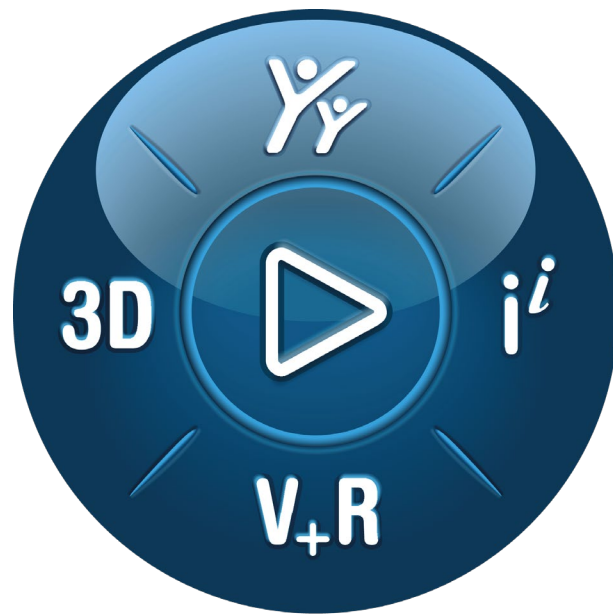


Access Advanced Abaqus Features Using 3DEXPERIENCE R2022x SIMULIA Apps



3DEXPERIENCE®

Version 2.0 - 6/15/2022

Written by: Yogesh UKE
Validated by: Asif KHAN
Edited by: Arati DESAI

3DEXPERIENCE Platform R2022x

Executive Summary

This Best Practices document demonstrates the use of advanced Abaqus features authored by a Mechanical Analyst using the External Solve feature of the Physics Simulation app. The Physics Simulation is exported as an Abaqus Input file and is updated interactively with additional Abaqus keywords using a simulation process authored in the simulation templates in the Process Composer app. The updated Abaqus Input file is executed using the Compute Orchestration Service. The results are imported back into the 3DEXPERIENCE Physics Results app for post-processing.

Refer to QA00000103244 for samples of final Abaqus Input file, Physics Simulation object, simulation process, and material property file.

Note: This Best Practice document is updated and valid for 3DEXPERIENCE R2022x GA level. From 3DEXPERIENCE R2022x FD01 level onwards, users can refer to the [Keyword Editor](#) feature.

Prerequisites and recommendations: The user referring to this document must be an expert in using the SIMULIA Mechanical Scenario Creation app and who has a knowledge of Abaqus Input files and the keywords used in it. The user should also know how to use the SIMULIA Process Composer app.

Target audience: Design Engineers, Mechanical Analysts, CFD Analysts, Methods Developers, and so on.

Table of contents

1. Introduction	4
2. Example of Physics Simulation	4
3. Simulation Process	5
3.1. Process Level Object Parameter	5
3.2. Simulation Activities.....	5
3.2.1. Manage Execution Paths	6
3.2.2. Parameter Mapping Using Dataflow 1.....	7
3.2.3. Generate Abaqus Input File Activity	7
3.2.4. Parameter Mapping Using Dataflow 2.....	10
3.2.5. Execute Input File and Import Results Activity	11
4. Simulation Template	12
4.1. Create Simulation Template	12
4.2. Maturity of the Simulation Template	14
4.2.1. Change the Maturity of Simulation Template from ‘Development’ State to ‘Released’ State	14
5. Use External Solve for Execution	15
6. Post-Processing Results	17
6.1. Access the Analysis Case	17
6.2. Create Field Plots.....	18
7. References	20
8. Document History	21

1. Introduction

This Best Practices document helps the user to leverage the advanced Abaqus features that are not available in the 3DEXPERIENCE Simulation applications. The Physics Simulation is exported to the Input file. The exported Input file is edited to include the required Abaqus keywords. The edited Input file is solved and the results are uploaded to the Physics Simulation Results section for visualizing the result.

The above methodology is automated using the SIMULIA Process Composer and made available within the Mechanical Scenario Creation app under the External Solve feature. The external solves are performed by associating the Physics Simulation with a simulation process and simulation template. The execution of the Input file is carried out using the Compute Orchestration Services on a station using the required solver. At the end, the results are uploaded back to the Physics Simulation Results section for post-processing.

2. Example of Physics Simulation

A Physics Simulation of a steel projectile impinging a steel plate is taken as an example here. This example uses an existing Physics Simulation where the Model section specifies the CATIA product representation of the steel plate and penetrator. The Scenario section specifies the impact velocity and the simulation step as shown below.

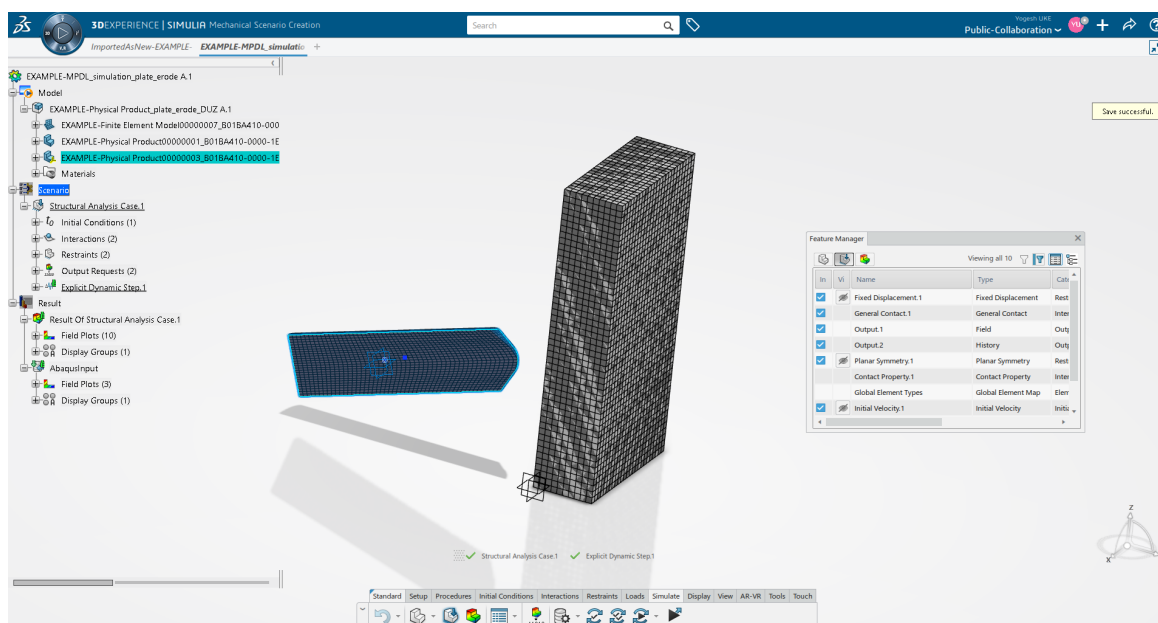


Figure 1. Simulation of a steel projectile impinging a steel plate

After the model and the scenario is ready, the simulation is submitted using the External Solve feature to:

- Export the Input file
- Perform keyword edits
- Execute the edited Input file
- Store simulation results

The External Solve calls a simulation template for instantiation that has a simulation process underneath.

3. Simulation Process

A simulation process with supporting simulation activities and steps is constructed to demonstrate the example in this document. New instances of the simulation process are created using the simulation template. Specific inputs and execution behavior is specified in the simulation template based on the execution options. The following sections describe the construction of the simulation process and simulation template configuration process.

3.1. Process Level Object Parameter

The simulation process contains an object parameter created at the Process level and is mapped to the activity that is responsible for generation of the Abaqus Input file. This object parameter is referenced to the Physics Simulation object and configured to use the “Export simulation to input file” batch program under the 3DX Utility adapter to write out the Abaqus Input file.

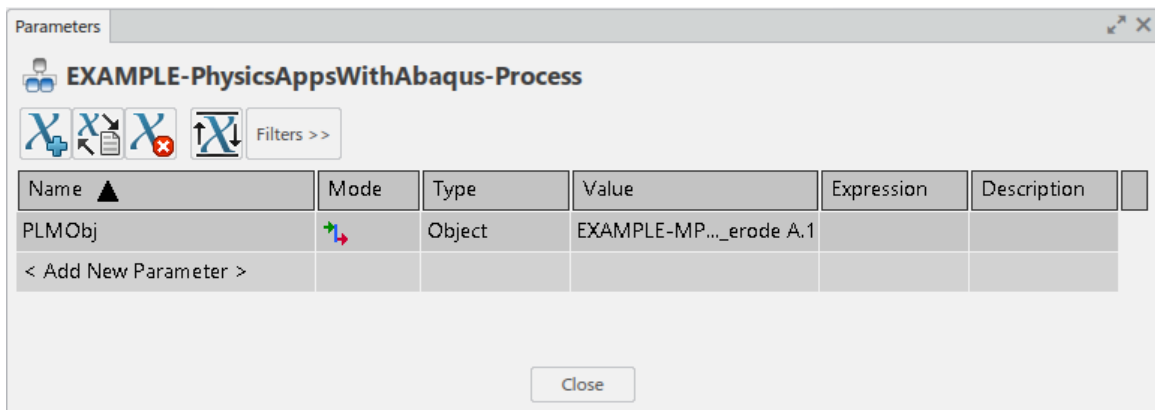


Figure 2. Process Level object parameter

3.2. Simulation Activities

The following Simulation Activities are created to achieve the scenarios described above and shown in Figure 3:

- Manage Execution Paths
- Generate Abaqus Input File
- Execute Input File and Import Results

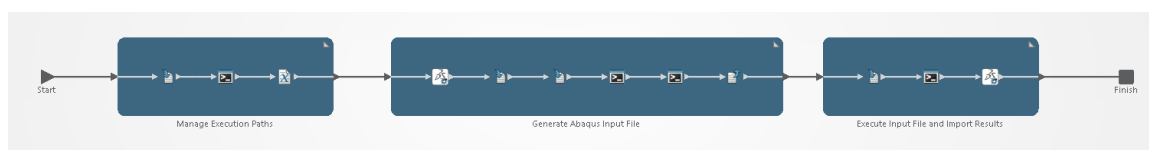


Figure 3. Simulation process overview

Within each activity, steps are added to accomplish the objective. The supporting content and parameters are defined as required to complete the Activity configuration.

3.2.1. Manage Execution Paths

The Manage Execution Paths activity helps the user to set the text editor executable path, the Abaqus launcher path, and the Input file name as shown below.

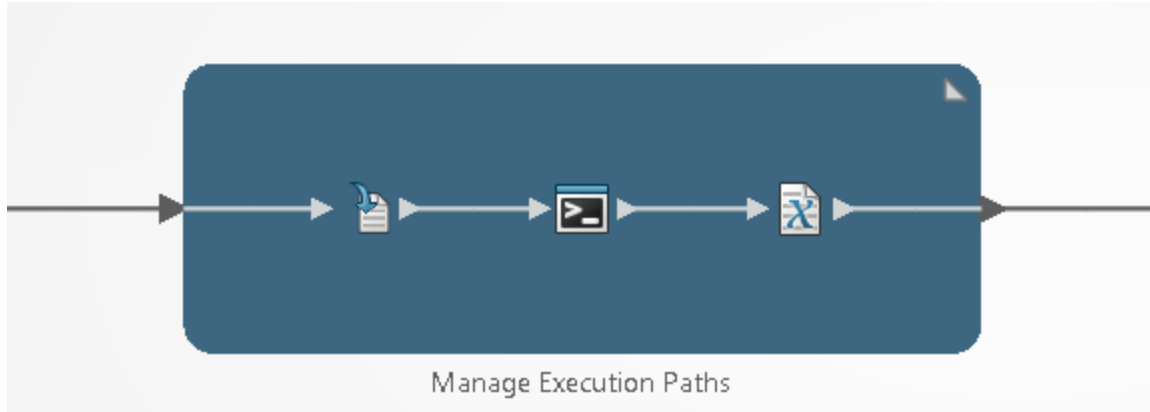


Figure 4. Manage Execution Paths activity

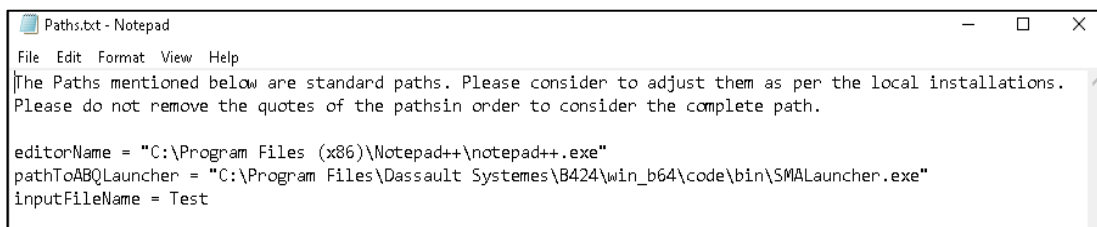
The Manage Execution Paths activity consists of the following three steps that capture the user provided local machine paths as parameter values.

1.  Download Execution Paths Step.

The Download Execution Paths step downloads the text file, Paths.txt to the execution directory containing the default executable paths of the text editor and Abaqus launcher, and the Input file name.

2.  Edit Execution Path Step.


This interactive Edit Execution Path step allows the user to manage the local installer text editor executable path, the Abaqus launcher path, and the Input file name. The Paths.txt file that was downloaded in the previous step will pop up automatically with the default notepad editor as shown in Figure 5. The user can set or change the default paths mentioned in this file with the local installation paths.



```
Paths.txt - Notepad
File Edit Format View Help
The Paths mentioned below are standard paths. Please consider to adjust them as per the local installations.
Please do not remove the quotes of the paths in order to consider the complete path.

editorName = "C:\Program Files (x86)\Notepad+\notepad+.exe"
pathToABQLauncher = "C:\Program Files\Dassault Systemes\B424\win_b64\code\bin\SMALauncher.exe"
inputFileName = Test
```

Figure 5. Paths.txt containing the path parameter values

3.  Set Execution Path Step.

The Set Execution Path step reads the execution paths set by the user as a parameter value of the parameters in the interactive step.

3.2.2. Parameter Mapping Using Dataflow 1

The parameter values set in the Manage Execution Paths activity are mapped to the Generate Abaqus Input File activity as shown below.

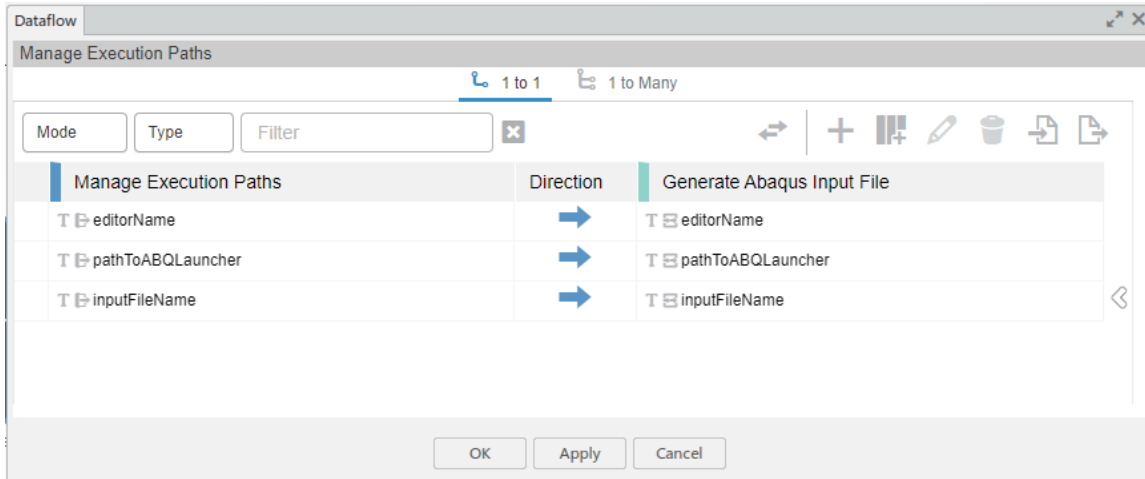


Figure 6. Parameter mapping using Dataflow 1

These parameter values are used in the Generate Abaqus Input File activity for naming the Abaqus Input file, to edit the Input file in the user selected text editor, and for the Abaqus launcher path.

3.2.3. Generate Abaqus Input File Activity

The Generate Abaqus Input File activity as shown in Figure 7 generates an Abaqus Input file. This interactive step allows the user to edit the Abaqus Input file and upload the updated Input file to the object parameter for execution.

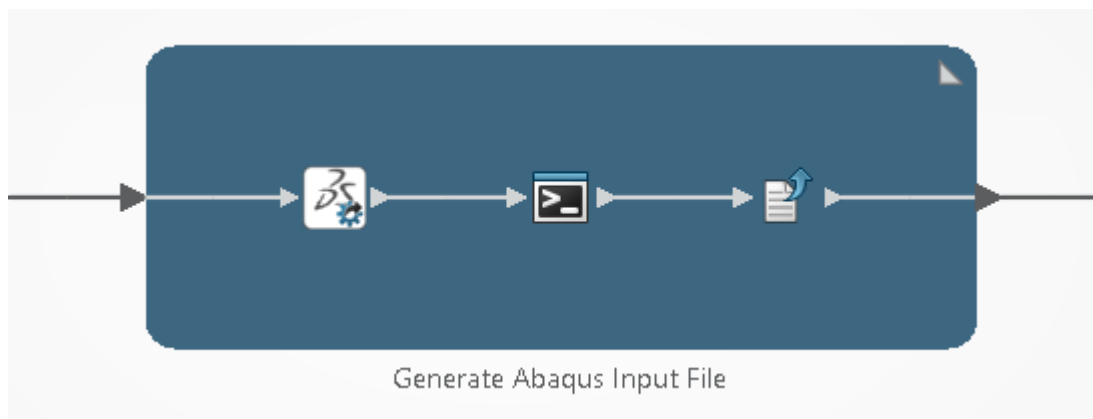



Figure 7. Generate Abaqus Input File activity

The Generate Abaqus Input File activity consists of three steps,

- Generate a Abaqus Input File
- Allow the user to edit the Input files and add the keywords
- Upload the updated Abaqus Input file to the object parameter.

The Generate Abaqus Input File activity steps are explained below:

1.  Write Input File Step.

This step generates an Abaqus Input file from the Physics Simulation that has been authored by the user. It uses the PLMObj parameter that refers to the Physics Simulation object and is configured to use the “Export simulation to Input file” batch program to write out the Input file using the name specified by the inputFileFileName parameter.

2.  Interactive Input File Modification Step.

This interactive step allows the user to edit the Abaqus Input file generated in the Write Input File step. It is configured to use the Command mode. The command line uses the parameters editorName and inputFileFileName to support editing of the Input file. For the example used in this document, make the following changes to the Abaqus Input file:

- a. Search for the text, “*PHYSICAL CONSTANTS, ABSOLUTE ZERO=0.” and add the following keywords just above the “*PHYSICAL CONSTANTS, ABSOLUTE ZERO=0.” line, (Note that the following text must be added as shown in Figure 8)

```
** KEYWORD EDIT NUMBER 1
```

```
*sectioncontrols, name=EC-1, hourglass=ENHANCED,  
kinematicsplit=ORTHOGONAL,distortion control,length ratio=0.1
```

```
1., 1., 1.
```

```
*RIGID BODY, REF NODE=27541, ISOTHERMAL=NO, POSITION=INPUT, ELSET="Rigid Body1-2-1"  
  
** KEYWORD EDIT NUMBER 1  
  
*sectioncontrols, name=EC-1, hourglass=ENHANCED, kinematicsplit=ORTHOGONAL,distortion control,length ratio=0.1  
  
1., 1., 1.  
  
*PHYSICAL CONSTANTS, ABSOLUTE ZERO=0.  
  
*MATERIAL, NAME="EXAMPLE--DUZSTEEL_820"
```

Figure 8. KEYWORD EDIT NUMBER 1 in the Abaqus Input file

- b. Search for the text, `"*MATERIAL, NAME="EXAMPLE--DUZSTEEL_820" "`, and add complete content from the `erode_material.inp` under this line as follows:

```
*PHYSICAL CONSTANTS, ABSOLUTE ZERO=0.  
  
*MATERIAL, NAME="EXAMPLE--DUZSTEEL_820"  
  
** KEYWORD EDIT NUMBER 2  
  
*Density  
  
7800  
  
*Elastic  
  
2.10E+11, 0.3
```

Figure 9. KEYWORD EDIT NUMBER 2

- c. Search for the text, `"Initial Velocity1", 3, 575.109976383422"` and add the following keywords under this line.

```
** KEYWORD EDIT NUMBER 3
```

```
*SURFACE, TYPE=ELEMENT, NAME=SURF1
```

```
,
```

```
"Solid Section1-1",interior
```

```
*surface,type=element,name=surf2
```

```
,
```

```
"Solid Section2-1",interior
```

- d. Search for the text, `"*STEP, NAME="Explicit Dynamic Step1", NLGEOM=YES"` card. Three lines after this text, add the following keywords as shown in Figure 10.

```
** KEYWORD EDIT NUMBER 4
```

```
*BULK VISCOSITY
```

```
0.06, 1.2
```

```
*STEP, NAME="Explicit Dynamic Step1", NLGEOM=YES
*DYNAMIC, EXPLICIT, SCALE FACTOR=1.
, 8e-06
** KEYWORD EDIT NUMBER 4
*BULK VISCOSITY
0.06, 1.2
*CONTACT, OP=NEW
*CONTACT INCLUSIONS, ALL EXTERIOR
```

Figure 10. KEYWORD EDIT NUMBER 4


- e. Search for the text, "*CONTACT INCLUSIONS, ALL EXTERIOR" and add the following keyword below this text. Refer to Figure 11.

** KEYWORD EDIT NUMBER 5

*contact controls assignment, nodal erosion=no

```
*CONTACT, OP=NEW
*CONTACT INCLUSIONS, ALL EXTERIOR
** KEYWORD EDIT NUMBER 5
*contact controls assignment, nodal erosion=no
```

Figure 11. KEYWORD EDIT NUMBER 5

3.  Store Input File Step.

This step uploads the modified Abaqus Input file to the object parameter inputFileObj with the help of a document.

3.2.4. Parameter Mapping Using Dataflow 2

The parameter values set in the Generate Abaqus Input File activity are mapped to the Execute Input File and Import Results activity as shown below.

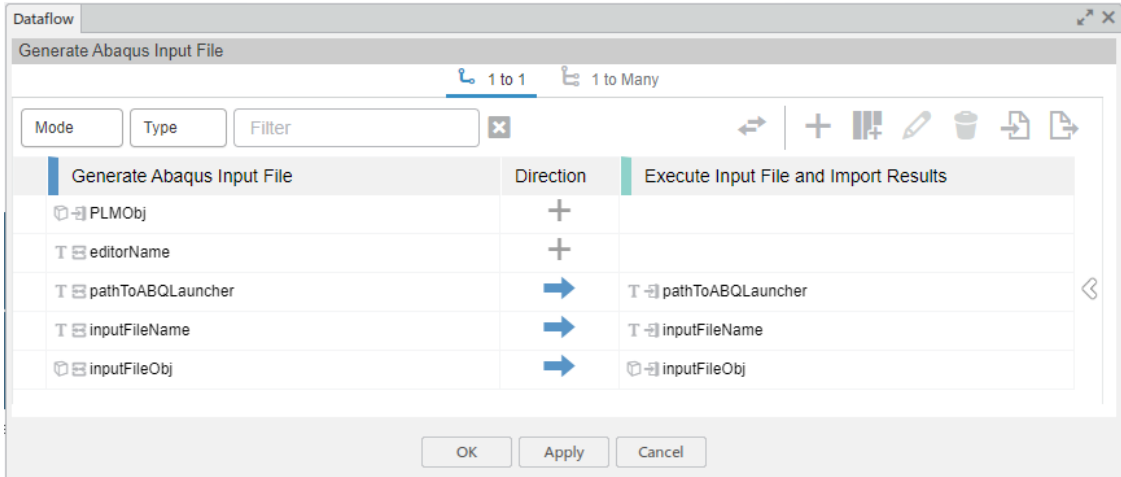


Figure 12. Parameter mapping using Dataflow 2

These parameter values are used in the Execute Input File and Import Results activity to:

- Download the Abaqus Input file,
- Execute the Input file, and
- Import it back in the **3DEXPERIENCE** Physics Results section for post-processing.

3.2.5. Execute Input File and Import Results Activity

The Execute Input File and Import Results activity as shown in Figure 13 downloads the Abaqus Input file updated by the user, executes the Input file, and uploads the results back to the **3DEXPERIENCE** platform for review.

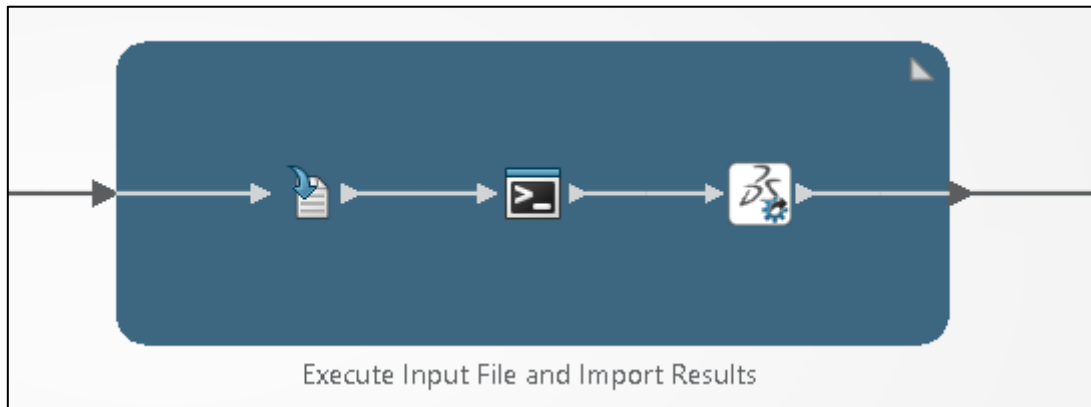


Figure 13. Execute Input File and Import Results activity

The Execute Input File and Import Results activity consists of three steps: download the Abaqus Input file, execute it, and upload the updated results to the **3DEXPERIENCE** platform.

Best Practices

1.  Download Input File Step.

The Download Input File step downloads the previously generated and modified Abaqus Input file to the execution directory that will have all the edits made by the user, interactively.

2.  Execute Abaqus Step.

This step submits the Abaqus Input file for execution. The step adapter is configured to execute the Input file using the command option.

3.  Import Abaqus Results Step.

This step stores the SIM file generated from the Abaqus job in the **3DEXPERIENCE** platform. The Import Simulation Results batch program imports the SIM file into the existing Physics Simulation object.

Note: Ensure that the whole simulation process is configured on the localhost.

4. Simulation Template

A simulation template is used to capture the best practices and intellectual property that are encapsulated in a well-designed simulation process or simulation activity in an organization. Once the simulation process is complete and ready for use, the simulation template can be created from the simulation process. After moving the simulation template from 'Development' state to 'Released' state, the best practices captured by the simulation template can be used by users who may not have engineering knowledge to build a simulation process. This simulation template is now available in the Physics Simulation app under the External Solve feature.

4.1. Create Simulation Template

The Process Experience Studio widget helps the user to create a simulation template that will be available in the Physics Simulation app under the Simulate External Solve feature.

1. Login to the 3DDashboard and add the 'Process Experience Studio' widget to the dashboard as shown below.

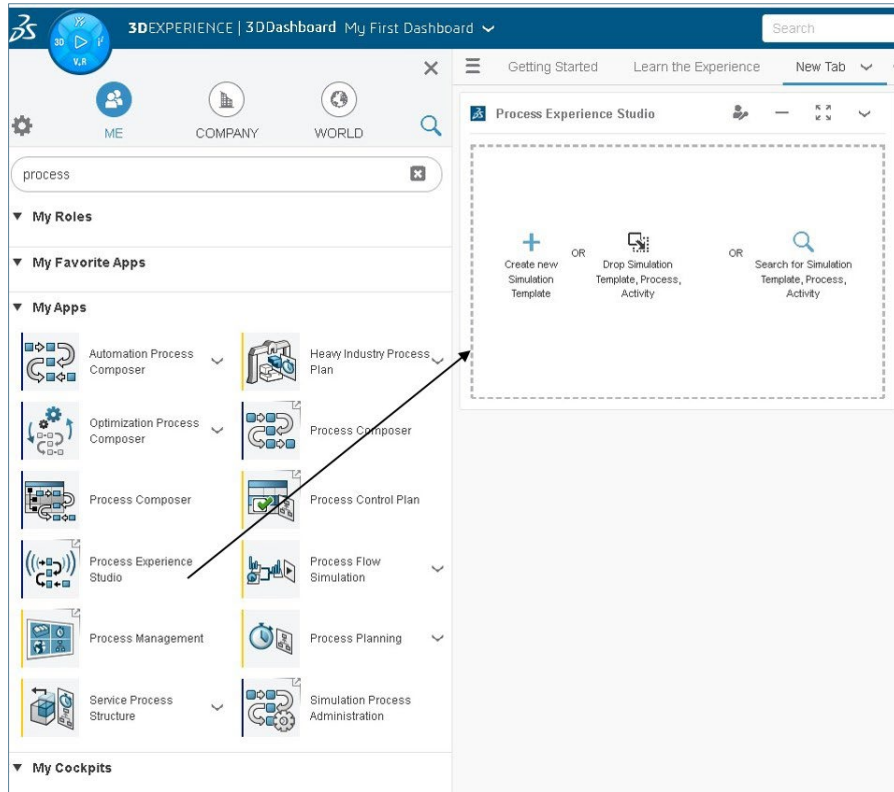


Figure 14. Process Studio widget

2. Search for the simulation process using the **Search for Simulation Template, Process, Activity** option.
3. Select the simulation process to create a simulation template and click **OK**. The **Create Simulation Template** dialog box opens as shown below.

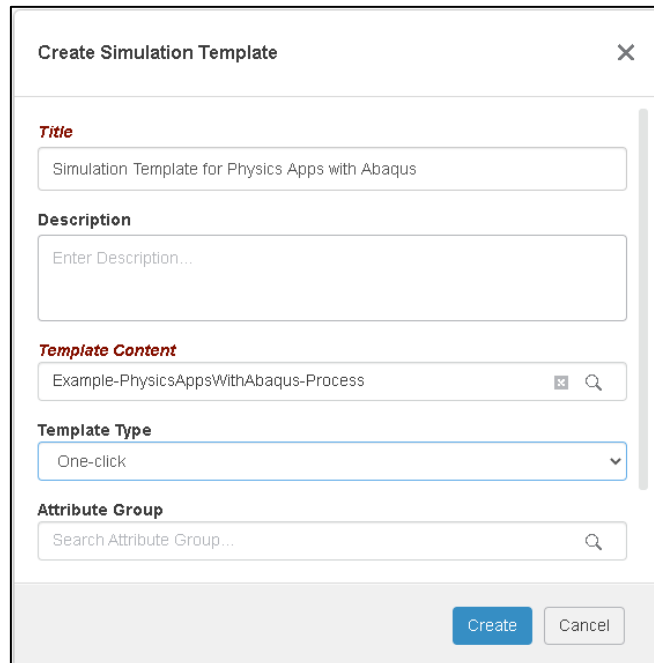


Figure 15. Create Simulation Template options


- Specify the Title for the simulation template, select the Template Type **One-click**, and click **Create**. A simulation template of type 'One-Click' is created.
- Remove all the editor paths from the template and save the changes of the simulation template. Note that the user can find these paths at 5 occasions in this simulation template.

4.2. Maturity of the Simulation Template

After creating the simulation template and it is ready for use, the user can change the maturity of the simulation template from 'Development' state to 'Released' state in the Process Experience Studio app. Changing the maturity to 'Released' state will make the simulation template available in the External Solve feature of the Physics Simulation app.

4.2.1. Change the Maturity of Simulation Template from 'Development' State to 'Released' State

Perform the following steps to change the maturity of the simulation template from 'Development' state to 'Released' state.

- In the newly created simulation template, click on the  icon to select the **Change maturity of Simulation Template** option.

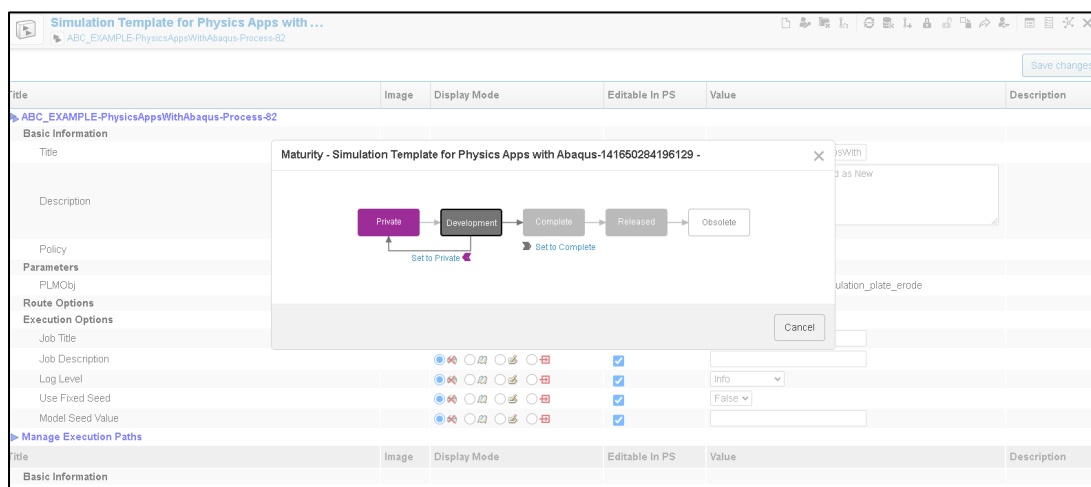


Figure 16. Simulation Template in Lifecycle Widget

- Change the maturity of the simulation template from 'Development' state to 'Complete' state using the **Set to Complete** option.
- Similarly, change the maturity of the simulation template from 'Complete' state to 'Released' state as shown below.

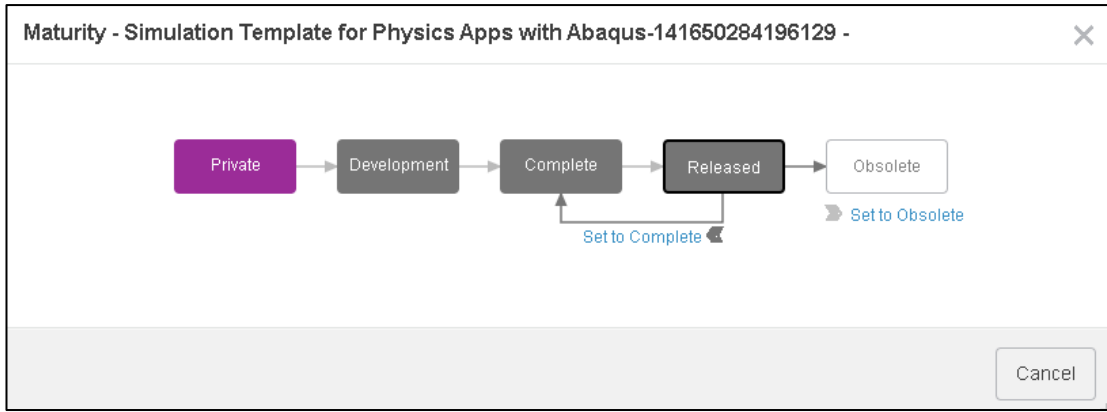


Figure 17. Simulation Template in the 'Released' state

After the simulation template is moved to the 'Released' state, the simulation template will be available in the Physics Simulation app under the External Solve feature.

5. Use External Solve for Execution

The External Solve feature is available under the Simulate section of the Physics Simulation app. The external solves are performed by associating the Physics Simulation with a simulation process and simulation template. The simulation process underneath is executed in a batch mode performing the specified operations in the activities.

1. Import and open the Example Physics Simulation object in the Mechanical Scenario Creation App.
2. Switch to Mechanical Scenario workbench and click the **Simulate** tab from the action bar.
3. Select the **External Solve** feature to fetch the simulation template as shown below.

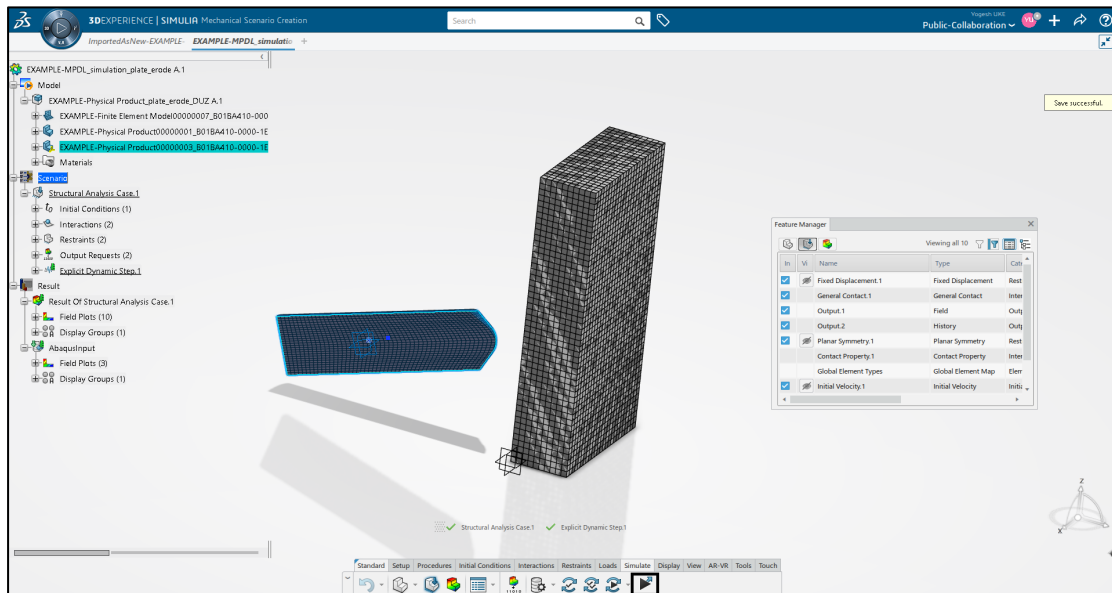


Figure 18. External Solve feature

- Using the **3DSearch** option, select the simulation template in the 'Released' state as shown below.

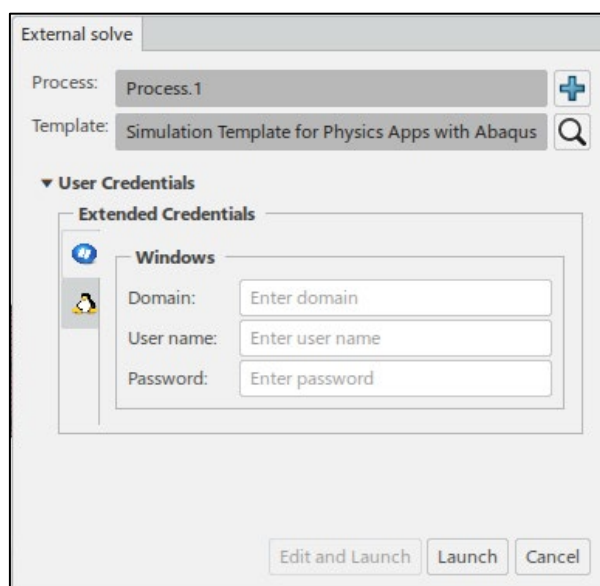


Figure 19. Select the simulation template using External Solve

- Select the **Launch** option to start the execution of the simulation process in batch mode.
- Once the execution starts, a text file with the name 'Paths.txt' opens in the default Notepad editor.
- Check the default execution paths and the Abaqus Input file name as explained in section 3.2.1. "Manage Execution Paths". Adjust the values according to the local machine configuration.
- Save the changes and close the Notepad editor.
- After some time, the Abaqus Input file opens in the user specified editor to include the user specific Abaqus keywords. For more information on adding keywords to the Abaqus Input file, refer to section 3.2.3. "Generate Abaqus Input File Activity".
- Save the changes and close the editor. The Abaqus Input file gets executed.
- Track the execution status in the Job Monitor wizard as shown below.

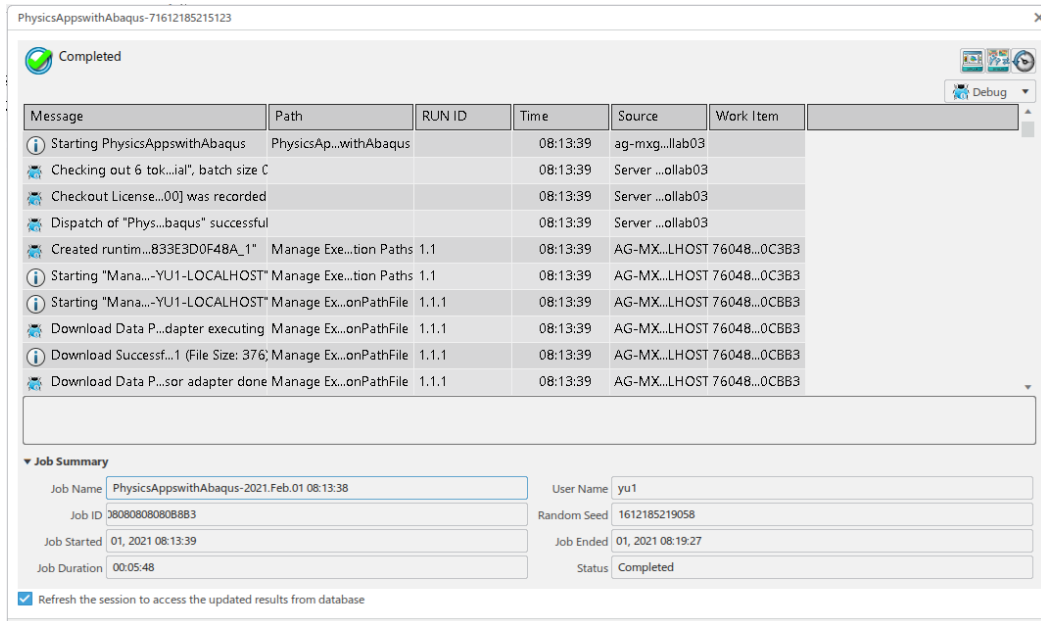


Figure 20. Job Monitor Wizard

- After the execution is complete, the results are imported back into the **3DEXPERIENCE** Physics Results section of the same Physics Simulation object for post-processing.

6. Post-Processing Results

In the **3DEXPERIENCE** Physics Results section, the user can evaluate different aspects of simulation. Using the simulation output data, the user can create and compare plots, reports, and animations.

6.1. Access the Analysis Case

Open the solved simulation object that is referred during the final activity of the simulation process. In the Execute Input File and Import Results activity, Abaqus will run the modified Input file and store the result in the simulation object referred to that particular activity as shown below.

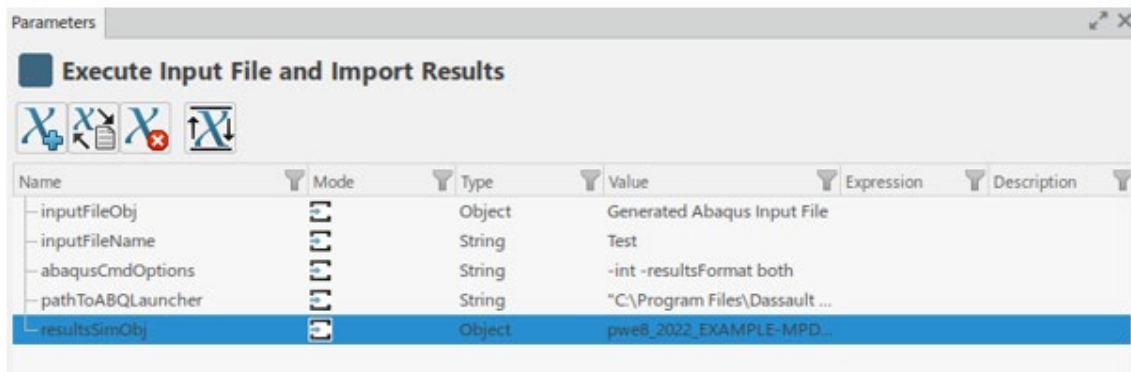


Figure 21. Solved Simulation Object

To visualize the results, the user can open the referred object in the 'resultsSimObj' parameter.

The user can use the Plots wizard to access the recently solved and imported user analysis case as shown below.

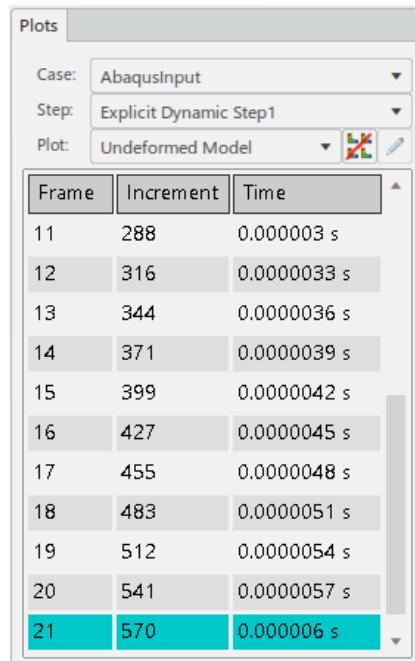


Figure 22. Select the Analysis case in the Plots

6.2. Create Field Plots

To create field plots, select the **Contour Plot** option in the **Plots** tab of the action bar. Select **Von Mises Stress** from the **Template** drop down list and click **OK** as shown below.

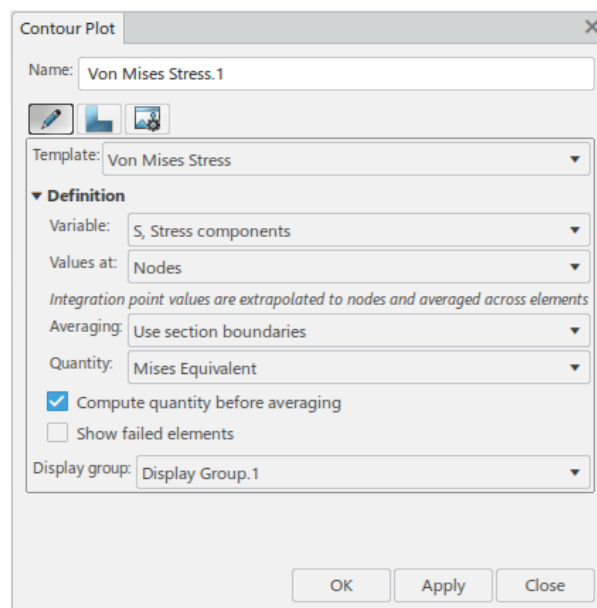


Figure 23. Von Mises Stress plot creation

Best Practices

The Von Mises Stress plots generated on the viewport help to understand the stress distribution as shown below.

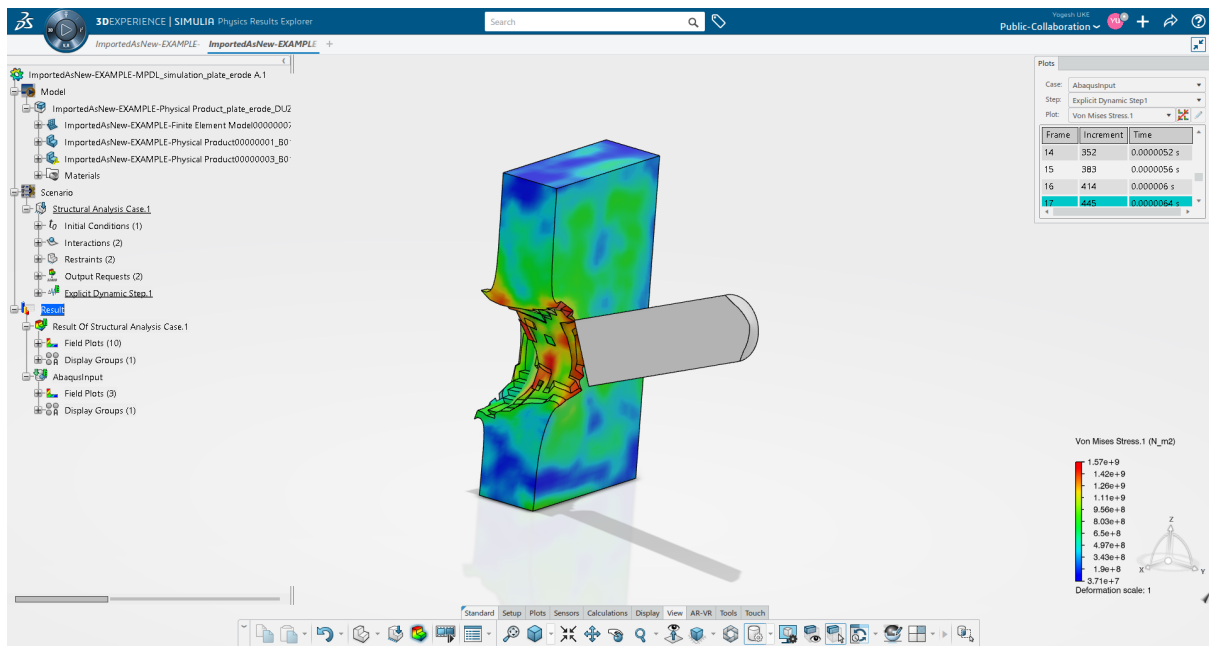


Figure 24. Von Mises Stress plots

Similarly, the user can also create other plots like displacement, Plastic Strain, and so on.

7. References

- QA00000069538
- [Structural Model Creation App User Guide](#)
- [Structural Scenario Creation App User Guide](#)
- [Process Composer App User Guide](#)
- [Understanding the 3DEXPERIENCE platform and Apps](#)

8. Document History

Document Revision	Date	Revised By	Changes/Notes
1.0	02/25/2021	Yogesh UKE	Original
2.0	06/15/2022	Yogesh UKE	Updated the following sections: 3.2.1. Manage Execution Paths 3.2.3. Generate Abaqus Input File Activity 4.2. Maturity of the Simulation Template