coreform IGA
for Abaqus

Thermal cycling of a hermetic glass-to-metal seal

Table of contents

Unreleased Alpha Functionality

This advanced example intentionally exercises capabilities that are not yet part of the supported Coreform IGA for Abaqus product scope. These include the combined thermomechanical material workflow, temperature-dependent material tables, a prescribed-temperature history on an immersed multicell part, and a distributed coupling on an IGA boundary.

The `NT` field-output request is now accepted without aborting translation, but an IGA temperature-result extractor is not yet implemented. The request is skipped with a warning for IGA regions.

Treat this page as an engineering-development workflow. Passing input translation or completing a solve does not establish accuracy. Check the Abaqus keyword support guide and capabilities matrix for the status of each capability in the product version you are using.

1 Description

1.1 Engineering problem

A hermetic glass-to-metal seal electrically isolates conductive pins while maintaining a leak-tight path through a metal header. The seal is assembled at high temperature, where the glass can flow and wet the adjoining metal surfaces. As the assembly cools, the glass, pins, and header attempt to contract by different amounts. Compatibility across the bonded interfaces converts that thermal-expansion mismatch into residual stress in the glass and elastic-plastic strain in the metals.

The residual state matters for two reasons. Tensile principal stress can initiate or grow flaws in the brittle glass, while yielding in the header or pins redistributes the stress carried by the glass. Tandon et al. showed that apparently modest changes to the metal plasticity model can materially change the predicted glass stress [1]. Later Sandia work characterized the temperature dependence, processing sensitivity, and rate dependence of the constituent materials [2], [3], [4], and assessed a related seal model against experimental measurements [5].

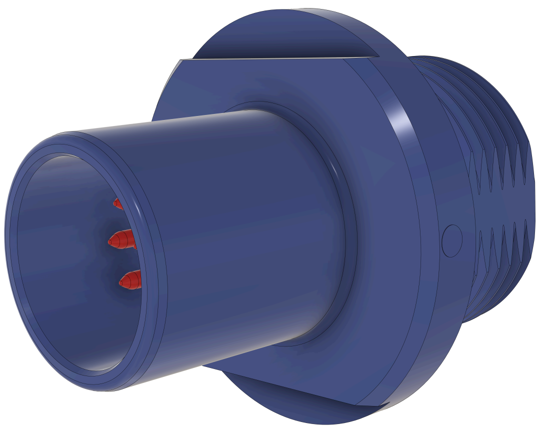
This example has two goals:

1. reproduce the engineering flow of the published finite-element assessment using public material data; and
2. provide a demanding development problem for thermomechanical, multicell immersed IGA workflows.

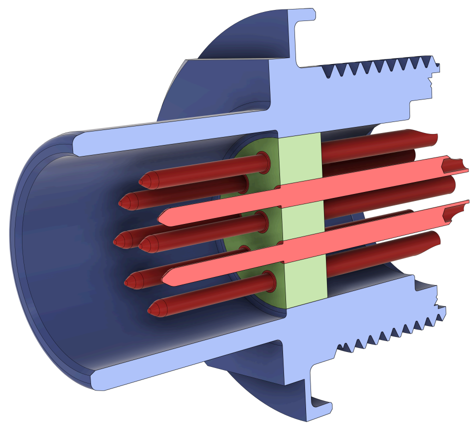
1.2 Geometry and components

The example STEP assembly is a substantially modified derivative of the Fischer Connectors SFE 104 A056-130 supplier model downloaded from 3D ContentCentral [6]. The citation documents the CAD model's provenance; because the source geometry was significantly changed, the example must not be interpreted as the manufacturer's as-supplied geometry.

> **i Required model file**
>
> Before beginning the Abaqus/CAE workflow, download the modified STEP assembly used by this example.



(a) Exterior isometric view.



(b) Isometric cross-section.

Figure 1: Exterior and isometric cross-section views of the modified connector geometry. The header is purple, the glass seal is green, and the pins are red.

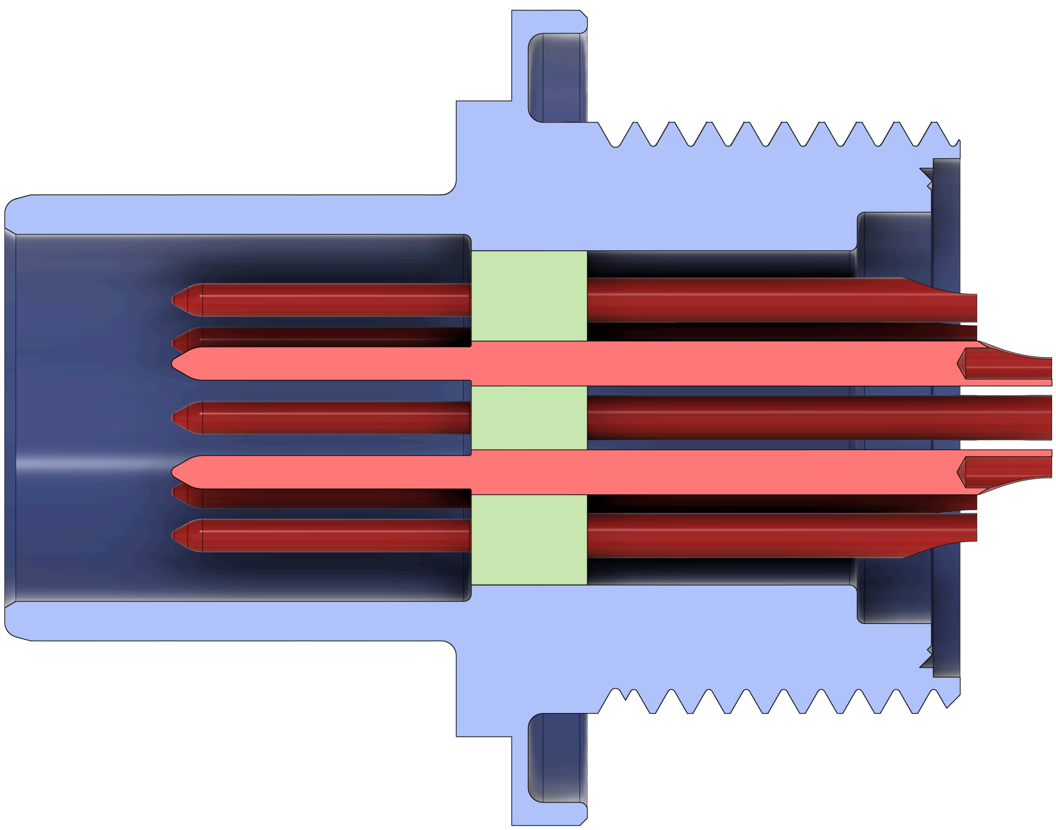


Figure 2: Longitudinal cross-section of the modified connector geometry. The header is purple, the glass seal is green, and the pins are red.

The modified STEP assembly contains 13 solid bodies. They are Boolean-merged with intersections retained, producing one deformable part with internal material boundaries:

Component	Material
Glass seal	Schott S-8061
Eleven electrical pins	Alloy 52 / UNS N14052
Metal header	Seal-conditioned 304L VAR stainless steel

Keeping a single merged part avoids artificial geometric gaps at the interfaces and permits a conforming spline field to span the shared boundaries. The initial model treats both glass-to-metal interfaces as perfectly bonded, consistent with the baseline model in the target paper [1]. Frictional separation is an important future sensitivity study rather than an assumption hidden in the baseline.

1.3 Thermal history and modeling scope

The analysis starts from a nominally stress-free state at 450 °C, near the reported S-8061 strain point. It then applies three spatially uniform temperature changes:

State	Temperature	Purpose
Initial	450 °C	Approximate the stress-free sealing state
`cool_to_room`	20 °C	Develop manufacturing residual stress
`cool_to_minus_55`	−55 °C	Apply the cold qualification excursion
`heat_to_200`	200 °C	Apply the hot qualification excursion

The temperature is prescribed uniformly, so this is a stress analysis rather than a transient heat-transfer analysis. It does not represent through-thickness thermal gradients, furnace dwell times, cooling rate, glass structural relaxation, or metal viscoplasticity. Instead, the thermo-elastic glass model begins at 450 °C and treats the high-temperature relaxation that created the stress-free state as already complete.

All dimensional values use the MMTS consistent unit system: millimeters, tonnes, seconds, newtons, megapascals, and degrees Celsius.

1.4 Engineering analysis approach

1.4.a Quantities of interest

The primary result is maximum principal stress in the glass, especially near the pin and header interfaces. Supporting results are:

- equivalent plastic strain (`PEEQ`) in the header and pins;
- plastic strain (`PE`) and logarithmic strain (`LE`) through the thermal cycle;

- displacement (`U`) of the seal and pins; and
- the prescribed nodal temperature (`NT`) when IGA temperature output becomes available.

Stress alone is not a deterministic glass-failure criterion. Predicting crack probability would require flaw-population statistics, a suitable Weibull or fracture-mechanics treatment, and relevant validation data [7], [8].

1.4.b Glass failure indicator

Use the positive maximum principal stress, $\sigma_1^+ = \max(\sigma_1, 0)$, as the primary screening field, not as a deterministic prediction of cracking. The recommended hierarchy is summarized in Table 1.

Table 1: Recommended hierarchy for evaluating glass failure.

Indicator or predictor	Recommended use
σ_1^+ contour and converged local peaks	Locate tensile regions and compare model variants
Tensile-stress surface-area and volume distributions	Quantify how much glass is highly stressed without reducing the result to one point
Surface- and volume-flaw Weibull probability	Predict fast-fracture probability after calibration to the relevant S-8061 lot, surface condition, and thermal process
Time-dependent fracture model	Assess environmentally assisted slow crack growth when dwell time, cycling rate, and service environment matter
Interface fracture or cohesive model	Assess debonding and loss of hermeticity separately from bulk-glass cracking

The Weibull calculation should use the complete multiaxial tensile stress field and distinguish surface from volume flaw populations. It also needs special care at idealized glass-to-metal corners: Sandia's FAILPROB work identifies such interfaces as potential stress singularities and shows why a raw maximum principal stress can remain mesh dependent [8]. Until relevant S-8061 Weibull parameters and a validated singularity treatment are available, report σ_1^+ and stressed-area or stressed-volume distributions as comparative indicators only.

1.5 Material properties

1.5.a Data strategy and provenance

The public reports do not contain the original measured dilatometry files or the complete production BCJ/SPEC parameter decks. An exact bit-for-bit reproduction of the Sandia analyses is therefore not possible from public information. The data used here distinguish four provenance classes:

Code	Meaning
R	Numerically reported by a source
D	Digitized from a published plot
C	Calculated or unit-converted from reported data
A	Analyst assumption or documented extrapolation

The codes are appended to the entries in the baseline material table below; multiple codes indicate that a value combines more than one provenance class. The baseline is a recommended public-data model rather than a claim that one particular production lot is represented exactly.

1.5.b Baseline material definitions

Property	S-8061 glass	Alloy 52 pins	304L VAR header
Density [tonne/mm³]	2.60×10^{-9} (**R/C**)	8.30×10^{-9} (**R/C**)	7.90×10^{-9} (**R/C**)
Poisson's ratio	0.22 (**R**)	0.29 (**R**)	0.30 (**R**)
Elastic modulus	Temperature-dependent digitized curve (**R/D/C**)	165,500 MPa (**R/C**)	Temperature-dependent Type 304 table (**R**)
Thermal expansion	9.3 ppm/°C (**R**)	9.95 ppm/°C (**R/C/A**)	Piecewise thermal-strain table (**R/C/A**)
Inelastic response	None (**A**)	Linear isotropic hardening (**D/C**)	Temperature-dependent multilinear J2 plasticity (**D/A**)

The glass density and catalog expansion are from S-8061 supplier data [9]. The elastic-modulus temperature dependence is the annealed specimen curve digitized from Chambers et al. and normalized to 69 GPa at 25 °C [2]. The sharp modulus decrease near 500 °C is rate dependent and should not be interpreted as a general elastic law above the glass transition.

Alloy 52 uses the manufacturer's 165.5 GPa modulus and 0.29 Poisson's ratio [10]. Its 9.95 ppm/°C expansion coefficient is the midpoint of the manufacturer's reported 30—450 °C range [11]. The baseline linear hardening definition is

$$\sigma_y = 234.4\,\mathrm{MPa} + 1924\,\mathrm{MPa}\,\bar{\varepsilon}^p.$$

The initial yield stress and slope are inferred from the curve published by Tandon et al. [1]. The supplier modulus is about 20% lower than the 207 GPa used in that paper, making the pin modulus an explicit sensitivity parameter.

The header uses seal-conditioned 304L VAR stress-plastic-strain curves digitized at 20, 100, 200, 300, 400, and 500 °C [2], [3]. The complete 20 °C curve is scaled by 1.2 at −55 °C, following the low-temperature trend used by Tandon et al.; this is an assumption, not a measurement of the later 304L VAR heat. The elastic-modulus values and accumulated thermal strain use public Type 304 data [12].

1.5.c Thermal strain

Figure 3 plots total linear thermal strain relative to 20 °C. The larger contraction of the stainless-steel header is the main source of thermal mismatch. Abaqus receives scalar expansion coefficients for S-8061 and Alloy 52, and a temperature-dependent secant-expansion table for 304L, while the analysis begins from the 450 °C initial temperature. Dotted segments mark Alloy 52 below 30 °C and above 450 °C, Type 304 below 0 °C, and S-8061 above the 450 °C limit of the baseline thermoelastic model.

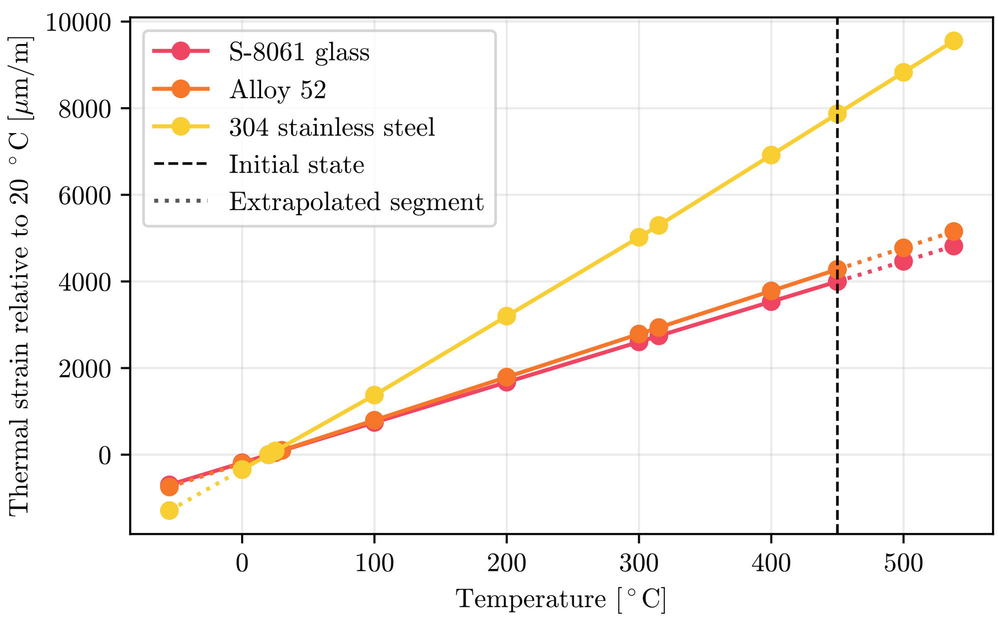


Figure 3: Recommended total linear thermal strain relative to 20 °C. Dotted portions are extrapolated beyond the stated data or model range.

1.5.d S-8061 elastic modulus

The measured temperature trend in Figure 4 is applied as piecewise-linear isotropic elasticity with constant Poisson's ratio. The baseline analysis does not include glass viscoelasticity or structural relaxation.

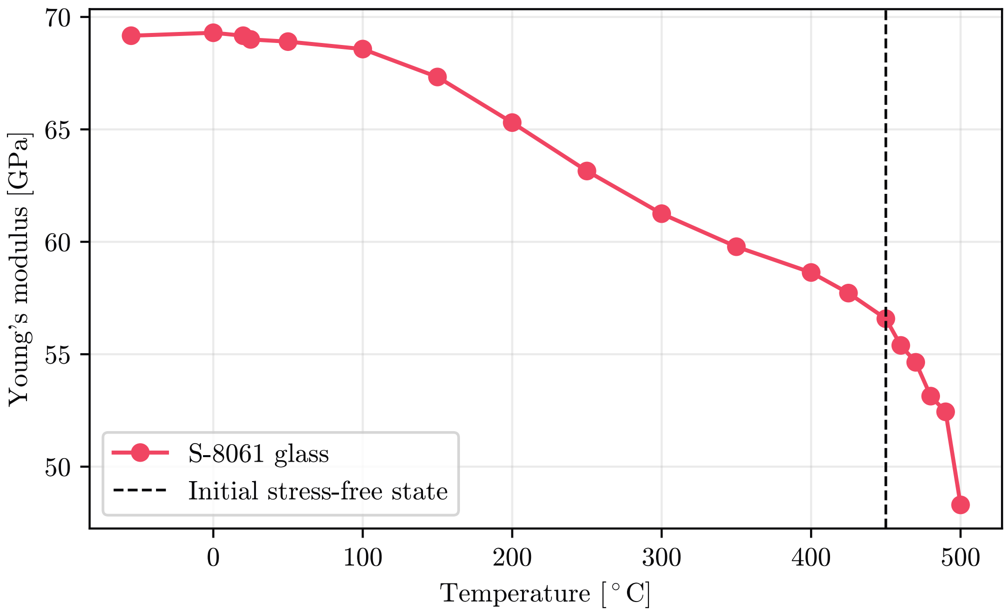


Figure 4: S-8061 Young's modulus digitized from the normalized annealed-specimen curve in Chambers et al. (2014).

1.5.e Metal plasticity

Figure 5 shows both sources of metal hardening data. The 304L curves are used directly as temperature-dependent multilinear plasticity. For Alloy 52, the initial example uses the simpler linear-hardening law and retains the digitized 2007 power-law curve for sensitivity studies. Each hardening table ends at its last digitized point: about 2.3% equivalent plastic strain for 304L VAR and 18% for Alloy 52. Abaqus holds the yield stress constant beyond the tabulated range, so any region that accumulates more plastic strain than this responds as perfectly plastic.

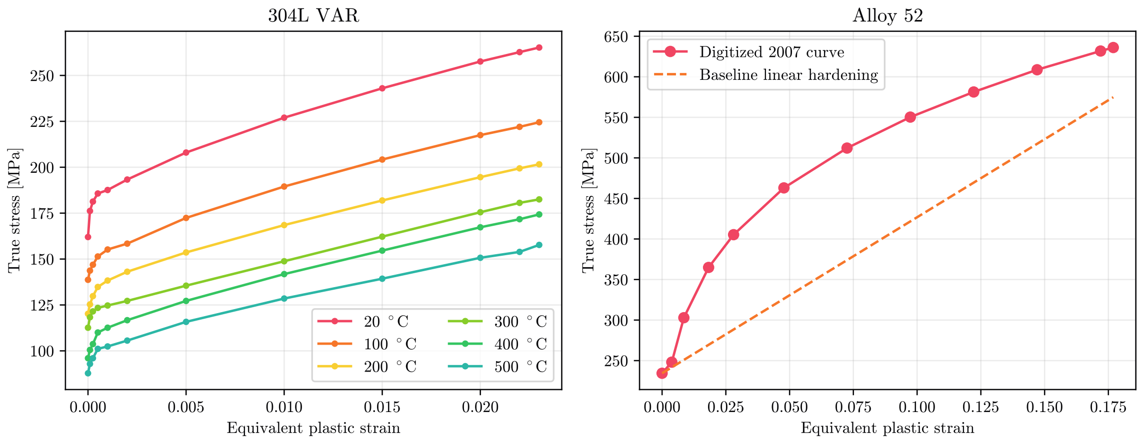


Figure 5: Metal plasticity data used by, or retained for sensitivity studies in, the example.

1.5.f Download the source and engineering data

The downloadable CSV files below preserve the engineering data used to build the plots and constitutive models. They are useful for reviewing the assumptions or developing alternative material fits, but their columns are **not** arranged for direct pasting into every Abaqus/CAE material editor. The Property module provides the exact Abaqus column order, a rendered table for every material behavior, and a headerless tab-delimited download that can be pasted directly into the corresponding dialog. Digitized values retain the precision needed to reproduce the tables, not an implied measurement accuracy.

Dataset	Download	Provenance and intended use
Recommended thermal strain	CSV	Derived from supplier and Type 304 mean-expansion data; includes documented extrapolations
S-8061 modulus	CSV	Digitized normalized annealed-glass curve from [2]
304L VAR multilinear plasticity	CSV	Digitized seal-conditioned curves from [2], [3]
Alloy 52 power-law curve	CSV	Digitized legacy sensitivity curve from [1]

2 Workflow in Abaqus/CAE

This section follows the same Abaqus/CAE module sequence as the introductory thin-plate example, but assumes that the reader already knows how to create materials, sets, steps, and jobs. The tables provide the exact object names and settings used by the completed baseline analysis.

The generated journal replays the complete Abaqus/CAE model-construction workflow, including the material tables described above.

Run the completed baseline analysis to provide its Abaqus/CAE journal.

2.1 Model definition

Begin with a new model database and create the model in Table 2. The description records the unit convention for anyone who opens the model later; Abaqus does not perform unit conversion or unit checking.

Table 2: Model database definition.

Setting	Value
Name	`Glass-to-Metal-Seal`
Description	`MMTS units: mm, N, s, tonne, MPa, deg C`
Do not use parts and assemblies in input files	On

2.2 Part module: import the geometry

2.2.a Import the STEP assembly

Import the downloaded `glass_to_metal_seal_connector.stp` using Table 3. With **Combine into a single part** off, the import creates one deformable part per solid body — 13 parts in total.

Table 3: STEP import settings.

Setting	Value
Modeling space	3D
Type	Deformable
Base feature	Solid
Combine into a single part	Off
Scale from file	Off

2.3 Assembly module: merge the solid bodies

Switch to the **Assembly** module for the Boolean merge. The imported solid bodies first need to be instanced because **Merge/Cut Instances** operates on assembly instances and creates a new part from the result.

2.3.a Merge the solid bodies

Instance the 13 imported parts, then combine them into a single part with **Merge/Cut Instances** using Table 4. Keeping the intersecting boundaries is essential: those faces separate the glass, pin, and header cells after the merge and allow each region to receive its own material.

Table 4: Boolean-merge settings.

Setting	Value
Name	`g2m_seal_merged`
Instances	All 13 dependent source instances
Original instances	Suppress
Keep intersecting boundaries	On
Domain	Geometry

2.4 Part module: create the component sets

Return to the **Part** module and make `g2m_seal_merged` current. The sets created here belong to the merged part, so they remain available on its instance when the assembly is regenerated.

2.4.a Create the component sets

Confirm that the merged part contains 13 cells before defining regions. If the STEP model is re-exported or repaired, identify each component from the geometry rather than relying on stored topology indices.

Create the four part sets defined in Table 5, Table 6, Table 7, and Table 8.

Table 5: Glass part set.

Setting	Value
Name	`SEAL_S8061`
Region	Glass seal shown in Figure 1 and Figure 2

Table 6: Pin part set.

Setting	Value
Name	`PINS_ALLOY52`
Region	Eleven electrical pins shown in Figure 1 and Figure 2

Table 7: Header part set.

Setting	Value
Name	`HEADER_SS304L`
Region	Metal header shown in Figure 1 and Figure 2

Table 8: Complete-part set.

Setting	Value
Name	`WHOLE_PART`
Region	All components shown in Figure 1 and Figure 2

The internal faces must remain present after the Boolean merge. They identify the material interfaces and preserve the separate cell assignments within the single part.

Temporarily hide the header set to inspect the two enclosed component families, as shown in Figure 6. This image is printed directly from an Abaqus/CAE viewport rather than reused from the CAD overview.

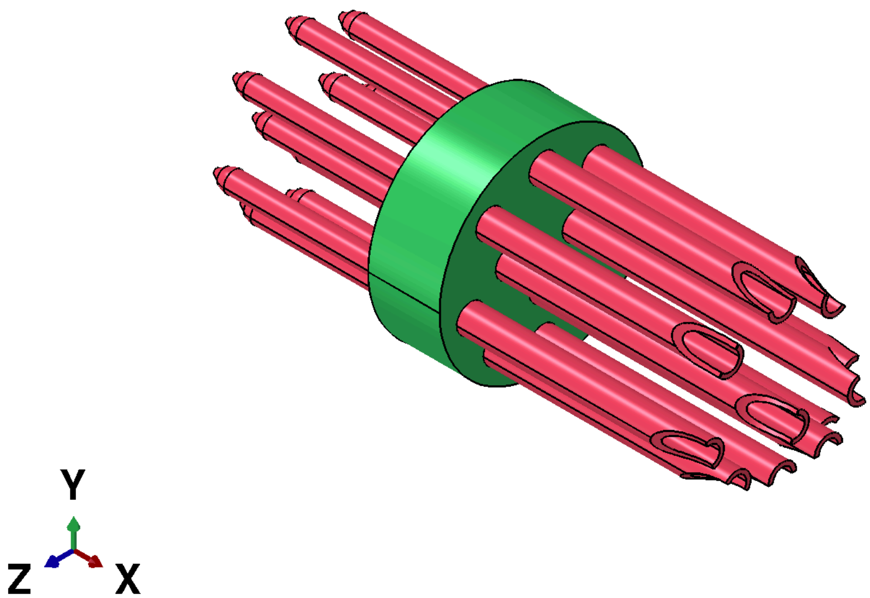


Figure 6: Abaqus/CAE viewport with the header temporarily hidden: the S-8061 glass-seal set is green and the Alloy 52 pin set is carnation.

2.5 Property module

The three materials have very different constitutive descriptions, so create and check them one at a time. The multi-row tables in this section reproduce the columns and row order expected by the corresponding Abaqus/CAE table editor. Each **Download data** link provides the same numeric rows in a headerless, tab-delimited file: open the file, copy all of its contents, select the upper-left data cell in Abaqus/CAE, and paste. Enable **Use temperature-dependent data** before pasting a table that includes a temperature column.

2.5.a Materials

2.5.a.a Create the S-8061 glass material

Create a material named `S8061_GLASS`, then add density, isotropic elasticity, and isotropic thermal expansion as summarized in Table 9. The glass is elastic in this baseline model; its stiffness varies with temperature, but its expansion coefficient is constant and no plasticity or viscoelastic relaxation is assigned.

Table 9: S-8061 glass material object.

Setting	Value
Name	`S8061_GLASS`
General > Density	2.60×10^{-9} tonne/mm^3
Mechanical > Elasticity > Elastic	Isotropic; temperature-dependent; see Table 10
Mechanical > Expansion	Isotropic; 9.30×10^{-6}/°C; zero temperature 20 °C

For **Mechanical > Elasticity > Elastic**, choose isotropic elasticity, enable temperature-dependent data, and paste Table 10.

Table 10: S-8061 isotropic elastic data entered in Abaqus/CAE.

Young's modulus [MPa]	Poisson's ratio	Temperature [°C]
69166	0.22	−55
69297	0.22	0
69166	0.22	20
69000	0.22	25
68903	0.22	50
68572	0.22	100
67330	0.22	150
65302	0.22	200
63149	0.22	250
61251	0.22	300
59782	0.22	350
58636	0.22	400
57719	0.22	425
56580	0.22	450
55393	0.22	460
54641	0.22	470
53137	0.22	480
52440	0.22	490
48300	0.22	500

Download data (headerless, tab-delimited)

For **Mechanical > Expansion**, choose isotropic expansion, enter 9.30×10^{-6}/°C, leave temperature dependence off, and enter 20 °C as the zero temperature.

2.5.a.b Create the Alloy 52 pin material

Create `ALLOY52` using Table 11. Unlike the glass, this material has constant elastic and expansion properties in the baseline model and a two-point isotropic-hardening law.

Table 11: Alloy 52 material object.

Setting	Value
Name	`ALLOY52`
General > Density	8.30×10^{-9} tonne/mm^3
Mechanical > Elasticity > Elastic	Isotropic; $E = 165,500$ MPa; $\nu = 0.29$
Mechanical > Expansion	Isotropic; 9.95×10^{-6}/°C; zero temperature 20 °C
Mechanical > Plasticity > Plastic	Isotropic hardening; see Table 12

Enter the scalar density, elastic, and expansion values directly from Table 11. Leave temperature dependence off for both elasticity and expansion, and enter 20 °C as the expansion zero temperature.

Add **Mechanical > Plasticity > Plastic**, leave temperature dependence off, and paste the true yield stress and equivalent plastic strain pairs from Table 12.

Table 12: Alloy 52 isotropic-hardening data entered in Abaqus/CAE.

Yield stress [MPa]	Plastic strain
234.4	0
580.72	0.18

Download data (headerless, tab-delimited)

2.5.a.c Create the 304L VAR header material

Create `SS304L_VAR` using Table 13. All three mechanical tables are temperature dependent, so confirm the temperature column is visible before pasting the elastic, expansion, or plastic data.

Table 13: 304L VAR material object.

Setting	Value
Name	`SS304L_VAR`
General > Density	7.90×10^{-9} tonne/mm^3
Mechanical > Elasticity > Elastic	Isotropic; temperature-dependent; see Table 14
Mechanical > Expansion	Isotropic; temperature-dependent; zero temperature 20 °C; see Table 15
Mechanical > Plasticity > Plastic	Isotropic hardening; temperature-dependent; see Table 16

For **Mechanical > Elasticity > Elastic**, choose isotropic elasticity, enable temperature-dependent data, and paste Table 14.

Table 14: 304L VAR isotropic elastic data entered in Abaqus/CAE.

Young's modulus [MPa]	Poisson's ratio	Temperature [°C]
193000	0.3	27
192000	0.3	93
187000	0.3	149
183000	0.3	204
179000	0.3	260
177000	0.3	316
170000	0.3	371
166000	0.3	427
160000	0.3	482
155000	0.3	538

Download data (headerless, tab-delimited)

For **Mechanical > Expansion**, choose isotropic expansion, enter 20 °C as the zero temperature, enable temperature-dependent data, and paste Table 15.

Table 15: 304L VAR isotropic thermal-expansion data entered in Abaqus/CAE.

Expansion coefficient [1/°C]	Temperature [°C]
1.72e-05	−55
1.72e-05	0
1.72e-05	20
1.72e-05	25
1.72e-05	100
1.77697666667e-05	200
1.79325571429e-05	300
1.79474576271e-05	315
1.82066684211e-05	400
1.83112744186e-05	450
1.83940854167e-05	500
1.84463320463e-05	538

Download data (headerless, tab-delimited)

For **Mechanical > Plasticity > Plastic**, select isotropic hardening, enable temperature-dependent data, and paste Table 16. Keep the rows grouped by temperature and in increasing plastic-strain order within each group.

Table 16: 304L VAR temperature-dependent isotropic-hardening data entered in Abaqus/CAE.

Yield stress [MPa]	Plastic strain	Temperature [°C]
194.4	0	−55
211.44	0.0001	−55
217.56	0.00025	−55
222.84	0.0005	−55
225.12	0.001	−55
231.96	0.002	−55
249.6	0.005	−55
272.4	0.01	−55
291.6	0.015	−55
309.12	0.02	−55
315.24	0.022	−55
318.24	0.023	−55
162	0	20
176.2	0.0001	20
181.3	0.00025	20
185.7	0.0005	20
187.6	0.001	20
193.3	0.002	20
208	0.005	20
227	0.01	20
243	0.015	20
257.6	0.02	20
262.7	0.022	20
265.2	0.023	20
138.7	0	100
143.7	0.0001	100
146.9	0.00025	100
151.4	0.0005	100
155.2	0.001	100
158.4	0.002	100
[illegible]	[illegible]	[illegible]

Download data (headerless, tab-delimited)

2.5.b Sections

With all three materials complete, create one homogeneous solid section for each component family. The section names make the later assignments easy to check in the Model Tree.

2.5.b.a Create the S-8061 glass section

Create the homogeneous solid section in Table 17 and associate it with `S8061_GLASS`.

Table 17: S-8061 homogeneous solid section.

Setting	Value
Name	`SECTION_S8061`
Category	Solid
Type	Homogeneous
Material	`S8061_GLASS`

2.5.b.b Create the Alloy 52 pin section

Create the homogeneous solid section in Table 18 and associate it with `ALLOY52`.

Table 18: Alloy 52 homogeneous solid section.

Setting	Value
Name	`SECTION_ALLOY52`
Category	Solid
Type	Homogeneous
Material	`ALLOY52`

2.5.b.c Create the 304L VAR header section

Create the homogeneous solid section in Table 19 and associate it with `SS304L_VAR`.

Table 19: 304L VAR homogeneous solid section.

Setting	Value
Name	`SECTION_SS304L`
Category	Solid
Type	Homogeneous
Material	`SS304L_VAR`

2.5.c Section assignments

Assign the sections to the part sets created earlier. After all three assignments, color-coding the viewport by material is a quick way to confirm that the glass, pins, and header are distinct and that no cell remains unassigned.

2.5.c.a Assign the S-8061 glass section

Select the glass region shown in Figure 1 and use Table 20.

Table 20: S-8061 section assignment.

Setting	Value
Region	`SEAL_S8061`
Section	`SECTION_S8061`

2.5.c.b Assign the Alloy 52 pin section

Select all eleven pins shown in Figure 1 and use Table 21.

Table 21: Alloy 52 section assignment.

Setting	Value
Region	`PINS_ALLOY52`
Section	`SECTION_ALLOY52`

2.5.c.c Assign the 304L VAR header section

Select the header shown in Figure 1 and use Table 22.

Table 22: 304L VAR section assignment.

Setting	Value
Region	`HEADER_SS304L`
Section	`SECTION_SS304L`

Color the viewport by **Section** to check the completed assignments. Figure 7 shows the resulting isometric Abaqus/CAE viewport; the 304L VAR header is purple and the Alloy 52 pins are carnation. The glass is enclosed by the header in this view, so hide the header temporarily and confirm the green seal region shown in Figure 6.

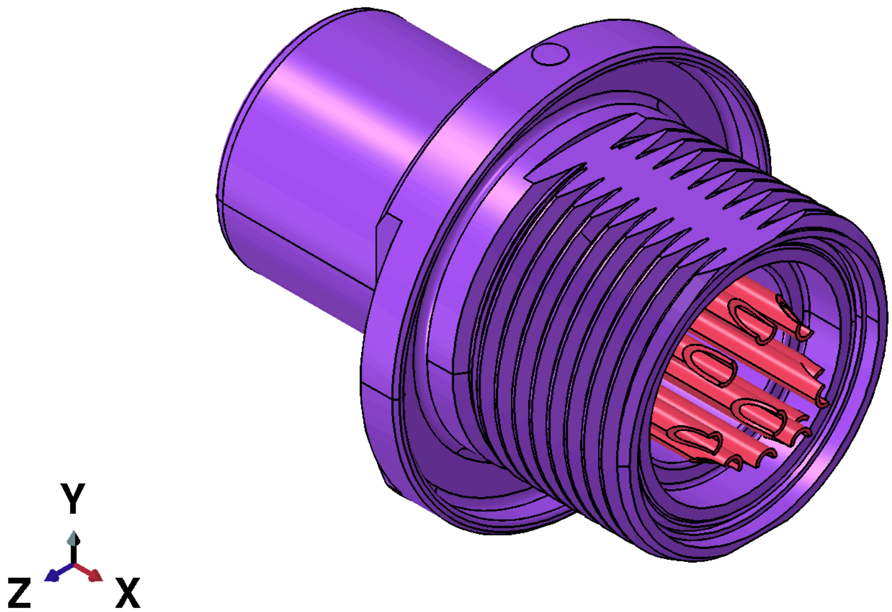


Figure 7: Abaqus/CAE viewport color-coded by section in an isometric view.

All three expansion definitions use 20 °C as their reference temperature; only the 304L definition is temperature dependent. The initial temperature condition introduced later establishes the 450 °C stress-free starting state from which Abaqus computes the subsequent thermal strains.

2.6 Assembly module: create the analysis regions

Return to the **Assembly** module to define the assembly-level regions used by the temperature condition and distributing constraint.

2.6.a Create the complete assembly set

Regenerate the assembly and confirm that the merged instance is named `g2m_seal_merged-1`. Create the complete-cell set in Table 23. This set will be used to apply one uniform temperature field to the entire connector.

Table 23: Complete-cell assembly set.

Setting	Value
Name	`WHOLE_ASSEMBLY`
Region	All cells of `g2m_seal_merged-1`

2.6.b Create the coupling surface and control point

Use a distributing constraint instead of fixing every point on the mounting surface. A fixed control point carries the resultant translational and rotational restraint, while uniform distributing weights transfer that restraint over the annular mounting surface. Individual points on the surface can still deform locally, which reduces the artificial stress and plastic strain produced by a face-wide pinned condition.

2.6.b.a Create the coupling surface

Create the mounting surface in Table 24. Coloring the assembly by **Surface** provides a useful check that only the planar mounting annulus has been selected, as shown in Figure 8.

Table 24: Coupled mounting-face surface.

Setting	Value
Name	`COUPLED_FACE_SURFACE`
Region	Planar annular mounting surface at the minimum-x end of the header

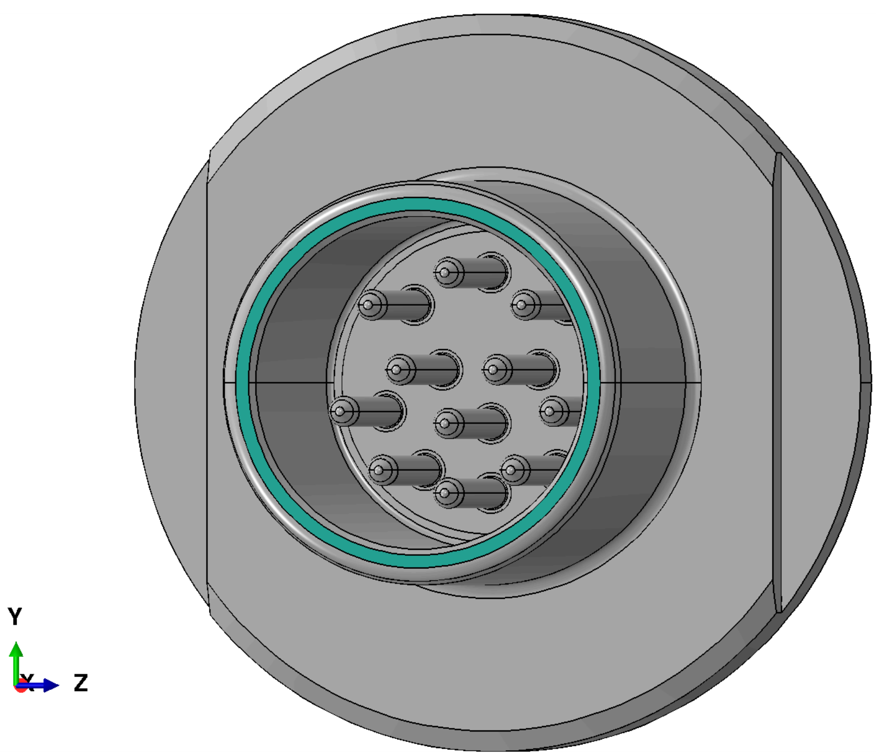


Figure 8: Abaqus/CAE assembly view with `COUPLED_FACE_SURFACE` highlighted in teal on the minimum-x mounting annulus.

2.6.b.b Create the coupling control point

Create a reference point at the centroid of `COUPLED_FACE_SURFACE`, then place it in the set defined by Table 25. The yellow `RP-1` marker in Figure 9 shows the created reference point; the set name used by the remaining workflow is `COUPLING_CONTROL_POINT`.

Table 25: Coupling control-point set.

Setting	Value
Name	`COUPLING_CONTROL_POINT`
Region	Reference point at the centroid of `COUPLED_FACE_SURFACE`

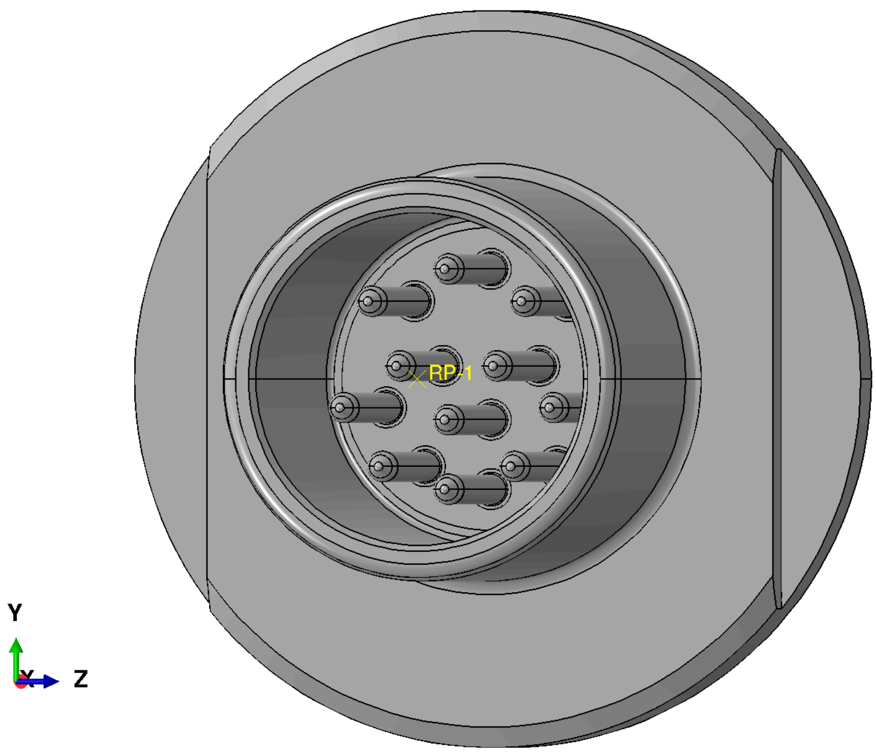


Figure 9: Abaqus/CAE assembly view showing the reference point at the centroid of the mounting surface.

The surface and control-point set will be used to create the distributing constraint in the Interaction module.

2.7 Step module

Create the three sequential **Static, General** step objects in Table 26, Table 27, and Table 28. Each step uses automatic incrementation and the same numerical controls, while the Load module supplies the temperature reached at the end of the step.

2.7.a Cool from the sealing state to room temperature

Create `cool_to_room` after the Initial step using Table 26. This is the manufacturing cooldown that develops the initial residual-stress state.

Table 26: Room-temperature cooling step.

Setting	Value
Name	`cool_to_room`
Previous step	Initial
Time period	1.0
Nlgeom	On
Initial increment	0.01
Minimum increment	10^{-6}
Maximum increment	0.1
Maximum increments	100
Equation solver	Direct
Matrix storage	Unsymmetric

2.7.b Apply the cold qualification excursion

Create `cool_to_minus_55` after `cool_to_room` using Table 27. The material state from the room-temperature cooldown carries forward automatically.

Table 27: Cold-excursion step.

Setting	Value
Name	`cool_to_minus_55`
Previous step	`cool_to_room`
Time period	1.0
Nlgeom	On
Initial increment	0.01
Minimum increment	10^{-6}
Maximum increment	0.1
Maximum increments	100
Equation solver	Direct
Matrix storage	Unsymmetric

2.7.c Apply the hot qualification excursion

Create `heat_to_200` after `cool_to_minus_55` using Table 28. This final step heats the already cooled assembly, so any plastic strain accumulated in either metal remains part of the starting state.

Table 28: Hot-excursion step.

Setting	Value
Name	`heat_to_200`
Previous step	`cool_to_minus_55`
Time period	1.0
Nlgeom	On
Initial increment	0.01
Minimum increment	10^{-6}
Maximum increment	0.1
Maximum increments	100
Equation solver	Direct
Matrix storage	Unsymmetric

Each step uses a time period of 1.0. Time is a load-proportional path parameter in this prescribed-temperature, rate-independent model; it does not represent the physical duration of the thermal cycle. The 0.1 maximum increment limits each accepted temperature change to one tenth of a step.

Unsymmetric storage is part of the baseline numerical definition: symmetric storage produced convergence difficulties during development, while the unsymmetric baseline completed the full thermal history without cutbacks.

2.7.d Request field output

Set the field-output variables to `LE`, `NT`, `PE`, `PEEQ`, `S`, and `U`, at averaged nodal position and every increment, as summarized in Table 29. Until IGA `NT` extraction is implemented, temperature output for IGA regions will be skipped with a warning; the remaining requested fields continue through translation.

Table 29: Field-output request.

Setting	Value
Name	F-Output-1
Variables	LE, NT, PE, PEEQ, S, and U
Position	Averaged at nodes
Frequency	Every increment

2.8 Interaction module

2.8.a Create the distributing constraint

Create the coupling constraint in the Interaction module using Table 30. This is the baseline model's one Interaction-module object. The continuum rotational formulation is used for this solid mounting surface, and all six coupling degrees of freedom remain active. Use the surface and control-point set created in the Assembly module as the coupling surface and control point, respectively.

Table 30: Distributed-coupling constraint.

Setting	Value
Name	DISTRIBUTED_COUPLING
Coupling type	Distributing
Control point	COUPLING_CONTROL_POINT
Surface	COUPLED_FACE_SURFACE
Influence radius	Whole surface
Weighting method	Uniform
Rotational coupling type	Continuum
U_1, U_2, and U_3	On
UR_1, UR_2, and UR_3	On

Turn on the constraint symbols and confirm that the distributing lines connect the control point to the complete mounting annulus, as shown in Figure 10.

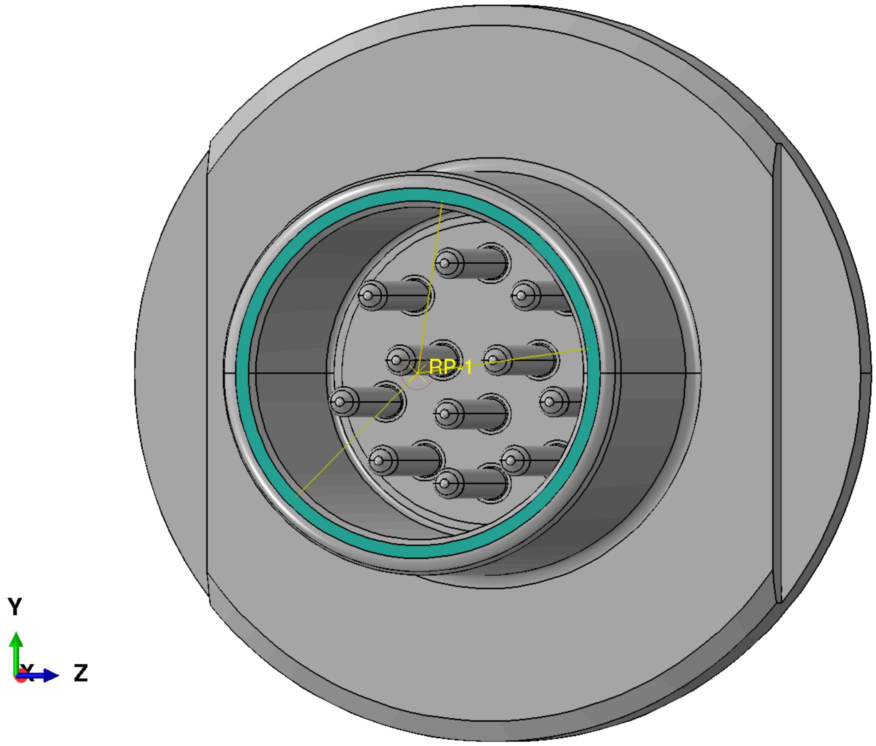


Figure 10: Abaqus/CAE constraint display showing `DISTRIBUTED_COUPLING` between the control point and the teal mounting surface.

After creating the constraint, restrain its control point in the Load module.

2.8.b Glass-to-metal interfaces

Do not create a separate contact or interface object for the glass-to-metal boundaries in the baseline model. The merged multicell part uses a shared spline field across its retained internal boundaries, so displacement compatibility is inherent in the discretization, just as it is across material boundaries in a contiguous finite-element mesh. The glass-to-metal interfaces are therefore perfectly bonded without an Abaqus interaction definition.

This assumption is appropriate for comparison with the target paper, but it is not universal. A model that permits separation or sliding would replace the shared-interface treatment with suitable contact or interface objects. Jamison et al. considered frictional contact with a coefficient of 0.3 for a related 304L VAR/S-8061 seal [5]; see Section 3.2 for the corresponding sensitivity study.

2.9 Load module

2.9.a Restrain the coupling control point

Apply the boundary condition in Table 31 to the control point in the Initial step. Because the surface motion is transferred through the distributing constraint, no face-wide displacement boundary condition is needed. This boundary condition restrains the control point used by DISTRIBUTED_COUPLING; the distributing constraint transfers that restraint to the annular mounting surface.

Table 31: Coupling control-point boundary condition.

Setting	Value
Name	COUPLING_CONTROL_POINT_BC
Step	Initial
Category	Mechanical
Type	Encastre
Region	COUPLING_CONTROL_POINT

With the Initial step active, turn on the boundary-condition symbols and confirm that the blue encastre glyphs surround the coupling control point, as shown in Figure 11.

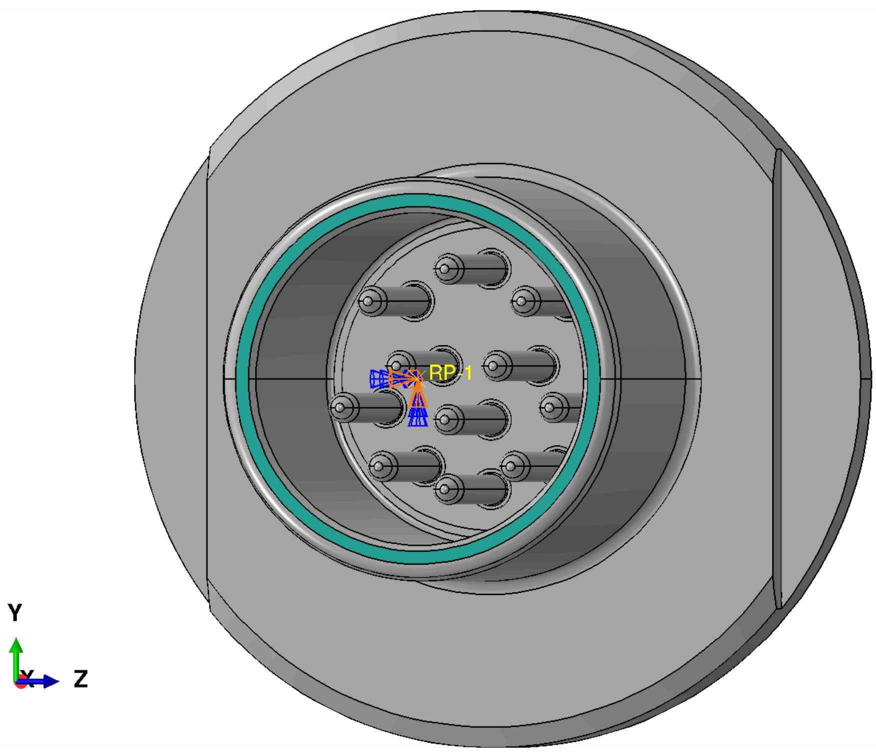


Figure 11: Abaqus/CAE boundary-condition display showing COUPLING_CONTROL_POINT_BC applied to the coupling control point.

2.9.b Prescribe the thermal history

Create the spatially uniform temperature predefined field in Table 32.

Table 32: Uniform-temperature predefined field.

Setting	Value
Name	`uniform_temperature`
Step	Initial
Category	Other
Type	Temperature
Region	`WHOLE_ASSEMBLY`
Distribution	Uniform
Cross-section distribution	Constant through thickness
Magnitude in Initial	450 °C
Magnitude in `cool_to_room`	20 °C
Magnitude in `cool_to_minus_55`	−55 °C
Magnitude in `heat_to_200`	200 °C

Abaqus ramps each changed predefined-field value through its step. Do not add a heat-transfer step, film condition, or conductivity unless the problem is being extended to solve the temperature field.

2.10 Mesh module

Create one **Immersed Rectilinear** Coreform IGA mesh on `g2m_seal_merged-1` using Table 33.

Table 33: Baseline immersed IGA mesh.

Setting	Value
Instance	`g2m_seal_merged-1`
Mesh Type	Immersed Rectilinear
Orientation	Axis-Aligned
Element Size — X	0.5 mm
Element Size — Y	0.5 mm
Element Size — Z	0.5 mm
Padding — Mode	Default
Degree	2
Continuity Mode	Smooth
Incompressibility	Off
Boundary Condition Enforcement — Setting	Medium
Small Cell Volume Ratio	0.1

Inspect the background grid around the thin glass and pin features before solving. A 0.5 mm mesh is an initial development discretization, not a mesh-converged recommendation. Refine the model until the glass stress and metal `PEEQ` quantities of interest are stable.

2.11 Job module

Create and submit the Coreform IGA for Abaqus job using Table 34.

Table 34: Coreform IGA for Abaqus job settings.

Setting	Value
Name	`glass_to_metal_seal`
Model	`Glass-to-Metal-Seal`
SIM Unit Licensing	Default
Run Mode	Background
Coreform IGA Mesh executable	Executable for the local Coreform installation
Scratch directory	Blank
Number of CPUs: Abaqus	16
Number of CPUs: IGA	16

The baseline performance reported in the Discussion was measured with these 16-CPU settings; adjust both counts to the local machine.

Submit the job through the Coreform job manager. Submission generates the IGA mesh, translates the model for Abaqus, and then runs Abaqus/Standard. Before interpreting results, review the translated model to confirm that all three material sections and their temperature-dependent tables are present, the distributed coupling and control-point boundary condition are retained, and the temperature field covers the complete assembly. The unsupported IGA `NT` extractor should issue a warning without preventing the remaining output variables from being written.

2.12 Visualization module

2.12.a Open and configure the output database

Open `glass_to_metal_seal_iga.odb` in the **Visualization** module and review each thermal state separately. The native IGA output database can be explored with the familiar Abaqus/CAE field-output, display-group, contour, and view-cut tools. Begin with the viewport settings in Table 35 so that the plotted deformation remains physical and the long connector is easy to read.

Table 35: Initial Visualization-module viewport settings.

Setting	Value
Plot state	Contours on deformed shape
Deformation scale factor	Uniform, 1.0
Visible edges	None
Projection	Parallel

The result views in Table 36 provide a useful first pass through the analysis.

Table 36: Recommended thermomechanical result views.

Region or display group	Field and invariant	Purpose
S-8061 glass	Maximum principal `S`	Screen tensile glass stress
304L VAR header	Mises `S` and `PEEQ`	Assess header yielding and redistribution
Alloy 52 pins	Mises `S` and `PEEQ`	Assess pin yielding and redistribution
Whole model	Magnitude of `U` and deformed shape	Check rigid-body motion and global deformation

2.12.b Plot tensile stress in the glass

Choose **Tools > Display Group > Create**, select **Part instances**, and retain only the S-8061 glass region using Table 37. The IGA output database contains one instance for each material cell, so confirm the selected region against Figure 1 and Figure 2 before plotting the field.

Table 37: Glass result display group.

Setting	Value
Name	`GLASS_ONLY`
Item	Part instances
Region	S-8061 glass seal

Open **Result > Field Output** and choose the stress invariant in Table 38. Maximum principal stress is the appropriate first screening field because glass fracture initiates in tension; compressive values are not interpreted as tensile failure demand.

Table 38: Glass tensile-stress field output.

Setting	Value
Primary variable	`S`
Position	Element centroid
Refinement	Invariant
Invariant	Max. Principal

In **Options > Contour**, specify the common limits in Table 39. Using the same limits at every thermal endpoint makes changes in the stress pattern visible without rescaling the legend from one frame to the next.

Table 39: Glass tensile-stress contour settings.

Setting	Value
Minimum	0 MPa
Maximum	450 MPa
Number of intervals	9

Open **Result > Options**, select the **Computation** tab, and retain the averaging settings in Table 40 so the viewport and legend state the threshold explicitly.

Table 40: Glass tensile-stress result options.

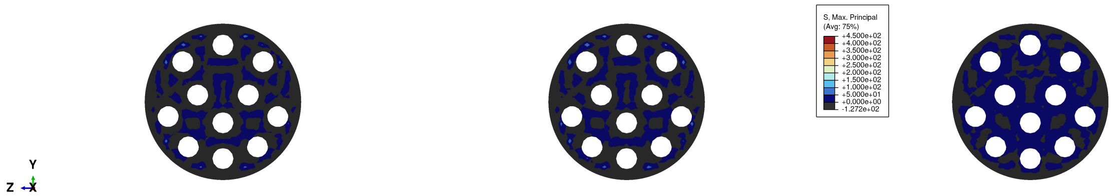
Setting	Value
Compute averages	On
Averaging threshold	75%

Use **Result > Step/Frame** to move among the last frames listed in Table 41.

Table 41: Thermal-cycle endpoint frames.

Thermal state	Step and frame
Room-temperature state	Last frame of `cool_to_room`
Cold qualification state	Last frame of `cool_to_minus_55`
Hot qualification state	Last frame of `heat_to_200`

Figure 12 shows the resulting Abaqus/CAE viewports. All panels use a 0—450 MPa tensile-screening scale; values at or below zero appear in the below-limit color. This is the σ_1^+ screening field recommended in Table 1, but the local peaks still require the convergence and singularity checks discussed there.



(a) After cooling from 450 °C to 20 °C.

(b) After the cold excursion to −55 °C.

(c) After heating from −55 °C to 200 °C.

Figure 12: Maximum principal stress in the glass at the three thermal-cycle endpoints, printed from Abaqus/CAE. All panels use the same 0—450 MPa tensile-screening scale.

2.12.c Inspect internal stress with a view cut

Return to the display group containing the complete model, then choose **Tools > View Cut > Create** and define the longitudinal plane in Table 42. Choose the standard **Top** view after activating the cut so that the camera looks normal to the cut plane.

Table 42: Longitudinal Abaqus/CAE view-cut definition.

Setting	Value
Name	`Connector centerline`
Shape	Plane
Origin	(0, 0, 0) mm
Normal	Global y: (0, 1, 0)
Show model above cut	Off
Show model on cut	On
Show model below cut	On

Select the last frame of `cool_to_minus_55`, plot maximum principal stress for the complete model, and apply Table 43. This signed range helps distinguish tensile and compressive regions across the bonded assembly.

Table 43: Cold-state maximum-principal-stress view.

Setting	Value
Primary variable	`S`
Position	Element centroid
Refinement	Invariant, Max. Principal
Minimum	−650 MPa
Maximum	650 MPa
Number of intervals	9

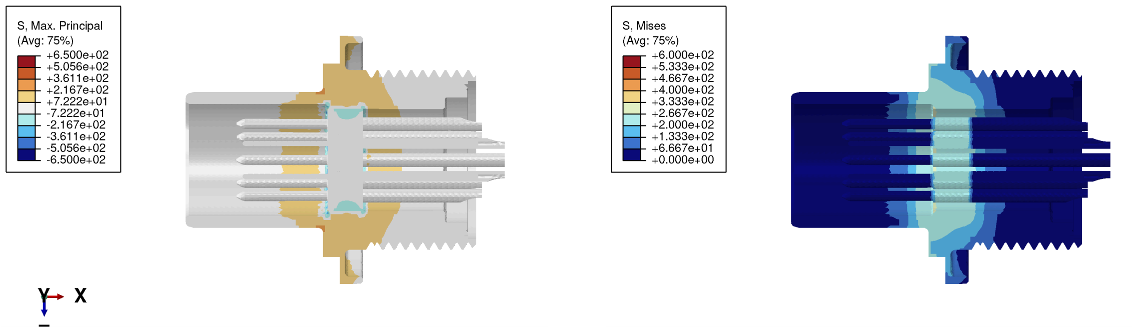
For the second view, create the metal-only display group in Table 44 and change the stress invariant to **Mises** using Table 45.

Table 44: Metal result display group.

Setting	Value
Name	`METALS_ONLY`
Item	Part instances
Region	304L VAR header and all Alloy 52 pins

Table 45: Cold-state metal von Mises stress view.

Setting	Value
Primary variable	`S`
Position	Element centroid
Refinement	Invariant, Mises
Minimum	0 MPa
Maximum	600 MPa
Number of intervals	9



(a) Maximum principal stress in all components; displayed range −650—650 MPa.

(b) Von Mises stress in the metal components; displayed range 0—600 MPa.

Figure 13: Cold-state internal stress views at −55 °C, printed from Abaqus/CAE using the centerline view cut.

The displayed ranges are intentionally limited so isolated support and interface peaks do not flatten the rest of the contour; the endpoint colors include values outside the displayed range. Use additional view cuts and material display groups to inspect the glass-to-pin and glass-to-header interfaces. Do not use a single global stress maximum as the only convergence measure: a local support artifact or boundary-tessellation feature can dominate it.

2.12.d Thermomechanical response

For each completed run, summarize the end of all three temperature steps in one table using the quantities in Table 46.

Table 46: Minimum thermomechanical response summary.

Quantity	Region	Reporting guidance
Maximum principal `S`	S-8061 glass	Report value and location
`PEEQ`	304L VAR header	Report the maximum and its relation to the support
`PEEQ`	Alloy 52 pins	Report the maximum and affected pins
Interface stress	Glass-to-pin and glass-to-header paths	Sample away from the support and geometric corners
Magnitude of `U`	Whole model	Report the maximum as a rigid-body-motion check

Pair the summary with material-filtered contour plots and section cuts through the most highly stressed pin and glass-to-header region. Report isolated global maxima separately from interface-path or small-volume statistics: a tessellation feature, material corner, or support-adjacent point can otherwise obscure the response of the seal itself.

The hot state must be interpreted as the continuation of the preceding cold excursion. Its stress and plastic strain therefore depend on the complete load history, not only on the final temperature of 200 °C.

Interpret the response in terms of thermal-expansion mismatch, metal yielding, and load redistribution between the header, glass, and pins. Stress or plasticity confined to the coupled mounting surface is more likely to reflect the support idealization than seal behavior, whereas repeatable peaks at the glass-to-metal interfaces warrant closer study.

The baseline remains limited by its assumed stress-free temperature, perfectly bonded interfaces, spatially uniform temperature, simplified Alloy 52 hardening, and omission of glass relaxation below the sealing state. Mesh, boundary tessellation, and penalty choices can also affect local extrema. These limitations should accompany any comparison with the published Sandia models or experimental measurements.

3 Discussion

3.1 Results

This section describes computational behavior. Engineering quantities used to assess the seal are discussed in the Visualization module.

3.1.a Computational performance

Problem size and performance are measured from the baseline run. The Abaqus/Standard row uses the solver's detailed performance summary.

Run the baseline analysis to populate this section.

A dash means that the current run predates the corresponding measurement or that the program did not report it. Abaqus solve time excludes model construction, mesh generation, and input translation. Performance comparisons require the same hardware, parallelism, output requests, and solver settings. Requesting field output every increment can make output writing a material part of the runtime.

3.1.b Solver convergence history

The summary and figure below include every increment attempt. Rejected attempts count toward the reported equilibrium-iteration total because they consume solver work.

Run the baseline analysis to populate this section.

```
No matching increment history is available.
```

Figure 14

Cumulative analysis time is the appropriate horizontal coordinate because time is the load-path parameter for these rate-independent static steps. For a rejected attempt, the marker is placed at its attempted endpoint; Abaqus leaves the reported total time at the last accepted equilibrium state. The temperature curve shows the prescribed path, while the accepted and rejected markers show how much of that path the solver actually reached.

Completing all steps is necessary but not sufficient: repeated cutbacks, unusually small increments, or convergence changes caused by mesh refinement should be discussed before treating a result as credible.

3.2 Recommended follow-on studies

Before a result from this model is treated as accepted, complete the following checks:

1. compare the legacy material approximation with the stress trends in Tandon et al. [1];
2. demonstrate mesh and boundary-tessellation convergence for the glass principal-stress locations;
3. compare the distributed support against the original pinned-face support away from the constrained region;
4. verify that energy and displacement histories are smooth across all three temperature steps; and
5. compare at least one discretization against a conventional Abaqus finite-element model.

The initial sensitivity matrix is:

Case	Change from the public-data baseline	Engineering question
P0	None	Recommended public-data model
P1	Increase Alloy 52 modulus from 165.5 to 207 GPa	How strongly does pin stiffness change glass stress?
P2	Remove or vary the 1.2 low-temperature 304L scaling	How sensitive is the cold excursion to header yielding?
P3	Replace Alloy 52 linear hardening with the digitized 2007 curve	How strongly does pin hardening redistribute stress?
P4	Replace the bonded interface with frictional contact	How strongly does interface behavior affect the result?

Cases P1—P4 should be treated as separate model variants, not as uncertainty bounds on the baseline.

4 References

[1] R. Tandon, M. K. Neilsen, T. C. Jones, and J. F. Mahoney, “Glass-to-metal (GTM) seal development using finite element analysis: Assessment of material models and design changes,” Sandia National Laboratories, 2007. Available: https://www.osti.gov/servlets/purl/1267109

[2] R. S. Chambers, J. M. Emery, R. Tandon, B. R. Antoun, M. E. Stavig, and C. S. Newton, "Characterization & modeling of materials in glass-to-metal seals: Part i," Sandia National Laboratories, SAND2014-0192, 2014. doi: 10.2172/1323655.

[3] B. R. Antoun, R. S. Chambers, J. M. Emery, and R. Tandon, "Small strain plasticity behavior of 304L stainless steel in glass-to-metal seal applications," Sandia National Laboratories, 2014. Available: https://www.osti.gov/servlets/purl/1140842

[4] B. R. Antoun, R. S. Chambers, J. M. Emery, and A. A. Brown, "Temperature-dependent small strain plasticity behavior of 304L stainless steel," Sandia National Laboratories, 2016. Available: https://www.osti.gov/servlets/purl/1348107

[5] R. D. Jamison *et al.*, "Validation assessment of a glass-to-metal seal finite-element model," Sandia National Laboratories, SAND2017-10894, 2017. doi: 10.2172/1399340.

[6] Fischer Connectors, "Fischer SFE 104 A056-130." Dassault Systèmes 3D ContentCentral, Nov. 2010. Available: https://www.3dcontentcentral.com/secure/download-model.aspx?catalogid=596&id=216520

[7] J. A. Salem and R. Tandon, "Test method variability in slow crack growth properties of sealing glasses," *International Journal of Fatigue*, vol. 32, no. 3, pp. 557—564, 2010, doi: 10.1016/j.ijfatigue.2009.07.018.

[8] G. W. Wellman, "FAILPROB—a computer program to compute the probability of failure of a brittle component," Sandia National Laboratories, SAND2002-0409, 2002. doi: 10.2172/793332.

[9] Richland Glass Company, *Precision glass GTMS sealing component specialists*. 2019. Available: https://www.richlandglass.com/wp-content/uploads/2019/04/GTMS-Lit_042519.pdf

[10] Hamilton Precision Metals, *HPM ni 52, UNS N14052*. Available: https://www.hpmetals.com/products/materials/nickel-strip-foil/hpm-ni-52

[11] Deutsche Nickel GmbH, *Technical datasheet ni 52: Dilaton 51, UNS N14052*. 2022. Available: https://www.deutsche-nickel.de/fileadmin/user_upload/download/Datenblaetter_englisch/Deutsche-Nickel-GmbH-Data-Sheet-Imperial-Units-Ni-52.pdf

[12] American Iron and Steel Institute, *High-temperature characteristics of stainless steels*. Nickel Development Institute / Nickel Institute, 1979. Available: https://nickelinstitute.org/media/4657/ni_aisi_9004_hightemperaturecharacteristics.pdf