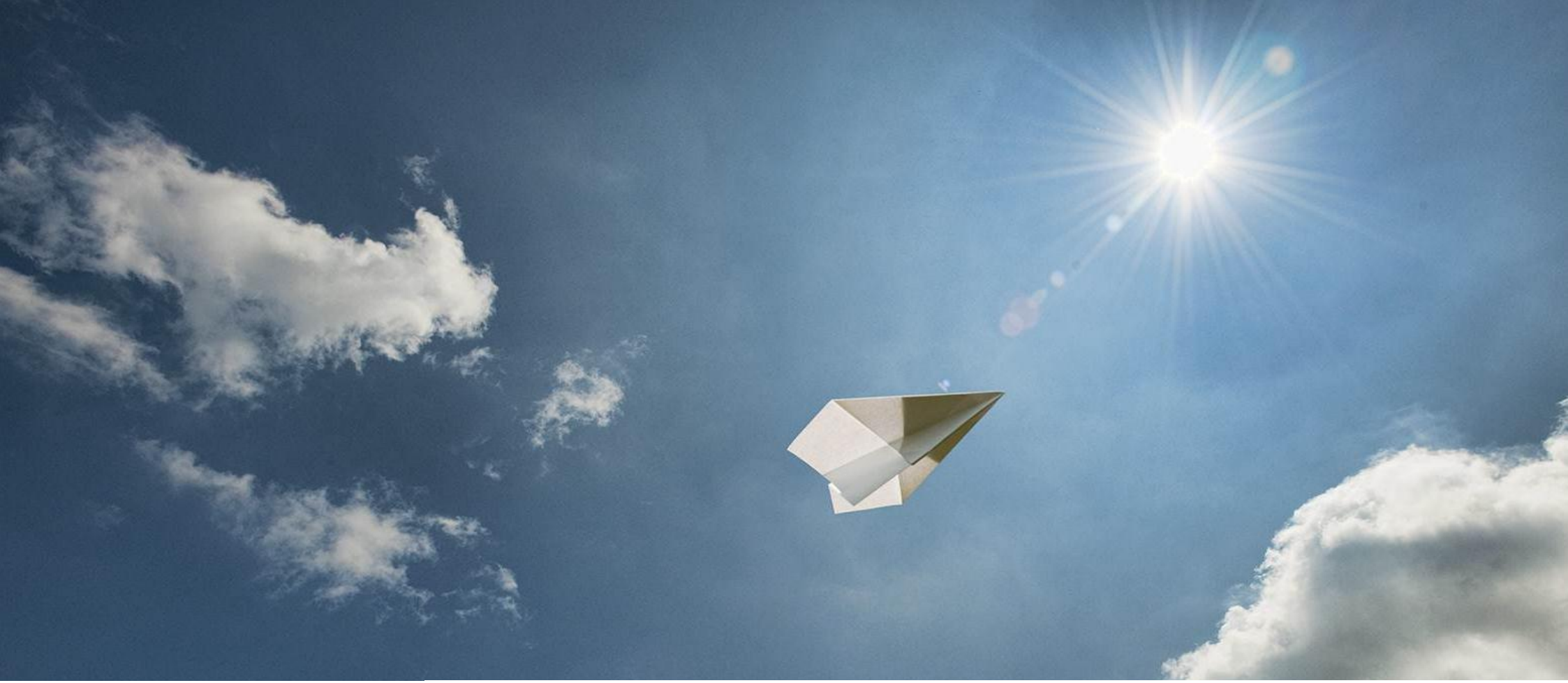


Testing and Validation of the MFRONT Interface for ABAQUS

Dominique Deloison, Fabrice Congourdeau, Airbus Group Innovations, Suresnes
dominique.deloison@airbus.com

AIRBUS GROUP

Interest in MFRONT

- ABAQUS is widely used within Airbus Group
- Within Airbus (commercial aircrafts) and Stelia Aerospace, ABAQUS is the official FE code for non linear simulations:
  - Impact and crashworthiness
  - Certification analyses (Full-scale testing)
  - Simulation of manufacturing processes

Interest in MFRONT

- Within Airbus Group Innovations, research work requires the implementation of advanced material models:

  - Anisotropic Plasticity (other than Hill-48)
  - Ductile Damage
  - Damage for Composite Materials

- ABAQUS library of material models is relatively poor, however user materials can be defined through the use of subroutines.
- The implementation has to be done for Implicit and Explicit Solvers (UMAT and VUMAT).

  Indeed, it is standard practice to switch from one solver to the other
  (eg Forming in Explicit, springback analysis in Implicit)
- UMAT and VUMAT conventions are significantly differents.
- Implementations are often painful and not very computationnaly efficient (not the primary skill of AGI researchers).

- **MFRONT for ABAQUS could bring more efficiency and allow to focus more on the model features rather than on the implementation**

AIRBUS GROUP

MFRONT/ABAQUS Interface

- MFRONT generates a library which is called by ABAQUS through a generic UMAT
- Example of input file:

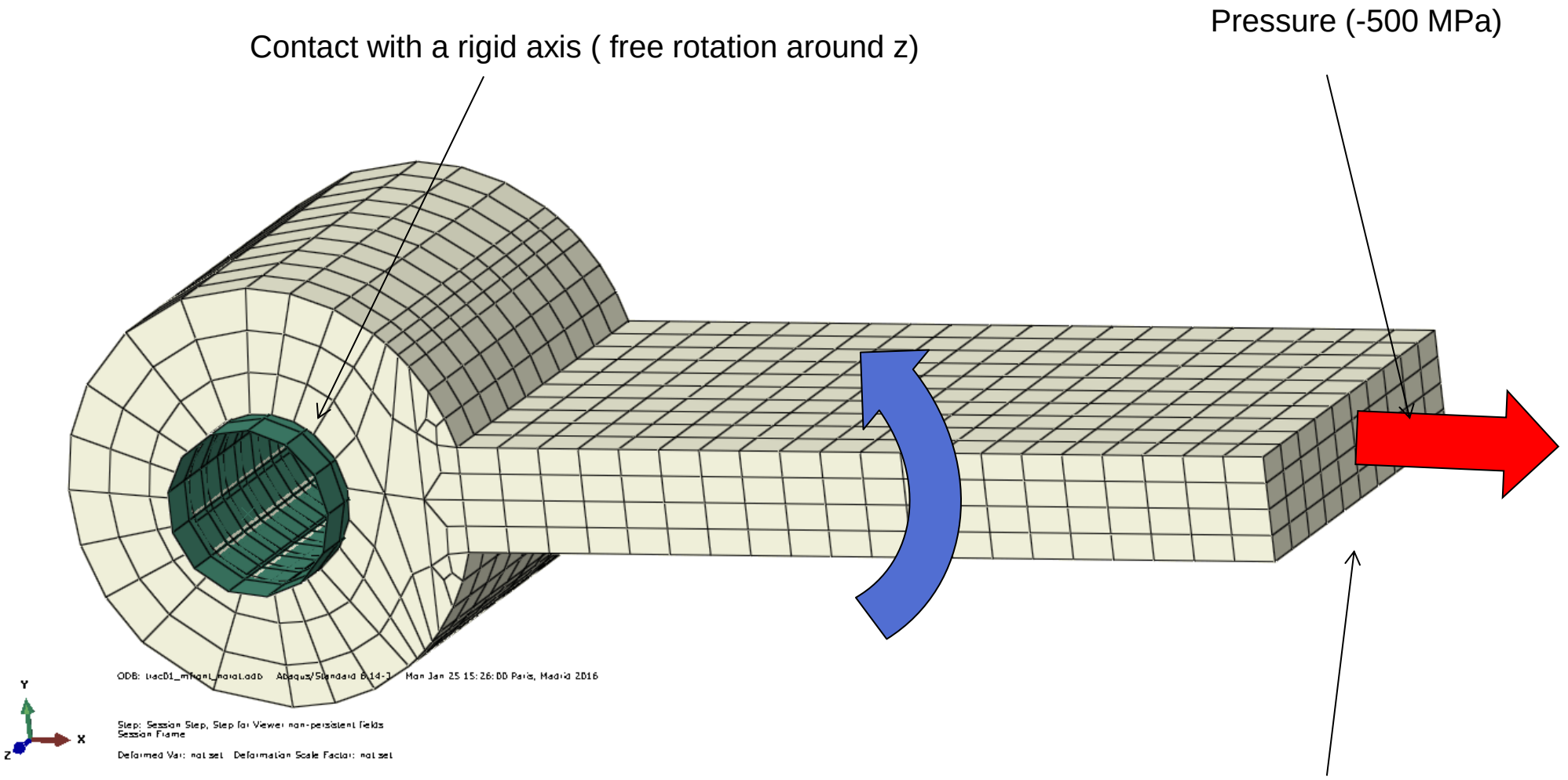
```
*Material, name=ABAQUSBEHAVIOUR_chaboche
*Depvar
31,
1, ElasticStrain_11
2, ElasticStrain_22
3, ElasticStrain_33
4, ElasticStrain_12
5, ElasticStrain_13
6, ElasticStrain_23
7, EquivalentPlasticStrain
…
*User Material, constants=11, unsymm
<YoungModulus>, <PoissonRatio>, <R_inf>, <R_0>, <b>, <C0>, <C1>, <C2>
<g0>, <g1>, <g2>
```


AIRBUS GROUP

MFRONT/ABAQUS Interface

- The UMAT behaviour has to be well understood to be sure of the consistency of the UMAT/MFRONT connection
- ABAQUS provide two ways to handle large transformations:
  - Implement in small transformations and let ABAQUS do the rest (recommended by SIMULIA)
  - Implement in large transformations.
- A test has been defined in large transformations to analyse ABAQUS behaviour

Test case 1



Calculations are performed with large displacements and rotations assumption

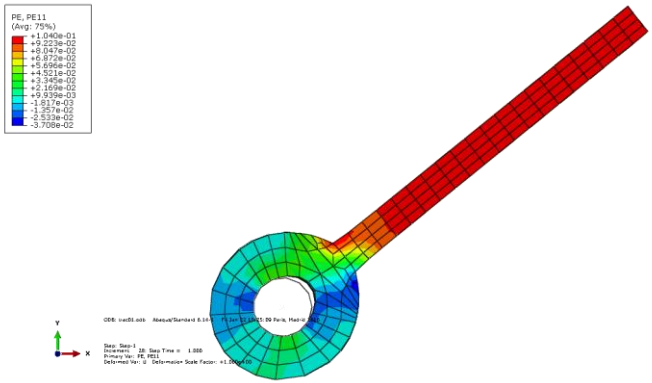
Large transformations are handled by ABAQUS. MFRONT law is written for small strains.

Test case 1

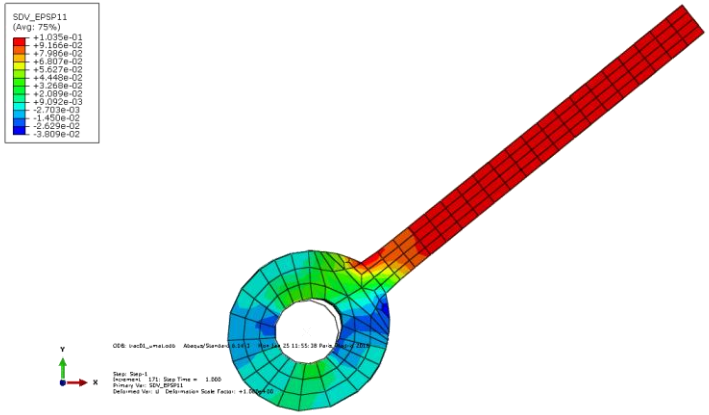
Objectives: 1st comparison between ABAQUS and ABAQUS/MFRONT
Check what happens in case of large transformations

	Case	**Rotation**
Test 1	ABAQUS Implicit Built-In Law	Yes
Test 2	ABAQUS UMAT	Yes
Test 3	ABAQUS UMAT (no ROTSIG)	Yes
Test 4	ABAQUS UMAT/MFRONT	Yes
Test 5	ABAQUS Implicit Built-In Law	No
Test 6	ABAQUS UMAT	No
Test 7	ABAQUS UMAT (no ROTSIG)	No
Test 8	ABAQUS UMAT/MFRONT	No

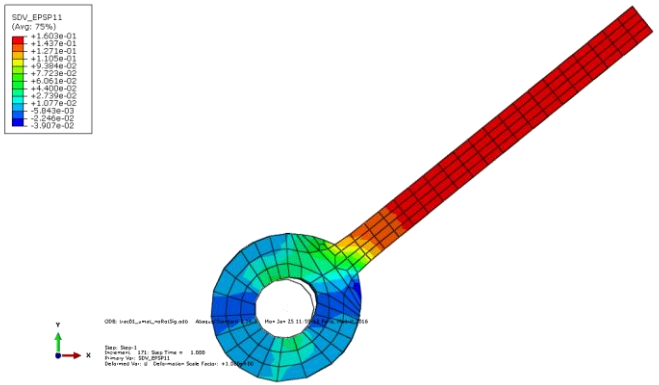
Abaqus reference solution



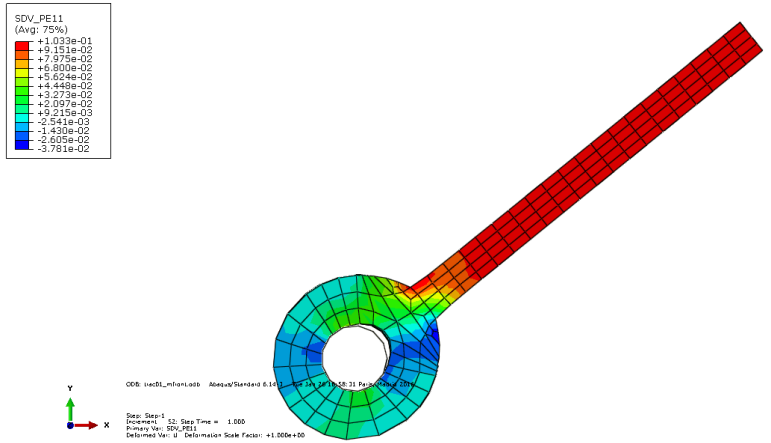
Test 1



Test 2



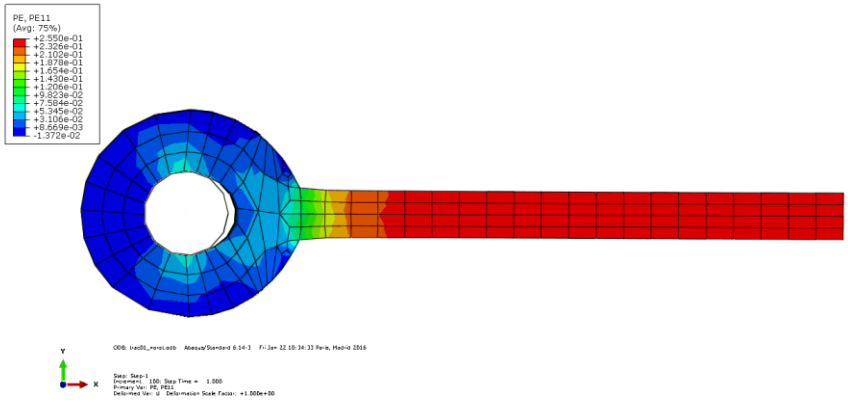
Test 3



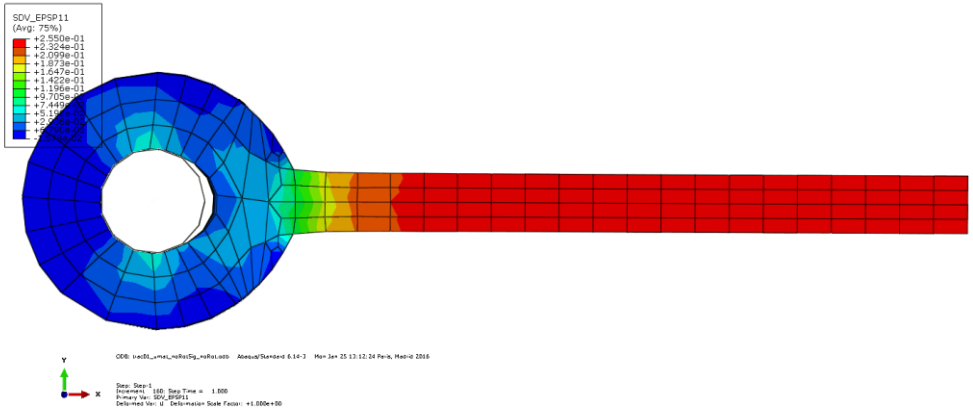
Test 4

AIRBUS GROUP

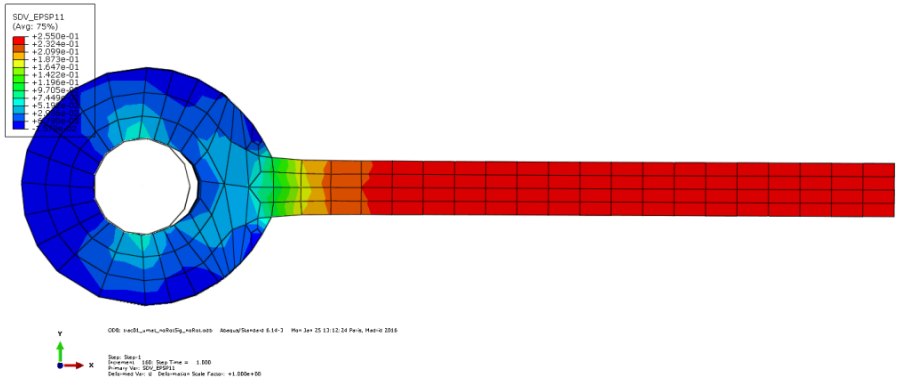
Abaqus reference solution



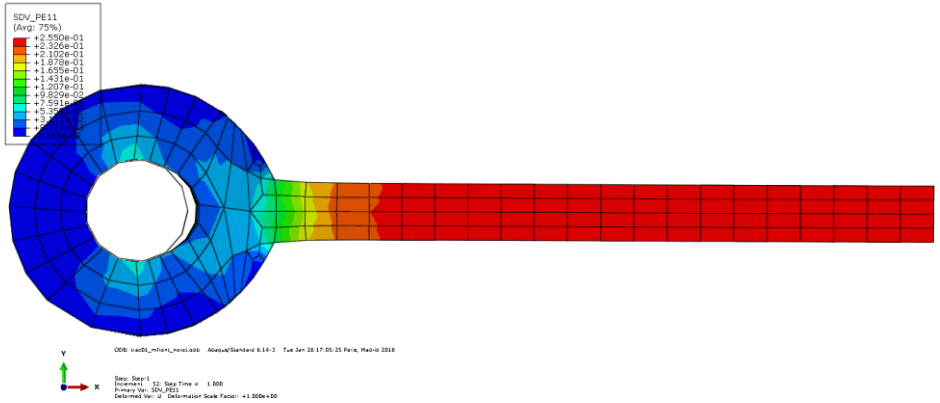
Test 5



Test 6



Test 7



Test 8

AIRBUS GROUP

Synthesis

	Rotation		NoRotation	
	PE11	Difference with respect to ref	PE11	
Ref	0.104	-	0.255	-
UMAT	0.1035	-0.5%		
UMAT no RotSig	0.1603	54.1%	0.255	0%
UMAT/MFRONT	0.1033	-0.7%	0.255	0%

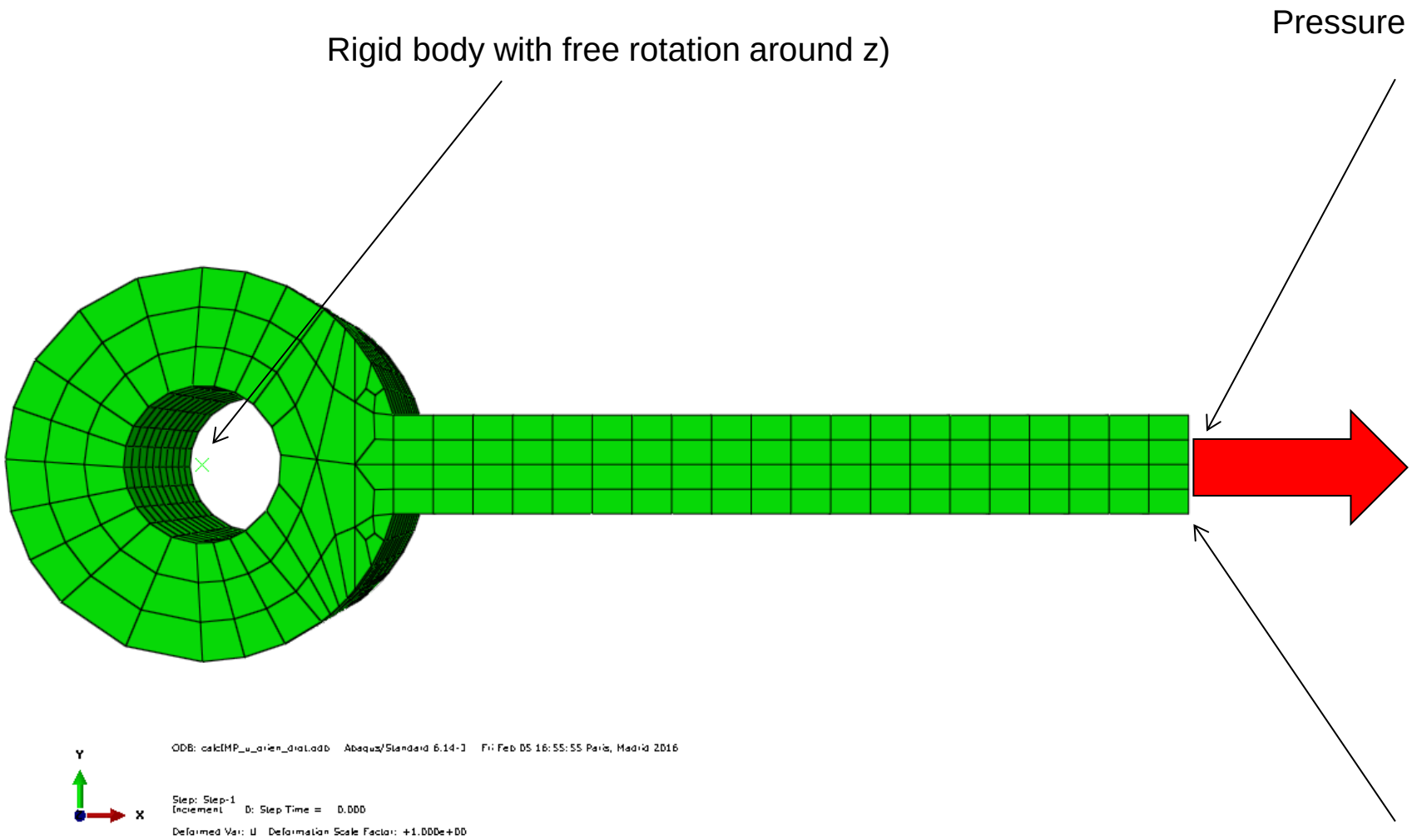
ABAQUS/MFRONT (small strain implementation)
gives very similar results than ABAQUS including when a large rotation occurs

	CPU Time (s)	Difference	Number of increments	Difference
ABAQUS	192	-	52	-
ABAQUS/UMAT(*)	1200	525%	295	467%
ABAQUS/MFRONT	201	5%	52	0%
(*) Numerical Jacobian				

MFRONT seems to be much more efficient than a badly implemented UMAT!!!

AIRBUS GROUP

Test case 2

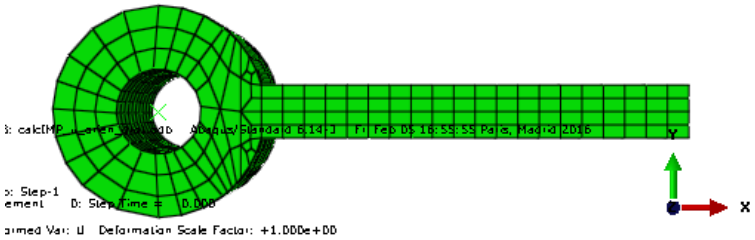


Calculations are performed with large transformations assumption

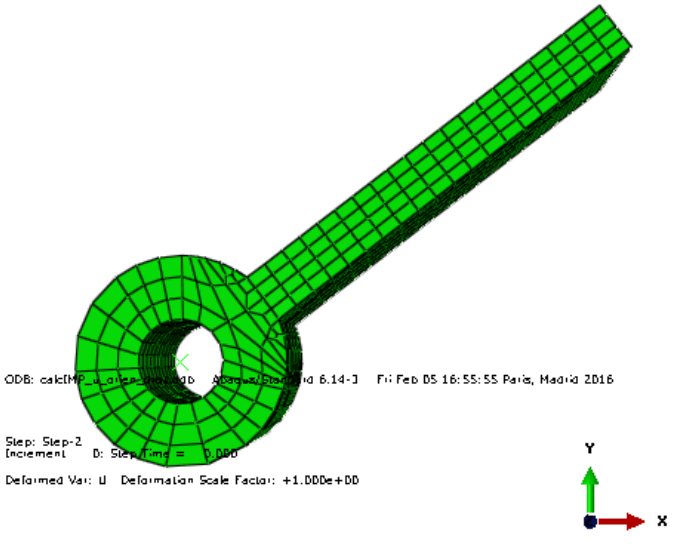
AIRBUS GROUP

Test case

- An elasto-plastic constitutive law is used (SY=200, H=1200)

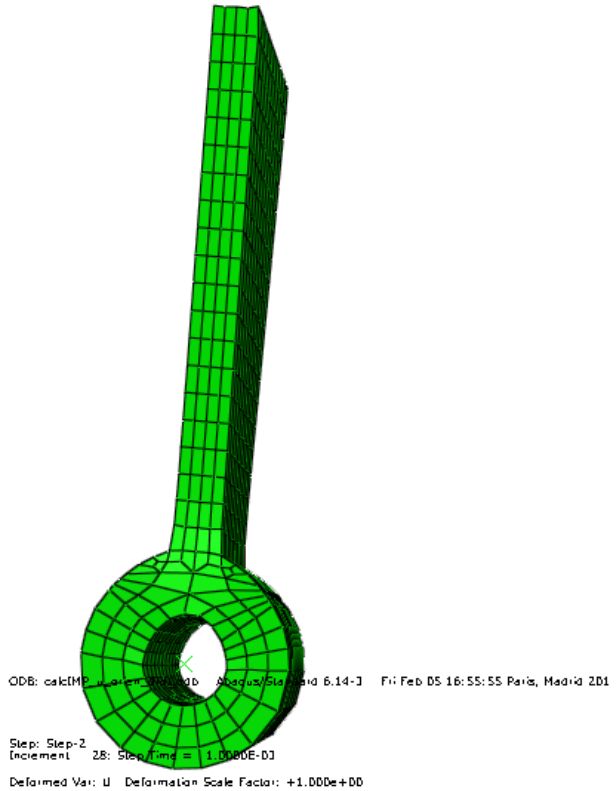


Beginning of step1



End of Step1

Beginning of Step 2



End of Step2

- Loading is gradually increased so that the specimen is stretched progressively over the 2 steps.
- For some calculations, a solver switch is performed between the two steps.

Test case objectives

- Check what happens with UMAT/VUMAT when large rotations occur
- Check compatibilities of tensorial internal variables.
- Switching between explicit and implicit solvers is of particular interest

Tested configurations (ABAQUS Only)

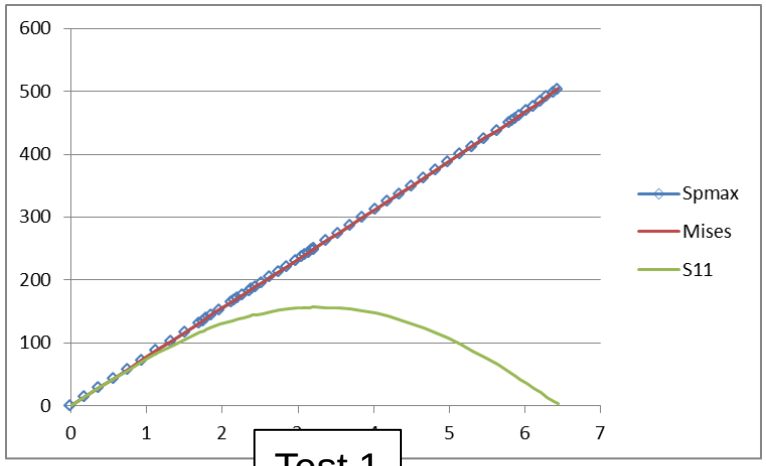
	Rotation 1 (0-->45°)	Rotation 2 (45-->90°)	Orientation
Test 1	ABAQUS Implicit Built-In Law	ABAQUS Implicit Built-In Law	No
Test 2	ABAQUS Explicit Built-In Law	ABAQUS Explicit Built-In Law	No
Test 3	ABAQUS Implicit Built-In Law	ABAQUS Explicit Built-In Law	No
Test 4	ABAQUS Explicit Built-In Law	ABAQUS Implicit Built-In Law	No
Test 5	ABAQUS UMAT	ABAQUS UMAT	No
Test 6	ABAQUS VUMAT	ABAQUS VUMAT	No
Test 7	ABAQUS UMAT	ABAQUS VUMAT	No
Test 8	ABAQUS VUMAT	ABAQUS UMAT	No
Test 9	ABAQUS Implicit Built-In Law	ABAQUS Implicit Built-In Law	Yes
Test 10	ABAQUS Explicit Built-In Law	ABAQUS Explicit Built-In Law	Yes
Test 11	ABAQUS Implicit Built-In Law	ABAQUS Explicit Built-In Law	Yes
Test 12	ABAQUS Explicit Built-In Law	ABAQUS Implicit Built-In Law	Yes
Test 13	ABAQUS UMAT	ABAQUS UMAT	Yes
Test 14	ABAQUS VUMAT	ABAQUS VUMAT	Yes
Test 15	ABAQUS UMAT	ABAQUS VUMAT	Yes
Test 16	ABAQUS VUMAT	ABAQUS UMAT	Yes

UMAT/VUMAT were coded in FORTRAN

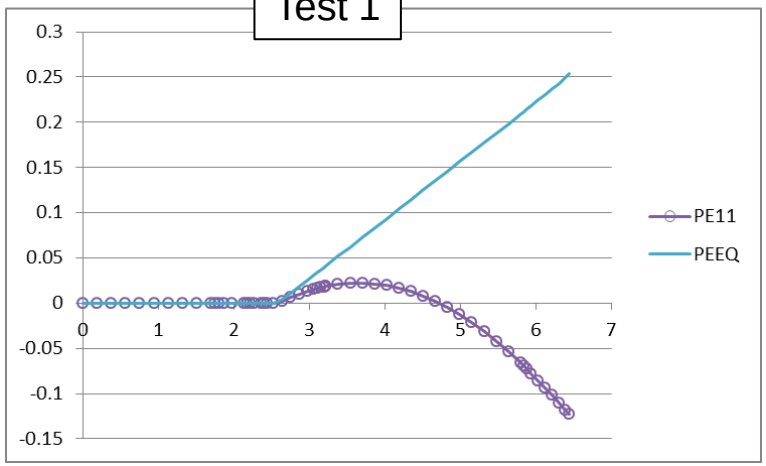
ORIENTATION is used for anisotropic materials

Results

Stress vs edge displacement



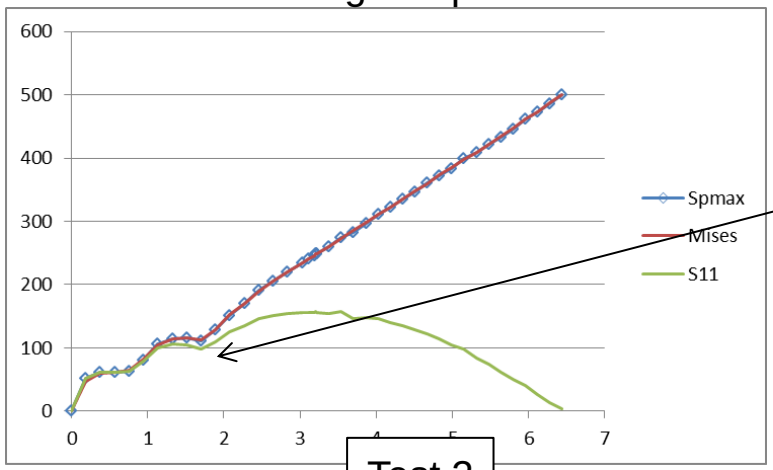
Test 1



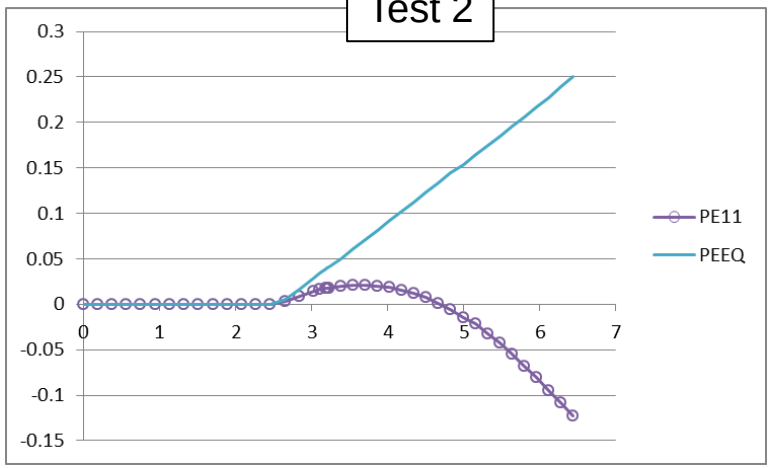
Strain vs edge displacement

Implicit solver + buil-in material law

Stress vs edge displacement



Test 2



Strain vs edge displacement

Explicit solver + buil-in material law

Non linear application of loading to minimize inertia effect at the end of step 1

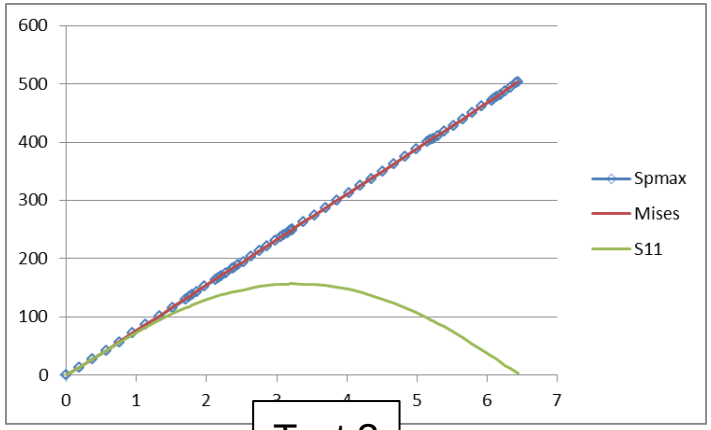
Results are similar with the 2 solvers.

Results are expressed in the global coordinate system.

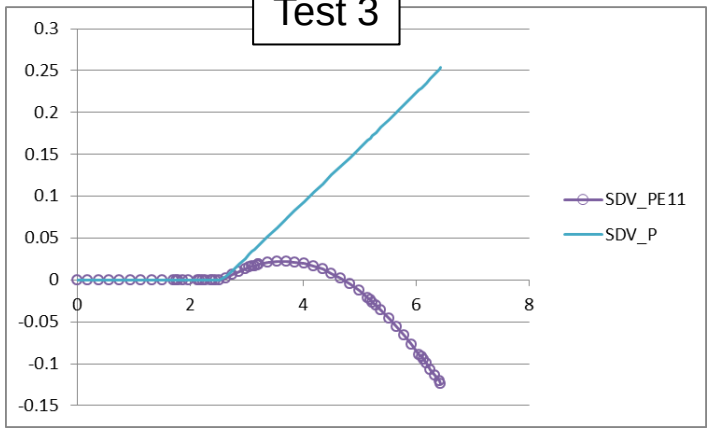
AIRBUS GROUP

Results

Stress vs edge displacement



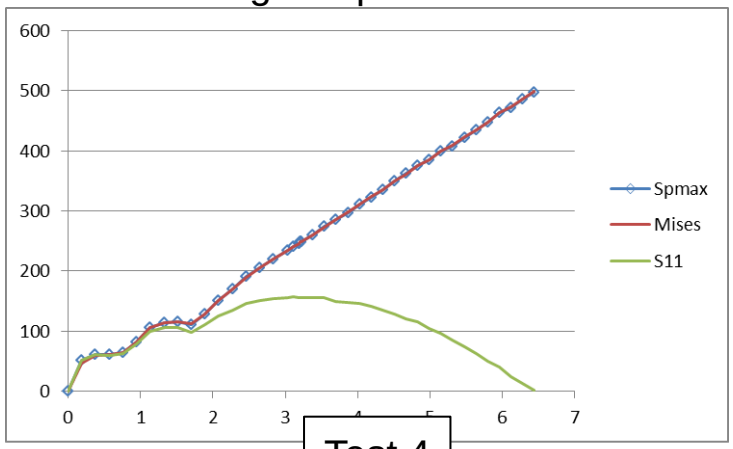
Test 3



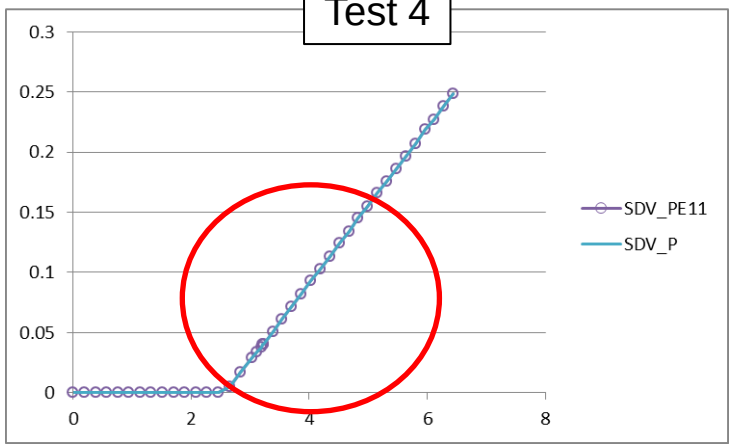
Strain vs edge displacement

Implicit solver + UMAT material law

Stress vs edge displacement



Test 4



Strain vs edge displacement

Explicit solver + VUMAT material law

Stresses are similar for the 2 solvers and consistant with the built-in laws.

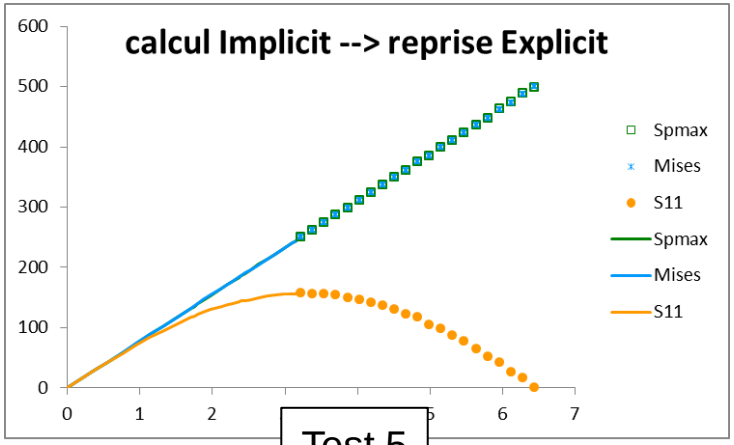
For UMAT, the plastic strain (tensorial internal variable) is consistant with the implicit solver.

For VUMAT, the plastic strain is expressed in the material coordinate system.

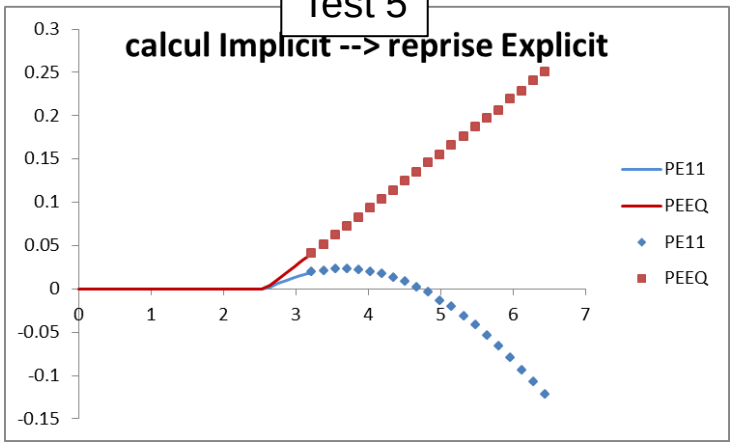
AIRBUS GROUP

Results

Stress vs edge displacement



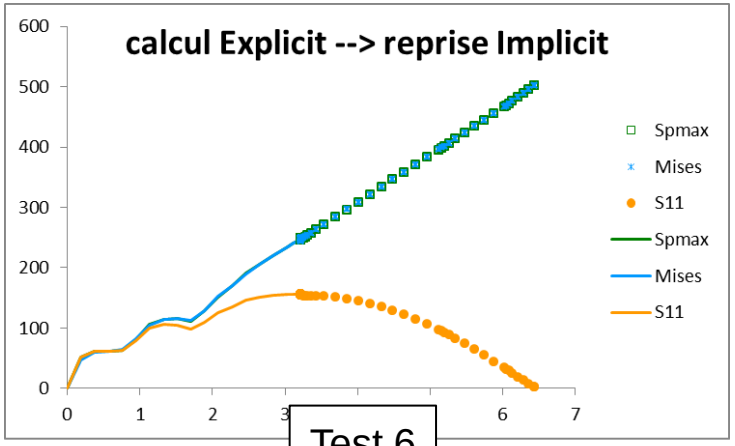
Test 5



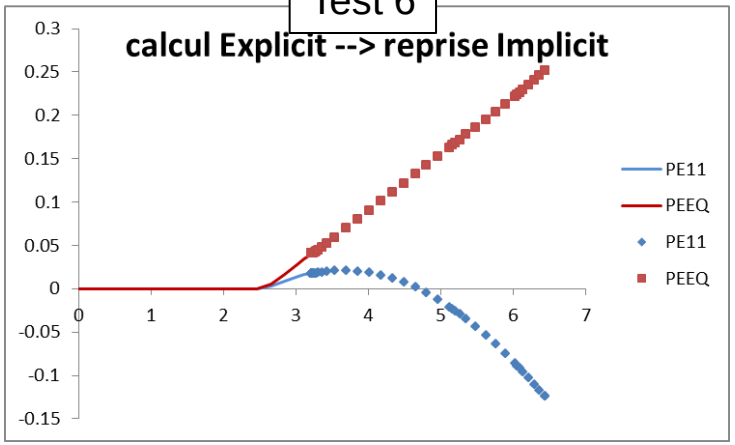
Strain vs edge displacement

Built-in material laws

Stress vs edge displacement



Test 6



Strain vs edge displacement

Butil-in material laws

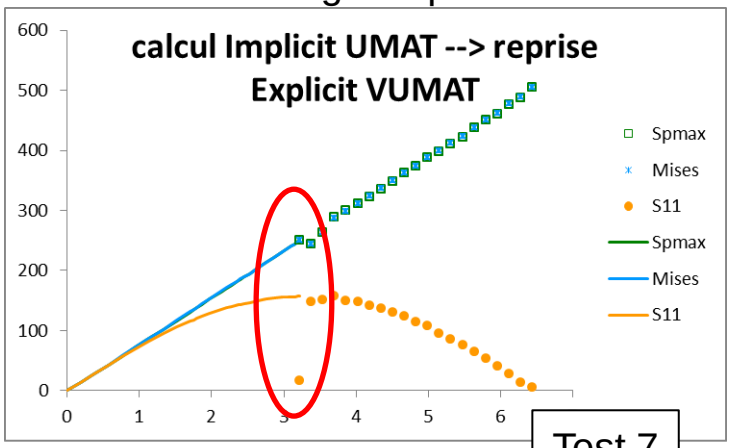
Switching between solvers does not change the results compare to single-solver calculations.

Results are still expressed in global coordinate systems.

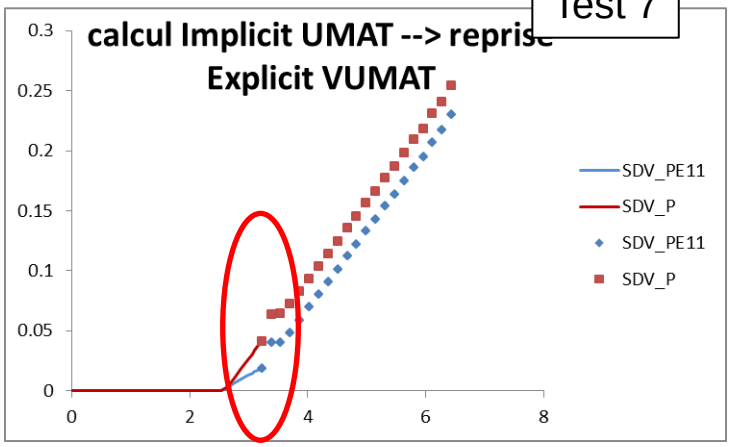
AIRBUS GROUP

Results

Stress vs edge displacement



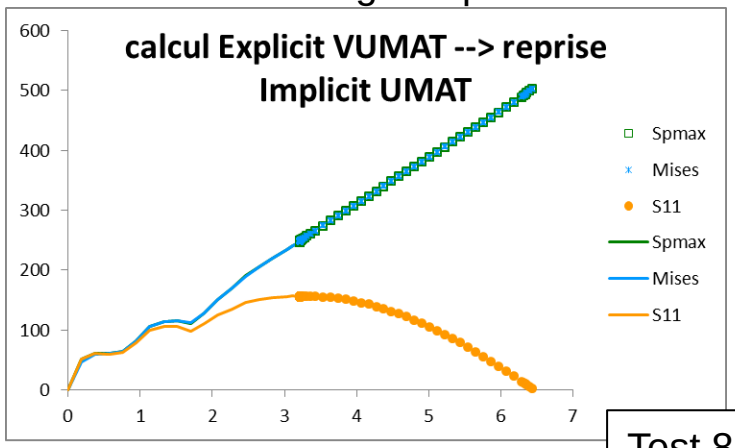
Test 7



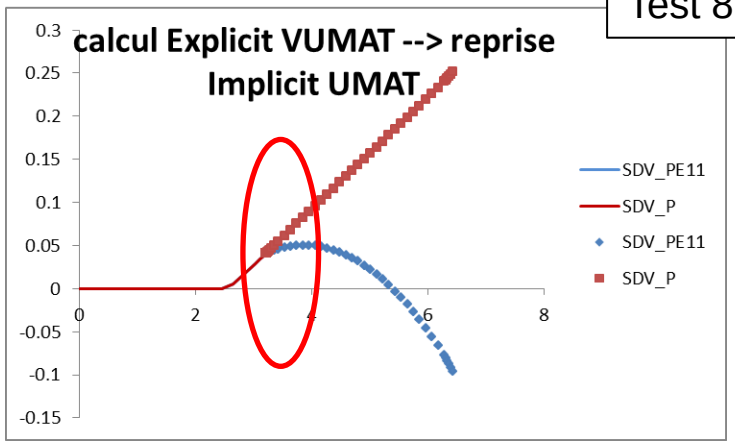
Strain vs edge displacement

UMAT/VUMAT material laws

Stress vs edge displacement



Test 8



Strain vs edge displacement

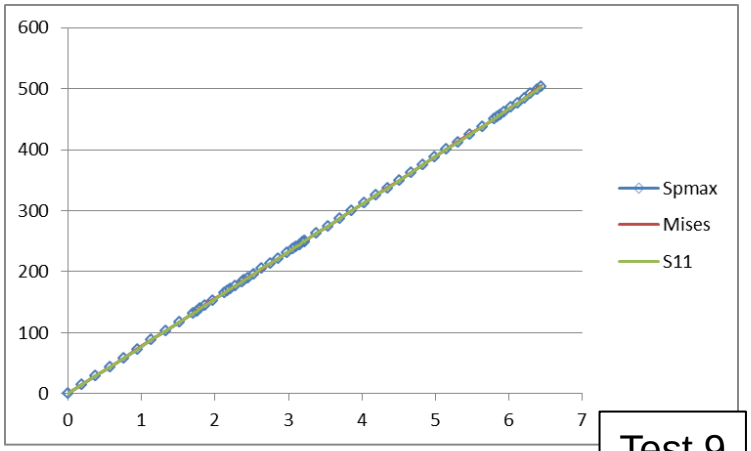
VUMAT/UMAT material laws

Results for tensorial variables are not expressed in the same coordinate systems, which creates significant discrepancies after switching.

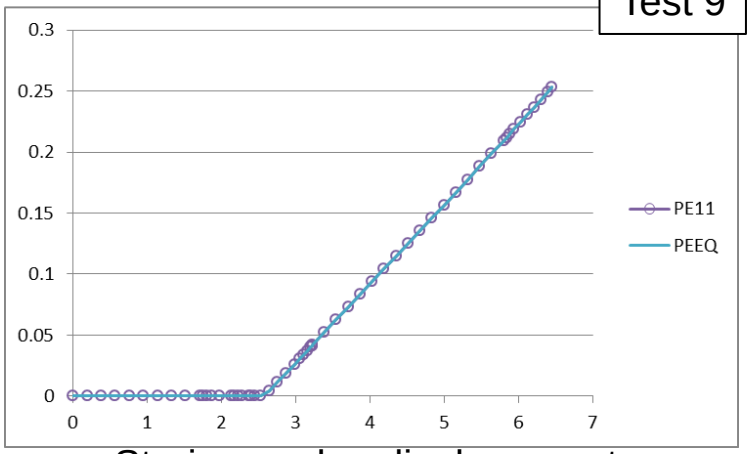
AIRBUS GROUP

Results

Stress vs edge displacement



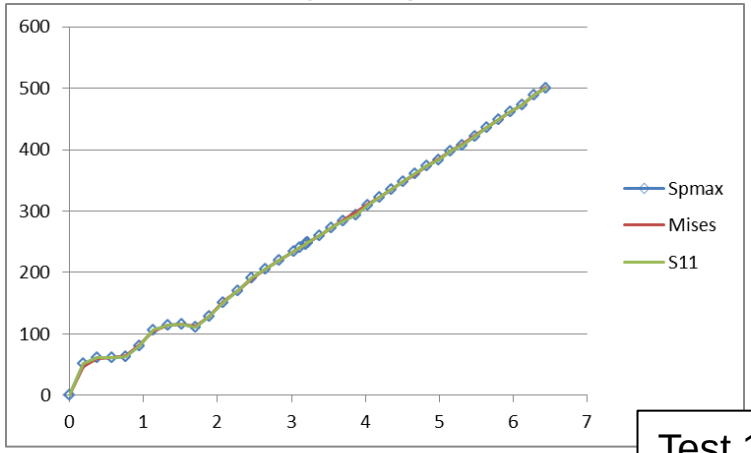
Test 9



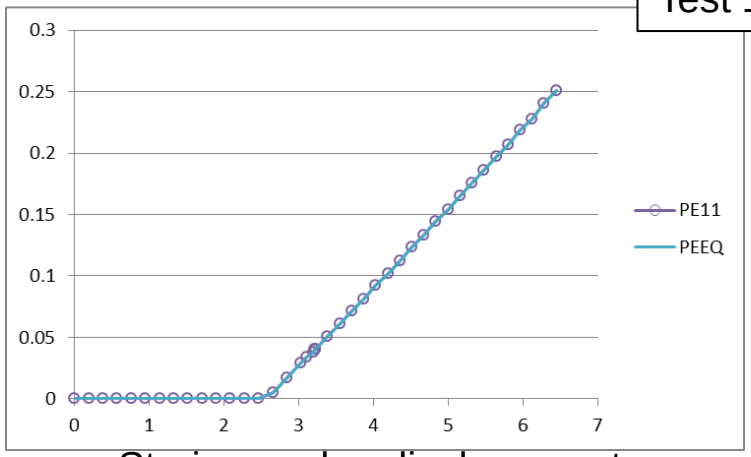
Strain vs edge displacement

Implicit solver + built-in material law + orientation

Stress vs edge displacement



Test 10



Strain vs edge displacement

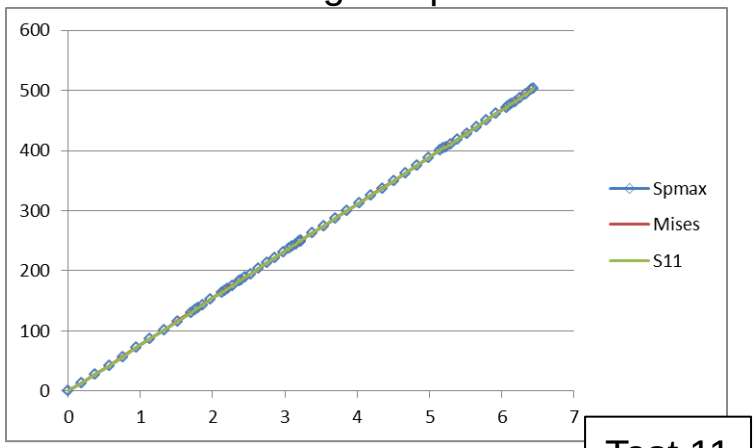
Explicit solver + built-in material law+ orientation

Results are all expressed in the material coordinate system (which rotates with the specimen)

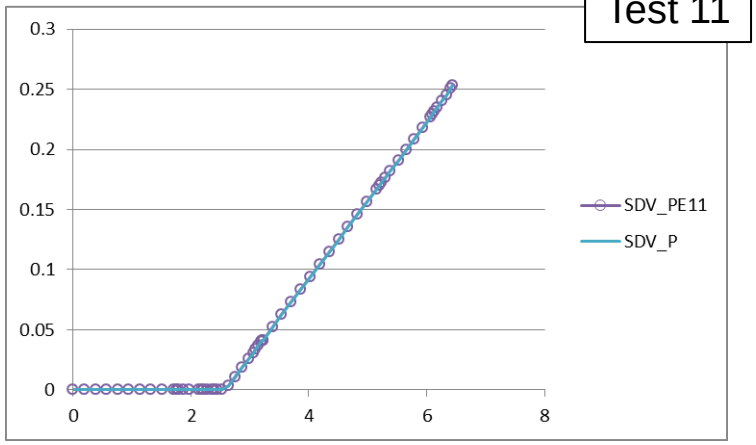
AIRBUS GROUP

Results

Stress vs edge displacement



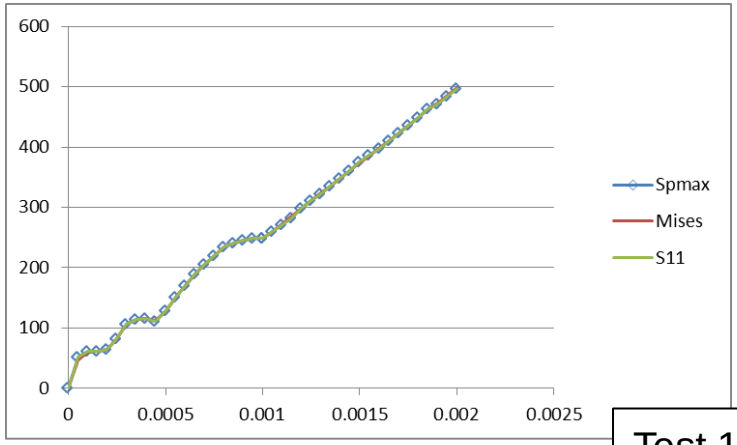
Test 11



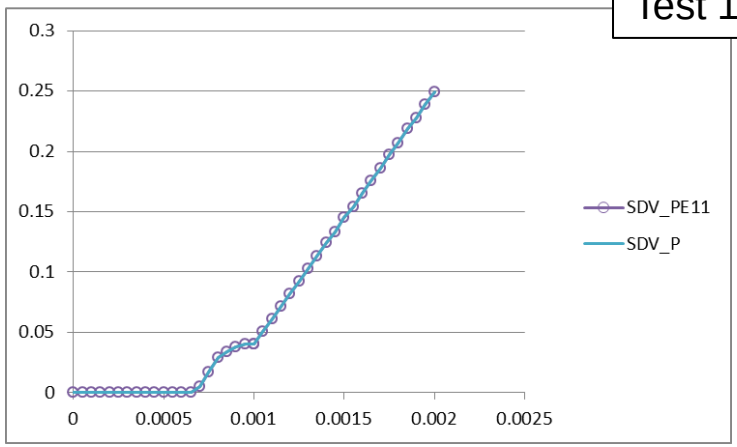
Strain vs edge displacement

Implicit solver + UMAT material law + orientation

Stress vs edge displacement



Test 12



Strain vs edge displacement

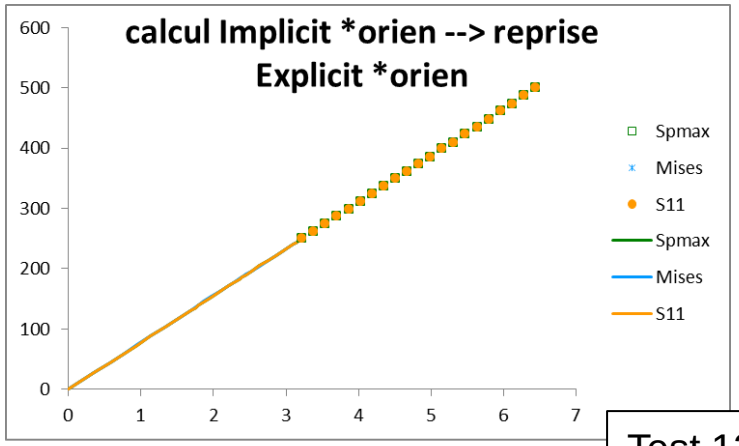
Explicit solver + VUMAT material law+ orientation

Results are all expressed in the material coordinate system (which rotates with the specimen). There is no difference any more between UMAT and VUMAT. Results are consistant with built-in material laws (with orientation).

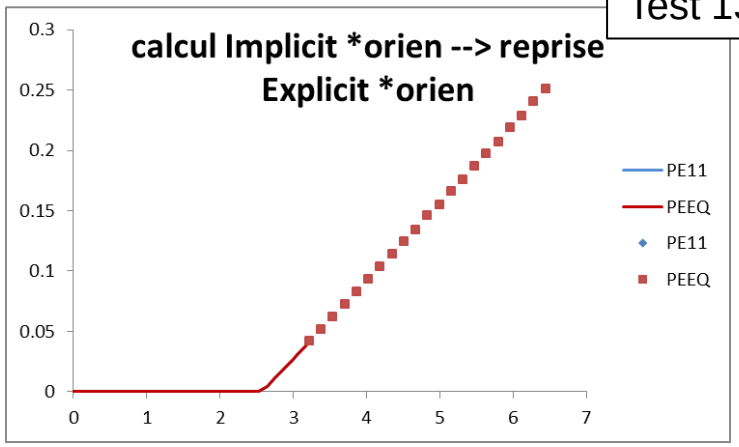
AIRBUS GROUP

Results

Stress vs edge displacement



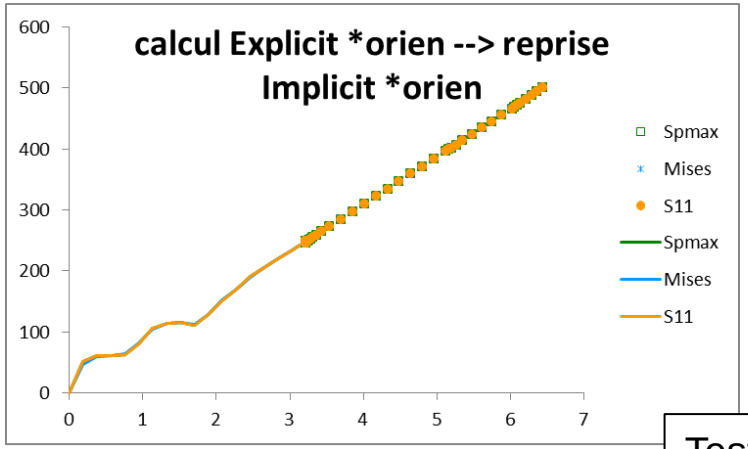
Test 13



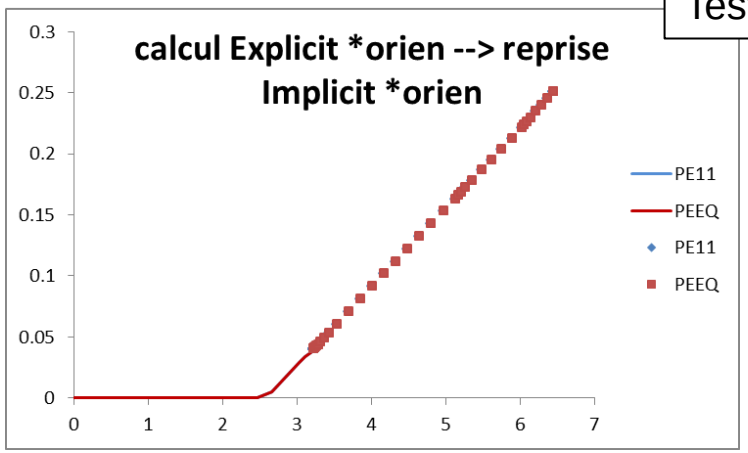
Strain vs edge displacement

Built-in material laws + orientation

Stress vs edge displacement



Test 14



Strain vs edge displacement

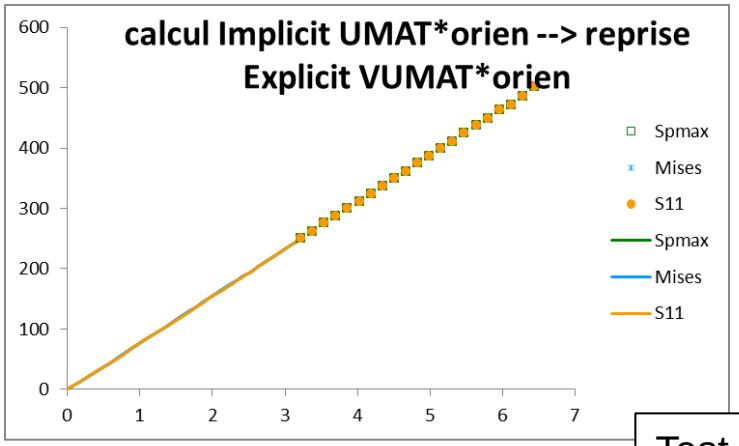
Built-in material laws + orientation

Results are all expressed in the material coordinate system (which rotates with the specimen).

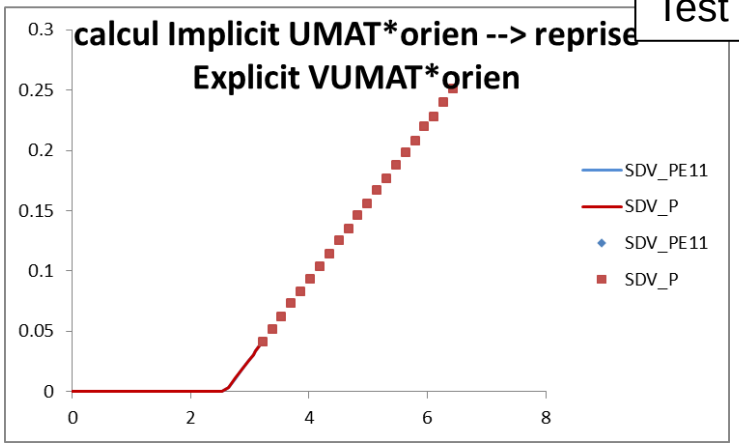
AIRBUS GROUP

Results

Stress vs edge displacement



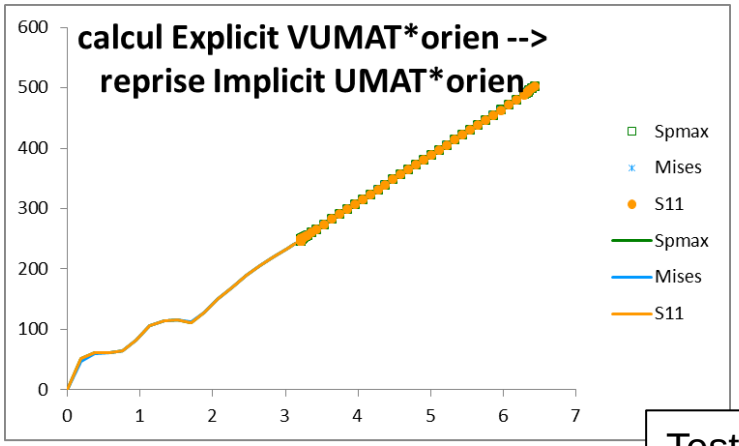
Test 15



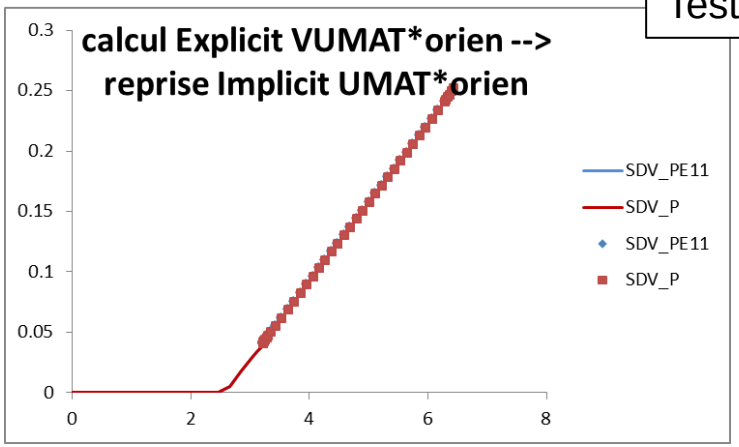
Strain vs edge displacement

UMAT/VUMAT material laws + orientation

Stress vs edge displacement



Test 16



Strain vs edge displacement

VUMAT/UMAT material law+ orientation

Results are all expressed in the material coordinate system (which rotates with the specimen). There is no difference any more between UMAT and VUMAT. Results are consistant with built-in material laws (with orientation).

AIRBUS GROUP

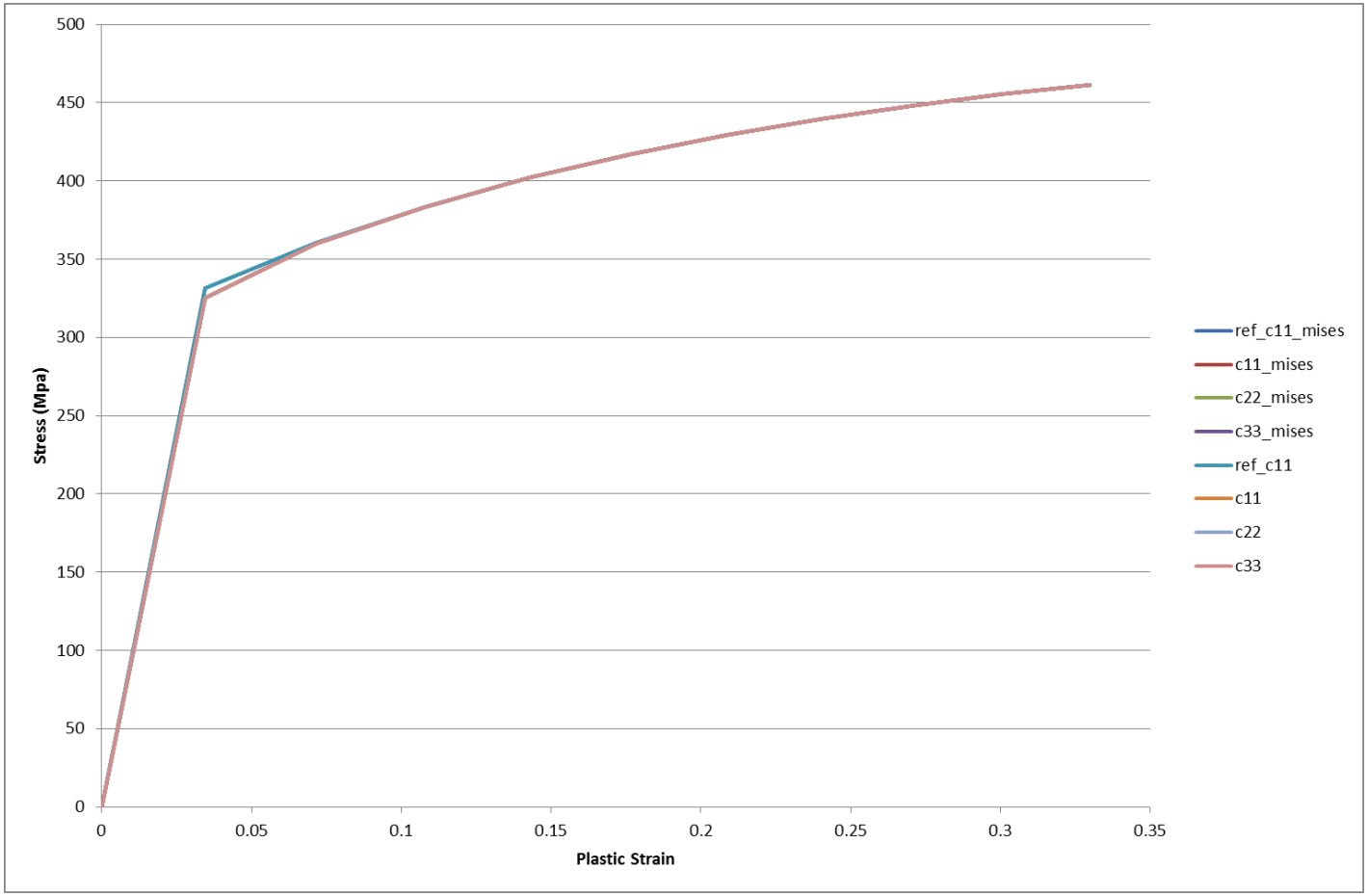
Test 2 Summary

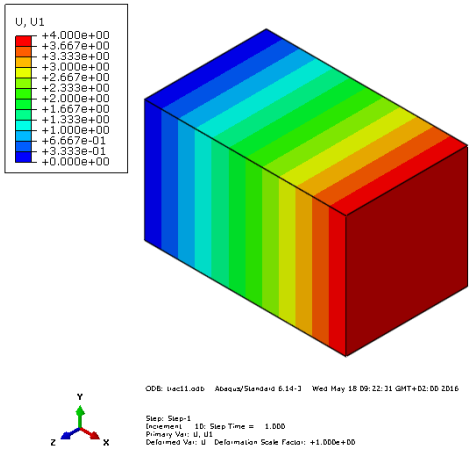
- When built-in material laws, results are all expressed in consistant coordinate systems. Either material or global (depending whether orientation is used or not). There is no issue when a switch between solvers is performed.
- When UMAT are used, results are consistant with built-in materials in terms of coordinate systems.
- When VUMAT are used, results for tensorial internal variables are always expressed in material coordinate systems, whether ORIENTATION is used or not.
- Switching between UMAT/VUMAT (and VUMAT/UMAT) is only valid when ORIENTATION is used.

Validation of the MFRONT/ABAQUS Interface

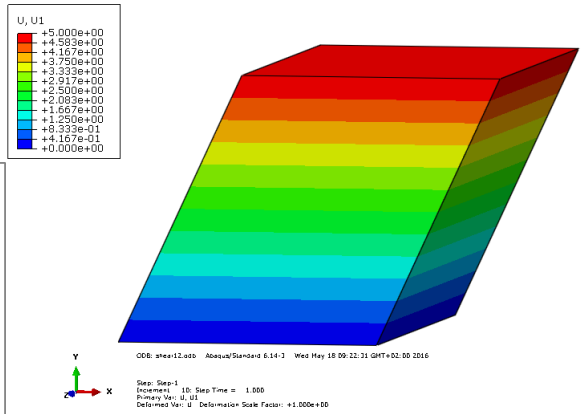
- Comparison between ABAQUS and MFRONT/ABAQUS
- Material model is a Chaboche non linear kinematic/isotropic model
- For the newton loop, a numerical jacobian is used (not optimized)
- Test complexity is gradual:
  - 1 brick (all directions and shearing)
  - 1 shell (all directions and shearing)
  - Cyclic test for 1 brick
  - Cyclic test for 1 shell
  - 1 tensile specimen model (shells)
  - 1 Failure Limit Diagram Model (3D)

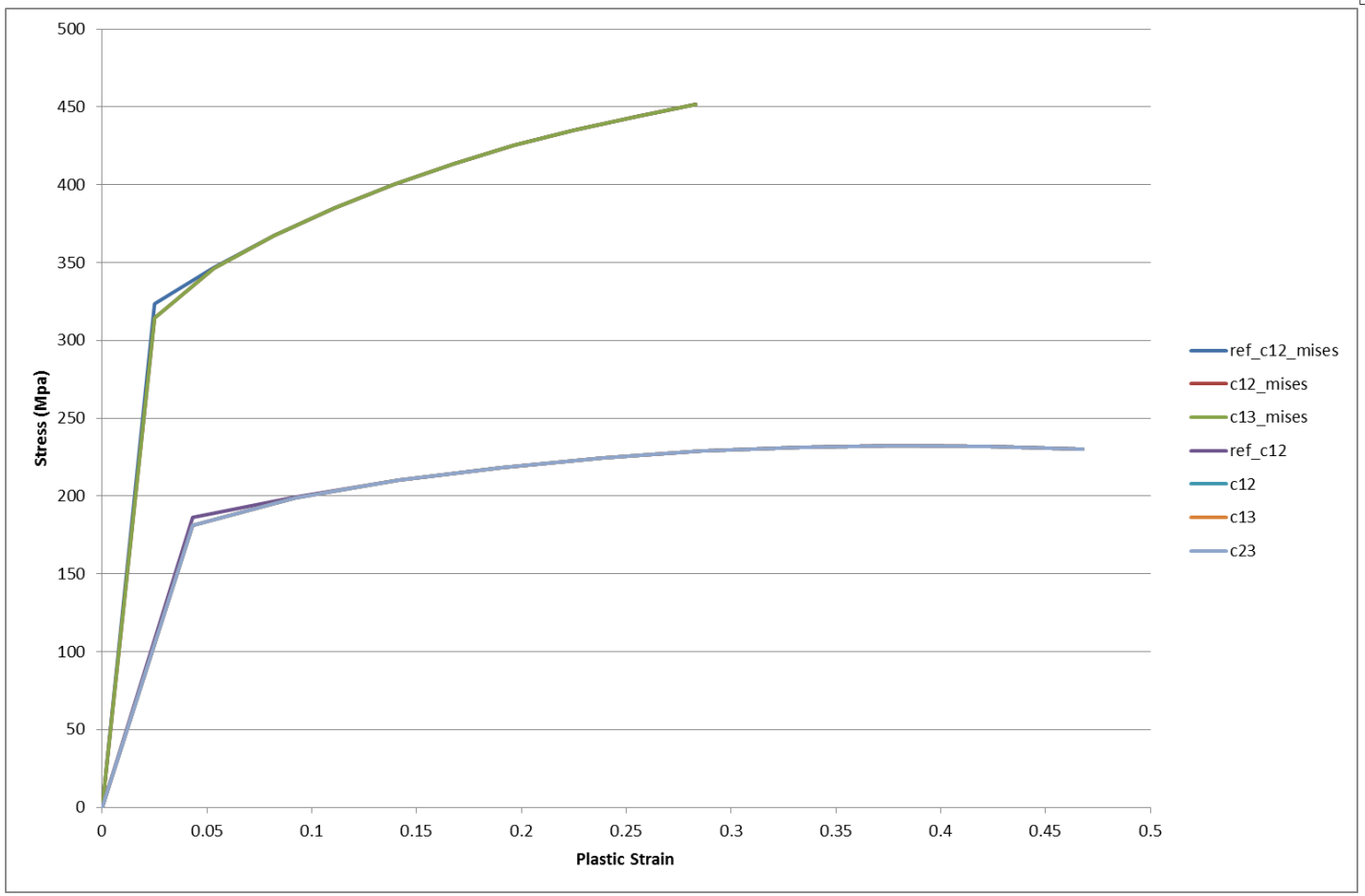
Validation of the MFRONT/ABAQUS Interface



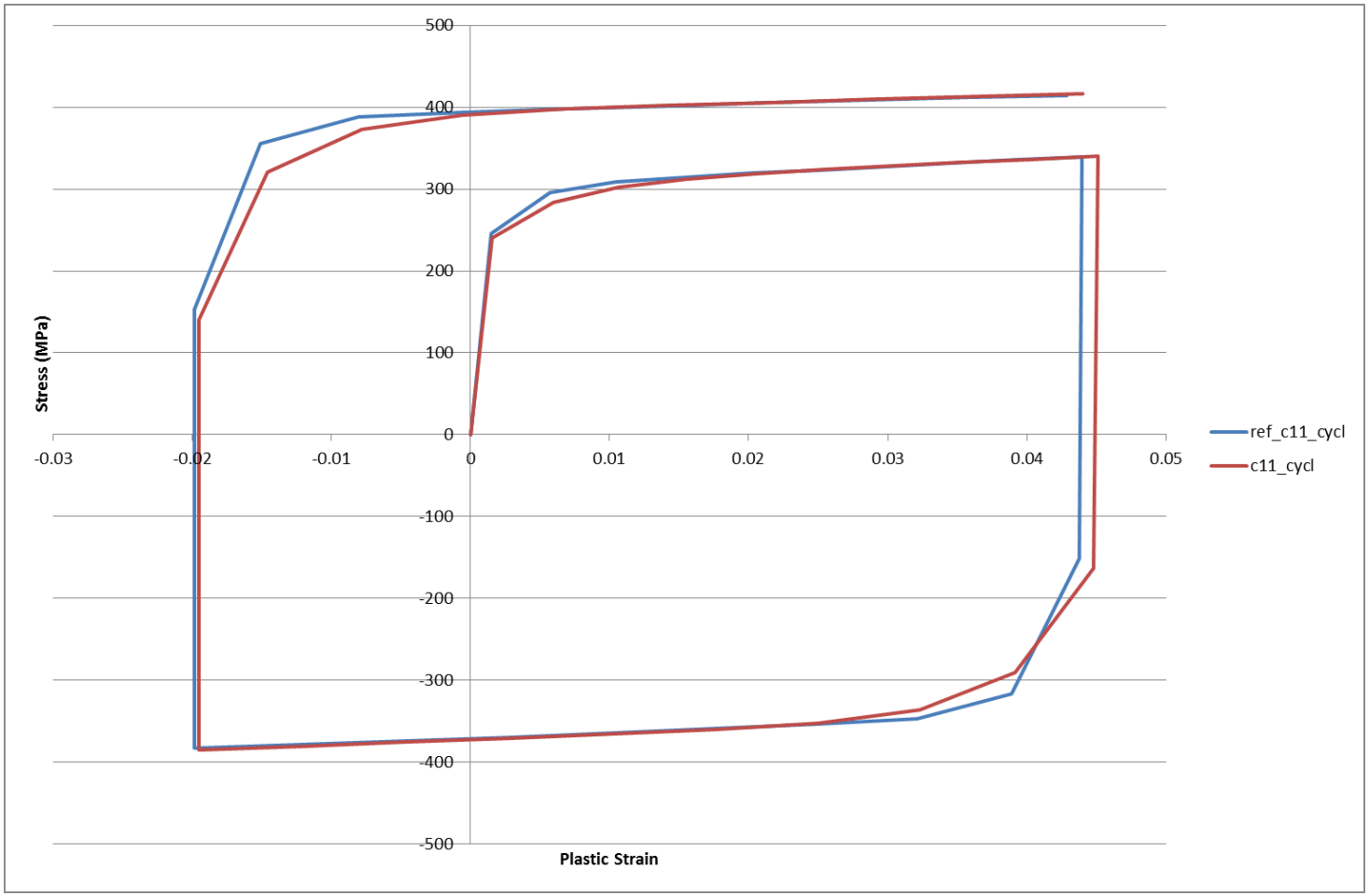


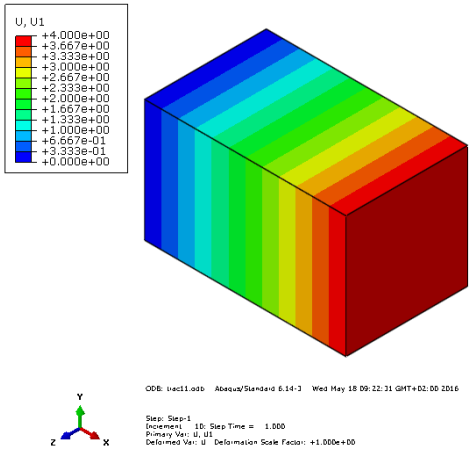
Validation of the MFRONT/ABAQUS Interface



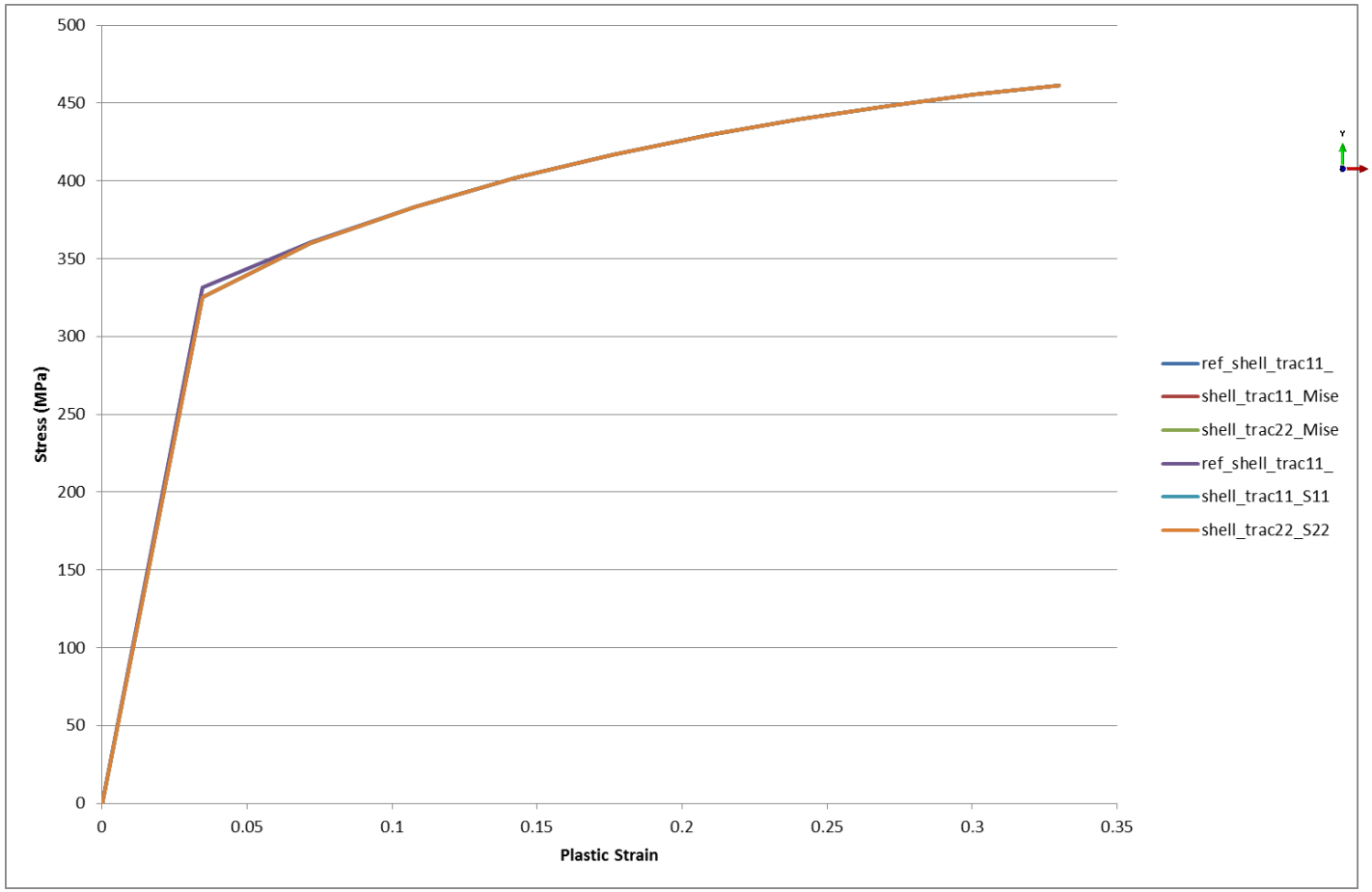


Validation of the MFRONT/ABAQUS Interface

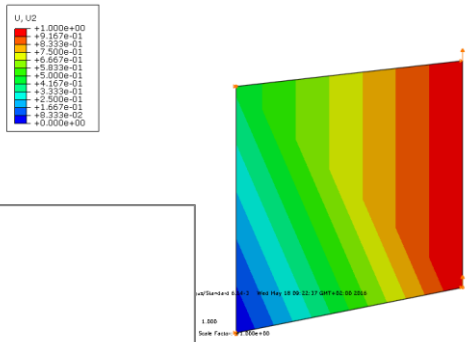


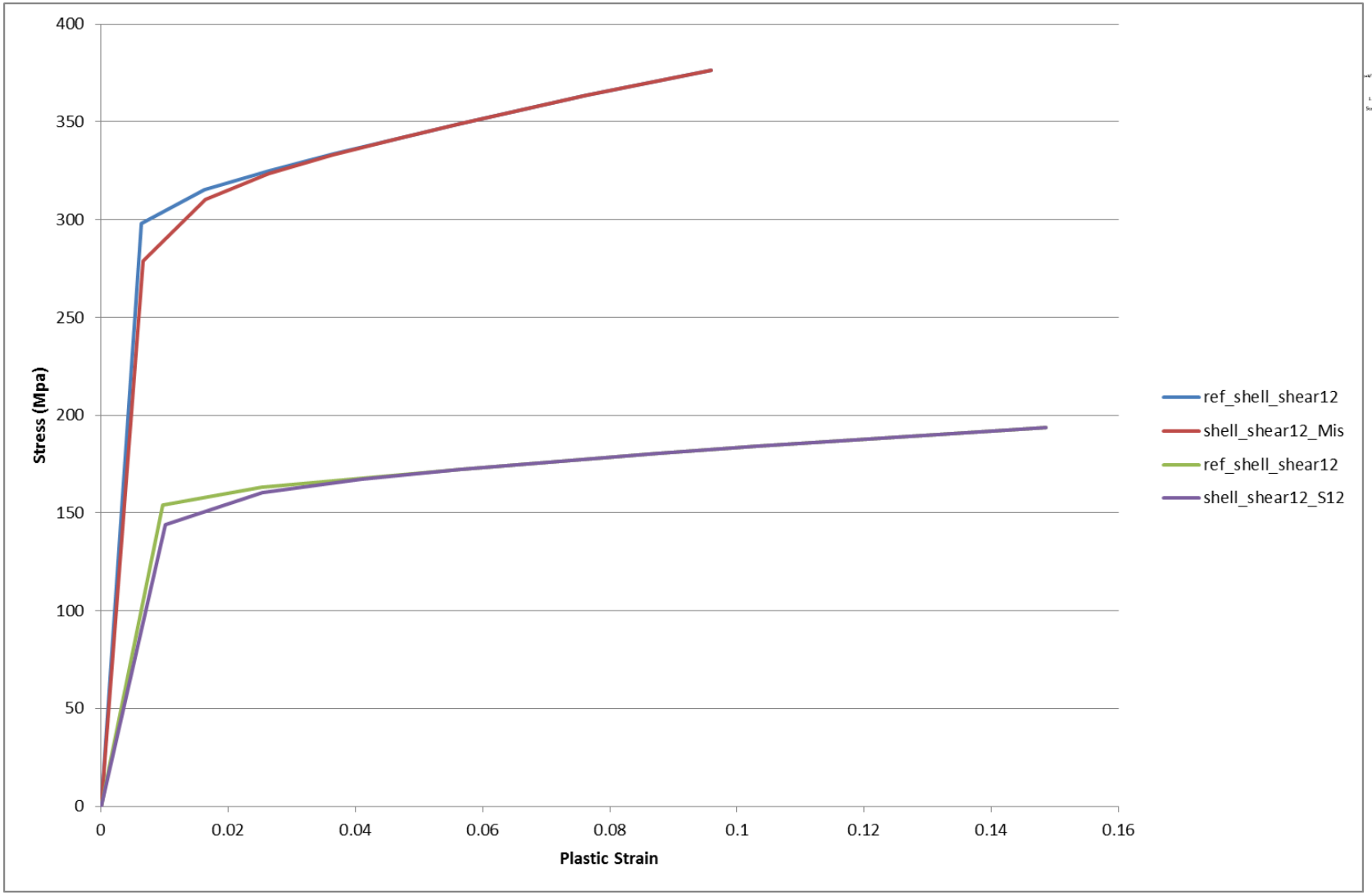


Validation of the MFRONT/ABAQUS Interface

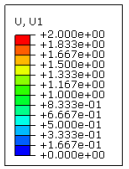


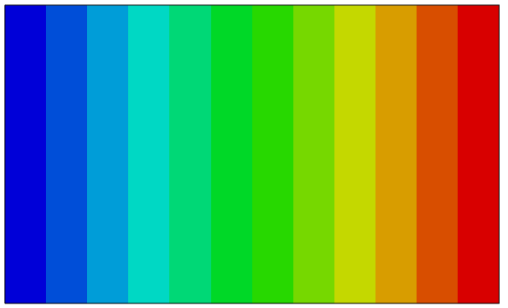
Validation of the MFRONT/ABAQUS Interface

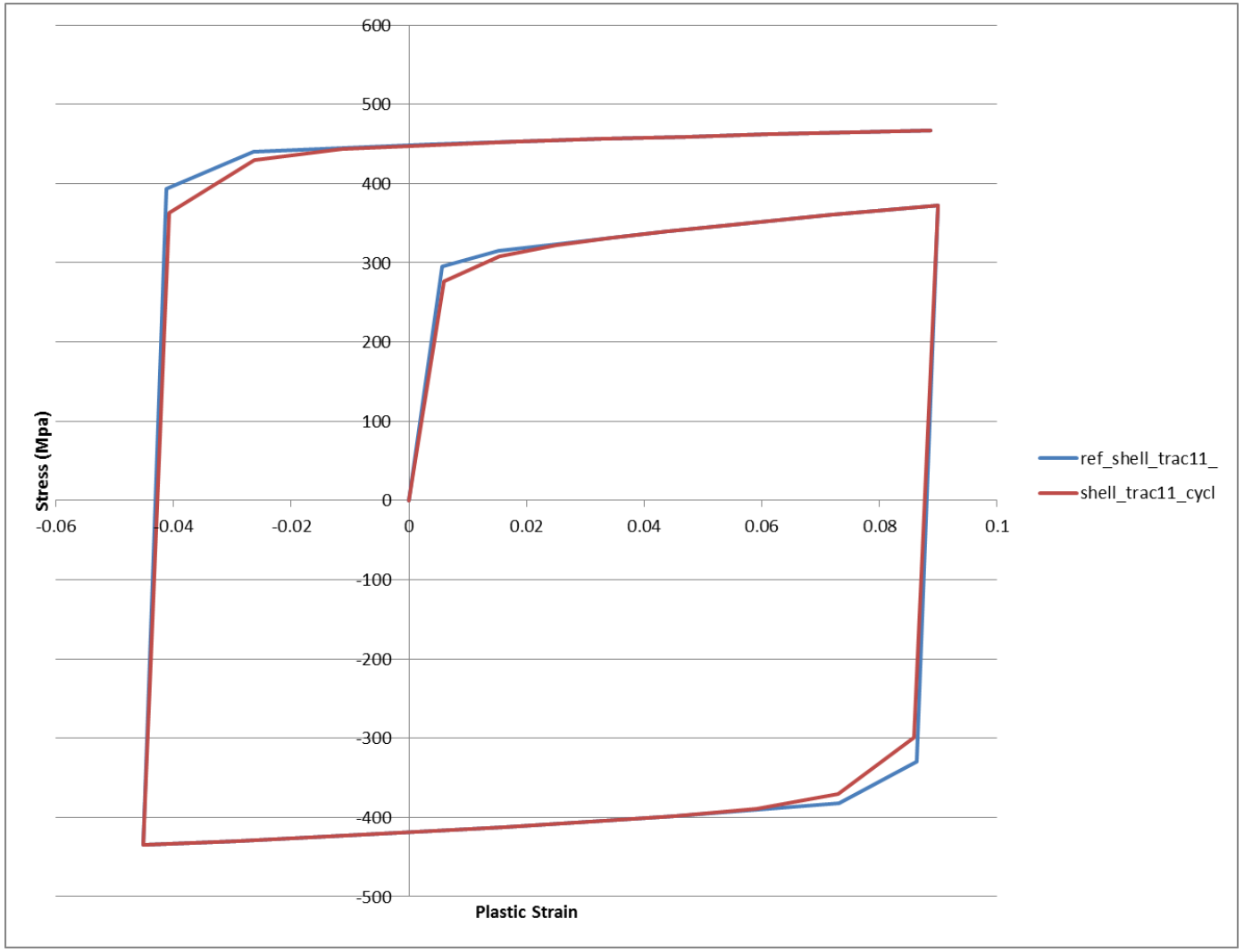




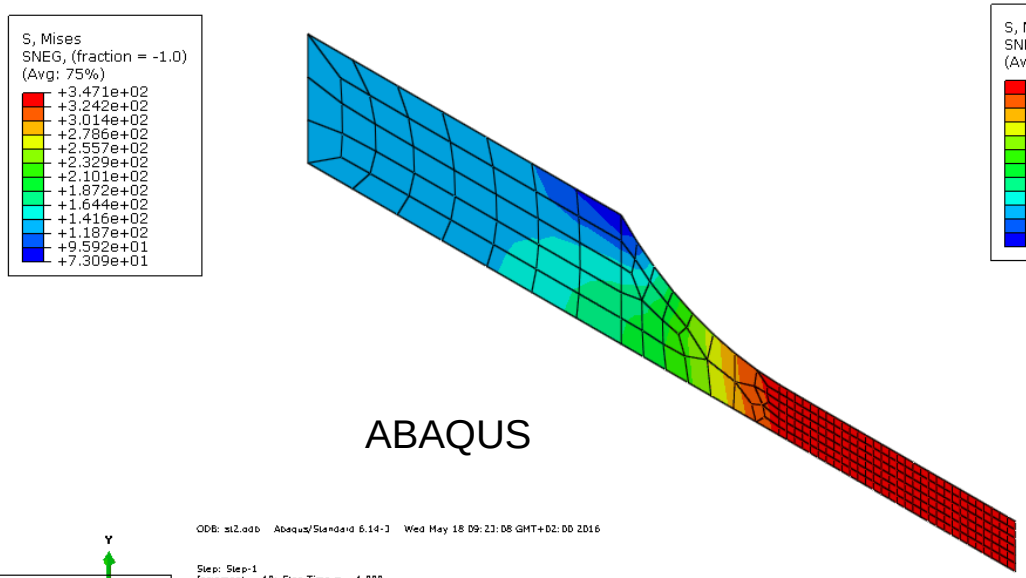
Validation of the MFRONT/ABAQUS Interface



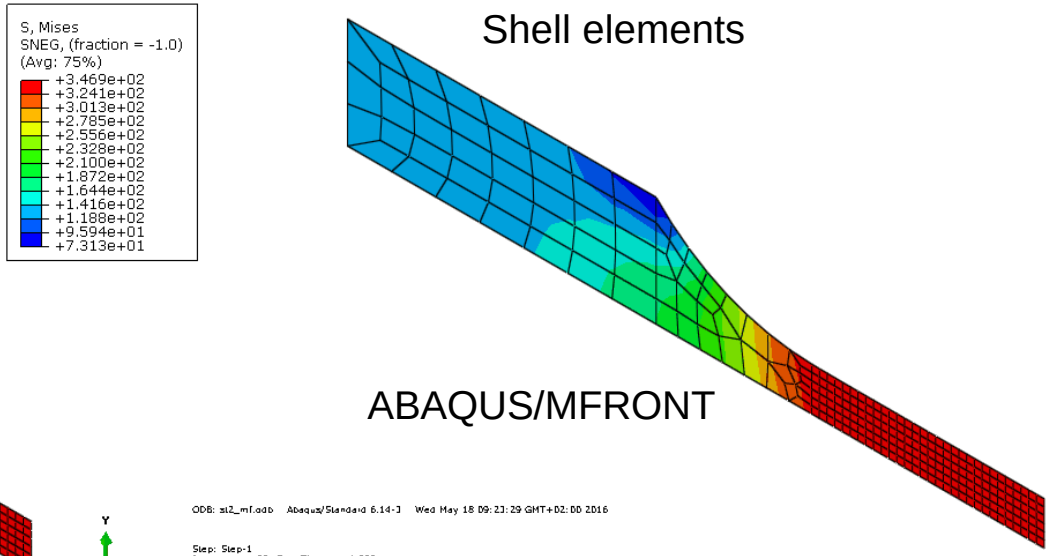




Validation of the MFRONT/ABAQUS Interface

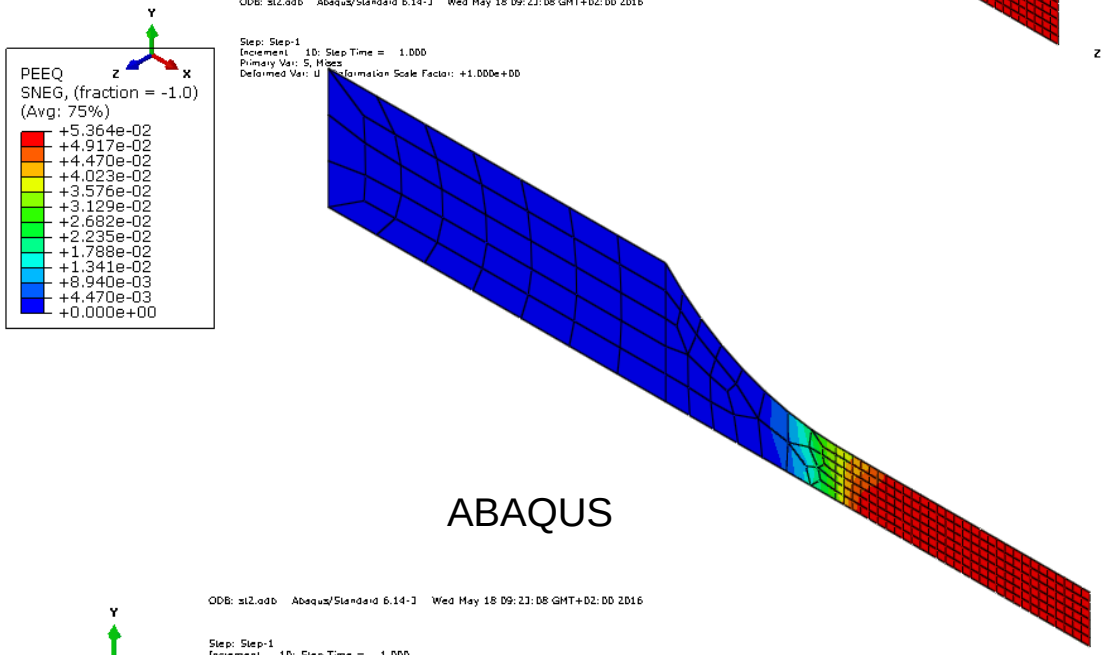


ABAQUS

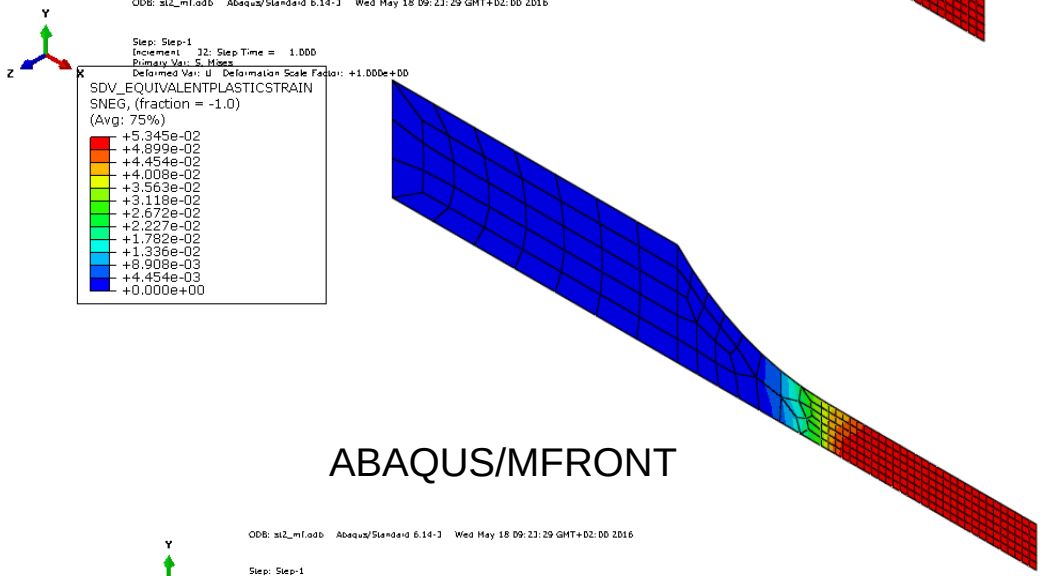


Shell elements

ABAQUS/MFRONT



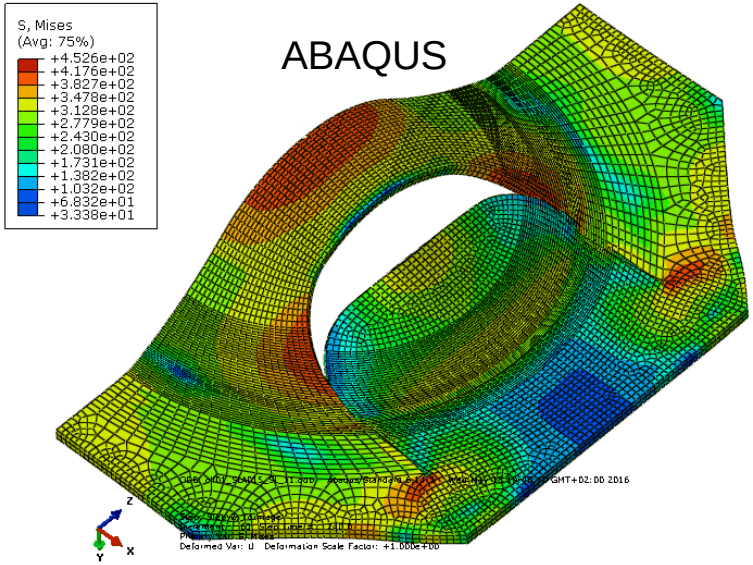
ABAQUS



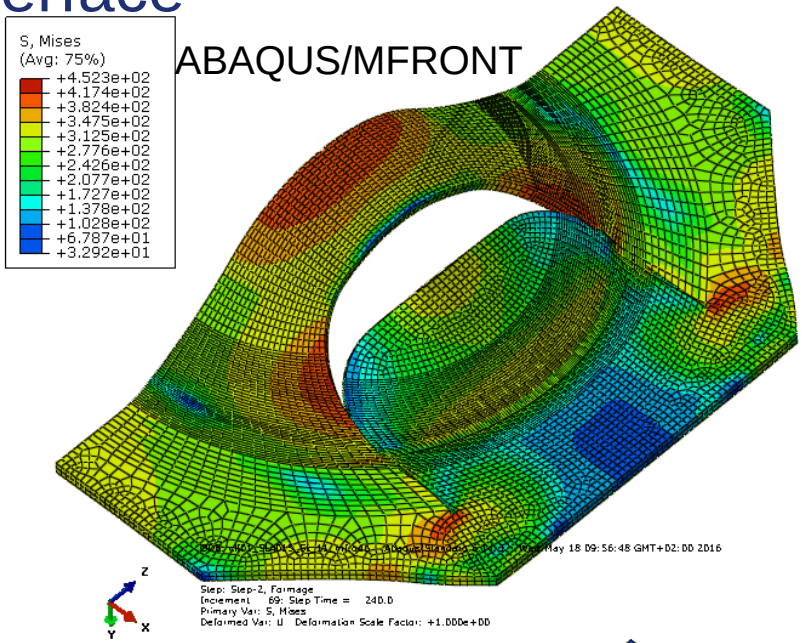
ABAQUS/MFRONT

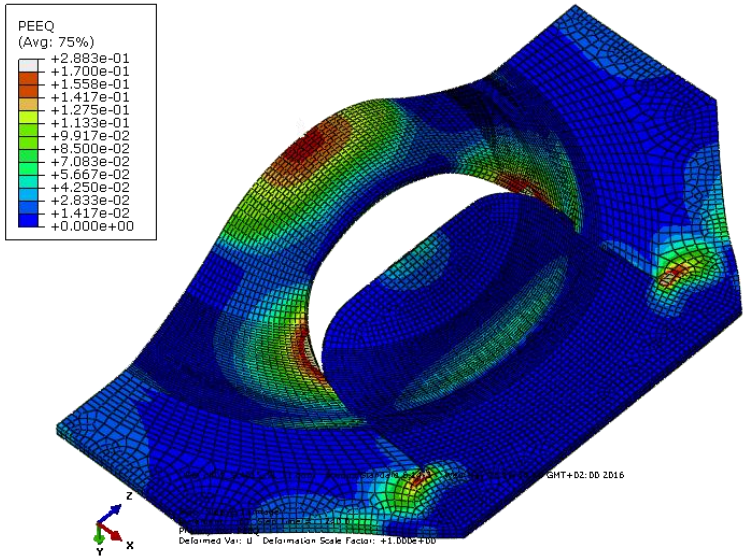
Validation of the MFRONT/ABAQUS Interface

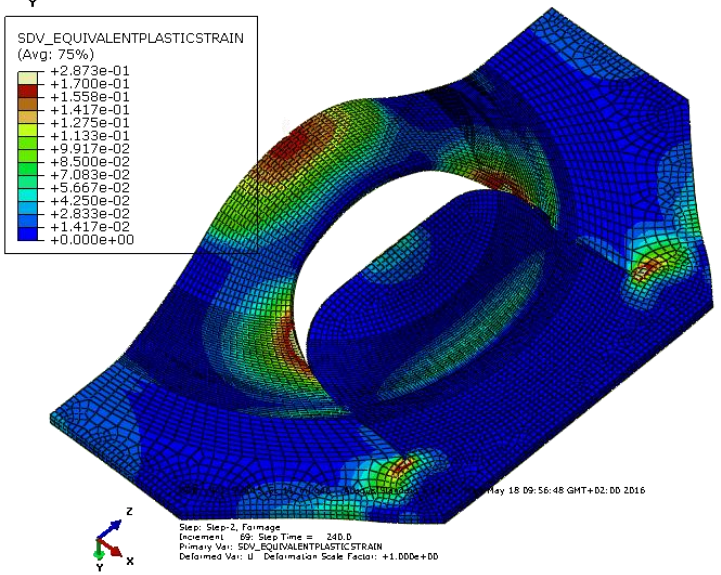
ABAQUS



ABAQUS/MFRONT





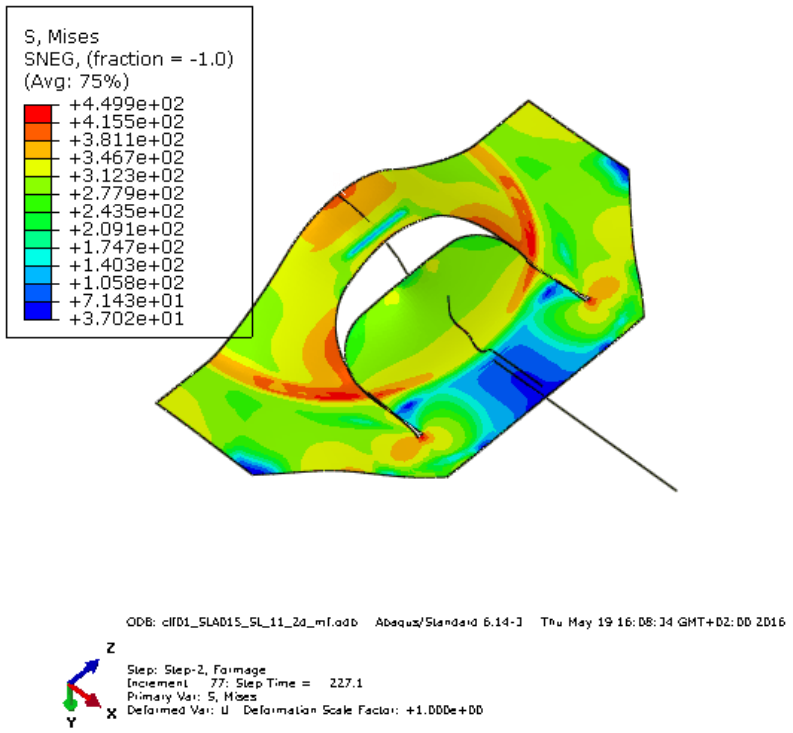


	CPU Time (s)	Number of increments
ABAQUS	4182.4	63
ABAQUS/MFRONT	5831.6	69

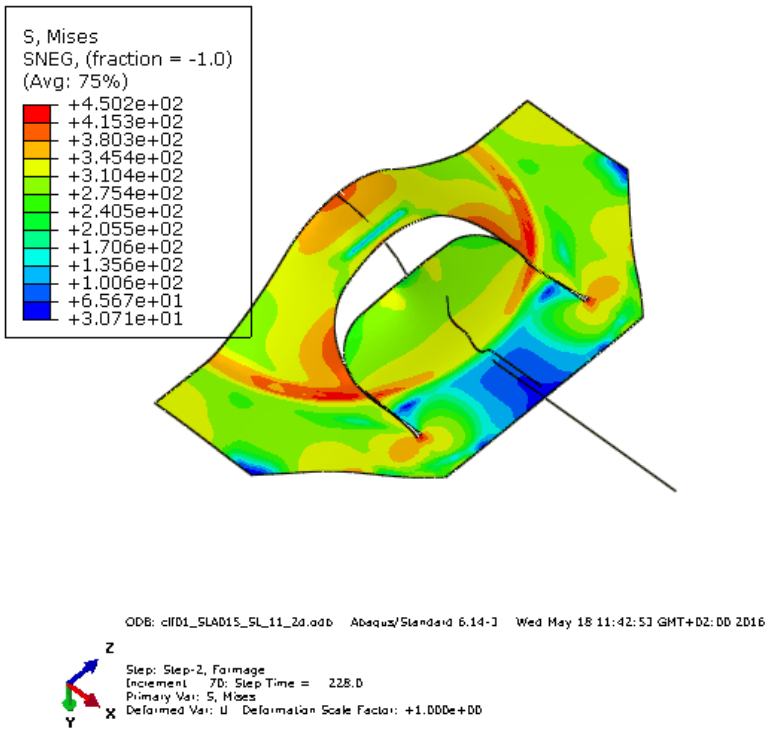
Validation of the MFRONT/ABAQUS Interface

Shell Elements

ABAQUS/MFRONT



ABAQUS



Summary

- MFRONT offers the possibility to implement material models more easily within ABAQUS
- The validation of the ABAQUS UMAT/MFRONT is well advanced but still a work-in progress

Perspectives

- The ABAQUS VUMAT/MFRONT is to be developped and validated.
- When available, both interfaces will give full potential and interest to MFRONT for ABAQUS
- The implementation of several material models is planned (anisotropic plasticity for metals and damage model for composite)
- Use of MTEST/MFRONT as a post-processor (within a Python script) to compute a fatigue criterion (Two-scale fatigue model)