coreform IGA
for Abaqus

Surface-to-surface contact in a door-lock assembly

Table of contents

Unreleased Alpha Functionality

This example uses unreleased functionality intended for evaluation by advanced users. The formulation, activation workflow, and results may change without notice. Validate results against a reference solution or a conventional Abaqus discretization before using them for engineering decisions.

1 Overview

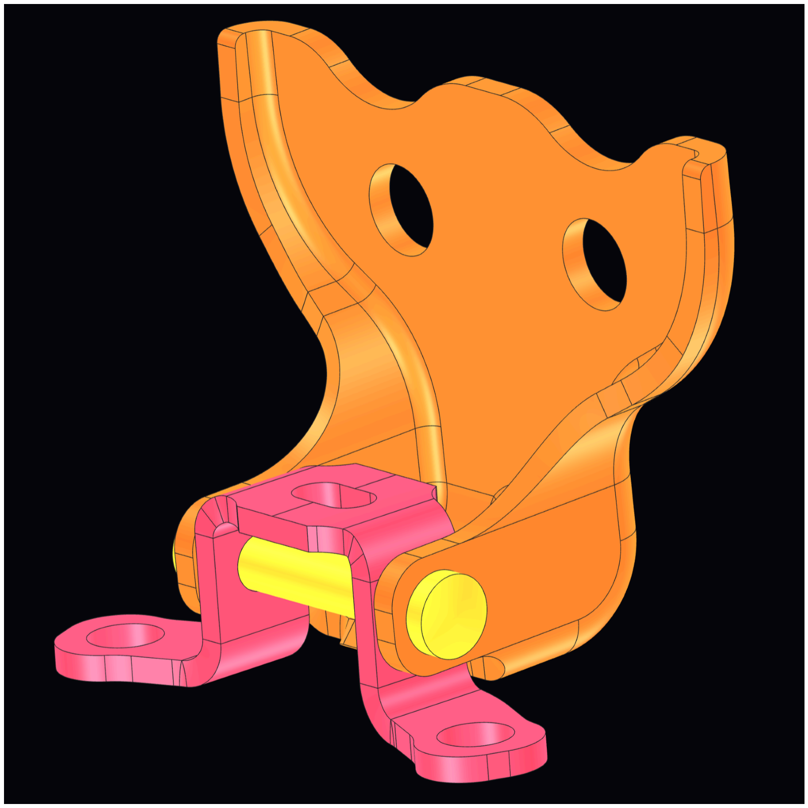

Figure 1: Door-lock assembly showing the hinge in orange, the link in pink, and the connecting pin in yellow.

1.1 Purpose

This example demonstrates contact-driven load transfer between three deformable, immersed isogeometric analysis (IGA) parts in an Abaqus/Standard static analysis. The assembly consists of a hinge, a link, and a connecting pin. A prescribed displacement pulls the hinge while the link is held in place, and the load is transferred through the connecting pin.

The problem exercises contact among multiple IGA parts, frictional sliding, large-deformation kinematics, elastic-plastic materials, and reaction-force output.

1.2 Supported contact scope

Coreform IGA for Abaqus currently supports only Abaqus/Standard surface-to-surface contact pairs. Immersed IGA surfaces may be paired with natively meshed Abaqus surfaces. You must explicitly select and pair the main and secondary surfaces for every intended interaction. Self-contact is not yet supported, so the main and secondary surfaces must belong to different bodies.

General contact is also not supported. Do not use General contact (Standard) or rely on an `ALL EXTERIOR` definition to discover possible contact automatically. Other contact formulations, including node-to-surface contact, are outside the currently supported scope.

The supported surface-to-surface capability remains experimental and may be sensitive to contact-surface selection, surface tessellation, mesh resolution, penalty parameters, and nonlinear solution controls. Convergence of this example does not establish that a different contact model will converge or produce an accurate result. Validate contact behavior, force transfer, and the quantities of interest against a reference solution or a conventional Abaqus discretization before using the results for engineering decisions.

1.3 How Coreform IGA for Abaqus handles contact

To set up a surface-to-surface contact problem, define the contact property, contact surfaces, and surface-to-surface interactions in Abaqus/CAE before creating the Coreform IGA for Abaqus job. During job generation, Coreform IGA for Abaqus retains the explicit contact-pair definitions and identifies the CAD faces referenced by each Abaqus surface. For each selected IGA boundary, the plugin evaluates the trimmed spline boundary and exports a triangulated layer of Abaqus `SFM3D3` surface elements. These surface elements provide Abaqus/Standard with conventional element-based surfaces on which its native surface-to-surface contact algorithm can operate.

The nodes of the exported contact surface are additional boundary degrees of freedom rather than control points of the IGA volume. Coreform IGA for Abaqus adds the relevant boundary nodes to the connected IGA user elements and couples their displacement to the spline displacement field with a penalty formulation. Contact forces calculated by Abaqus on the surface elements are therefore transferred through the boundary penalty coupling to the IGA volume. The Abaqus tangential contact penalty and the Coreform IGA for Abaqus boundary coupling penalty serve different purposes: the former enforces the selected friction law, while the latter couples the exported contact surface to the underlying IGA solution.

This construction allows Abaqus' contact machinery to interact with body-fitted or immersed IGA parts, but it introduces an approximation of the CAD boundary through the exported surface tessellation. Use a contact-surface tessellation and IGA mesh resolution that are fine enough to represent the local curvature and pressure distribution.

1.4 Model definition

The model uses the MMTS consistent unit system recorded in the test artifact and defines two steel materials. Both materials use isotropic elasticity and a two-point, linearly extrapolated plastic hardening definition.

Property	Weak steel	Strong steel
Density	8.00×10^{-9} tonne/mm^3	8.00×10^{-9} tonne/mm^3
Young's modulus	200×10^3 MPa	200×10^3 MPa
Poisson's ratio	0.29	0.29
Initial yield point	1.50×10^3 MPa at plastic strain 0	2.00×10^3 MPa at plastic strain 0
Second hardening point	150×10^3 MPa at plastic strain 10	200×10^3 MPa at plastic strain 10

The pin is partitioned into the cylindrical section and the two end caps. The cylindrical section uses the strong steel material and the two end caps use the weak steel material. The hinge and link use weak steel throughout.

Analysis setting	Value
Unit system	mm, N, MPa
Model	`Door-Lock`
Procedure	Static, General
Geometric nonlinearity	On
Initial increment	0.1
Minimum increment	10.0×10^{-9}
Link boundary condition	Pinned
Hinge displacement (U_1, U_2, U_3)	(0, 0, 5) mm

1.5 Contact definition

The model uses an isotropic penalty-friction interaction property. The normal contact behavior uses the Abaqus defaults.

Contact setting	Value
Formulation	surface-surface
Interaction type	friction
Friction coefficient	0.3
Maximum elastic-slip fraction	5.00×10^{-3}
Sliding	Finite
Include surface thickness	Yes

Contact setting	Value
Interaction	Main surface
—	—
pin_to_hinge_contact	Pin
pin_to_link_contact	Pin
hinge_to_link_contact	Hinge

All three interactions are active from the Initial step. They apply no initial overclosure adjustment and omit a user-specified initial clearance.

1.6 IGA discretization

Discretization setting	Value
Mesh	Immersed rectilinear
Spline degree	2
Continuity	C^1
Element size (x, y, z)	(2, 2, 2) mm
Padding elements (x, y, z)	(2, 2, 2)
Boundary tessellation size	$1\ mm$

The ratio between the nominal element size and boundary tessellation size is the recommended minimum for boundary tessellations. In some cases, a finer boundary tessellation may be needed for convergence, but the additional degrees of freedom will impact performance.

1.7 Output and test result

The field-output request and reaction-force history request are defined by the test:

Output	Value
Field variables	`S`, `U`, `PEEQ`
Field position	Averaged at nodes
History component	`RF3`
History region	`hold_link_surfaces`
Summed history regions	771

At the end of the prescribed displacement, the summed link reaction force is $-46.692\ kN$.

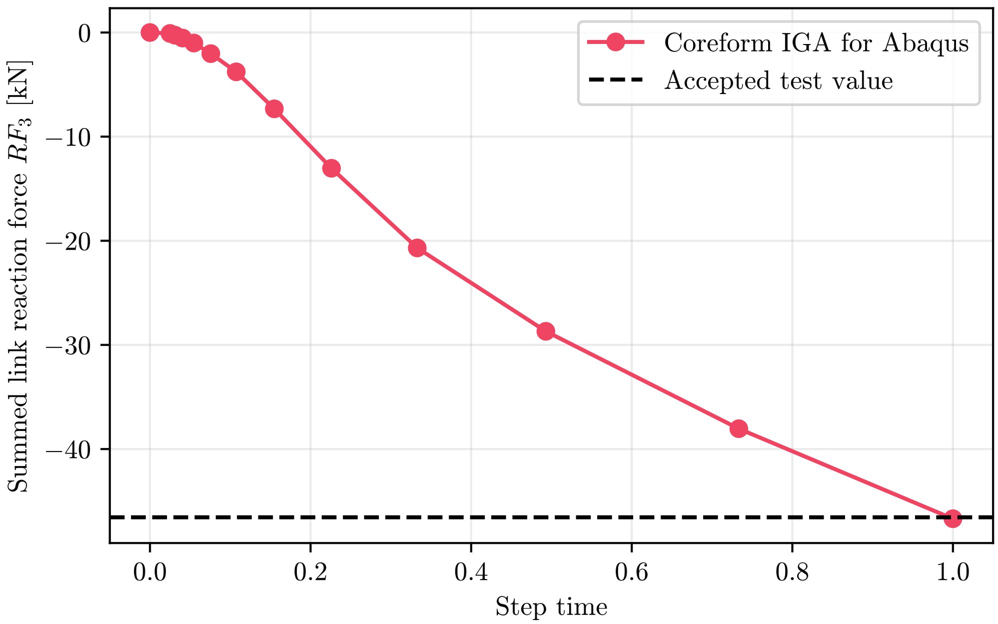

Figure 2: Summed link reaction force over the door-lock assembly load step.