3S SIMULA

Two-Pass Rolling Simulation

Summary

Hot rolling is a basic metal forming technique that is used to transform preformed shapes into final products or forms that are suitable for further processing. The process typically involves passing heated stock pieces through multiple sets of forming rolls until the desired cross-sectional shape is achieved. The important aspects of this manufacturing operation are the elongation and spread of the material during the rolling process. Abaqus contains all the features necessary to model this process and has been used extensively within the metal processing industry to optimize roll pass designs.

Background

A two-pass rolling simulation, which is part of the multi-pass operation required to form a rail, is discussed. The process starts with a heated preformed stock that is rolled between a pair of shaped rolls. During the rolling process the stock is cooled by spray-applied cooling fluid. After the first pass the stock cools before being passed through another pair of shaped rolls. The desired rail cross-section is produced by repeating this process with new roll shapes at each rolling stage of the multi-pass operation.

Finite Element Analysis Approach

The analysis steps used to model the two-pass rolling process are as follows:

1. The first roll pass in Abaqus/Explicit proceeds until the steady-state rolling solution is achieved.

2. A two-dimensional mesh of the deformed cross-section at the steady-state plane is extracted using Abaqus/CAE. The final temperatures on the steady-state plane are also extracted.

3. A two-dimensional transient thermal analysis in Abaqus/Standard is used to model the cooling during the 45 sec between roll passes.

4. A new model of the cross-section with a finer mesh is generated in Abaqus/CAE. In a second thermal analysis, the submodeling capability in Abaqus/Standard is used to transfer the temperatures from the cooling analysis of Step 3 to the finer mesh. The purpose of the analysis is only to transfer the temperatures to the new mesh.

5. The new cross-section mesh and temperature data from Step 4 are used to construct the three-dimensional mesh for the second roll pass in Abaqus/Explicit.
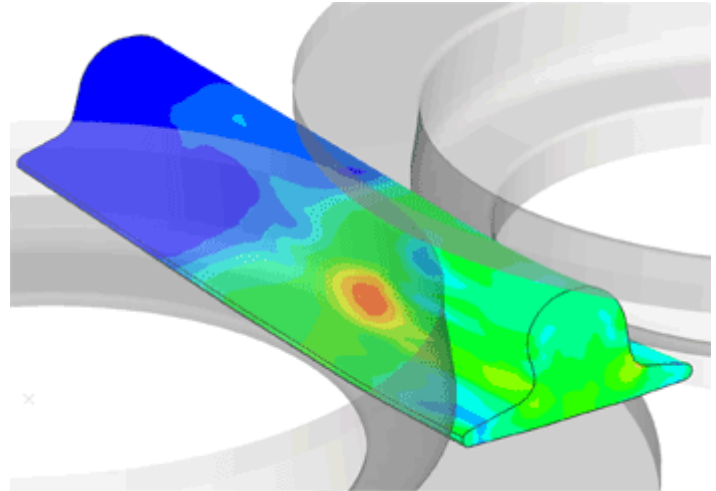
Key Abaqus Features and Benefits

- Optimal solution time achieved by combined use of Abaqus/Standard and Abaqus/Explicit

- Robust mechanical and thermal contact interactions

- Automatic mass scaling for bulk metal rolling

- Adaptive meshing

- Automatic solution termination when steady-state conditions detected

- Abaqus/CAE with Python scripting to build new models based on results of previous analyses

6. The second roll pass in Abaqus/Explicit proceeds until the steady-state rolling solution is achieved.

Roll Pass Analyses

For each roll pass analysis the rolls are modeled as isothermal, analytical rigid bodies, and heat transfer occurring between the rolls and the deformable rail is included. Heat can be generated in the rail due to both plastic work and frictional contact effects. The heat loss from the rail due to the fluid cooling is accounted for with convective heat transfer boundary conditions; radiation to the ambient environment is also included. Therefore, these analyses are performed using a fully coupled thermal-stress solution in Abaqus/Explicit.
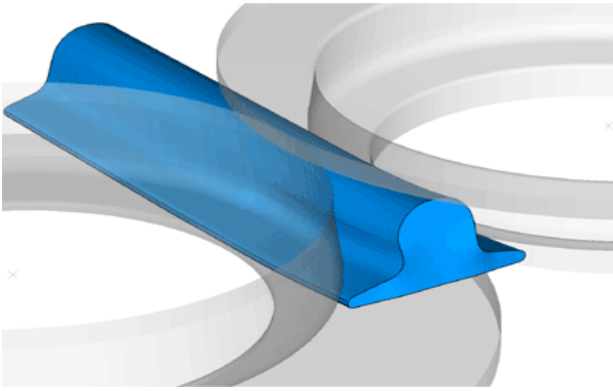
Figure 1: Deformed stock and rigid rolls.

Automatic adaptive meshing is used for the rail during the roll pass analyses in Abaqus/Explicit to maintain a high-quality mesh as the rail deforms. Mass scaling is applied to speed up the solution; Abaqus/Explicit has automatic mass scaling for bulk metal rolling that allows the optimum scaling to be selected based on several parameters of the rolling process. For maximum computational efficiency, the analysis is terminated automatically once the solution has achieved steady state; the criteria for determining whether steady state has been achieved is based on the force and torque on the rigid rolls and the spread and plastic strain in the cross-section. Once the steady-state criteria are satisfied for a plane of elements, a set containing those elements is generated automatically in the output database file.

Thermal Analyses

The mesh for the inter-pass cooling analysis was extracted from the output database of the first roll pass analysis using Abaqus/CAE. A script extracts the nodes on the leading face of the elements in the steady-state element set and projects them onto a plane to give the required mesh. The script also maps the temperatures from the three-dimensional first roll pass analysis to the two-dimensional inter-pass heat transfer mesh.

A new, finer cross-section mesh was used for the second roll pass analysis; to map the temperatures from the first heat transfer analysis, a second heat transfer analysis using the new mesh was required. The new mesh for this second heat transfer analysis was also generated in Abaqus/CAE using a script. This script takes the output database from the first heat transfer analysis as an input, reads in the mesh, and generates a geometric part from that mesh. This new part was meshed to give the required new mesh for the second heat transfer analysis. Because the nodal locations in the new two-dimensional mesh are different than those in the mesh used for the inter-pass heat transfer analysis, the temperatures are interpolated to the nodes of the new mesh. A heat transfer analysis in Abaqus/Standard using the submodeling technique is used for this purpose.

The final temperatures from the second heat transfer analysis are used as the initial temperatures for the second roll pass. The input file for the second roll pass was constructed by hand, based on the mesh from the second heat transfer analysis. Again, scripting methods can automate this process. The second roll pass is run in the same way as the first, and it terminates when a steady-state solution is achieved.
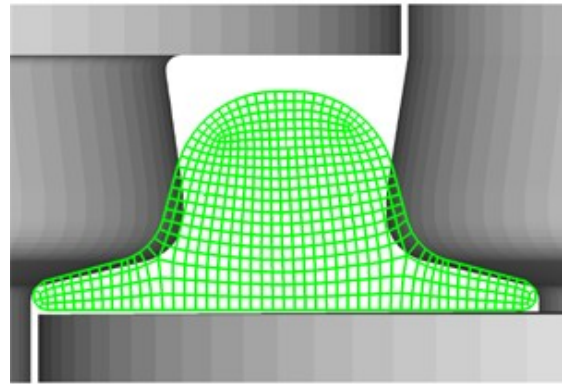

Figure 2: Undeformed cross-section mesh of the stock and rigid roll profiles for the first roll pass.
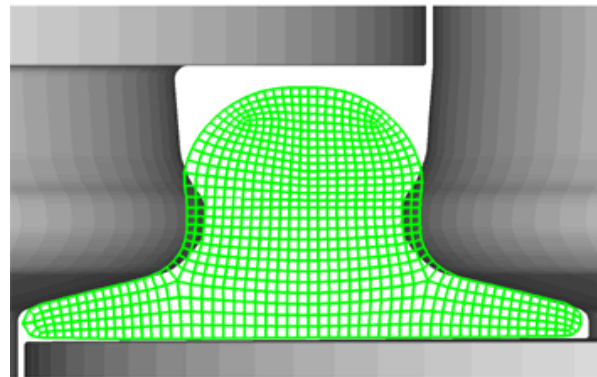

Figure 3: Cross-section mesh and rigid rolls for the second roll pass.

Analysis Results and Discussion

The first roll pass terminates once user-defined steady-state rolling conditions are met, reducing the time that would otherwise be required for the analysis. The maximum stresses on the surface of the rail are located under the contact points of the rolls, as shown in Figure 4.

As the analysis proceeds, the deformation approaches a steady-state condition. If the analysis continued beyond this point, the deformation of the rail cross-section would be reproduced as more of the rail passed between the rolls. As shown in Figure 5, the largest plastic strains occur under the rolls where the deformation is the largest.
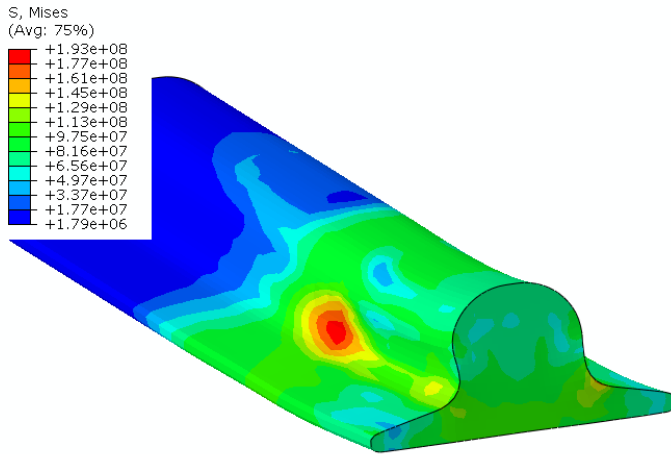
Figure 4: Mises stress in the stock at the end of the first roll pass analysis.
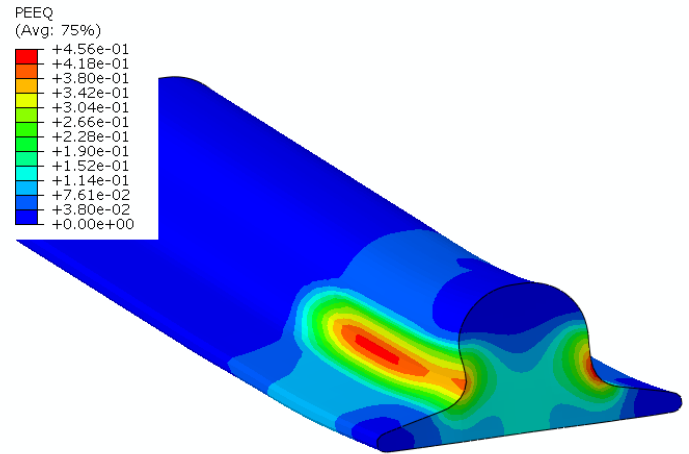

Figure 5: Equivalent plastic strain in the stock at the end of the first roll pass analysis.

Plastic deformation causes heat generation. The peak temperatures, shown in Figure 6, are very localized and correspond to the peaks in the plastic strains.

Between the roll passes the stock cools significantly and the gradient of temperature across the product reduces. Figure 7 shows the temperatures at the end of the heat transfer analysis modeling the cool down in Abaqus/Standard.

The second roll pass is terminated when steady-state conditions are reached; again, the entire mesh will not have passed between the rolls when the user-defined steady-state conditions are met.

The Mises stress, equivalent plastic strain, and temperatures at the end of the second roll pass are shown in Figures 8, 9, and 10. These plots show the same general characteristics as those from the first roll pass.

Further modification of the rail profile can be achieved by repeating the sequence of roll passes with new roll profiles. Given the repetitive nature of the roll forming process, the analytical simulation is ideally suited for automation. Abaqus, Inc., and Corus have developed a fully integrated rolling mill simulation system for long products. This system automates a number of the processes described here. For more information, see www.simulia.com/services/serv_ex_corus.html.

With the release of Version 6.4, Abaqus now includes a toolkit for the customization of the Abaqus/CAE GUI interface. The Abaqus GUI Toolkit is one of the Abaqus Process Automation tools that allow users to modify and extend the capabilities of Abaqus/CAE and enable a wide range of users to generate more efficient Abaqus solutions.
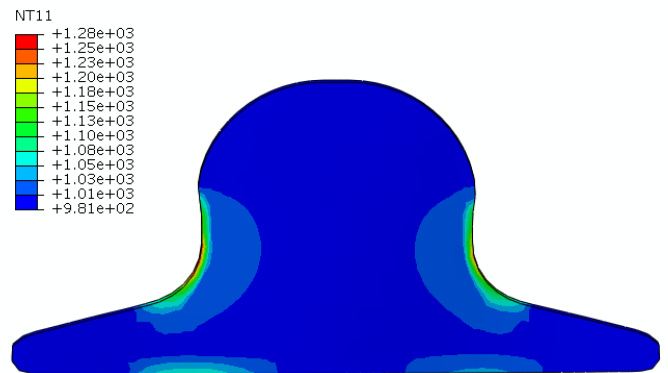

Figure 6: Temperatures on the steady-state plane at the end of the first roll pass analysis.
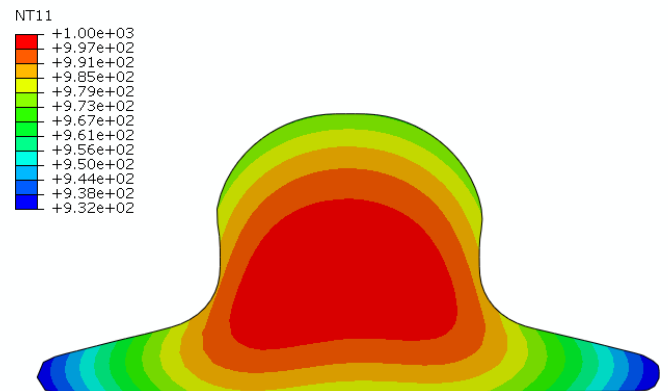

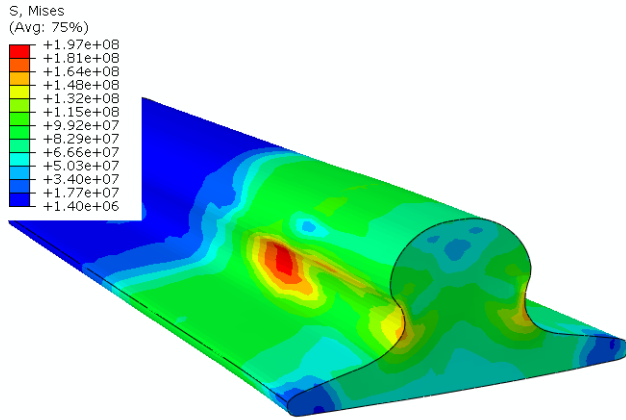Figure 7: Temperatures on the steady-state plane after the inter-pass heat transfer analysis.

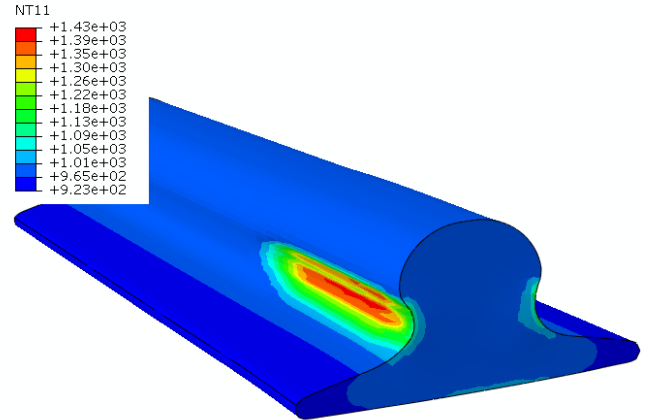Figure 8: Mises stress at the end of the second roll pass

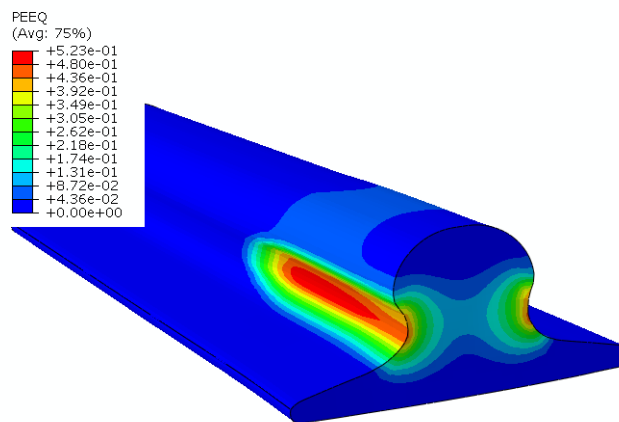Figure 10: Temperature in the stock at the end of the second roll pass analysis.

Figure 9: Equivalent plastic strain in the stock at the end of the second roll pass analysis

Abaqus References

For additional information on the capabilities and techniques outlined above, see the following references to the Abaqus 6.12 documentation:

- Analysis User's Manual
  - "Fully coupled thermal-stress analysis," Section 6.5.3
  - "Submodeling: overview," Section 10.2.1
  - "Mass scaling," Section 11.6.1
  - "ALE Adaptive meshing: overview," Section 12.2.1

- Example Problems Manual
  - "Rolling of thick plates," Section 1.3.6
  - "Flat rolling: transient and steady-state," Section 1.3.11
  - "Section rolling," Section 1.3.12

- Scripting User's Manual
  - "Creating a part," Section 3.1
  - "Reading from an output database," Section 3.2

Abaqus Input Files and Scripts

The following input files are available:

`ROLLPASS_1.inp`   Fully coupled first roll pass that runs until steady state is reached.

`INTERPASS_HT1.inp`   Transient heat transfer analysis using the same cross-section mesh as `ROLLPASS_1.inp`.

`INTERPASS_HT2.inp`   Transient heat transfer analysis using a new, finer cross-section mesh. This is a submodel analysis that transfers the results of the first heat transfer analysis and needs `INTERPASS_HT1.odb`.

`ROLLPASS_2.inp`   Fully coupled second roll pass that runs until steady state is reached.

The input files listed above constitute a complete two-pass rolling analysis. A set of Python scripts are also available to facilitate any modifications to these analyses. The scripts must be run in Abaqus/CAE, and they can be used in the process of generating the models and input files for the modified analyses. The following scripts are available:

`S1_BUILD_INTERPASS_HT1.py`   Python script that extracts the two-dimensional mesh from `ROLLPASS_1.odb` and then completes the definition of the first inter-pass heat transfer analysis.

`S2_BUILD_INTERPASS_HT2.py`   Python script that extracts the two-dimensional mesh from `INTERPASS_HT1.odb` and then generates an associated geometric part. This part can be used to define a new mesh and input file manually in Abaqus/CAE for the second heat transfer analysis.

`S3_BUILD_ROLLPASS_2.py`   Python script that extracts the two-dimensional mesh from `INTERPASS_HT2.odb`, extrudes it to give a three-dimensional mesh, and then completes the definition of the second roll pass analysis. If the second roll pass analysis is to be generated using the same mesh as in the first roll pass (i.e., bypassing the thermal submodeling step), this script can be edited to read in the mesh from `INTERPASS_HT1.odb`. Note that the surface RP2_Outer must be re-defined manually in Abaqus/CAE.

Visit the Resource Center for more SIMULIA customer papers