M.: "CAD-Integrated Design and Analysis of Shell Structures", PhD Thesis, Technical University of Munich, 2016.

© 2024 Copyright by Ansys Inc.