



Workshop 1

Compression of a Rubber Seal

Interactive Version

Note: This workshop provides instructions in terms of the Abaqus GUI interface. If you wish to use the Abaqus Keywords interface instead, please see the “Keywords” version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Define surface-to-surface contact and self-contact.
- Perform a large displacement analysis with Abaqus/Standard.
- Use the Visualization module to create a compression load-deflection curve.

Introduction

In this workshop, a compression analysis of a rubber seal is performed to determine the seal's performance. The goal is to determine the seal's compression load-deflection (CLD) curve, deformation and stresses. The analysis will be performed using Abaqus/Standard.

As shown in Figure W1-1, the top outer surface of the seal is covered with a polymer layer, and the seal is compressed between two rigid surfaces (the upper one is displaced along the negative 2-direction; the lower one is fixed). During compression, the cover contacts the top rigid surface; the outer surface of the seal is in contact with the cover and the bottom rigid surface; in addition the inner surface of the seal may come into contact with itself.

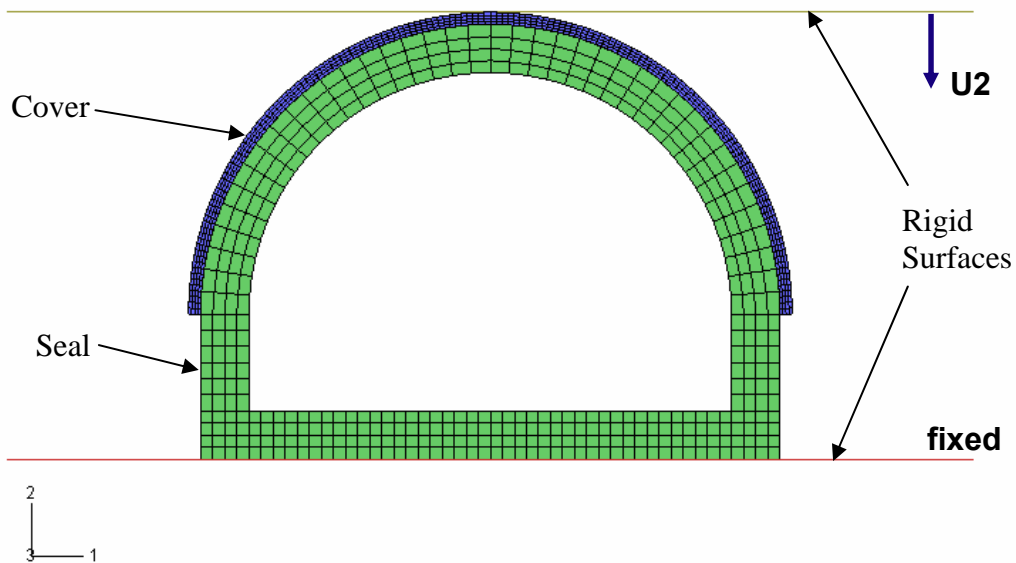


Figure W1-1. Seal model: meshed assembly

Preliminaries

1. Enter the working directory for this workshop
`../contact/interactive/seal`
2. Run the script `ws_contact_seal.py` using the following command:
`abaqus cae startup=ws_contact_seal.py.`

The above command creates an Abaqus/CAE database named `seal.cae` in the current directory. The geometry, mesh, and material definitions are included in the model named `seal`. You will add the necessary data to complete the model, run the job, and finally postprocess the results.

Defining the step and contact interactions

1. In the Model Tree, double-click the **Steps** container to create a static, general step named `PushDown` with a time period of `1`. Turn on **Nlgeom**. In the **Incrementation** tabbed page of the step editor, enter a value of `0.005` for **Initial Increment size** and `200` for **Maximum number of increments**.
2. Define a surface-to-surface contact interaction between the seal and the bottom rigid surface.
 - a. In the Model Tree, double-click the **Interactions** container. In the **Create Interaction** dialog box, name the interaction `BotSeal` and select the step `PushDown` and **Surface-to-surface contact (Standard)**. Click **Continue**.
 - b. You will be prompted to select a master surface. In the prompt area, click **Surfaces**. In the **Region Selection** dialog box that appears, select the

predefined surface **Bottom** and toggle on **Highlight selections in viewport** to view this surface. Click **Continue**.

- c. In the prompt area, select **Surface** as the slave surface type. In the **Region Selection** dialog box that appears, select the predefined surface **SealOuter** and visualize this surface. Click **Continue**.

The interaction editor appears.

- d. In the **Edit Interaction** dialog box, enter a value of **0.001** for **Specify tolerance for the adjustment zone** and accept all other defaults. Click **OK** to exit the interaction editor.

Note that Abaqus/CAE automatically assigns the predefined (also the only available) interaction property **frictionless** to this interaction definition.

3. Using a similar procedure, define the following surface-to-surface contact pairs as listed in Table W1–1 with the interaction property **frictionless** and the adjustment value of **0.001**.

Table W1–1. Contact pairs

Interaction Name	Master Surface	Slave Surface
TopCover	Top	Cover
SealCover	SealOuter	Cover

Question W1–1: In the interaction **SealCover**, why do we choose **SealOuter** as the master surface?

4. Create a self-contact interaction for the inner surface of the seal.
 - a. In the Model Tree, double-click the **Interactions** container. In the **Create Interaction** dialog box, name the interaction **SealSelf** and select the step **PushDown** and **Self-contact (Standard)**. Click **Continue**.
 - b. In the **Region Selection** dialog box that appears, select the predefined surface **SealInner** and visualize this surface. Click **Continue**.
The interaction editor appears.
 - c. In the **Edit Interaction** dialogue box, accept all defaults and click **OK**.

Defining boundary conditions and output requests

Asymmetric lateral sliding of the model is prevented by constraining the seal and the cover along their vertical symmetry axes in the 1-direction. The bottom rigid surface is fixed, and a displacement of -6 units is applied to the top rigid surface along the 2-direction to compress the seal between the two surfaces. To complete these boundary conditions:

1. In the Model Tree, double-click the **BCs** container to create a **Displacement/Rotation** type boundary condition named **Fix1** in the step


- PushDown.** When prompted to select the region, click **Sets** in the prompt area (if necessary). In the **Region Selection** dialog box, select the predefined set **Fix1**, toggle on **Highlight selections in viewport** to visualize the selection, and click **Continue**. In the **Edit Boundary Condition** dialog box, toggle on **U1**, accept the default value of 0, and click **OK**.
2. Create a **Symmetry/Antisymmetric/Encastre** type boundary condition named **FixBot** to encastre the predefined set **BotRP** (the reference node of the bottom rigid surface).
 3. Create a **Displacement/Rotation** type boundary condition named **PushDown** in the step **PushDown** to define the displacement of the top rigid surface. Select the predefined set **TopRP** (the reference node of the top rigid surface). Specify a value of 0 for **U1** and **UR3**, and **-6** for **U2**.
 4. Edit the field output request named **F-Output-1** to include the nominal strain, **NE**.
 5. Create a new history output request in the step **PushDown** for the set **TopRP** to write the history of the variables **Displacements: U** and **Forces: RF** to the output database file.

Running the job and postprocessing the results

1. Create a job named **sea1** for the model **Seal**.
2. Save your model database, submit the job for analysis, and monitor the job's process.

When the job is complete, open the output database file **sea1.odb** in the Visualization module and postprocess the results.

3. Plot the undeformed and the deformed model shapes. To distinguish between the different instances, color code the model based on instances.

Tip: From the toolbar, select **Part instance** from the color-coding pull down menu, as shown in Figure W1-4 (or use the **Color Code Dialog** tool  to customize the color for each instance).

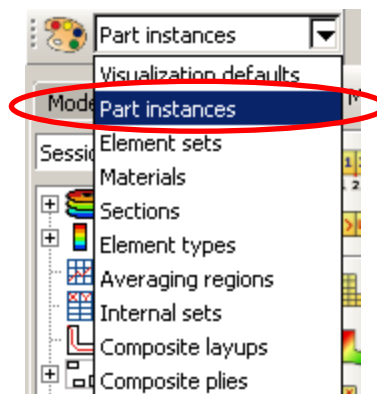



Figure W1-4. Color-coding pull down menu

4. Use the **Animate: Time History** tool  to animate the deformation history.

5. Display only the seal. In the Results Tree, expand the **Instances** container underneath the output database file named **sea1.odb**. Click mouse button 3 on the instance **SEAL-1** and select **Replace** from the menu that appears.

Abaqus/CAE now displays only this instance.

6. Contour the Mises stress of the seal on the deformed shape. If necessary, use the frame selector  in the context bar to select the final increment.

The contour plot is shown in Figure W1–5.

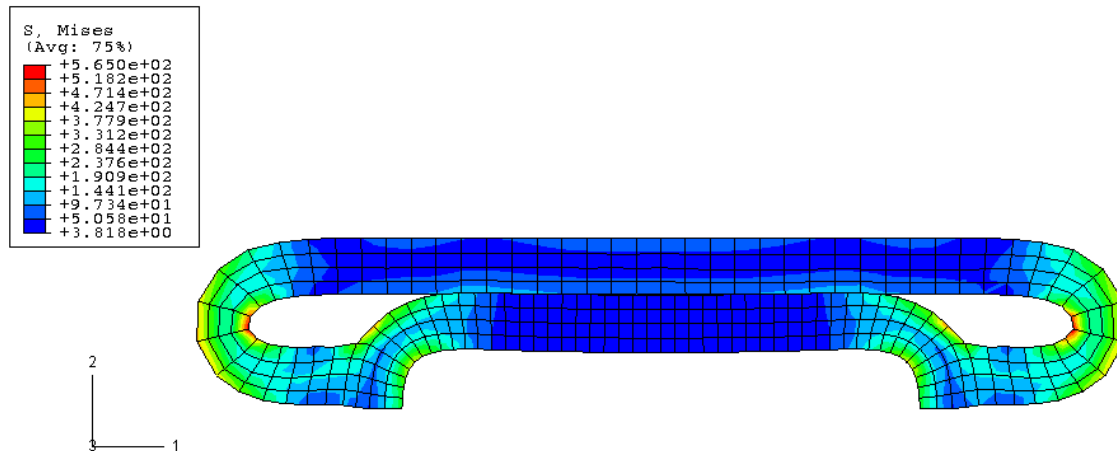



Figure W1–5. Mises contour plot

7. Contour the minimum and maximum principal nominal strains. Notice that strains can be very high for hyperelastic materials. In comparison, the linear elastic material model is not appropriate for elastic strains greater than 5%.
8. Contour the contact pressures. Note that the mesh obscures the contours in the region of self-contact. Extrude the mesh and use display groups as indicated below to make the mesh translucent:
 - a. From the main menu bar, select **View**→**ODB Display Options**. In the dialog box that appears, switch to the **Sweep/Extrude** tabbed page and toggle on **Extrude elements**. Click **OK**.
 - b. Use the **Common Plot Options** dialog box to set the deformation scale factor to **0.96** (this will further aid visualization).
 - c. In the toolbar, click the **Create Display Group** tool .
 - d. In the **Create Display Group** dialog box, click **Save As** at the bottom of the dialog box. Name the display group **mesh**.
 - e. In the **Create Display Group** dialog box, select **Surfaces** as the item.
 - f. Select **SEALINNER** and **SEALOUTER**. Click **Replace** and then **Save As** at the bottom of the dialog box. Name the display group **surfaces**.
 - g. Dismiss the dialog box.
 - h. In the toolbar, click the **Display Group Manager** tool .

- i. In the display group manager, select **mesh** and click **Plot**.
- j. Change the plot state to display the deformed shape of the seal.
- k. Open the **Common Plot Options** dialog box and do the following:
 - In the **Basic** tabbed page of the dialog box, select the **Filled** render style, and make only **Feature edges** visible.
 - Change the fill color to light grey (under the **Color & Style** tabbed page of the dialog box).
 - Activate the **Translucency** option (under the **Other** tabbed page of the dialog box); set the translucency level to **0.15** and activate **Use depth sorting**.
- l. In the display group manager, click **Lock** (next to **mesh**) in the **Display Group Instance** field to freeze the mesh display.
- m. In the display group manager, select **surfaces** and click **Add**. Lock the surface display. The plot appears as shown in Figure W1–6.

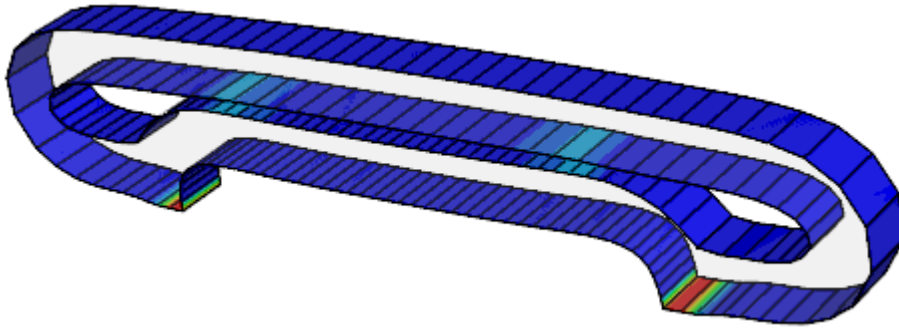



Figure W1–6. CPRESS contour plot

9. Animate the contour plot (click  in the toolbox).
10. Display the reaction force history at the reference node of the top rigid surface. In the Results Tree, expand the **History Output** container underneath the output database file named **sea1.odb** and double-click **Reaction force: RF2 PI: TOP-1 Node 1 in NSET TOPRP**.
11. You will now create the CLD curve.
 - a. In the list of the **History Output** branch in the Results Tree, click mouse button 3 on **Reaction force: RF2 PI: TOP-1 Node 1 in NSET TOPRP** and select **Save As** from the menu that appears. Save the data as **Force**.
 - b. Click mouse button 3 on **Spatial displacement: U2 PI: TOP-1 Node 1 in NSET TOPRP** and select **Save As** from the menu that appears. Save the data as **Disp**.
 - c. In the Results Tree, double-click **XYData**. In the **Create XY Data** dialog box that appears, select the **Operate on XY Data** source and click **Continue**.

The **Operate on XY Data** dialog box appears.

- d. From the **Operators** listed in the **Operate on XY Data** dialog box, select **combine(X, X)** and then **abs()**. Note that the **abs()** operator is used to obtain the absolute values. In the **XY Data** field, double-click the curve **Disp**. The current expression reads **combine(abs("Disp"))**. Move the cursor before the far-right bracket, enter a comma, and then select the operator **abs()**. In the **XY Data** field, double-click the curve **Force**. The final expression reads **combine(abs("Disp"), abs("Force"))**. Click **Plot Expression** to plot this expression. Save this plot as **CLD**.

The final plot appears as shown in Figure W1-7. (Note this plot has been customized.)

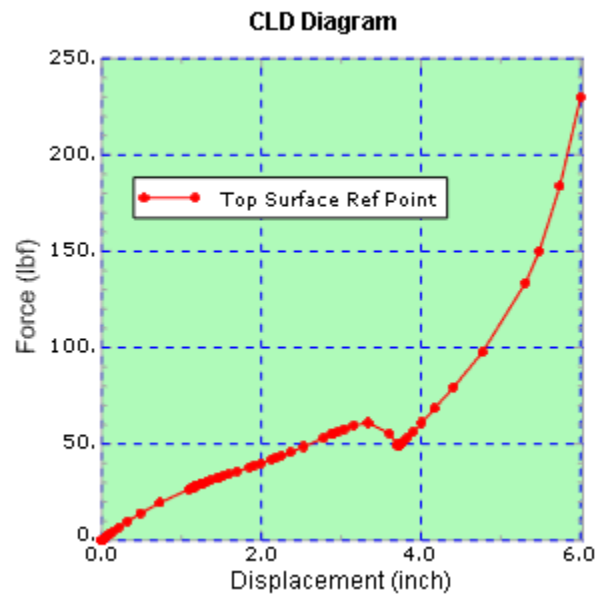


Figure W1-7. Compression load deflection diagram

Question W1-2: What does the inverted peak near 4 inches of deflection represent?

Note: A script that creates the complete model described in these instructions is available for your convenience. Run this script if you encounter difficulties following the instructions or if you wish to check your work. The script is named `ws_contact_seal_answer.py` and is available using the Abaqus fetch utility.