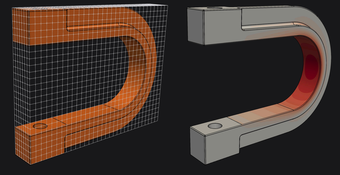
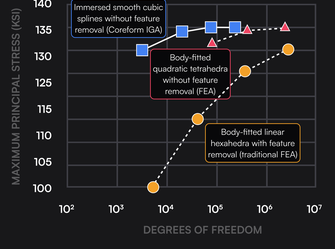
More accurate than traditional FEA

Coreform IGA is mathematically proven to be more robust and accurate per degree of freedom than traditional FEA.

Coreform IGA enables simulation directly on CAD with fully automated meshing.





Coreform IGA's high-order, smooth spline basis functions allow engineers to efficiently obtain accurate and useful solutions.

Isogeometric analysis runs FEA directly on CAD geometry.

Traditional finite element analysis (FEA) discretizes your CAD into hex or tet elements. Isogeometric analysis (IGA) is a finite element method that uses the spline basis that already defines the CAD as the analysis basis itself. Learn more at coreform.com/iga.

20 years. 4,000 papers.

Isogeometric analysis is not experimental. More than 4,000 peer-reviewed papers have been published since 2005, and the field has its own annual conference. Coreform is the lead sponsor of IGA 2026, October 11—14 in Tokyo.

(Register at iga2026.usacm.org)

Request a Benchmark

Send us your hardest model. We'll run it, share what we find, and talk through how we can help you today.

coreform

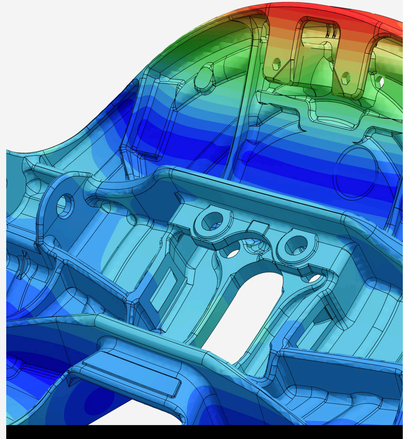
1427 South 550 East, Orem, UT 84097
801.717.2296
info@coreform.com
https://www.coreform.com

coreform IGA
for Abaqus

ISOGEOMETRIC ANALYSIS / IGA

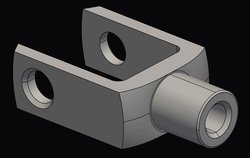
Fully-featured CAD. Nonlinear FEA.

- Abaqus / Standard support
- Works inside Abaqus / CAE
- Nonlinear, contact, modal, optimization

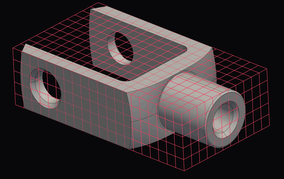


coreform

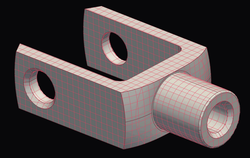
How Coreform IGA for Abaqus works



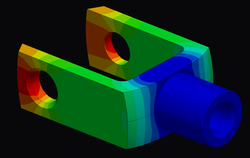
CAD Model
Fully-featured CAD, as designed.



IGA Meshing
Drop in a smooth spline grid.



Simulation Model
Trim to create volumetric CAD model.



Simulation Results
Run Abaqus on the volumetric CAD.

1 — 2 — 3 — 4

THE PROBLEM

~~Defeature. Mesh. Repeat.~~

For 50 years, FEA has started the same way: simplify the CAD, mesh it, lose accuracy, and hope the simplifications didn't matter.

We started Coreform to fix that. Engineers should spend their time on engineering, not meshing.

THE SOLUTION

Run Abaqus directly on fully-featured CAD.

Coreform IGA for Abaqus is a plugin that enables you to use Abaqus to simulate your geometry as designed.

Analyze more designs. Find better ones.

Dozens of complex CAD models from the GrabCAD Alcoa bracket challenge were analyzed with IGA with zero manual meshing time.

CAPABILITIES

Built for the Abaqus you already know.

- Abaqus / Standard support
- Works inside Abaqus / CAE
- Nonlinear, contact, modal, optimization
- Supports mixed IGA / FEA assemblies
- Fully scriptable Python API