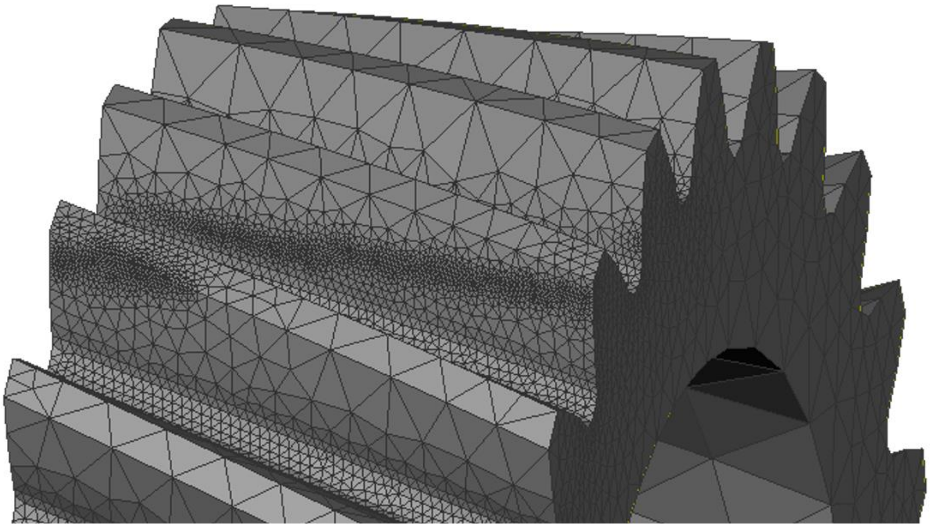
Gecko
Design for IGA-type discretization workflows

Gecko Technical Report 1

DC10 – Wei LI



This project has received funding from the European Union's Horizon Europe research and innovation program under grant agreement No 101073106
Call: HORIZON-MSCA-2021-DN-01

Funded by the European Union

Executive summary

This report outlines the preliminary advancements achieved in the GECKO project under the Horizon 2020 program, focusing on Isogeometric Analysis (IGA) techniques. Significant progress was made in FEM and IGA simulation studies using advanced workflows, including ANSA-Epilysis-META, ANSA-LSDYNA-META, and ANSA-Kratos-Rhino, for tensile test specimen analysis. The results, validated against theoretical and experimental data, demonstrated IGA's high precision and efficiency. Additionally, 1D, 2D, and 3D IGA solvers were developed for Poisson equations and tension problems, exhibiting excellent accuracy and providing a strong foundation for advanced applications, such as gear contact simulations.

Future research will focus on the IGA of contacts, with particular attention to the study of concentrated (Hertzian) contacts due to their critical role in machine elements such as rolling element bearings and gears. These efforts aim to advance the understanding and application of IGA in solving complex engineering challenges.

Table of contents

List of figures

List of tables

List of abbreviations

IGA	*Iso Geometric Analysis*
FEM	*Finite Element Method*
NURBS	*Non-uniform rational B-spline*
CAD	*Computer Aided Design*
CAE	*Computer-Aided Engineering*

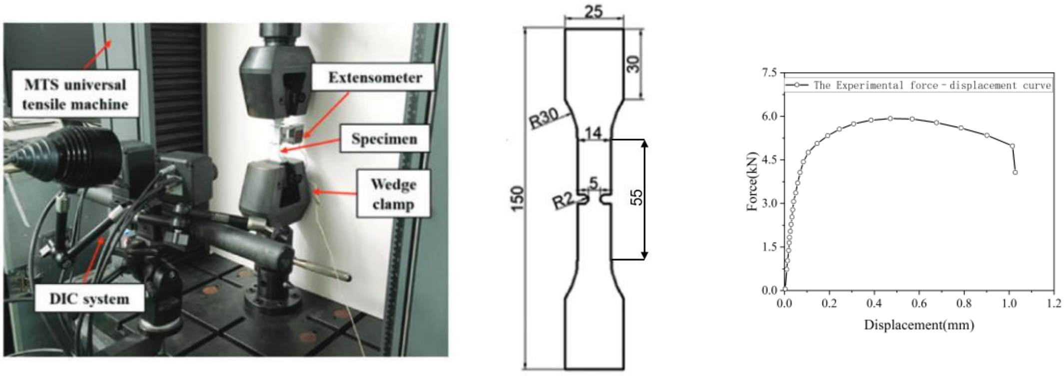
Introduction

The report begins with an overview of simulation studies performed using FEM and IGA workflows. Subsequently, the report also details the development of 1D, 2D, and 3D IGA solvers designed for solving Poisson equations and tension problems.

1 Test different workflows

According to [Tuo21], the test specimens, shown in Fig 1(b), are fabricated from a DP900 steel sheet with a thickness of 1.5mm, a gauge length 55mm, a Young's modulus E=207,600MPa, and a Poisson's ratio v=0.3. The uniaxial tensile test was conducted on an MTS universal tensile machine with a maximum load capacity of 100kN, Fig 1(a). Boundary conditions were applied such that one end of the specimen was fully fixed, while the opposite end was subjected to axial displacement at a loading rate of 2.0 mm/min. The tensile specimen exhibits complex mechanical behavior under tensile loading, including elastic deformation, yielding, plastic deformation, and fracture. The force–displacement curves is shown in Fig 1(c).

Our study focuses on the elastic deformation stage. As shown in Fig 1(c), for displacements ≤0.04 mm and uniaxial forces ≤3kN, the response demonstrates a clear linear elastic behavior.



(a) Experimental equipment

(b) specimen (mm)

(c) Force–displacement curve of uniaxial tensile test

Figure 1: Test specimen model [1]

Experimentally, Fig 1(c) shows when F = 3KN, the displacement is 0.0398mm. While Fig 1(c) is obtained by point-pick from the reference paper, which contains acceptable physical error.

Theoretically, the elongation Δx and stress σ are calculated by formula:

$$\Delta x = \frac{FL}{EA} \tag{1}$$

$$\sigma = \frac{F}{A} \tag{2}$$

Where F is the tensile force, L is the effective length, E is the Young's modulus, A is the initial cross-sectional area.

According to Fig 1(a), only the elongation of the gauge length 55 mm is considered. The theoretical elongation calculation is organized by three parts, as shown in table 1.

Table 1: Theoretical elongation calculation

Cross-sectional area (mm^2)	Effective length (mm)	Elongation Δx (mm)
14mm*1.5mm	(55-4)/2 = 25.5	0.0175
5mm *1.5mm	4	0.0077
14mm*1.5mm	(55-4)/2 = 25.5	0.0175
sum	/	0.0427

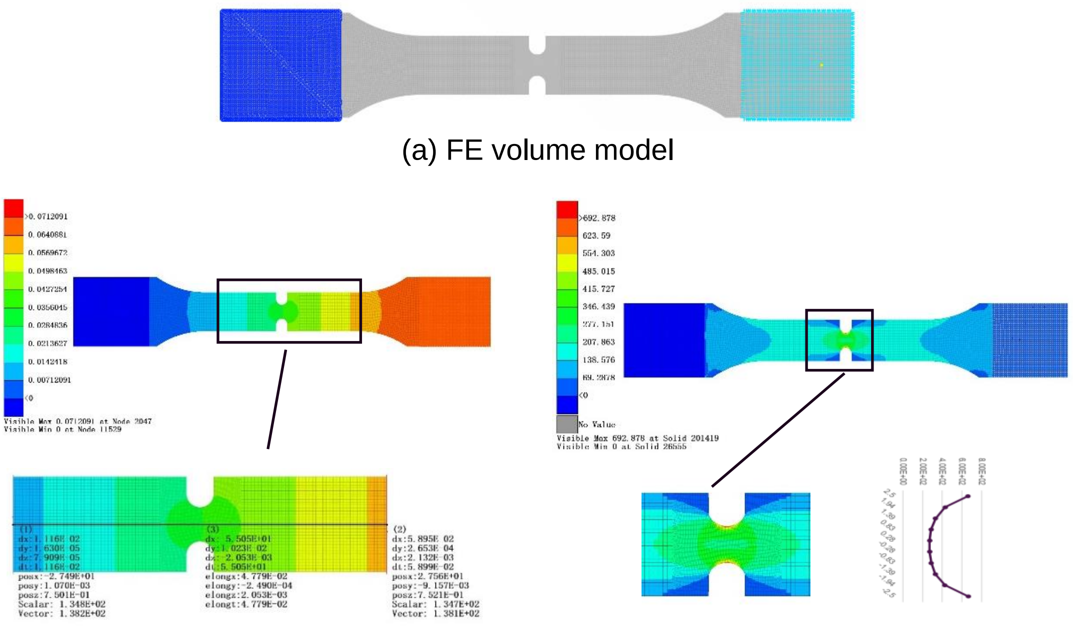
The nominal stress at the notch cross-section is 400 N/mm^2. The stress concentration factor 1.62[Pe08] should be considered, so the maximum stress at the notch is 648 N/mm^2.

In the subsequent analysis, the elastic deformation of the notched tensile specimen will be simulated using different workflows.

1.1 ANSA-Epilysis-META Workflow

In this section, a comprehensive FEM process ANSA–Epilysis (NastranSOL101)–Meta workflow is used to simulate the linear static analysis of the specimen. Fig 2(a) illustrates the boundary conditions and loading, where left end of the specimen is fully fixed, and a 3kN force is applied along the axial direction.

First, according to [Tuo21], the volume mesh is used. The minimum mesh size of the notched part is 0.15mm. Fig 2(b) shows the elongation of the gauge is 0.04779mm. Fig 2(c) shows the maximum stress in the notch region reaches 692.878 N/mm^2, the stress distribution along the symmetry plane typically resembles a parabolic curve. As the notch root is approached, the stress increases significantly due to the localized stress concentration. Since the specimen is symmetric, the stress distribution on both sides of the symmetry plane is mirror symmetric.



(a) FE volume model

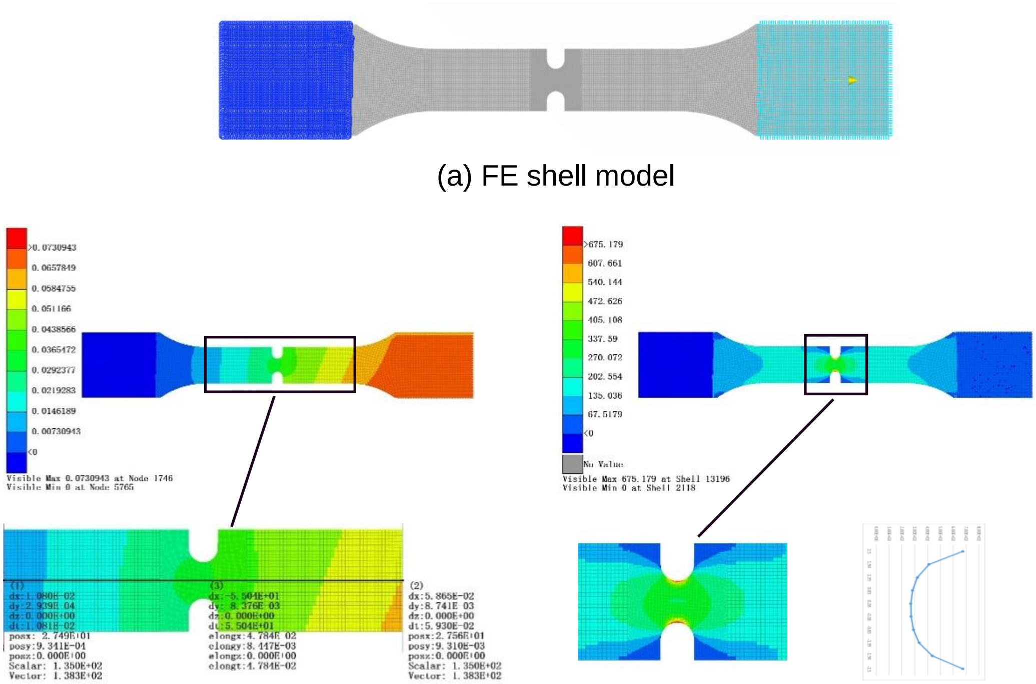
(b) Elongation

(c) Stress distribution

Figure 2: ANSA-Epilysis-META workflow simulation results with volume mesh

Second, the specimen mid-surface is extracted, and shell element is used. The min mesh size of the notched part is 0.15mm. Fig 3(b) shows the elongation of the gauge is 0.04795mm. Fig 3(c) shows the maximum stress in the notch region reaches 675.179 N/mm^2, the stress distribution along the symmetry plane exhibits a parabolic curve.

This section shows that the results of shell element and of volume element agree with each other, and compline with theory result. Therefore, shell element simulation will fulfill the analysis requirement.



(a) FE shell model

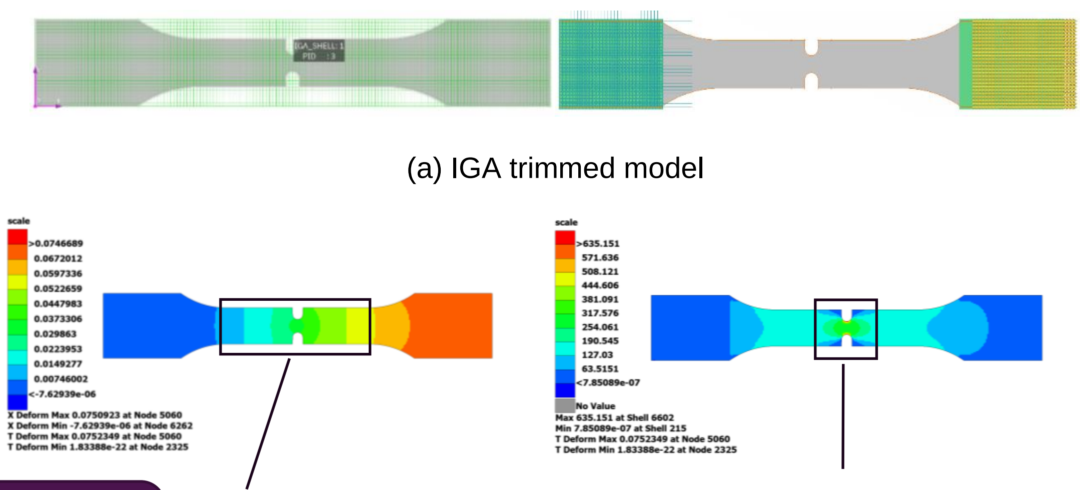
(b) Elongation

(c) Stress distribution

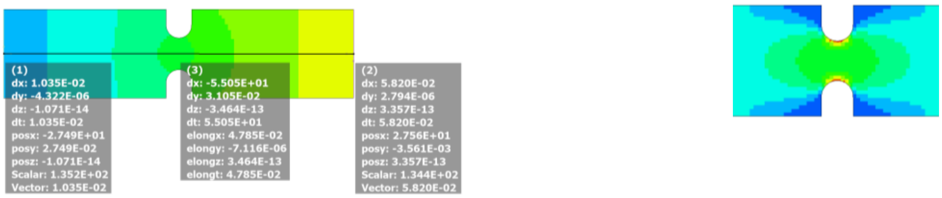
Figure 3: ANSA-Epilysis-META workflow simulation results with shell mesh

1.2 ANSA-Lsdyna-Meta workflow

In this section, IGA-based ANSA-Lsdyna-Meta workflow is used. Fig 4(a) illustrates the IGA trimmed model, and the boundary conditions which are consistent with Section 1.1.1. Lower polynomial order and refined mesh can be a good strategy to avoid the cross-talk effect in the notched part. In this case, biquadratic polynomial order and 1mm span are used. Then ANSA output the .key file to Lsdyna. Fig 4(b) shows the elongation of the gauge is 0.04785mm. Fig 4(c) shows the maximum stress in the notch region reaches 635.151 N/mm^2.



(a) IGA trimmed model



(b) Elongation (c) Stress distribution

Figure 4: ANSA-Lsdyna-Meta workflow simulation results

1.3 ANSA- Kratos -Rhino workflow

Additionally, IGA-based ANSA–Kratos–Rhino workflow is used. The IGA trimmed model and the boundary conditions are consistent with Section 1.1.2. As shown in Fig 5, the ANSA–Kratos plugin generates the input data (5 JSON files) required by the Kratos solver. Then Kratos performs the simulation and produces the result files which will be visualized by Rhino–Crocodile plugin. Fig 6(a) shows the displacement of whole model. Fig 6(b) shows the maximum stress in the notch region reaches 655.35MPa. The elongation 0.0476mm is obtained by calculating the displacement difference of specific parametric positions, see details in Table 2.

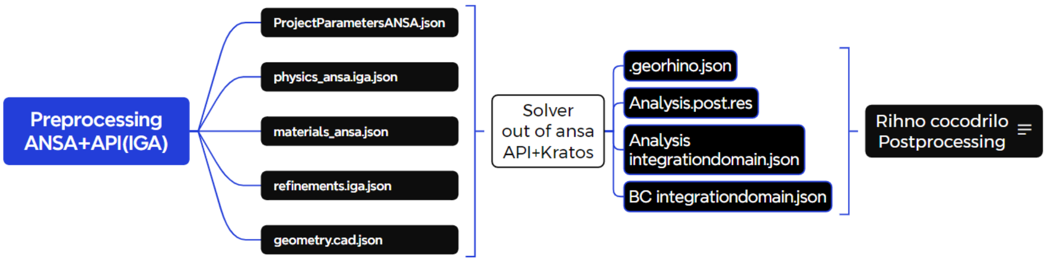


Figure 5: ANSA-Kratos-Rhino workflow

(a) Displacement (b) Stress distribution

Figure 6: Tension results in ANSA-Kratos-Rhino workflow

Table 2: Elongation calculation in ANSA- Kratos -Rhino workflow

Parametric position	Displacement (mm)	Elongation (mm)
[0.3167, 0]	0.010038	0.0476
[0.6833, 0]	0.057644	

To validate the performance of IGA method, the results obtained from FEM and IGA are compared with the theoretical solution and experimental data in table 3. The findings demonstrate that IGA provides acceptable results, closely matching the experimental data.

Table 3: Results from experiment, theory and different workflows

Method	Min mesh size (mm)	Elongation (mm)	Max stress (N/mm^2)	Deviation from Experimental elongation (%)
Theoretical	/	0.0427	648	7.29
Experiment	/	0.0398	/	0
FEM Volume elements	0.15	0.04779	692.878	20.08
FEM Shell elements	0.15	0.04784	675.179	20.2
IGA (LSDYNA)	1	0.04785	635.151	20.23
IGA (Kratos)	1	0.0476	655.35	19.6

2 IGA untrimmed 1D, 2D and 3D model test

In this section, solvers for 1D, 2D and 3D models are built according to IGA book [Co09]. Being compared with theory solution. The IGA solvers show good performance.

2.1 IGA 1D model test

There's 1D Poisson equation [Ng15] with the Dirichlet boundary conditions *u(0)* = *u(1)* = 0:

$$\frac{d^2u(x)}{dx^2}+b(x)=0, x\in(0,1) \tag{3}$$

Choosing b(x) = x, the exact solution is:

$$u(x)=-\frac{1}{6}x^3+\frac{1}{6}x \tag{4}$$

Fig 7 shows quadratic NURBS simulation for 1D Poisson equation. The 1D NURBS solver accurately reproduces theoretical solution across simple linear domains. This precision highlights the method's ability to capture smooth field variations with minimal computational resources.

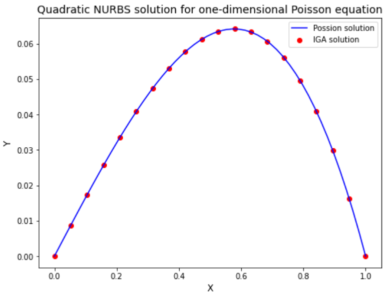


Figure 7: Quadratic NURBS simulation for 1D Poisson equation

2.2 IGA 2D and 3D model test

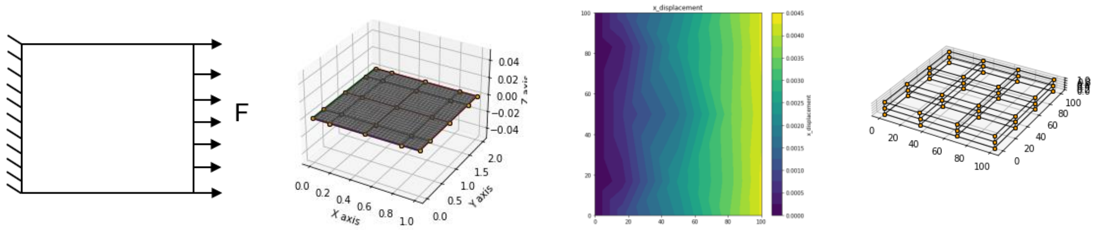
A square plate in Fig 8(a) with dimensions 100mm×100mm×1mm is analyzed under distinct simulation setups to validate the accuracy of 2D and 3D NURBS-based numerical methods. The plate is modeled with a Young's modulus E=207,600MPa, and a Poisson's ratio v=0.3. The left edge is fully constrained, the right edge experiences force 1000N:

- 2D Simulation: A line load uniformly distributed along the edge.
- 3D Simulation: A pressure load uniformly distributed over the surface corresponding to the right edge.

Theoretically, according to Eq (1), the elongation along x direction is 0.0048mm.

As shown in Fig 8(b), Bicubic NURBS and 4 elements are used to describe the geometry. Fig 8(c) shows the displacement area. The displacement along the right end is 0.0045 mm, which are consistent with theoretical solution. The coding implementation for the 2D problem serves as a foundational step for extending the methodology to 3D solid analyses.

Fig 8(d) depicts the control net of the 3D model. Cubic NURBS and 2 elements are used in x and y direction separately, quadratic NURBS and 1 element in z direction. The displacement along the right end is 0.00466mm. However, displacement plot for the 3D model is not included yet, as Python's limitations in volume rendering hinder visualization. Future work will explore the use of ANSA as a tool for visualizing 3D models effectively.



(a) Square plate (b) Bicubic 2D model (c) 2D displacement (c) 3D model

Figure 8: IGA untrimmed 2D, 3D models solution

Both 2D and 3D NURBS solvers accurately replicate the theoretical solution for displacement fields in a thin square plate. The maximum displacement occurs at the right edge due to the applied loading. The theoretical solution validates the numerical results, confirming the reliability of both 2D line load and 3D pressure load approaches for modeling boundary conditions.

3 CONCLUSIONS

This report highlights the significant advancements in Isogeometric Analysis (IGA) achieved within the GECKO project. Key accomplishments include precise FEM and IGA workflow simulations, and the development of accurate 1D, 2D, and 3D IGA solvers. These achievements establish a strong foundation for advanced applications, particularly in gear contact simulations. Future research will focus on the IGA of concentrated (Hertzian) contacts, addressing critical engineering challenges in machine elements.

4 REFERENCES

[Tuo21] Tuo, Z., Yue, Z., Zhuang, X., Min, X., Badreddine, H., Qiu, L., Gao, J., 2021. Comparison of two uncoupled ductile damage initiation models applied to DP900 steel sheet under various loading paths. International Journal of Damage Mechanics 30, 25–45.

[Pe08] Peterson, R. E., & Pilkey, W. D., 2008. Peterson's stress concentration factors (3rd ed., pp. 57–134). Wiley.

[Co09] Cottrell, J. A., Hughes, T. J., & Bazilevs, Y., 2009. Isogeometric analysis: toward integration of CAD and FEA. John Wiley & Sons.
[Ng15] Nguyen, V.P., Anitescu, C., Bordas, S.P.A., Rabczuk, T., 2015. Isogeometric analysis: An overview and computer implementation aspects. Mathematics and Computers in Simulation 117, 89–116.