coreform IGA
for Abaqus

Coreform IGA for Abaqus 2026.3

Table of contents

1 Initial Abaqus integration . . . 2
 1.1 Purpose-built Abaqus/CAE application . . . 2
 1.2 Automated setup for Abaqus 2025 . . . 2
2 IGA mesh generation . . . 2
 2.1 Immersed rectilinear meshes . . . 2
 2.2 Limited body-fitted IGA workflow . . . 2
 2.3 Standalone and parallel preprocessing . . . 2
3 Abaqus/Standard analysis . . . 3
 3.1 Linear, nonlinear, and modal analysis . . . 3
 3.2 Loads, boundary conditions, and constraints . . . 3
 3.3 Assemblies and material interfaces . . . 3
 3.4 Early surface-to-surface contact . . . 3
4 Results and probes . . . 3
 4.1 IGA results in Abaqus/CAE . . . 3
 4.2 Point and line probes . . . 4

Coreform IGA for Abaqus 2026.3 was the initial, limited-distribution release of the Coreform IGA for Abaqus product line. It introduced an integrated workflow for defining IGA discretizations in Abaqus/CAE, generating them with Coreform IGA Mesh, solving them with Abaqus/Standard, and inspecting IGA results.

Released: 17 March 2026

> i Historical, non-public release
>
> This release was shared with a limited audience and was not broadly announced or marketed. This page provides a concise historical summary of the capabilities available to that audience.
>
> Version 2026.3 predates the current Capabilities matrix, whose support classifications begin with 2026.7. The current **Production**, **Beta**, and **Alpha** classifications are therefore not applied retroactively on this page. Consult the matrix and Current capabilities and limitations for the support status of a current release.

1 Initial Abaqus integration

1.1 Purpose-built Abaqus/CAE application

This release introduced the Coreform IGA for Abaqus custom application. It retained the familiar Abaqus/CAE modeling modules and added Coreform-aware mesh, probe, and job workflows.

The initial application could create and manage IGA mesh definitions on assembly instances, preview the generated discretization, define result probes, and run the Coreform IGA Mesh and Abaqus/Standard stages as one job workflow. Job controls included separate processor counts for mesh generation and the Abaqus solve.

1.2 Automated setup for Abaqus 2025

The initial setup wizard configured the Abaqus environment, installed the Python dependency used by the integration, and authenticated the Coreform IGA for Abaqus license. This release targeted Abaqus 2025; Abaqus 2024 was not formally qualified.

2 IGA mesh generation

2.1 Immersed rectilinear meshes

Coreform IGA Mesh generated trimmed, immersed IGA discretizations directly from the model geometry exported by Abaqus/CAE. Users could control the background-grid element size, choose axis-aligned, object-aligned, or user-defined orientations, and adjust the frame and padding around the geometry.

The mesh authoring interface exposed linear, quadratic, and cubic spline bases with continuous, maximally smooth, or user-defined continuity. It also included penalty enforcement for immersed boundaries and a selective-integration option for nearly incompressible analysis.

2.2 Limited body-fitted IGA workflow

An initial body-fitted IGA path converted an existing Abaqus mesh into an IGA discretization. This path was limited to fully structured, mapped, all-hexahedral meshes; bottom-up and sweep meshes were not supported by the 2026.3 interface.

2.3 Standalone and parallel preprocessing

The new `coreform_iga_mesh` executable separated IGA mesh generation and persistence from the legacy solver workflow. It processed selected model parts, generated their trimmed quadrature and boundary data, and stored the resulting IGA mesh in a SQLite database for Coreform IGA Interop and the Abaqus user subroutines. The Abaqus job workflow could assign multiple processors to this preprocessing stage independently of the solver configuration.

3 Abaqus/Standard analysis

3.1 Linear, nonlinear, and modal analysis

The initial Abaqus/Standard integration supplied the user-element and material interfaces needed to solve body-fitted and immersed IGA models. The release included workflows for linear static analysis, multistep static analysis, geometrically nonlinear static analysis, natural-frequency extraction, and early direct implicit dynamics.

Available material behavior included linear elasticity, hyperelastic response, and isotropic metal plasticity. The nonlinear implementation included large-deformation kinematics, and selective integration was available for locking-sensitive, nearly incompressible models.

3.2 Loads, boundary conditions, and constraints

Coreform IGA Interop provided initial translation paths for prescribed and pinned boundary conditions, concentrated forces, uniform pressure, surface traction, body force, gravity, and moments transferred through kinematic couplings. These definitions were authored with standard Abaqus/CAE objects and translated where they targeted an IGA discretization.

3.3 Assemblies and material interfaces

The release included assembly workflows containing multiple IGA parts, mixed native finite-element and IGA parts, tied material interfaces, and both matching and nonmatching interfaces. An initial cellular-material workflow also read cell-based section assignments from Abaqus/CAE Boolean merges created with retained intersections, allowing distinct material definitions on the retained cells.

3.4 Early surface-to-surface contact

An early Abaqus/Standard surface-to-surface contact path generated conventional boundary elements on IGA surfaces so Abaqus could apply its native contact algorithm. The release included an early two-part, finite-sliding contact workflow, but contact remained an emerging capability rather than a broadly qualified production workflow.

4 Results and probes

4.1 IGA results in Abaqus/CAE

Coreform IGA Interop created a companion `<job-name>_iga.odb` containing a visualization mesh for the IGA geometry. It mapped displacement, stress, and equivalent plastic-strain results onto that mesh for supported time-domain analyses, allowing the results to be viewed with standard Abaqus/CAE visualization tools.

4.2 Point and line probes

The custom application introduced point and line probes for sampling IGA results at physical locations independent of visualization-mesh nodes. Users could preview probe locations in Abaqus/CAE, manage probe definitions with the model, and export displacement, stress, and equivalent plastic-strain histories from the solve.