

Single Pass Rolling of a Thick Plate

Introduction

Rolling is a basic manufacturing technique used to transform preformed shapes into a form suitable for further processing. The rolling process is essentially quasi-static because rolling is normally performed at relatively low speeds (typically 1 m/s) and, therefore, inertia effects are not significant.

Representative rolling geometries generally require three-dimensional modeling, resulting in very large models, and include both nonlinear material behavior and nonlinear boundary effects (contact and friction). For this reason the explicit dynamics approach is often less expensive computationally and more reliable than an implicit quasi-static solution if mass scaling is used.

This workshop involves simulating quasi-static rolling of a plate using ABAQUS/Explicit. The rolling process reduces the thickness of the plate by 42% in a single rolling pass. Mass scaling will be used to reduce the solution time for analysis. For the purpose of comparison, the simulation will also be performed using the implicit solver available with ABAQUS/Standard.

This workshop is based in part on “Rolling of thick plates,” Section 1.3.6 of the ABAQUS Example Problems Manual.

Geometry and model

A half-symmetry plane strain model will be used with the plate having a length of 0.092 m and a half-thickness of 0.02 m. A 90° segment of the roller is constructed as an analytical rigid surface.

1. Start a new session of ABAQUS/CAE from the `../IntroClass/workshops/rolling` directory and create a new database.

A default model name is assigned by ABAQUS/CAE (**Model-1**). However, since the explicit dynamics model will later form the basis of the implicit analysis model, you should rename the current model to give it a more descriptive name.

2. In the Model Tree, click mouse button 3 on **Model-1** and select **Rename** from the menu that appears. Rename the model **Explicit**.
3. In the Model Tree, double-click **Parts** to create a new part. Name the part **plate**, choose **2D Planar** as the modeling space, and set the approximate size of the sketch area to **0.5**. Review the other defaults, and click **Continue**.

4. Using the **Create Lines: Rectangle** tool, sketch a rectangle 0.092 m long and 0.020 m high, as shown in Figure W15–1.

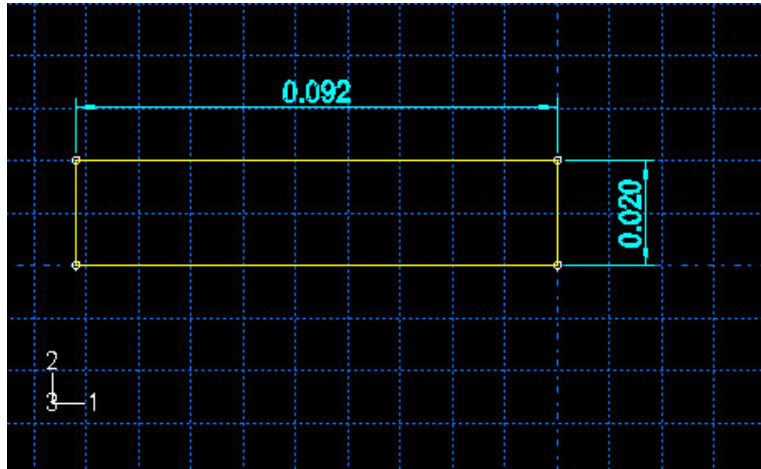


Figure W15–1 Plate geometry.

5. In the Model Tree, double-click **Parts** to create another part. Name the part **roller**. Choose the **2D Planar** modeling space and the **Analytical rigid** part type. Set the approximate size of the sketch area to 2.0, and click **Continue**.
6. Using the **Create Isolated Point** tool, create two points with the coordinates (-0.170, 0.170) and (0, 0.170).
7. Using the **Create Arc** tool, sketch a 90° arc centered at the point (0, 0.170), as shown in Figure W15–2.

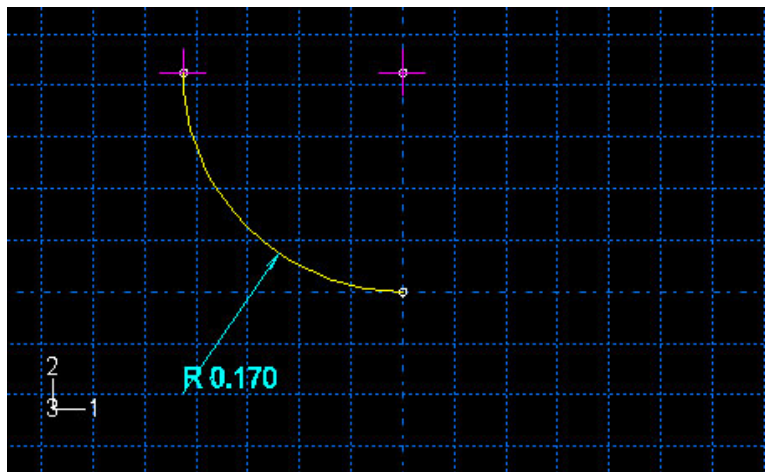


Figure W15–2 Roller geometry.

8. From the main menu bar, select **Tools→Reference Point** and choose the center of the arc to create a reference point for the part.

Materials and section

The steel plate is assumed to be elastic-plastic. Although rate-dependent plastic behavior is usually considered in rolling simulations, it is not considered here.

- In the Model Tree, double-click **Materials**. Create a material named **Steel** with the following properties:
 - Modulus of elasticity: **1.5E11 Pa**
 - Poisson's ratio: **0.3**
 - Density: **7850 kg/m³**
 - The initial yield stress is 168.72 MPa rising to 448.45 MPa at 100% strain. Use the plasticity data provided in Table W15–1:

Yield stress (Pa)	Plastic strain
1.6872 E8	0.0
2.1933 E8	0.1
2.7202 E8	0.2
3.0853 E8	0.3
3.3737 E8	0.4
3.6158 E8	0.5
3.8265 E8	0.6
4.0142 E8	0.7
4.1842 E8	0.8
4.3401 E8	0.9
4.4845 E8	1.0

Table W15–1 Yield stress-plastic strain data.

- In the Model Tree, double-click **Sections** and create a homogeneous solid section named **plateSection** with the default thickness of **1.0**.
- In the Model Tree, expand the **plate** item underneath the **Parts** container and double-click **Section Assignments** in the list that appears. Assign **plateSection** to the part.

Assembly

You must now instance the parts and position the plate relative to the roller.

- In the Model Tree, expand the **Assembly** container and double-click **Instances**.
- In the **Create Instance** dialog box, select both parts and click **OK**.

To position the plate relative to the roller, it is necessary to identify the location on the roller where contact will first occur between the parts. Then datum geometry can be used to facilitate the positioning. In this problem the top-right corner of the plate will first make contact with the roller at a point that is located approximately 20% of the distance along the edge of the roller (relative to the bottom vertex of the roller).

3. From the main menu bar, select **Tools→Datum** to define a datum point using the **Enter parameter** method. Create the point on the roller edge, and specify a normalized edge parameter of **0.2**.
4. From the main menu bar, select **Instance→Translate** to modify the position of the plate. Define the translation vector using the start and end points shown in Figure W15–3.

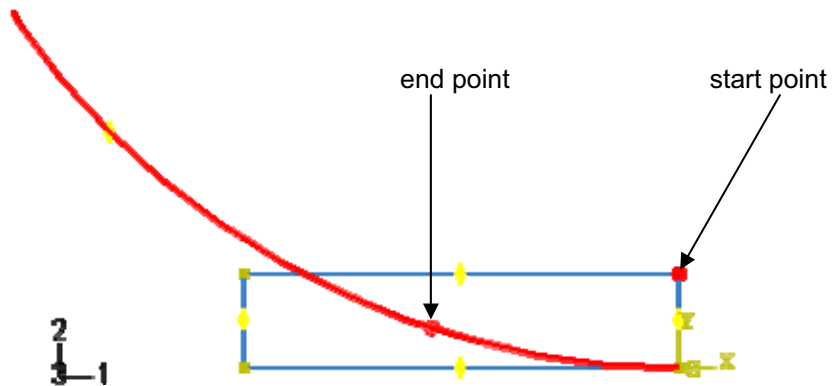


Figure W15–3 Part positioning.

Sets and surfaces

To facilitate the application of loads, contact, etc., you will create geometry sets and surfaces.

1. In the Model Tree, double-click the **Sets** item underneath the **Assembly** container. Create the following sets corresponding to the regions shown in Figure W15–4:
 - left:** The left edge of the plate
 - bottom:** The bottom edge of the plate
 - plate:** The entire plate
 - refPt:** The reference point of the roller

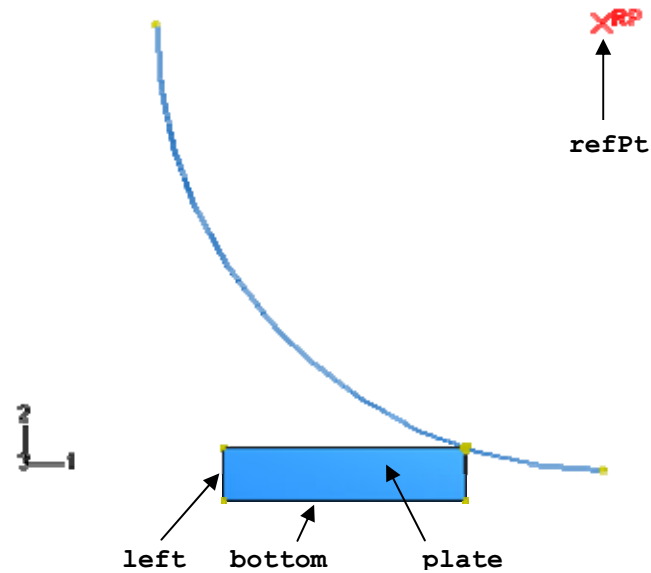


Figure W15-4 Geometry sets.

2. In the Model Tree, double-click the **Surfaces** item underneath the **Assembly** container. Create the following surfaces corresponding to the regions shown in Figure W15-5:

roller: Side of the roller facing the plate

topPlate: Top and right edges of the plate

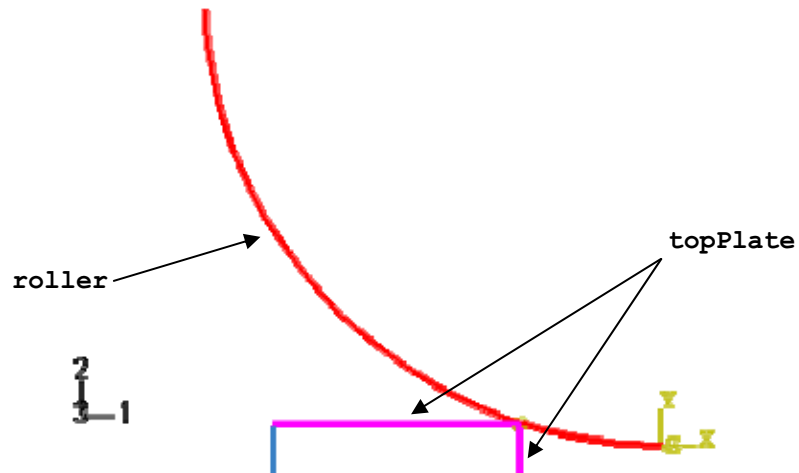


Figure W15-5 Surfaces.

Analysis step and output requests

The rolling simulation will be performed using a single explicit dynamics analysis step.

1. In the Model Tree, double-click **Steps** to create a **Dynamic, Explicit** step named **Single Pass Rolling** with a time period of **0.1** seconds. Accept all defaults for the time incrementation and other parameters.

Because rate effects are usually considered in rolling problems, the only method available to speed up the simulation effectively is mass scaling.

2. Open the **Mass Scaling** tabbed page of the **Edit Step** dialog box. In this page, select **Use scaling definitions below** then click **Create**.
3. In the **Edit mass scaling** dialog box, choose **Set** as the application region; and from the pull down menu, select **plate**. Enter a **Scale by factor** value of **1600**. Click **OK** in the **Edit mass scaling** and **Edit Step** dialog boxes.
4. Accept the default field output requests. In the Model Tree, double-click **History Output Requests** to create a history output request for the reaction moment at the roller reference point. In the **Edit History Output Request** dialog box:

Select **Set** in the **Domain** field and select **refPt** from the list of sets that appears.

Request stored history output at **100** equally spaced time intervals during the analysis.

From the list of available output variables, click the arrows next to **Forces/Reactions** and **RF, Reaction forces and moments** to expand the variable lists. Toggle on **RM3**.

Click **OK**.

Very large deformations are anticipated in this analysis. Use the adaptive meshing capability available in ABAQUS/Explicit to reduce the amount of mesh distortion and maintain a high quality mesh throughout the analysis.

5. From the main menu bar, select **Other→Adaptive Mesh Domain→Manager**.
 - a. In the **Adaptive Mesh Domain Manager**, click **Edit**.
 - b. In the **Edit Adaptive Mesh Domain** dialog box, toggle on **Use the adaptive mesh domain below**. Select **plate** as the region. Specify a frequency of **10** and **3** mesh sweeps per increment.
 - c. Toggle on **Adaptive mesh controls**, and click **Create** in the right side of the dialog box. Define adaptive mesh controls using all supplied default parameters.

Contact

Surface-to-surface contact will be defined between the top surface of the **plate** and the outer surface of the roller. Coulomb friction with a coefficient of 0.4 will be used.

1. In the Model Tree, double-click **Interaction Properties** to create a **Contact** property with the default name **IntProp-1**.
2. In the **Edit Contact Property** dialog box, select **Mechanical→Tangential Behavior**, and choose the **Penalty** friction formulation.
3. Enter a friction coefficient of **0.4**. Click **OK**.
4. In the Model Tree, double-click **Interactions**.
5. In the **Create Interaction** dialog box, select **Single Pass Rolling** as the step and select **Surface-to-surface contact (Explicit)** as the interaction type. Click **Continue**.
6. When prompted for the surfaces involved in the contact interaction, click **Surfaces** in the prompt area to open the **Region Selection** dialog box. Select **roller** as the first contact surface and **topPlate** as the second surface.
7. In the **Edit Interaction** dialog box, review the default interaction settings and click **OK**.

Boundary conditions

The roller spins about the global Z-axis with an angular velocity of 6.2832 radians/sec. The plate will be given an initial velocity of 1.01587m/sec in the global X-direction, which is equal to the X-direction velocity component of the roller at the point where contact first occurs. This minimizes the initial transient disturbance since the initial acceleration in the X-direction at the contact point is zero.

1. In the Model Tree, click mouse button 3 on the **BCs** container and select **Manager** from the menu that appears.
2. Create the boundary conditions listed in Table W15–2:
 - a. In the **Boundary Condition Manager** dialog box, click **Create**.
 - b. In the **Create Boundary Condition** dialog box, name the boundary condition and choose the appropriate step. For the displacement boundary conditions select **Displacement/Rotation** as the type. For the angular velocity boundary condition select **Velocity/Angular velocity** as the type. Click **Continue**.
 - c. Select the boundary condition region.
 - d. Define the boundary condition constraint in the **Edit Boundary Condition** dialog box.

BC name	Step	Region	Boundary condition
bottom	Initial	bottom	U2=0
fixRoll	Initial	refPt	U1=U2=0
rotateRoll	Single pass rolling	refPt	VR3=6.2832

Table W15–2 Boundary conditions.

Initial conditions

As mentioned earlier, the plate is given an initial velocity of 1.01587 m/sec in the global *X*-direction.

1. In the Model Tree, double-click **Fields**.
2. In the **Create Field** dialog box, select the **Initial** step, the **Mechanical** category, and the **Velocity** type. Name the field **initvel** and click **Continue**.
3. Select the set **plate** as the region for the initial velocity field.
4. In the **Edit Field** box, enter a value of **1.01587** m/sec for the velocity component **V1**.

Mesh

The plate will be meshed with 150 linear, plane strain, reduced integration elements (CPE4R).

1. In the Model Tree, expand the **plate** item underneath the **Parts** container and double-click **Mesh** in the list that appears.
2. From the main menu bar, select **Mesh→Element Type**. Examine the various options available in the **Element type** dialog box, change the element library to **Explicit** and the element family to **Plane Strain**, and accept the element type **CPE4R**.
3. From the main menu bar, select **Seed→Part** and specify a seed size of **0.002**.
4. From the main menu bar, select **Mesh→Part** and click **Yes** in the prompt area to mesh the part. The mesh is shown in Figure W15–6.

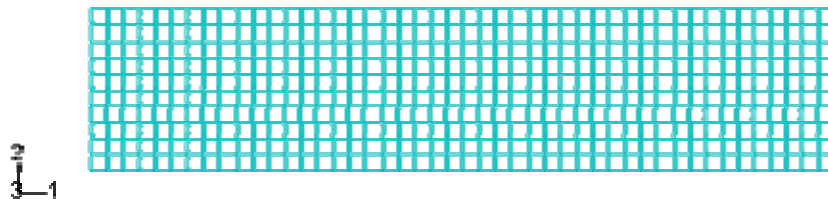


Figure W15–6 Plate mesh.

Analysis

1. In the Model Tree, double-click **Jobs** to create a job named **roll-xpl**. Accept the default job parameters.
2. From the main menu bar, select **File→Save As** and save the model database as **roll.cae**.
3. In the Model Tree, click mouse button 3 on the job **roll-xpl** and select **Submit** from the menu that appears to submit your job for analysis.
4. Monitor the progress of the job by clicking mouse button 3 on the job **roll-xpl** and selecting **Monitor** from the menu that appears.

Visualization

1. Once the analysis completes successfully, click mouse button 3 on the job **roll-xpl** in the Model Tree and select **Results** from the menu that appears.
2. Plot the undeformed and the deformed model shapes.
3. From the main menu bar, select **View→ODB Display Options**.
4. In the **ODB Display Options** dialog box, choose the **Extra Fine** curve refinement level. Click the **Sweep and Extrude** tab and set the extrusion depth to **0.01**. Click **OK**.
5. Turn on shaded render style, and rotate the model.
6. Plot the contours for **MISES** stress and plastic strain **PEEQ**. The contours for Mises stress are shown in Figure W15–7.

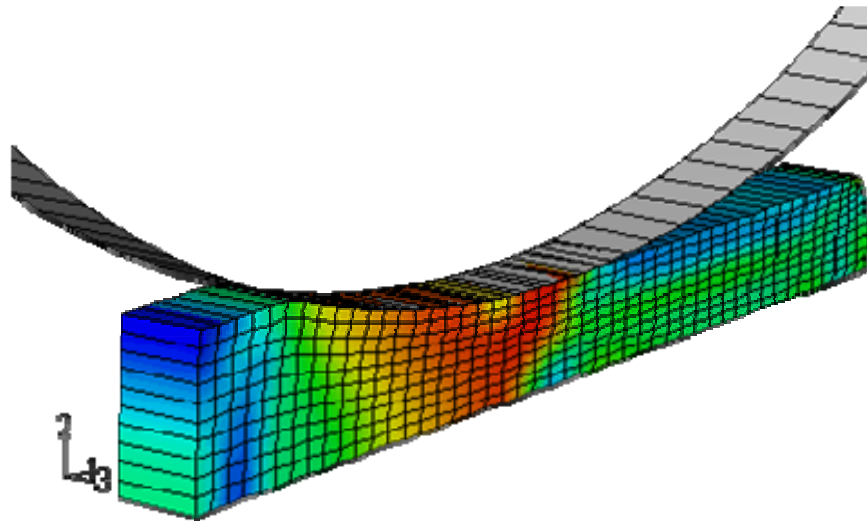


Figure W15–7 Mises stress contours.

7. From the main menu bar, select **Animate→Time History** to animate the history of the Mises stress distribution and the resulting deformation.
8. From the main menu bar, select **Result→History output** to create *X–Y* plots for the model kinetic energy (ALLKE), external work (ALLWK), internal energy (ALLIE), and total energy (ETOTAL). In the **History Output** dialog box, click **Plot** to display the curves and click **Dismiss** to close the box. The plot is shown in Figure W15–8.

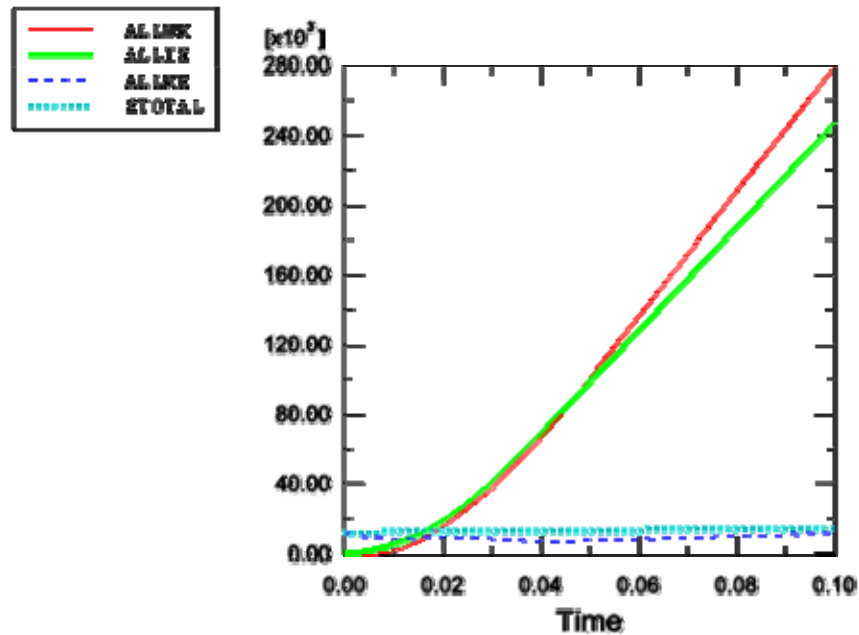


Figure W15–8 Model energies.

9. Plot the history of the reaction moment **RM3**, and save the curve as **RM3 XPL**.

Implicit analysis model

Now perform the simulation using ABAQUS/Standard. You will copy the model for the ABAQUS/Explicit analysis and modify the new model as described below.

1. In the Model Tree, click mouse button 3 on the model **Explicit** and select **Copy Model** from the menu that appears. In the **Copy Model** dialog box, name the new model **standard**. Click **OK**.
2. Replace the **Single Pass Rolling** step with a **Static, General** step.

For the implicit analysis, the static analysis procedure will be used. Note, however, that the simple analysis technique used earlier (whereby the initial plate velocity was prescribed and a single analysis step was used) will not yield the desired results in ABAQUS/Standard. The reason is that, in a static step material inertia is neglected. Thus, two steps are needed to perform the simulation. In the first step firm contact must be established between the plate and the roller. This then allows the roller to draw the plate in the second step, where the roll pass event will be simulated.

The adaptive meshing capability in ABAQUS/Standard is not intended to be used for general large-deformation problems, such as bulk metal forming. Deactivate adaptive meshing for this step as follows:

3. From the main menu bar, select **Other→Adaptive mesh domain→Edit→Establish Contact** to edit the adaptive mesh domain for this step.
4. In the **Edit Adaptive Meshing Domain** dialog box, choose **No adaptive mesh domain for this step** and click **OK**.
5. Rename the step **Establish Contact** with the following parameters:
 - Total time period = 0.001 sec
 - Maximum number of increments = 100
 - Initial time increment size = 0.0001 sec
 - Minimum increment size = 1E-8 sec
 - Maximum increment size = 0.001 sec
 - **NLGEOM** toggled on
6. Create a second **Static, General** named **Roll Pass** with the following parameters:
 - Total time period = 0.099 sec
 - Maximum number of increments = 500
 - Initial time increment size = 0.0001 sec
 - Minimum increment size = 9.9E-7 sec
 - Maximum increment size = 0.099 sec
7. From the main menu bar, select **Output→Restart Requests** and specify a restart frequency of 250 for each step.
8. Edit the field output requests so that preselected output is written every 25 increments during each analysis step.

 For problems dominated by highly discontinuous nonlinearities (such as contact and friction), nondefault analysis controls may aid the convergence behavior. In this problem allow extra equilibrium iterations for the resolution of these discontinuities, as described below.
9. From the main menu bar, select **Other→General Solution Controls→Edit→Roll Pass** to edit the solution controls for this step.
10. In the solution controls editor, select **Specify**. In the **Time Incrementation** tabbed page of this dialog box, toggle on **Discontinuous analysis**. Click **OK** to close the dialog box.
11. In the Model Tree, double-click **BCs** to create the **Velocity/Angular velocity** type boundary condition listed in Table W15–3.

BC name	Step	Region	Boundary condition
feed piece	Establish Contact	left	V1=1.01587

Table W15–3 New boundary condition.

12. In the Model Tree, click mouse button 3 on the **BCs** container and select **Manager** from the menu that appears.
13. In the **Boundary Condition Manager** dialog box, select cell for the **feed piece** boundary condition in the **Roll Pass** step, and click **Deactivate**.
14. In the Model Tree, expand the **plate** item underneath the **Parts** container and double-click **Mesh** in the list that appears.
15. From the main menu bar, select **Mesh→Element Type**; and, in the **Element type** dialog box, change the element library to **Standard** and click **OK**.
16. Save the model database.
17. Create a job named **roll-std** for the model **Standard**.
18. Submit the job for analysis.
19. Monitor the progress of the job by clicking mouse button 3 on the **roll-std** item underneath the **Jobs** container and selecting **Monitor** in the menu that appears.

Visualization

1. Once the analysis completes successfully, click mouse button 3 on the job **roll-std** in the Model Tree and select **Results** in the menu that appears.
2. Plot the undeformed and the deformed model shapes.
3. Plot the contours for **MISES** stress and plastic strain **PEEQ**.
4. Create a second viewport by selecting **Viewport→Create** from the main menu bar. Tile the viewports horizontally (**Viewport→Tile Horizontally**).
5. Switch to the output database file **roll-xp1.odb**, and plot the contours for **PEEQ**. The contours for **PEEQ** for both analyses are shown in Figure W15–9.

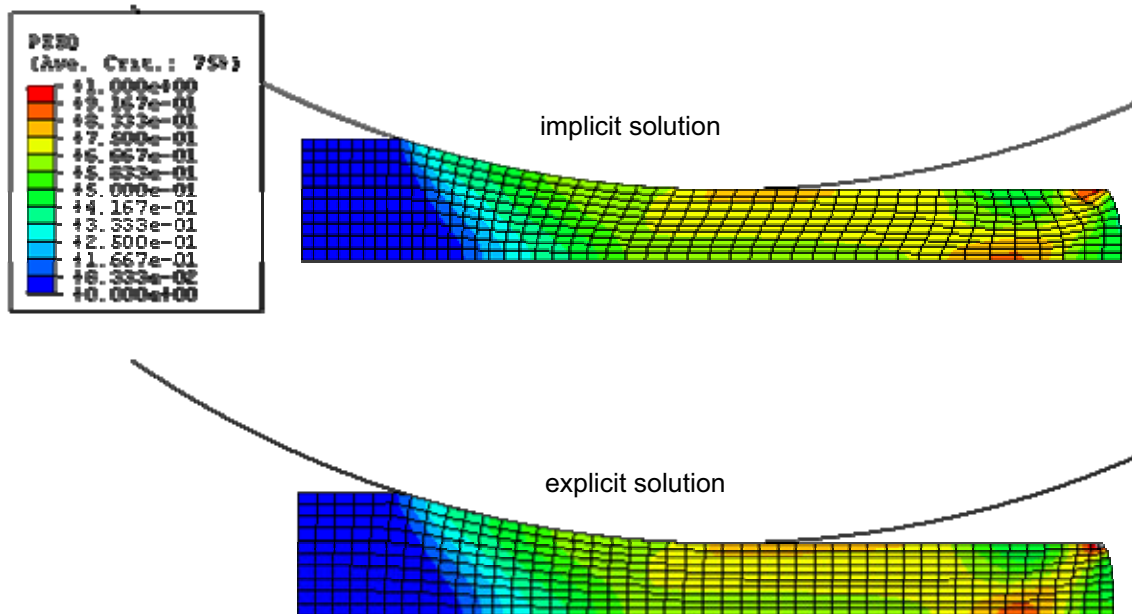


Figure W15-9 PEEQ for the implicit and explicit solutions.

- Save the time history for the reaction moment **RM3** as **RM3 STD**. Plot both curves as shown in Figure W15-10.

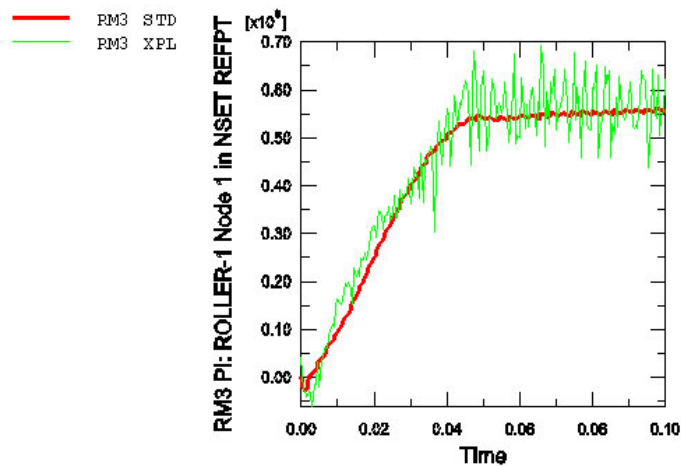


Figure W15-10 Comparison of the reaction moment history.

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 outlined here or if you wish to check your work. The script is named `ws_intro_rolling.py` and is available using the ABAQUS fetch utility.