

Example Problems

Table of contents

Getting started	1
Advanced capabilities	1
How examples are organized	2

These examples provide complete Abaqus/CAE workflows for preparing, solving, and post-processing Coreform IGA for Abaqus simulations. Choose an example by experience level and by the capability you want to explore.

Difficulty and feature maturity are independent: an advanced example may use released functionality, while an alpha example may require an advanced activation or authoring workflow.

Getting started

Absolute beginner Solid mechanics

Stress concentration in a thin plate with a hole

Build and solve a complete Coreform IGA model from an empty Abaqus/CAE database. This example assumes no previous Abaqus experience and introduces each module in sequence.

[Start the first example →](#)

Advanced capabilities

Advanced Thermomechanics Unsupported development

Thermal cycling of a hermetic glass-to-metal seal

Evaluate residual glass stress and metal plasticity through a qualification thermal cycle using temperature-dependent public material data.

[Open the advanced example →](#)

Advanced Solid mechanics Unreleased alpha

Nearly incompressible elasticity — Cook's membrane

Exercise the alpha locking-free formulation in a large-deformation Cook's membrane problem, including its environment-variable activation workflow.

[Open the advanced example →](#)

Advanced Solid mechanics Unreleased alpha

Surface-to-surface contact — door-lock assembly

Explore alpha surface-to-surface contact in a three-part immersed assembly with frictional sliding, large-deformation kinematics, and elastic-plastic materials.

[Open the advanced example →](#)

How examples are organized

While customers may develop their own prepared workflows, these examples will generally follow the “modules” workflow in Abaqus/CAE in a linear manner. Each example problem will begin with any relevant model database operations such as creating the model or defining physical constants. Thus the overall workflow for each example problem will generally follow:

1. Model definition
 - a. Create a new model database & rename the default model
 - b. Add a description to the model attributes, including the [consistent unit system](#)
2. Part module
 - a. Creation of base geometry
3. Property module
 - a. Material and material section creation
 - b. Assignment of material sections to geometry
4. Assembly module
 - a. Root assembly definition
 - b. Creation of merged-geometry parts
5. Step module
 - a. Step definition and Abaqus outputs
 - b. **[Coreform IGA]** Creation of Coreform IGA probes for accurate output
6. Interaction module
7. Load module
8. Mesh module
 - a. **[Coreform IGA]** Creation of Coreform IGA meshes
9. Job module
 - a. **[Coreform IGA]** Creation of Coreform IGA for Abaqus jobs
10. Visualization module
 - a. Visualizing field results
 - b. **[Coreform IGA]** Importing Coreform IGA probe data and plotting as XY-Data

Some examples extend this workflow with command-line steps between job creation and solving — for instance, [local refinement toward a feature](#) drives the `refinecf` utility on the generated model file. Each such example states its extra steps explicitly.