DynELA Finite Element Code

v.3.0

Olivier PANTALE

vonMises
time: 8.000E-05

+1.002E+03
+9.386E+02
+8.749E+02
+8.112E+02
+7.474E+02
+6.837E+02
+6.199E+02
+5.562E+02
+4.924E+02
+4.287E+02
+3.650E+02
+3.012E+02
+2.375E+02
+1.737E+02
+1.100E+02
+4.627E+01
-1.747E+01

Contents

List of Figures

List of Tables

Notations

From a general point of view, it is usual to observe that one of the main difficulties in the field of mechanics, as in other fields, is the non-homogeneity of notations between the various authors. It is then easy to make completely incomprehensible the slightest theory when one decides to change notation. As the notion of universal notation is not yet valid (even if certain conventions can be assimilated to universal concepts), then we present below the set of notations used throughout this document and in a broader way in all the other documents in that series.

Notations Conventions

a	Scalar
$\vec{a}$	Vector
A	2nd order Tensor or matrix
$\mathcal{A}$	3rd order Tensor
$\mathbb{A}$	4th order Tensor

Linear Algebra and Mathematical Operators

$\vec{a} \cdot \vec{b}$	Dot product of the vectors $\vec{a}$ and $\vec{b}$
$\vec{a} \otimes \vec{b}$	Tensor (or Dyadic) product of the vectors $\vec{a}$ and $\vec{b}$
$\vec{a} \wedge \vec{b}$	Vectorial product of the vectors $\vec{a}$ and $\vec{b}$
A : B	Double contracted product of the two tensors A et B
$\dot{\square}$	Time derivative of quantity $\square$
$\ddot{\square}$	Second order time derivative of quantity $\square$
$\square_{,\square}$	Partial derivative of quantity $\square$ with respect to $\square$
$\square^T$	Transpose of a matrix or a vector $\square$
tr $\square$	Trace of a matrix or a tensor $\square$ (tr $\square = \sum \square_{ii}$)
dev $\square$	Deviatoric part of a tensor $\square$ (dev $\square = \square - \frac{1}{3}$ tr $\square\, 1$)
δ_{ij}	Kronecker delta identity
1	Unity matrix or second order tensor
$\mathbb{I}$	Unity fourth order tensor

Basic Continuum Mechanics

$\vec{x} = \begin{bmatrix} x & y & z \end{bmatrix}^T$	Coordinates in the physical domain
$\vec{u} = \begin{bmatrix} u & v & w \end{bmatrix}^T$	Displacement field

$\vec{\omega} = \begin{bmatrix} \omega_x & \omega_y & \omega_z \end{bmatrix}^T$ Rotation field

$\boldsymbol{\Omega}$ Arbitrary body in the current configuration

$\boldsymbol{\Gamma}$ Boundary of an arbitrary body $\boldsymbol{\Omega}$ in the current configuration

ρ

E Young's modulus of a material

ν Poisson's ratio of a material

K Bulk modulus of a material

λ Lamé's first parameter of a material

$\mu = G$ Lamé's second parameter / Coulomb's shear modulus

$\vec{F}$ External load vector

$\vec{f}$ External load vector

$\boldsymbol{\varepsilon}$ Green–Lagrange strain tensor

$\boldsymbol{\sigma}$ Cauchy stress tensor

s Deviatoric part of the Cauchy stress tensor

$\boldsymbol{\alpha}$ Backstress tensor

$\boldsymbol{\phi}$

Constitutive laws

f

n Direction of the plastic flow

q Heredity variables in an elastoplastic behavior

$\overline{\sigma}$ von Mises equivalent stress

$\overline{\varepsilon}^p$ Equivalent plastic strain

$\dot{\overline{\varepsilon}}^p$ Equivalent plastic strain rate

Λ Norm of the plastic strain

σ^v

σ^v_0

σ^v_∞

Large Deformations

$\vec{X} = \begin{bmatrix} X & Y & Z \end{bmatrix}^T$ Coordinates in the reference domain

E Green–Lagrange deformation tensor

F Deformation gradient tensor

U, V Right and left pure deformation tensors

R Rotation tensor

L Deformation speed tensor

D Symmetric part of the L tensor

W Skew-symmetric part of the L tensor

Finite Element Data Structures

N Shape functions matrix

$\vec{\xi} = \begin{bmatrix} \xi & \eta & \zeta \end{bmatrix}^T$ Coordinates in the parent domain

B Derivatives of the shape functions

$\square^e$ Quantity $\square$ related to element e

J Jacobian matrix

M	Mass matrix
K	Stiffness matrix
$\overline{\mathsf{F}}$	External surfacic load vector
F	External load vector
$\overline{\mathsf{f}}$	External volumic load vector
q	Nodal unknowns vector
n_g	Number of nodes of the current element
n_Q	Number of integration points of the current element

Chapter 1

Installation of the DynELA Finite Element Code

The DynELA Finite Element Code is an Explicit FEM code written in C++ using a Python's interface for creating the Finite Element Models. This is a new version of the early proposed v.2 code written between 1996 and 2010. The previous version has been included into the CAE Linux distribution some year ago and the corresponding work has been published in some Scientific Journals [1–7] , some international conferences [8–13] and served to support for some Ph.D. theses [14,15] . The aim of v.3.0 is to provide an enhanced version of the code with enhancements concerning the constitutive laws, a new programming interface based on Python 3 formalism, along with some documentation. The DynELA Finite Element Code is developped under Linux (an Ubuntu 18.04 LTS is currently used for the development). All source code and material can be downloaded from the following github website:

https://github.com/pantale/DynELA

The DynELA Finite Element Code is licensed under BSD-3-Clause license.

1.1 Prerequisites

Compilation of the DynELA Finite Element Code requires a number of libraries. Generation of Makefiles for DynELA compilation is based on the use of the CMake tool. CMake is a cross-platform, open-source build system generator. Under Ubuntu it can be installed with the following command:

```
sudo apt install cmake
```

DynELA is written in C++ and Python 3.x therefore it needs a C++ compiler and some Python 3.x libraries. Under Ubuntu those libraries can be installed with the following command:

```
sudo apt install build-essential swig zlib1g-dev liblapacke-dev python3-dev
```

It also needs some Python 3.x modules to run properly and at least numpy, matplotlib:

```
sudo apt install python3-numpy python3-matplotlib texlive dvipng \
texlive-latex-extra texlive-fonts-recommended
```

1.2   Download and compilation

Download of the source code from the github repository, compilation and installation of the software into a sub–directory can be done using the following procedure:

```
git clone https://github.com/pantale/DynELA.git
cd DynELA
mkdir Build
cd Build
cmake ../Sources
make
```

After downloading and compilation, there is no need to install the executable or something similar to use the FEM code. You just have to modify the .bashrc file and add the following lines where *path_to_DynELA* points to the top directory of your DynELA Finite Element Code installation:

```
export DYNELA="path_to_DynELA"
export PATH=$PATH:$DYNELA/bin
export DYNELA_BIN=$DYNELA/Build/bin
export DYNELA_LIB=$DYNELA/Build/lib
export PYTHONPATH="$DYNELA_BIN:$PYTHONPATH"
export PYTHONPATH="$DYNELA_LIB:$PYTHONPATH"
export LD_LIBRARY_PATH=$DYNELA_LIB:$LD_LIBRARY_PATH
```

1.3   Testing and usage

Testing of the installation can be done by running one of the provided samples. All samples of the DynELA Finite Element Code are located into the sub-directories of the Samples folder. Running a simulation is done using the following command in one of the Samples sub-directories:

```
python sample.py
```

Running the tests in the Samples directories can also be done with regard to the Makefiles contained in the Samples directories. Benchmark tests can be run from any sub-directory of the Sample folder using the following command:

```
make
```

The DynELA Finite Element Code now has a class for direct export of contourplot results using SVG vectorial format for a 2D or 3D mesh and time-history curves through the Python command interface. See the documentation for all instructions concerning SVG and time-history outputs and the examples included in the Samples directories.

The DynELA FEM code can generate VTK files for the results. I'm using the Paraview postprocessor to visualize those results. Paraview is available here:

https://www.paraview.org

Part I

DynELA FEM code theory

Part II

DynELA Programming

Chapter 1

DynELA programming language

This chapter deals about the DynELA Finite Element Code programming language. This language is based on Python 3 and all models must be described using this formalism. Therefore, this chapter will describe step by step how to build a Finite Element Model for the DynELA Finite Element Code, using the Python 3 language.

1.1   Introduction and basic knowledge

1.1.1   Calling the python interpreter

After the installation and compilation phase of the code[1], the DynELA Finite Element Code can be run using the following command:

```
python model.py
```

where **model.py** is the Python 3 source file defining the Finite Element Model. The **model.py** file contains the definition of the Finite Element Model using a Python 3 language and calls to specific DynELA methods written in C++.

[1]See the installation instructions in chapter 1 of the preamble, page 5

1.1.2   Formalism of a DynELA python file

To build a Finite Element Model, it is mandatory to import the **dnlPython** interpreter from the **.py** script. Conforming to this formalism, we give hereafter the minimal piece of Python code to set up a Finite Element Model in the DynELA Finite Element Code.

```python
1  #!/usr/bin/env python3
2  import dnlPython as dnl  # Imports the dnlPython library as dnl
3  model = dnl.DynELA()     # Creates the main Object
4  ...                      # Set of instructions to build the FE model
5  ...                      # conforming to the DynELA language and Python 3
6  model.solve()            # Runs the solver
7  ...                      # Set of instructions to postprocess the FE model
```

In the preceding piece of code, line 2 is used to load into the namespace dnl the **dnlPython** module containing the interface to all C++ methods of the DynELA Finite Element Code, based on the use of the SWIG Python interface. Therefore, all public methods of the DynELA Finite Element Code written in C++ can be called from the Python script to build the Finite Element Model, launch the solver, produce output results,…

In the proposed piece of code, line 3 is used to create an object of type **DynELA** (the higher object type in the DynELA Finite Element Code library) and instantiate it as the **model** object[2], while line 6, the solver of the DynELA Finite Element Code library is called to solve the problem and produce the results.

As the interpreter of the DynELA Finite Element Code is based on Python 3 language, we can use all instructions valid in Python 3 along with the specific DynELA instructions. In the rest of this documentation, we assume that the notions of programming in Python are mastered, and we will focus only on the functions specific to the DynELA Finite Element Code.

1.2   The Kernel library

#include "LogFile.h"

#include "MacAddress.h"

#include "Settings.h"

#include "String.h"

#include "System.h"

#include "Timer.h"

#include "Field.h"

1.3   The Maths library

#include "DiscreteFunction.h"

#include "DiscreteFunctionSet.h"

---

[2]For the rest of this chapter, we assume that the name of the instantiated **DynELA** object is **model**.

#include "Function.h"

#include "Matrices.h"

#include "Matrix.h"

#include "MatrixDiag.h"

#include "PolynomialFunction.h"

#include "RampFunction.h"

#include "SinusFunction.h"

#include "SymTensor2.h"

#include "Tensor2.h"

#include "Tensor3.h"

#include "Tensor4.h"

#include "Vec3D.h"

#include "Vector.h"

#include "ColorMap.h"

1.4 Model, Nodes and Elements

All Finite Element Models involves nodes and elements. The very first part of the model is therefore to create the nodes and the elements of the structure to set up a Finite Element Model. The DynELA Finite Element Code library doesn't include any meshing procedure yet, therefore, it is mandatory to create all elements and all nodes by hand or using Python loops in case it can be used. Another way is to use an external meshing program and convert the output of this program to produce the ad hoc lines of Python to describe the elements and the nodes of the model. This has been used many times by the author, and the Abaqus Finite Element code is an efficient way to create the mesh using the .inp text file generated by the CAE Abaqus program.

1.4.1 Model

Definition of a model in the DynELA Finite Element Code is done by creating an instance of the **DynELA** object into memory. This is done by calling the **dnlPython.DynELA()** method that returns an object of type **DynELA** as presented hereafter.

```
1 import dnlPython as dnl       # Imports the dnlPython library as dnl
2 model = dnl.DynELA("Taylor")  # Creates the main Object model named Taylor
```

In line 2 of the preceding piece of code, a reference name[3] **Taylor** is associated to the model object during creation. Once the model is created, one can then define all nodes, elements, materials, constitutive laws, boundary conditions,...

---

[3]The reference name is a string used to identify the object, this is completely optional but useful for debugging purposes for example as one can know the associated name to an object.

1.4.2   Nodes

1.4.2.1   Definition of the nodes

In the DynELA Finite Element Code, creation of nodes is done by calling the DynELA.createNode()
method. Therefore, a node is created by calling the **createNode()** method and giving the new node
number and the x, y and z coordinates of the new node as presented just below.

```
1  model.createNode(1, 0.0, 0.0, 0.0)  # Creates node 1, coordinates [0.0, 0.0, 0.0]
2  model.createNode(2, 1.0, 2.0, -1.0) # Creates node 2, coordinates [1.0, 2.0, -1.0]
```

An alternative method can be used if the coordinates of the node are already stored into a Vec3D
object as presented hereafter.

```
1  vect = dnl.Vec3D(1.0, 2.0, -1.0) # Creates a Vec3D object [1.0, 2.0, -1.0]
2  model.createNode(1, vect)        # Creates node 1 with coordinates vect
```

A check of the total number of nodes of the structure can be done using the DynELA.getNodesNumber()
method that returns the total number of nodes created.

1.4.2.2   Definition of the Nodes sets

Manipulation of nodes, application of boundaries conditions, etc,...is done through the definition of
nodes sets. Such nodes sets are used to group nodes under a **NodeSet** object for further use. A
NodeSet object contains a reference name and a list of nodes. Creation of a **NodeSet** is done using
the DynELA.NodeSet() method that returns an new **NodeSet** instance. The **NodeSet** can be named
during the creation by specifying its name as a string.

```
1  nset = dnl.NodeSet("NS_All")
```

When the **NodeSet** has been created, one can now define the list of nodes constituting the **NodeSet**
with the generic DynELA.add() method with the following formalism:

DynELA.add(nodeset, start, end, increment)

Hereafter is some self explaining examples to illustrate this process.

```
1  nset = dnl.NodeSet("NS_All")
2  model.add(nset, 2)       # Add node number 2 to node set
3  model.add(nset, 1, 4)    # Add nodes number 1-4 to node set
4  model.add(nset, 1, 4, 2) # Add nodes number 1 and 3 to node set
```

1.4.3   Elements

1.4.3.1   Definition of the elements

Creation of elements is done by calling the DynELA.createElement() method. An element is created
by calling the **createNode()** method and giving the new element number and the list of nodes defining

the element shape separated by comas and ordered thanks to the element definition as presented just hereafter.

DynELA.createElement(elementNumber, node1, node2,...)

Before creating the very first element of the structure, it is necessary to define the element shape using the DynELA.setDefaultElement() method. An example of element creation combining the two preceding methods is presented hereafter.

```
1  model.setDefaultElement(dnl.Element.ElQua4N2D) # Defines the default element
2  model.createElement(1, 1, 2, 3, 4)             # Creates element 1 with nodes 1,2,3,4
```

The following elements are available in the DynELA Finite Element Code.

ElQua4n2D : 4 nodes bi-linear 2D quadrilateral element.

ElQua4NAx : 4 nodes bi-linear axisymmetric quadrilateral element.

ElTri3N2D : 3 nodes 2D triangular element.

ElHex8N3D : 8 nodes 3D hexahedral element.

ElTet4N3D : 4 nodes 3D tetrahedral element.

ElTet10N3D : 10 nodes 3D tetrahedral element.

The total number of elements of the structure can be checked using the DynELA.getElementsNumber() method that returns the total number of elements created.

1.4.3.2   Definition of the Element sets

Declaration of materials, boundaries conditions, etc...is done through the definition of elements sets. Such elements sets are used to group elements under an **ElementSet** object for further use. An **ElementSet** object contains a reference to a name and a list of elements. Creation of an **ElementSet** is done using the DynELA.ElementSet() method that returns a new **ElementSet** instance. The **ElementSet** can be named during the creation by specifying its name as a string.

```
1  eset = dnl.ElementSet("ES_All")
```

When the **ElementSet** has been created, one can now define the list of elements constituting the **ElementSet** with the generic DynELA.add() method according to the following formalism:

DynELA.add(elementset, start, end, increment)

Hereafter is some self explaining examples to illustrate this process.

```
1  eset = dnl.ElementSet("ES_All")
2  model.add(eset, 2)        # Add element number 2 to element set
3  model.add(eset, 1, 4)     # Add elements number 1-4 to element set
4  model.add(eset, 1, 4, 2)  # Add elements number 1 and 3 to element set
```

1.4.4   Coordinates transformations

When the mesh has been created, it is always possible to modify the geometry of the structure by applying some geometrical operations such as translations, rotations and change of scale. Those operations apply on a **NodeSet**.

1.4.4.1 Translations

One can define a translation of the whole model or a part of the model by defining a translation vector (an instance of the DynELA Finite Element Code Vec3D) and apply this translation to the whole structure (without specifying the NodeSet) or a NodeSet using the DynELA.translate() method with the following syntax.

```
1  vector = dnl.Vec3D(1, 0, 0)    # Defines the translation vector
2  model.translate(vector)        # Translates the whole model along [1, 0, 0]
3  model.translate(vector, nset)  # Translates the NodeSet nset along [1, 0, 0]
```

1.4.4.2 Rotations

One can define a rotation of the whole model or a part of the model by defining a rotation vector (global axes $\vec{x}$, $\vec{y}$, $\vec{z}$ or an instance of the DynELA Finite Element Code Vec3D) and an angle α then apply this rotation to the whole structure (without specifying the NodeSet) or a NodeSet using the DynELA.rotate() method with the following syntax.

```
1  model.rotate('X', angle)          # Rotation of the whole structure around X
2  model.rotate('X', angle, nset)    # Rotation of NodeSet nset around X
3  axis = dnl.Vec3D(1.0, 1.0, 1.0)   # Defines the axis of rotation
4  model.rotate(axis, angle)         # Rotation of the whole structure around axis
5  model.rotate(axis, angle, nset)   # Rotation of NodeSet nset around axis
```

1.4.4.3 Scaling

One can define a scaling of the whole model or a part of the model by defining a scale factor or a scale vector (an instance of the DynELA Finite Element Code Vec3D) and apply this scaling operation to the whole structure (without specifying the NodeSet) or a NodeSet using the DynELA.scale() method with the following syntax.

```
1  model.scale(value)                # Scales the whole structure by factor value
2  model.scale(value, nset           # Scales the NodeSet nset by factor value
3  vec = dnl.Vec3D(2.0, 1.0, 1.0)    # Defines the scale vector
4  model.scale(vec)                  # Scales the whole structure by a factor of 2.0 on x
5  model.scale(vec, nset)            # Scales the NodeSet nset by a factor of 2.0 on x
```

1.5 Materials

1.5.1 Declaration of materials

1.5.1.1 Material declaration

Creation of a Material is done using the DynELA.Material() method. It is possible to give a name to a material during the creation process by specifying it through a string during the declaration. This can be used further.

```
1  # Creates the material
2  steel = dnl.Material("Steel")
```

1.5.1.2 General properties of materials

General properties of materials in DynELA Finite Element Code concerns the general constants such as Young's modulus, Poisson's ratio, density,... The complete list of parameters is reported in Table II.1.1. After creating an instance of the object **dnl.Material**, on can apply the prescribed values to all

Name	Symbol	Unit	description
youngModulus	E	MPa	Young modulus
poissonRatio	ν		Poisson ratio
density	ρ	kg/m^3	Density
heatCapacity	C_p	$J/°C$	Heat capacity
taylorQuinney	η		Taylor-Quinney coefficient
initialTemperature	T_0	$°C$	Initial temperature

Table II.1.1 – General properties of materials

those parameters using the following syntax.

```
1  # Creates the material
2  steel = dnl.Material("Steel")
3  # Apply all parameters
4  steel.youngModulus = 206e9
5  steel.poissonRatio = 0.3
6  steel.density = 7830
7  steel.heatCapacity = 46
8  steel.taylorQuinney = 0.9
9  steel.initialTemperature = 25
```

1.5.1.3 Material affectation to a set of elements

And, the material can be affected to the elements of the model by the **DynELA.add()** method as proposed hereafter.

```
1  # Creates the material
2  steel = dnl.Material("Steel")
3  # Apply all parameters
4  ...
5  # Affect the material to the element set eset
6  model.add(steel, eset)
```

1.5.2 Johnson–Cook constitutive law

The Johnson–Cook constitutive law is an hardening law defining the yield stress $\sigma^y(\bar{\varepsilon}^p, \dot{\bar{\varepsilon}}^p, T)$ by the following equation:

$$\sigma^y = \left(A + B\bar{\varepsilon}^{p^n}\right)\left[1 + C\ln\left(\frac{\dot{\bar{\varepsilon}}^p}{\dot{\bar{\varepsilon}}_0}\right)\right]\left[1 - \left(\frac{T - T_0}{T_m - T_0}\right)^m\right] \tag{1.1}$$

where $\dot{\bar{\varepsilon}}_0$ is the reference strain rate, T_0 and T_m are the reference temperature and the melting temperature of the material respectively and A, B, C, n and m are the five constitutive flow law parameters. Therefore, this kind of hardening law can be defined by using the following piece of code:

```
1  hardLaw = dnl.JohnsonCookLaw()                       # Hardening law
2  hardLaw.setParameters(A, B, C, n, m, depsp0, Tm, T0) # Parameters of the law
```

Once the hardening law has been created, one have to link this hardening law to a material already defined using the following piece of code:

```
1  # Creates the material
2  steel = dnl.Material("Steel")
3  # Creates the hardening law
4  hardLaw = dnl.JohnsonCookLaw()
5  # Attach hardening law to material
6  steel.setHardeningLaw(hardLaw)
```

1.6   Boundaries conditions

1.6.1   Restrain boundary condition

```
1  # Declaration of a boundary condition for top part
2  topBC = dnl.BoundaryRestrain('BC_top')
3  topBC.setValue(0, 1, 1)
4  model.attachConstantBC(topBC, topNS)
```

1.6.2   Amplitude

```
1  # Declaration of a ramp function to apply the load
2  ramp = dnl.RampFunction("constantFunction")
3  ramp.set(dnl.RampFunction.Constant, 0, stopTime)
```

1.6.3   Constant speed

```
1  # Declaration of a boundary condition for top part
2  topSpeed = dnl.BoundarySpeed()
3  topSpeed.setValue(displacement, 0, 0)
4  topSpeed.setFunction(ramp)
5  model.attachConstantBC(topSpeed, topNS)
```

1.6.4 Initial speed

```
1  # Declaration of a ramp function to apply the load
2  ramp = dnl.RampFunction("constantFunction")
3  ramp.set(dnl.RampFunction.Constant, 0, stopTime)
```

1.7 Fields

1.7.1 Nodal fields

Nodal fields are defined at nodes and cover types defined in table II.1.2. Concerning those fields, some of them are directely defined at nodes, some other are extrapolated from integration points and transfered to nodes as reported in column loc of table II.1.2. Concerning types, scalars, vec3D and tensors are available. Depending in the type of data, different methods can be used to acces those data:

scalar : Direct access to the value as it is unique.

vec3D : Access to all 3 components of a vec3D using nameX, nameY, nameZ or the norm of the vec3D using name.

tensor : Access to all 9 components of a tensor using nameXX, nameXY,…, nameZZ or the norm of the tensor using name.

1.7.2 Element fields

Element fields are defined at integration points and cover types defined in table II.1.3. Concerning types, scalars, vec3D and tensors are available. Depending in the type of data, different methods can be used to acces those data:

scalar : Direct access to the value as it is unique.

vec3D : Access to all 3 components of a vec3D using nameX, nameY, nameZ or the norm of the vec3D using name.

tensor : Access to all 9 components of a tensor using nameXX, nameXY,…, nameZZ or the norm of the tensor using name.

1.7.3 Global fields

1.8 Data Output during computation

1.8.1 VTK Data files

```
1  model.setSaveTimes(0, stopTime, stopTime/nbreSaves)
```

name	type	label	loc	description
density	scalar		IntPt	
displacementIncrement	vec3D		node	
displacement	vec3D		node	
energyIncrement	scalar		IntPt	
energy	scalar		IntPt	
gammaCumulate	scalar		IntPt	
gamma	scalar		IntPt	
internalEnergy	scalar		IntPt	
mass	scalar		node	
nodeCoordinate	vec3D		node	
normal	vec3D		node	
PlasticStrainInc	tensor		IntPt	
plasticStrainRate	scalar		IntPt	
plasticStrain	scalar		IntPt	
PlasticStrain	tensor		IntPt	
pressure	scalar		IntPt	
speedIncrement	vec3D		node	
speed	vec3D		node	
StrainInc	tensor		IntPt	
Strain	tensor		IntPt	
Stress	tensor		IntPt	
temperature	scalar		IntPt	
vonMises	scalar		IntPt	
yieldStress	scalar		IntPt	

Table II.1.2 – *Nodal fields*

1.8.2   History files

```
1  dtHist = dnl.HistoryFile("dtHistory")
2  dtHist.setFileName(dnl.String("dt.plot"))
3  dtHist.add(dnl.Field.timeStep)
4  dtHist.setSaveTime(stopTime/nbrePoints)
5  model.add(dtHist)
```

1.9   Solvers

```
1  # Declaration of the explicit solver
2  solver = dnl.Explicit("Solver")
3  solver.setTimes(0, stopTime)
4  solver.setTimeStepSafetyFactor(1.0)
5  model.add(solver)
```

name	type	label	description
density	scalar		
gammaCumulate	scalar		
gamma	scalar		
internalEnergy	scalar		
plasticStrainRate	scalar		
plasticStrain	scalar		
PlasticStrain	tensor		
PlaticStrainInc	tensor		
pressure	scalar		
StrainInc	tensor		
Strain	tensor		
Stress	tensor		
temperature	scalar		
vonMises	scalar		
yieldStress	scalar		

Table II.1.3 – Element fields

name	type	label	description
kineticEnergy	scalar		
realTimeStep	scalar		
timeStep	scalar		

Table II.1.4 – Global fields

1.9.1 Parallel solver

```
1  # Parallel solver with two cores
2  model.parallel.setCores(2)
```

1.9.2 Solving procedure

```
1  # Run the main solver
2  model.solve()
```

1.10 Vectorial SVG contourplots

<div align="right">

Chapter 2

</div>

Curves utility

This chapter deals about the curves utility of the DynELA Finite Element Code. This curve utility is a Python library used to produce plots (vector or bitmap plots) from DynELA Finite Element Code history files by reading a configuration file describing the way to produce those plots from a simple syntax language. The core of the library uses the matplotlib library implemented using the Python3 language. With this curve utility library, it is therefore possible to produce output graphics and curves directly from the python model file at the end of the solve.

2.1   Introduction and presentation of the script

2.1.1   Usage of the Python script

The Curve library is called from a Python script with the following minimal form.

```python
import dnlCurves                   # Import the dnlCurve library
curves = dnlCurves.Curves()        # Creates an instance of the Curve object
curves.plotFile('Curves.ex')       # Plot the curves defined by the 'Curves.ex' file
```

2.1.2   Datafile format

The datafile format is the plot file used by DynELA Finite Element Code. This file is a text file with $\vec{x}$ and $\vec{y}$ datas defined by two or more series of floating point data on two or more columns separated by spaces. This file should contain two header lines on top and should conform to the following for a datafile containing one data.

```
1  #DynELA_plot history file
2  #plotted :timeStep
3  1.0596474E-06  1.0596474E-06
4  1.0058495E-04  1.0572496E-06
5  \ldots\ldots
6  1.0000000E-02  1.4838518E-07
```

In this file, the very first line is ignored by the script, while the second one is used to define the default name of the variable to be plotted (here 'timeStep' in this case). Here after is a example of a datafile containing more columns.

```
1  #DynELA_plot history file
2  #plotted :s11 s22 s33
3  1.0596474E-06  1.0596474E+06  2.0596474E+06  -1.0596474E+06
4  1.0058495E-04  1.0572496E+06  2.0572496E+06  -1.0572496E+06
5  \ldots\ldots
6  1.0000000E-02  1.4838518E+07  2.4838518E+07  -1.4838518E+07
```

In this case, one can specify which columns will be used for plotting the curves as it will be presented just later.

2.2   Configuration file to define the plots

A configuration file is used to define the plots and is read by the **curves.plotFile**() method of the Curves object. Global parameters are defined using the keyword **Parameters** at the beginning of a line of the parameter file. The keyword parameter is followed by a set of commands used to set some global parameters for all the subsequent generated graphs by overwriting the default parameters, before generating the very first graph.

```
1  Parameters, xname=Displacement x, crop=True
```

Then all parameters can also be altered within the definition of a plot. A definition of a plot is done by a line without the **Parameters** keyword. The first element on this line is the name of the generated graph, followed by a set of parameters defining the generated plot.

```
1  # Plot of Temp.plot to Temp.svg file
2  Temp, name=$T$, Temp.plot
3  # Plot of column 1 of Stress.plot file to S11.svg file
4  S11, name=$\sigma_{xx}$, legendlocate=topleft, Stress.plot[0:1]
5  # Plot of column 2 of Stress.plot file to S22.svg file
6  S22, name=$\sigma_{yy}$, legendlocate=bottomleft, Stress.plot[0:2]
7  # Plot of dt.plot and Abaqus/Step.plot files to TimeStep.svg file
8  TimeStep, name=$\Delta t$, dt.plot, name=$Abaqus \Delta t$, Abaqus/Step.plot
```

In the proposed example we have the following.

- Line 2: plots the content of the Temp.plot file with legend T in a Temp.svg file using default parameters.

- Line 4: plots the content of the Stress.plot file with legend σ_{xx} in a S11.svg file using data from columns 0 for $\vec{x}$ and from column 1 for $\vec{y}$ and with a legend located in the top left part of the figure.

- Line 6: plots the content of the Stress.plot file with legend σ_{yy} in a S22.svg file using data from columns 0 for $\vec{x}$ and from column 2 for $\vec{y}$ with a legend located in the bottom left part of the figure.

- Line 8: plots the content of the dt.plot file with legend Δt and Abaqus/Step.plot file with legend $Abaqus\,\Delta t$ in a TimeStep.svg file.

As presented just before, LaTeX can be used for names and legends by using the escape character $ and remembering that space character has to be defined by the combination of an escape '\' character followed by the space ' ' so that '\ '.

Comments can be inserted as presented briefly earlier by using the '#' character. Everything after the '#' character and up to the end of the line is ignored and treated as comment. Blank lines are also allowed in the file to help human readability of the file.

2.2.1   Global parameters for a graph

outputformat = <'format'> : is used to define the format of the output file, *i.e.* pdf, svg, png,... (default value is svg file).

transparent = <True, False> : is used to define if the background should be or not transparent (default value is False).

crop = <True, False> : is used to define if the output image should be or not cropped (default value is False).

grid = <True, False> : is used to define if the output graph should contain or not a grid (default value is True).

title = <'name'> : defines the title of the graph on the top part of the plot (default value 'Default title of the graph')

xname, yname = <'name'> : defines the names of the $\vec{x}$ and $\vec{y}$ axis for the plot (default value are 'x-axis' and 'y-axis').

titlefontsize = <size> : defines the font size of the title on the top part of the plot (default value is 20).

xfontsize, yfontsize = <size> : defines the font size of the $\vec{x}$ and $\vec{y}$ axis names for the plot (default value is 20).

xlabelfontsize, ylabelfontsize = <size> : defines the font size of the values along the $\vec{x}$ and $\vec{y}$ axis for the plot (default value is 16).

xrange, yrange = <range> : remaps the range of the curves for the $\vec{x}$ and $\vec{y}$ axis to the given value. For example, setting **xrange=10** will remap the x-data within the range $[0, 10]$.

xscale, yscale = <range> : multiply the $\vec{x}$ or the $\vec{y}$ data by a given factor. For example, setting **xscale=10** will multiply all x-data by a factor of 10.

2.2.2   Plotting curve parameters

name = <'name'> : defines a new name for the next curve in the legend and overrides the one defined by the datafile.

removename = <'name'> : removes from the names of the curves a given pattern for the legends. Example: **removename** = –S11 will convert 'data-S11' into 'data'. This is not a very useful method and one should prefer the **name** command to redefine the name of the plotting curve.

linewidth = <'width'> : defines the line width for all subsequent plots (default value is 2).

marks = <'symbol'> : is used to define the next marker to use for all subsequent plots (default value is "). If the value is void, markers are cycled through the default markers list.

marksnumber = <number> : defines the total number of markers on the curves for all subsequent plots (default value is 20).

markersize = <size> : defines the size of the marker symbols to use for all subsequent plots (default value is 10).

2.2.3 Legend definition

legendcolumns = <columns> : defines the number of columns to use for the legend (default value is 1).

legendshadow = <True, False> : is used to define if the output graph legend should contain or not a shadow box (default value is True).

legendanchor = <'position'> : defines the position of the legend anchor independently from the legend position parameter.

legendposition = <'position'> : defines the position of the legend position independently from the legend anchor parameter.

legendlocate = <'position'> : defines the position of the legend with the following four options: 'topleft', 'topright', 'bottomleft' and 'bottomright' (default value is 'topright').

legendfontsize = <size> : defines the font size of the text in the legend (default value is 16).

Chapter 3

Abaqus extractor utility

This chapter deals about the Abaqus extractor utility of the DynELA Finite Element Code. This extractor is a Python library used to extract results from an Abaqus **odb** datafile by reading a configuration file describing the way to produce those extracts from a simple syntax language.

3.1 Introduction and presentation of the script

The **AbaqusExtract** library is called from a Python script with the following minimal form.

```
1  # import the AbaqusExtract library
2  import AbaqusExtract
3  # Extract data using datafile Extract.ex
4  AbaqusExtract.AbaqusExtract('Extract.ex')
```

Assuming that the Python script containing this piece of code is named '**Extract.py**', therefore this script must be called from the Abaqus main program using the following command:

```
1  abaqus python Extract.py
```

3.2 Syntax of the configuration file

Different forms are used depending on the nature of the data to extract, but everything conforms to the Abaqus Python command syntax. The **AbaqusExtract** library uses some keywords to define the nature of the data to extract as proposed here after.

TimeHistory : is a keyword defining that the line contains the instructions to extract a time history from an Abaqus **odb** file.

job : is used to define the name of the odb file to use for the extraction of the results *i.e.* it's the name of the **odb** file without the extension **.odb**.

value : is the nature of the variable to extract conforming to the variables names of Abaqus.

name : is the given name of the plotting file to generate, *i.e.* the name of the plotting file without the **.plot** extension.

region : is used to define the location of the data to extract. This one is more complex, as it can be a global location, a node location or an integration point location as described hereafter.

operate : is an optional parameter defining what to do when multiple regions are defined.

As an example, the following piece of code gives some application examples.

```
1  # Extraction of a time history node data
2  TimeHistory, job=Torus, value=U1, name=dispX, region=Node PART-1.3
3  # Extraction of a time history integration point data
4  TimeHistory, job=Torus, value=MISES, name=vM, region=Element PART-1.25 Int Point 1
5  # Extraction of a time history global variable
6  TimeHistory, job=Torus, value=DT, name=timeStep, region=Assembly ASSEMBLY
```

In the previous example:

- line 2 is used to extract the nodal displacement **U1** from the odb file **Torus.odb** and produce the **dispX.plot** file for node 3 of the **PART-1** piece.

- line 4 is used to extract the integration point value **MISES** from the odb file **Torus.odb** and produce the **vM.plot** file for integration point 1 of element 25 of the **PART-1** piece.

- line 6 is used to extract the global timestep **DT** from the odb file **Torus.odb** and produce the **timeStep.plot** file.

Concerning the definition of the region to use, it is possible to define more than 1 region by using the '+' sign in the definition of the region. When this is used, the optional parameter operate is used to define what to do with this multiple data. The operation is defined by:

operate = none : or no operate parameter defined, therefore, all values are reported to the plotting file separated by a white space.

operate = mean : the mean value is computed and reported to the plotting file.

operate = sum : the sum of the values is computed and reported= to the plotting file.

Part III

DynELA Samples

Chapter 1

DynELA single element sample cases

This chapter deals with some numerical applications of the DynELA Finite Element Code for dynamic applications in 2D, axi–symmetric and 3D cases. In the subsequent tests, if not specified, a Johnson-Cook constitutive law is used to model the behavior of the material. The Johnson-Cook hardening flow law is probably the most widely used flow law for the simulation of high strain rate deformation processes taking into account plastic strain, plastic strain rate and temperature effects. Since a lot of efforts have been made in the past to identify the constitutive flow law parameters for many materials, it is implemented in numerous Finite Element codes such as Abaqus [?]. The general formulation of the Johnson-Cook law $\sigma^y(\overline{\varepsilon}^p, \dot{\overline{\varepsilon}}^p, T)$ is given by the following equation:

$$\sigma^y = \left(A + B\overline{\varepsilon}^{p^n}\right) \left[1 + C\ln\left(\frac{\dot{\overline{\varepsilon}}^p}{\dot{\overline{\varepsilon}}_0}\right)\right] \left[1 - \left(\frac{T - T_0}{T_m - T_0}\right)^m\right] \tag{1.1}$$

where $\dot{\overline{\varepsilon}}_0$ is the reference strain rate, T_0 and T_m are the reference temperature and the melting temperature of the material respectively and A, B, C, n and m are the five constitutive flow law parameters. A 42CrMo4 steel following the Johnson-Cook behavior law has been selected for all those tests, and material properties are reported in Table III.3.1.

E (Gpa)	ν	A (MPa)	B (MPa)	C	n	m
206.9	0.3	806	614	0.0089	0.168	1.1
ρ (kg/m^3)	λ $(W/m°C)$	C_p $(J/Kg°C)$	η	$\dot{\overline{\varepsilon}}_0$ (s^{-1})	T_0 $(°C)$	T_m $(°C)$
7830	34.0	460	0.9	1.0	20	1540

Table III.1.1 – *Material parameters of the Johnson–Cook behavior for the numerical tests*

1.1 Uniaxial tensile tests

1.1.1 Element plane tensile test

The uniaxial one element tensile test is a numerical test where an plane element (with a square prescribed shape) is subjected to pure tensile as presented in figure III.1.1. The initial shape of
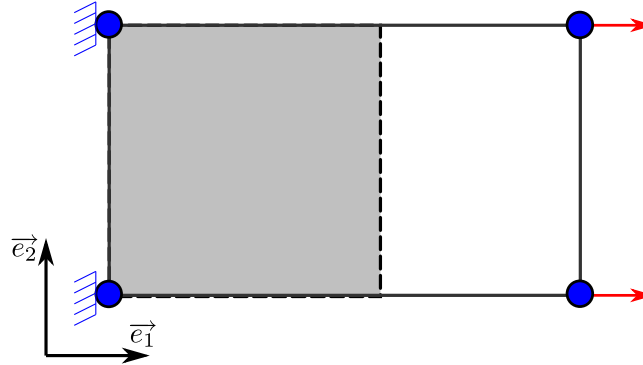
Figure III.1.1 – *Numerical model for the one element tensile test*

the specimen is $10\,mm \times 10\,mm$, the two left nodes of the element are encastred and a prescribed horizontal displacement $d = 10\,mm$ is applied on the two right nodes of the same element as illustrated in Figure III.1.1. As we are using an explicit integration scheme, the total simulation time is set to $t = 0.01\,s$.

All the properties of the constitutive law reported in Table III.3.1 are used and the material is assumed to follow the Johnson-Cook behavior described by equation 3.1.

Figure III.1.2 shows the comparison of the DynELA solver results (plotted in red) and the Abaqus numerical results (plotted in blue) concerning the evolution of the stress components σ_{11}, σ_{22}, σ_{12}, $\overline{\sigma}$, $\overline{\varepsilon}^p$ and T *vs.* the horizontal displacement of the right edge of the specimen along the horizontal axis. For the Abaqus model, the exact same mesh has been used. Comparison of Abaqus and DynELA results is made by averaging the DynELA results on the 4 integration points of the element. This has been done because DynELA Finite Element Code uses full integrated elements while Abaqus have reduced integrated elements only but in fact, the DynELA results at the 4 integration points are the same in this benchmark test.
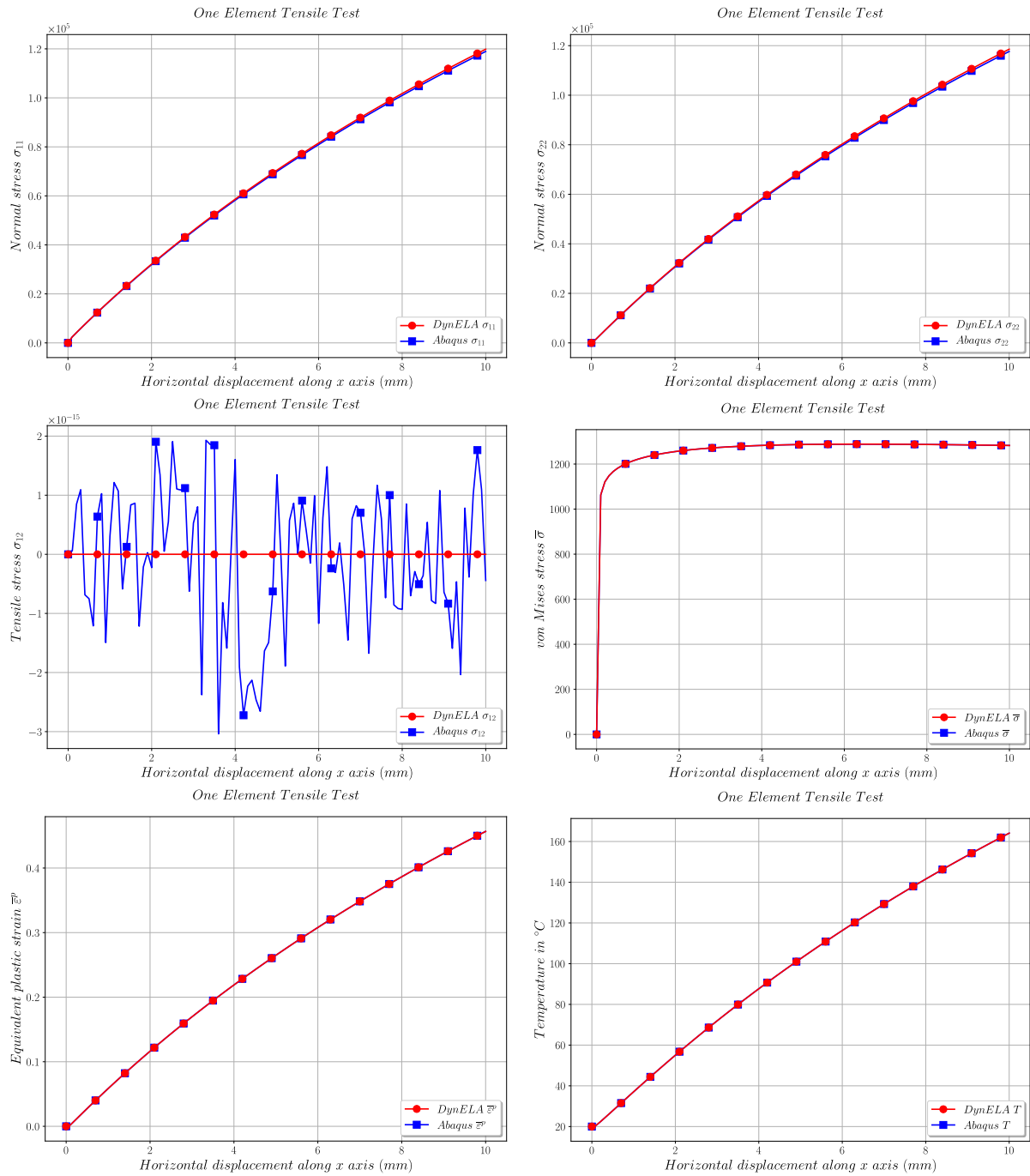
Figure III.1.2 – *Comparison of numerical and analytical results for the one element tensile test*

1.1.2   Element 3D tensile test

The uniaxial one element 3D tensile test is a numerical test where a 3D brick element (with a cubic prescribed shape) is subjected to radial tensile as presented in figure III.1.3. No boundary condition has been prescribed on the $\vec{z}$ direction, so that the reduction of the width during the test can occur. The initial shape of the specimen is $10\,mm \times 10\,mm \times 10\,mm$ and the the two left nodes of the element are restrained for their displacements along the $\vec{x}$ and $\vec{y}$ directions and a prescribed horizontal displacement $d = 10\,mm$ is applied on the two right nodes of the same element as illustrated in Figure III.1.3. As we are using an explicit integration scheme, the total simulation time is set to $t = 0.01\,s$.
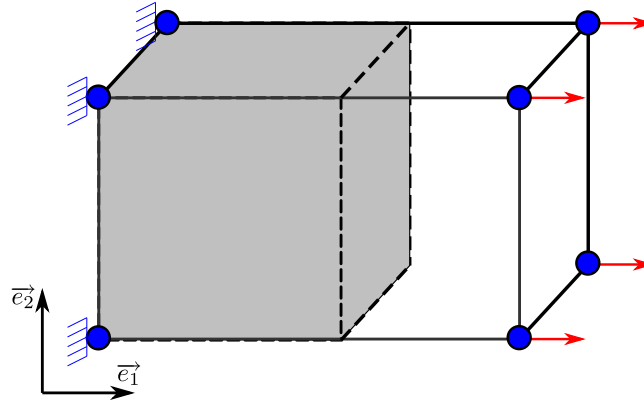

Figure III.1.3 – Numerical model for the one element tensile test

All the properties of the constitutive law reported in Table III.3.1 are used and the material is assumed to follow the Johnson–Cook behavior described by equation 3.1.

Figure III.1.4 shows the comparison of the DynELA solver results (plotted in red) and the Abaqus numerical results (plotted in blue) concerning the evolution of the stress components σ_{11}, σ_{22}, σ_{12}, $\overline{\sigma}$, $\overline{\varepsilon}^p$ and T *vs.* the horizontal displacement of the right edge of the specimen along the horizontal axis. For the Abaqus model, the exact same mesh has been used. Comparison of Abaqus and DynELA results is made by averaging the DynELA and the Abaqus results on the 8 integration points of the element.

1.1.3   Element radial tensile test

The uniaxial one element radial tensile test is a numerical test where an axisymmetric element (with a square prescribed shape) is subjected to pure radial tensile as presented in figure III.1.1. The initial shape of the specimen is $10\,mm \times 10\,mm$ and the the two left nodes of the element are encastred and a prescribed horizontal displacement $d = 10\,mm$ is applied on the two right nodes of the same element as illustrated in Figure III.1.1. As we are using an explicit integration scheme, the total simulation time is set to $t = 0.01\,s$.

All the properties of the constitutive law reported in Table III.3.1 are used and the material is assumed to follow the Johnson–Cook behavior described by equation 3.1.

Figure III.1.5 shows the comparison of the DynELA solver results (plotted in red) and the Abaqus numerical results (plotted in blue) concerning the evolution of the stress components σ_{11}, σ_{22}, σ_{12}, $\overline{\sigma}$, $\overline{\varepsilon}^p$ and T *vs.* the horizontal displacement of the right edge of the specimen along the horizontal axis. Again, comparison of Abaqus and DynELA results is made by averaging the DynELA results on the 4 integration points of the element because DynELA Finite Element Code uses full integrated

elements while Abaqus have reduced integrated elements only but in fact, the DynELA results at the 4 integration points are the same in this benchmark test.
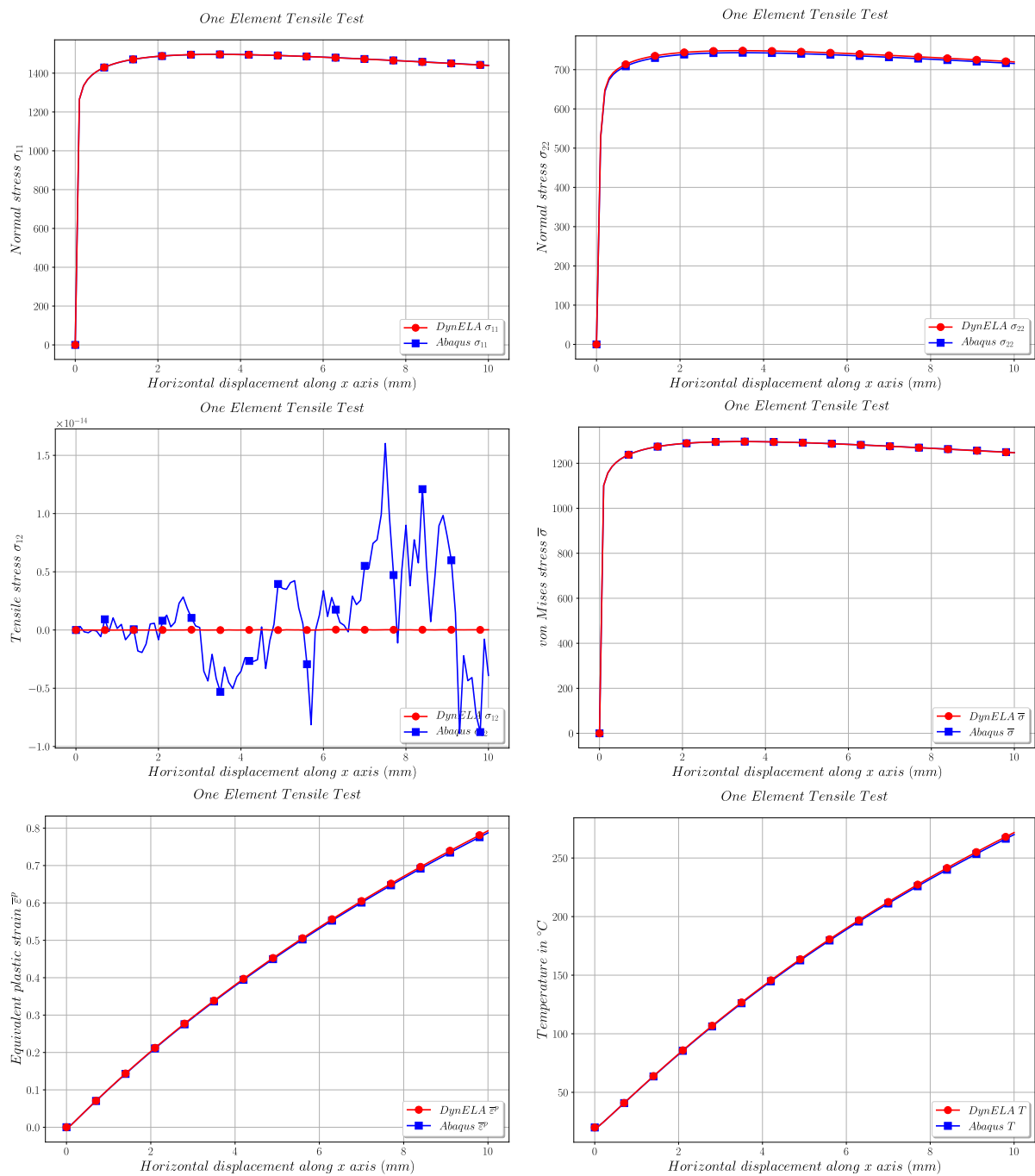
Figure III.1.4 – *Comparison of numerical and analytical results for the 3D one element tensile test*
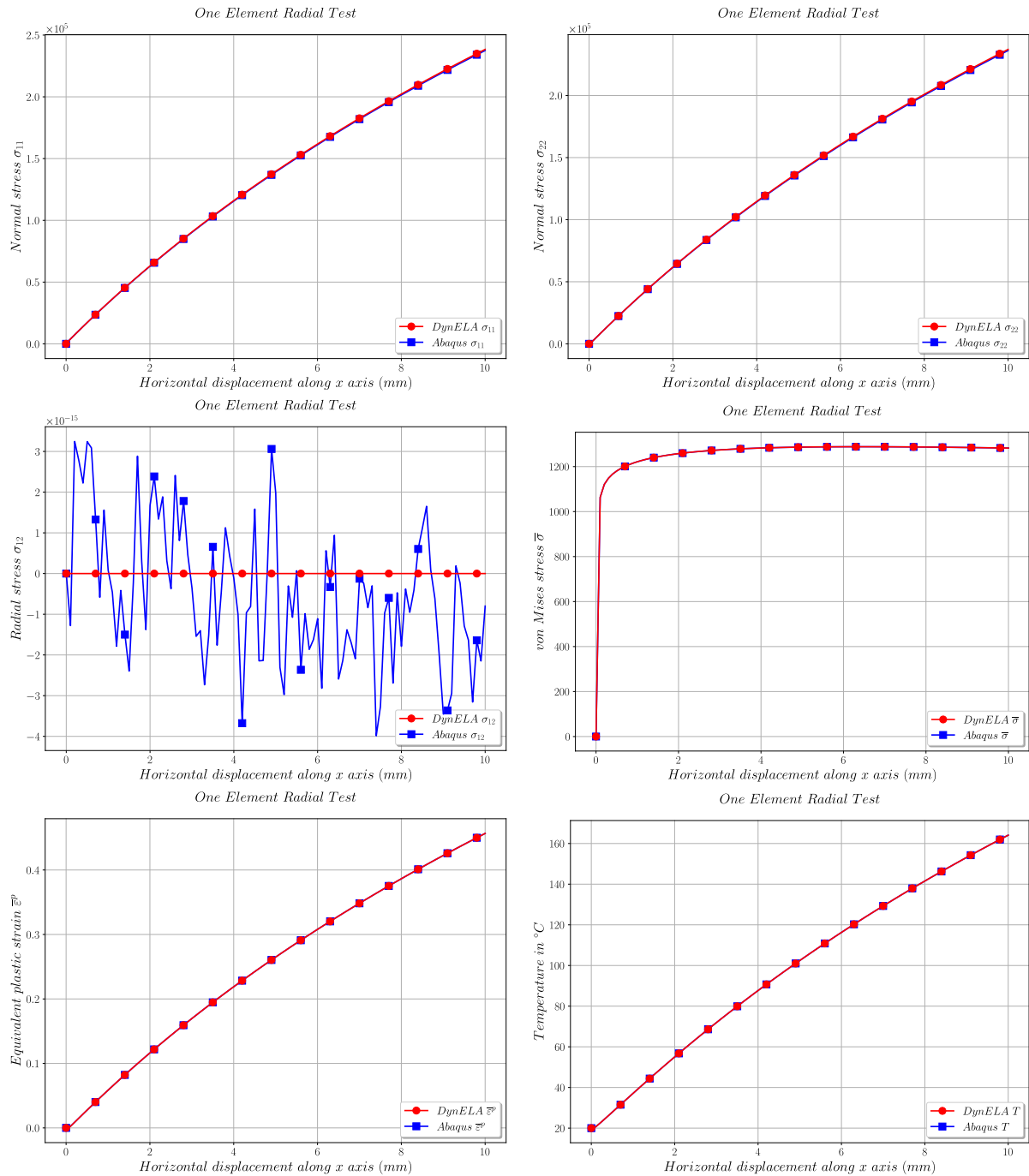
Figure III.1.5 – *Comparison of numerical and analytical results for the one element radial tensile test*

1.1.4   Element radial torus test

The uniaxial one element torus tensile test is a numerical test where an axisymmetric element (with a square prescribed shape) is subjected to radial tensile as presented in figure III.1.6. The difference
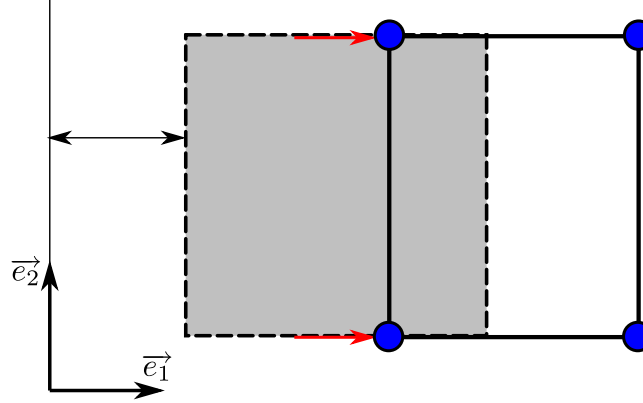

Figure III.1.6 – Numerical model for the one element torus test

with the previous test is that the left edge of the specimen in not aligned with the symmetry axis, the radial coordinate of the left edge is $r = 10\,mm$. The initial shape of the specimen is $10\,mm \times 10\,mm$ and a prescribed horizontal displacement $d = 10\,mm$ is applied on the two left nodes of the element as illustrated in Figure III.1.6. As we are using an explicit integration scheme, the total simulation time is set to $t = 0.01\,s$. All the properties of the constitutive law reported in Table III.3.1 are used and the material is assumed to follow the Johnson–Cook behavior described by equation 3.1.

As a global comparison, Figure III.1.7 show the comparison of the right edge displacement *vs.* the left edge displacement for both the DynELA and the Abaqus simulations.
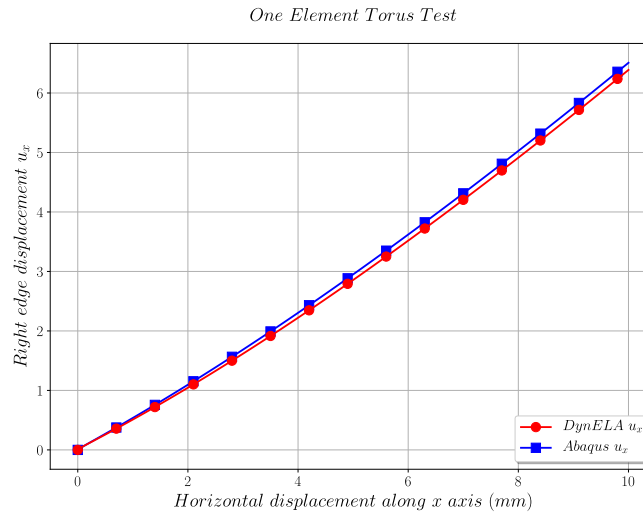

Figure III.1.7 – Comparison of the right edge displacement for the one element torus test

Figure III.1.9 shows the comparison of the DynELA solver results (plotted in red) and the Abaqus numerical results (plotted in blue) concerning the evolution of the stress components σ_{11}, σ_{22}, σ_{12}, $\overline{\sigma}$, $\overline{\varepsilon}^p$ and T *vs.* the horizontal displacement of the right edge of the specimen along the horizontal axis. As the Abaqus software only provides under–integrated elements, we are using a 2×2 elements mesh for the Abaqus model, as presented in Figure III.1.8, to compare the results. On abaqus, the

mean value of the results of the 4 elements is computed and compared to the mean value of the 4 integration points of the DynELA simulation.
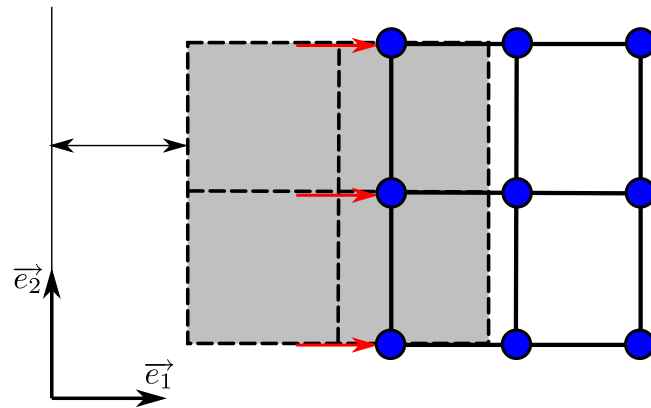
Figure III.1.8 – *Numerical model for the one element torus test under Abaqus*
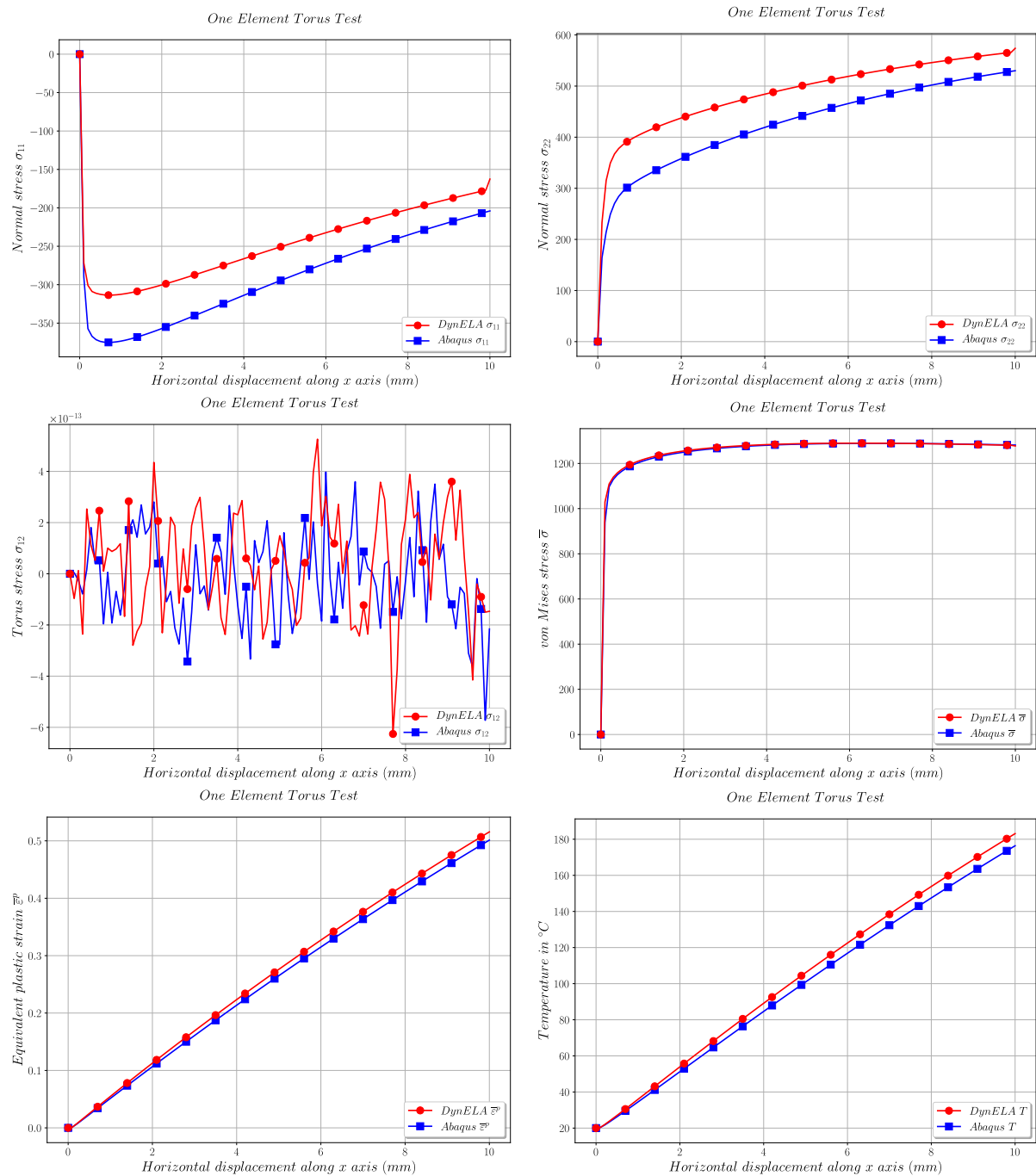
Figure III.1.9 – *Comparison of numerical results for the one element torus tensile test*

1.2 Uniaxial one element shear test

The uniaxial one element shear test is a numerical test where an element (with a square prescribed shape) is subjected to pure shear test as presented in figure III.1.10. The initial shape of the specimen
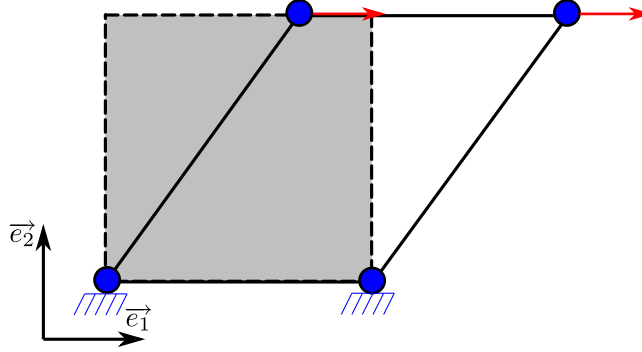

Figure III.1.10 – Numerical model for the one element shear test

is $10\,mm \times 10\,mm$ and the the two bottom nodes of the element are encastred and a prescribed horizontal displacement $d = 100\,mm$ is applied on the two upper nodes of the same element as illustrated in Figure III.1.10. As we are using an explicit integration scheme, the total simulation time is set to $t = 0.01\,s$. Mechanical properties of the material are reported in Table III.3.1.

Comparison of Abaqus and DynELA results is made by averaging the DynELA results on the 4 integration points of the element. This has been done because DynELA Finite Element Code uses full integrated elements while Abaqus have reduced integrated elements only but in fact, the DynELA results at the 4 integration points are the same in this benchmark test.

1.2.1 Elastic case

In this case, only the elastic properties of the constitutive law reported in Table III.3.1 are used and the material is assumed to be hyper-elastic. As we are using a Jaumann objective rate within the DynELA Finite Element Code, one can obtain, using an analytical development, the following results for the proposed case:

$$\boldsymbol{\sigma} = G \begin{bmatrix} 1 - \cos e & \sin e & 0 \\ & \cos e - 1 & 0 \\ sym & & 0 \end{bmatrix}, \tag{1.2}$$

where $G = \mu = \frac{E}{2(1+\nu)}$ is the shear modulus of the material and e is the elongation along the horizontal axis with $e_{max} = 10$ conforming to the prescribed boundaries conditions. Figure III.1.11 shows the comparison of the DynELA solver results (plotted in red) and both the analytical (plotted in blue) and the Abaqus numerical results (plotted in green) concerning the evolution of the stress components σ_{11}, σ_{22}, σ_{12} and $\overline{\sigma}$ vs. the horizontal displacement of the top edge of the specimen along the horizontal axis. A perfect match between all those results can be seen from this later.
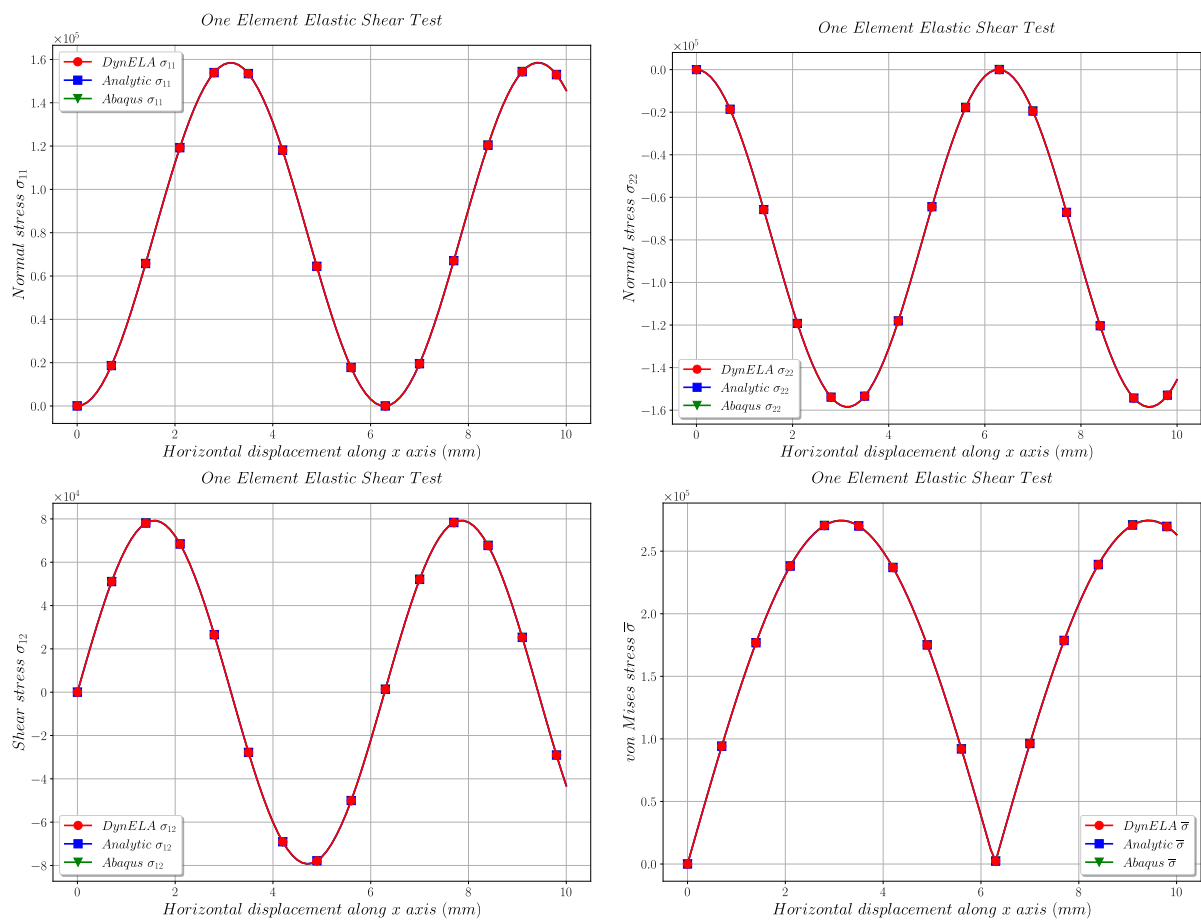
Figure III.1.11 – *Comparison of numerical and analytical results for the one element elastic shear test*

1.2.2 Johnson–Cook plastic behaviour

In this case, all the properties of the constitutive law reported in Table III.3.1 are used and the material is assumed to follow the Johnson-Cook behavior described by equation 3.1. Figure III.1.12 shows the comparison of the DynELA solver results (plotted in red) and the Abaqus numerical results (plotted in blue) concerning the evolution of the stress components σ_{11}, σ_{22}, σ_{12}, $\overline{\sigma}$, $\overline{\varepsilon}^p$ and T *vs.* the horizontal displacement of the top edge of the specimen along the horizontal axis.
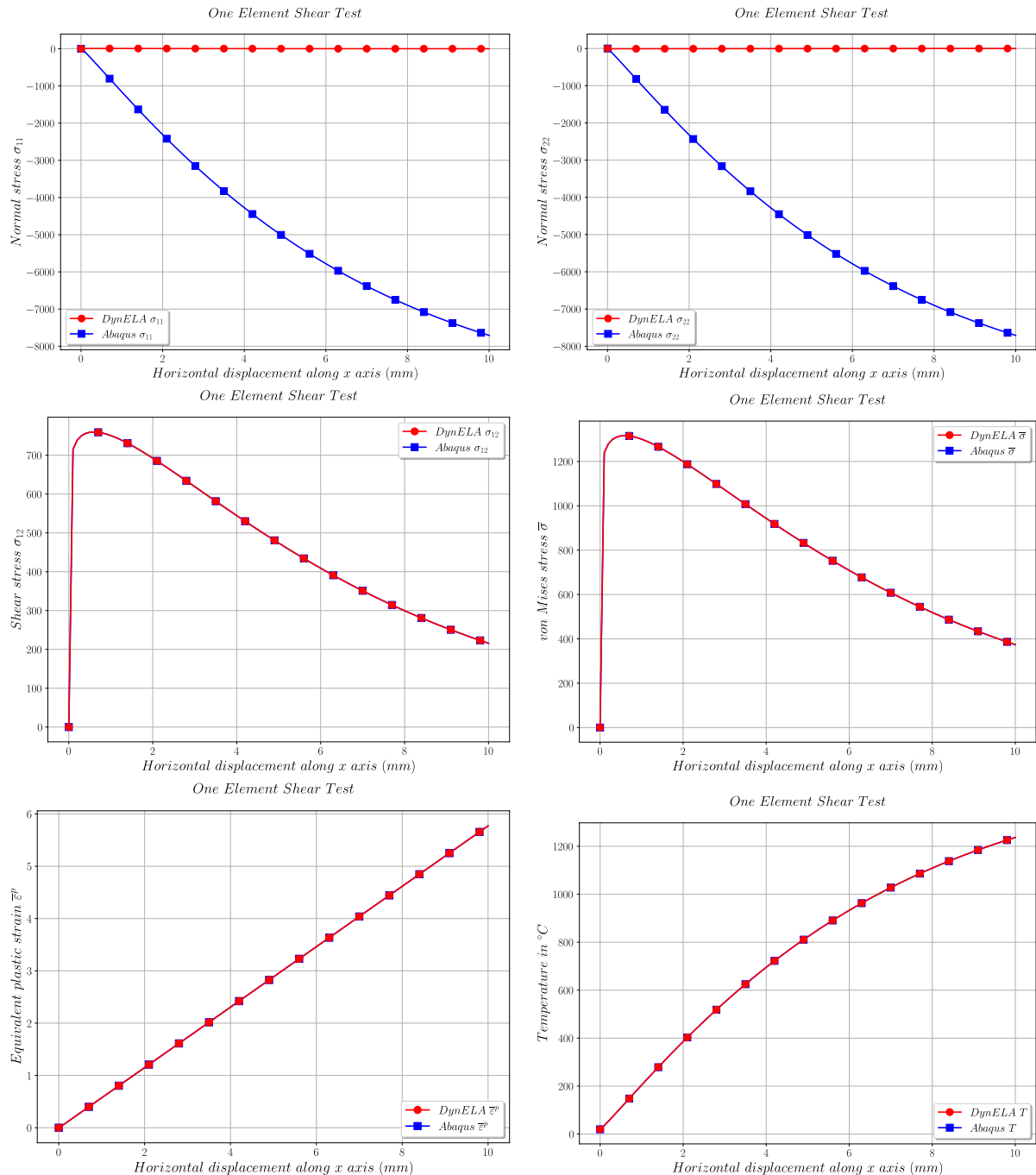

Figure III.1.12 – *Comparison of numerical and analytical results for the one element shear test*

Chapter 2

DynELA plasticity sample cases

This chapter deals with some numerical applications of the DynELA Finite Element Code for plasticity applications in 2D, axi-symmetric and 3D cases. In the subsequent tests, if not specified, a Johnson-Cook constitutive law is used to model the behavior of the material. The Johnson-Cook hardening flow law is probably the most widely used flow law for the simulation of high strain rate deformation processes taking into account plastic strain, plastic strain rate and temperature effects. Since a lot of efforts have been made in the past to identify the constitutive flow law parameters for many materials, it is implemented in numerous Finite Element codes such as Abaqus [?]. The general formulation of the Johnson-Cook law $\sigma^y(\overline{\varepsilon}^p, \dot{\overline{\varepsilon}}^p, T)$ is given by the following equation:

$$\sigma^y = \left(A + B\overline{\varepsilon}^{p^n}\right) \left[1 + C\ln\left(\frac{\dot{\overline{\varepsilon}}^p}{\dot{\overline{\varepsilon}}_0}\right)\right] \left[1 - \left(\frac{T - T_0}{T_m - T_0}\right)^m\right] \tag{2.1}$$

where $\dot{\overline{\varepsilon}}_0$ is the reference strain rate, T_0 and T_m are the reference temperature and the melting temperature of the material respectively and A, B, C, n and m are the five constitutive flow law parameters. A 42CrMo4 steel following the Johnson-Cook behavior law has been selected for all those tests, and material properties are reported in Table III.3.1.

E (Gpa)	ν	A (MPa)	B (MPa)	C	n	m
206.9	0.3	806	614	0.0089	0.168	1.1
ρ (kg/m³)	λ (W/m°C)	C_p (J/Kg°C)	η	$\dot{\overline{\varepsilon}}_0$ (s⁻¹)	T_0 (°C)	T_m (°C)
7830	34.0	460	0.9	1.0	20	1540

Table III.2.1 – *Material parameters of the Johnson–Cook behavior for the numerical tests*

2.1   Necking of a circular bar

2.1.1   Axisymmetric Bar Necking

The necking of a circular bar test is useful to evaluate the performance of VUMAT subroutine for materials in presence of plasticity and large deformation [16,17]. Because of the symmetric structure, an axisymmetric quarter model of the specimen is established. Dimensions of the specimen are reported in Figure III.2.1. The loading is realized through an imposed total displacement of $7\,mm$ along the $\vec{z}$ axis on the left side of the specimen while the radial displacement of the same edge is supposed to remain zero. On the opposite side the axial displacement is restrained while the radial displacement is free. The mesh consists of 400 CAX4RT elements with a refined zone of 200 elements on the right side on $1/3$ of the total height. Again, the total simulation time is set to $t = 0.01\,s$ because of the explicit integration scheme adopted.
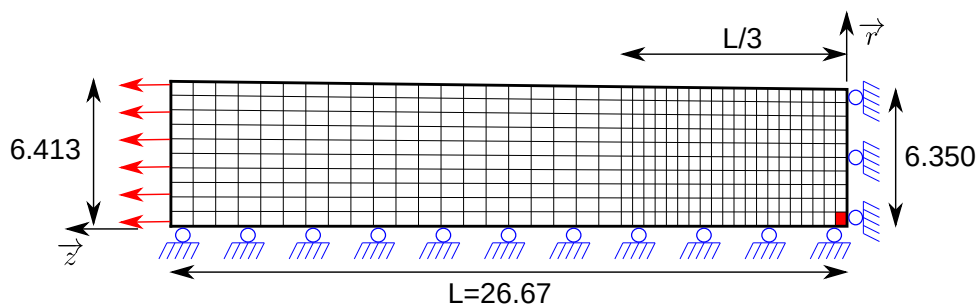

Figure III.2.1 – *Numerical model for the necking of a circular bar*

Figures III.2.2 and III.2.3 shows the temperature T and the von Mises stress $\bar{\sigma}$ contourplots of the deformed rod for both the DynELA Finite Element Code and Abaqus. The distributions of the temperatures and stresses are almost the same for both models. The maximum temperature T is located into the center element of the model (the red element in Figure III.2.1) and the models give quite the same results as reported in Table III.2.2 for $\bar{\varepsilon}^p$, T and the final dimensions of the specimen D_f (final diameter of the necking zone).

Figure III.2.4 shows the evolution of the the final dimensions of the specimen L_f (final length) and R_f (final radius of the impacting face), the equivalent plastic strain $\bar{\varepsilon}^p$, the temperature T, the von Mises stress $\bar{\sigma}$ and the timeStep Δt for the different models for the element at the center of the impacting face (the red element in Figure III.2.1).

As reported in this figure and according to the results presented in Table III.2.2, a quite good agreement between the results is obtained.

code	D_f	T	$\bar{\varepsilon}^p$
	(mm)	$(°C)$	
DynELA	2.71	642.78	2.07
Abaqus	2.35	658.91	2.12

Table III.2.2 – *Comparison of numerical results for the necking of a circular bar test*
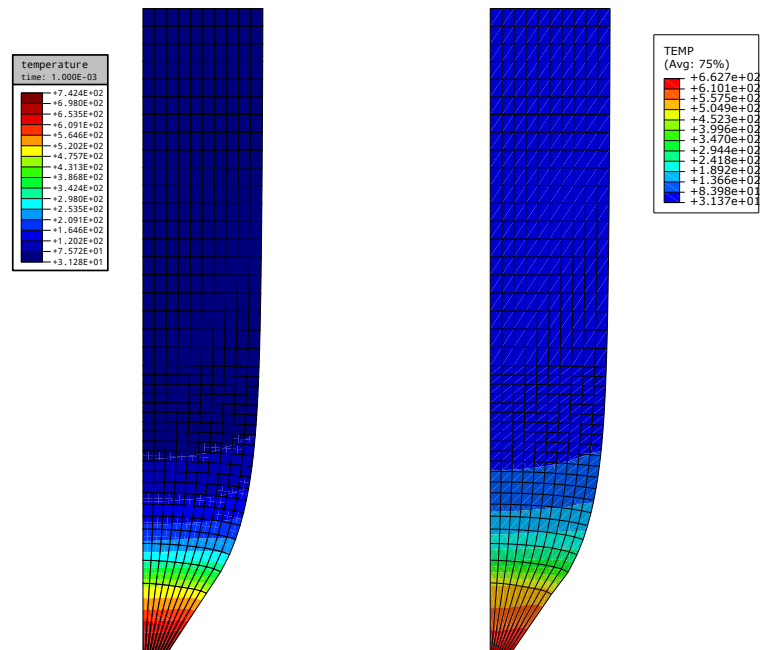
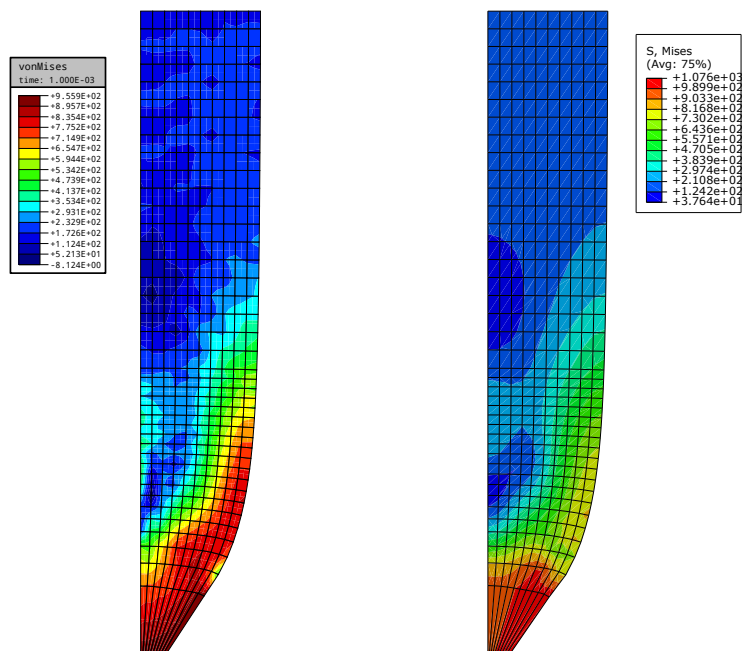**Figure III.2.2** – *Temperature T contourplot for the necking of a circular bar (DynELA left and Abaqus right)*

Figure III.2.3 – *von Mises stress $\overline{\sigma}$ contourplot for the necking of a circular bar (DynELA left and Abaqus right)*

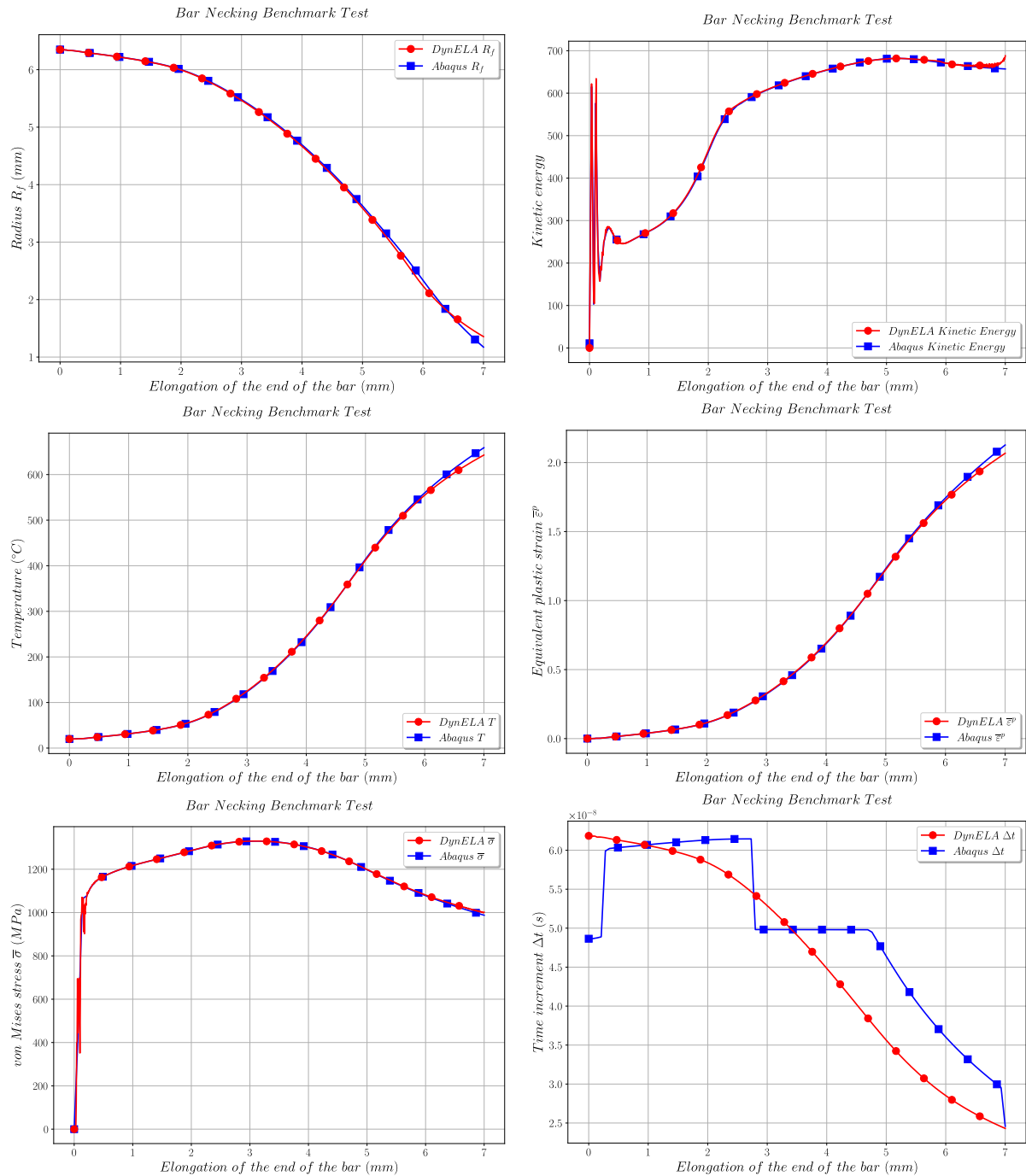**Figure III.2.4** – *Comparison of numerical and analytical results necking of a circular bar test*

Chapter 3

DynELA impact sample cases

This chapter deals with some numerical applications of the DynELA Finite Element Code for impact applications in 2D, axi–symmetric and 3D cases. In the subsequent tests, if not specified, a Johnson–Cook constitutive law is used to model the behavior of the material. The Johnson–Cook hardening flow law is probably the most widely used flow law for the simulation of high strain rate deformation processes taking into account plastic strain, plastic strain rate and temperature effects. Since a lot of efforts have been made in the past to identify the constitutive flow law parameters for many materials, it is implemented in numerous Finite Element codes such as Abaqus [?]. The general formulation of the Johnson–Cook law $\sigma^y(\overline{\varepsilon}^p, \dot{\overline{\varepsilon}}^p, T)$ is given by the following equation:

$$\sigma^y = \left(A + B\overline{\varepsilon}^{p^n} \right) \left[1 + C \ln \left(\frac{\dot{\overline{\varepsilon}}^p}{\dot{\overline{\varepsilon}}_0} \right) \right] \left[1 - \left(\frac{T - T_0}{T_m - T_0} \right)^m \right] \tag{3.1}$$

where $\dot{\overline{\varepsilon}}_0$ is the reference strain rate, T_0 and T_m are the reference temperature and the melting temperature of the material respectively and A, B, C, n and m are the five constitutive flow law parameters. A 42CrMo4 steel following the Johnson–Cook behavior law has been selected for all those tests, and material properties are reported in Table III.3.1.

E (GPa)	ν	A (MPa)	B (MPa)	C	n	m
206.9	0.3	806	614	0.0089	0.168	1.1
ρ (kg/m^3)	λ $(W/m°C)$	C_p $(J/Kg°C)$	η	$\dot{\overline{\varepsilon}}_0$ (s^{-1})	T_0 $(°C)$	T_m $(°C)$
7830	34.0	460	0.9	1.0	20	1540

Table III.3.1 – *Material parameters of the Johnson–Cook behavior for the numerical tests*

3.1   Taylor impact sample

3.1.1   Axisymmetric Taylor impact

The performance of the proposed code is validated under high deformation rate with the simulation of the Taylor impact test [18]. In the Taylor impact test, a cylindrical specimen is launched to impact a rigid target with a prescribed initial velocity. The numerical model, reported in Figure III.3.1 is established as axisymmetric. The height is $32.4\,mm$ and the radius is $3.2\,mm$. The axial displacement is restrained on the right side of the specimen while the radial displacement is free (to figure a perfect contact without friction of the projectile onto the target). A predefined velocity of $V_c = 287\,m/s$ is imposed on the specimen. The mesh consists of 250 elements (5×50 elements). The total simulation time for the Taylor impact test is $t = 8.0\,10^{-5}s$.
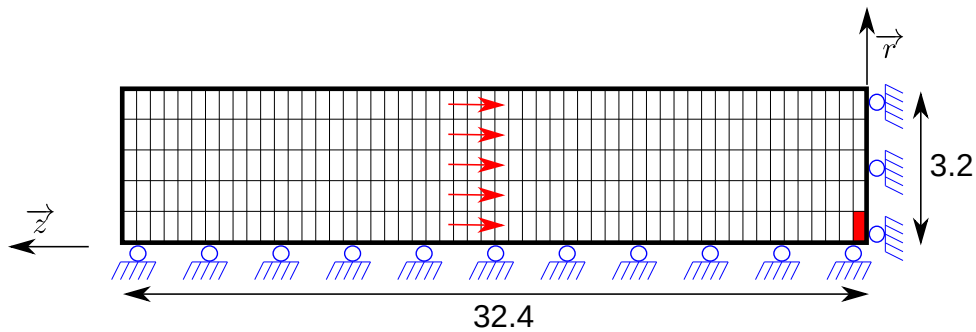

Figure III.3.1 – *Numerical model for the Axisymmetric Taylor impact test*

Figure III.3.2 shows the temperature contourplot of the deformed rod for both the DynELA Finite Element Code and Abaqus. The distributions of the temperatures are almost the same for both models. The maximum temperature T is located into the center element of the model (the red element in Figure III.3.1) and the models give quite the same results as reported in Table III.3.2 for $\bar{\varepsilon}^p$, T and the final dimensions of the specimen L_f (final length) and D_f (final diameter of the impacting face).

Figure III.3.3 shows the evolution of the the final dimensions of the specimen L_f (final length) and R_f (final radius of the impacting face), the equivalent plastic strain $\bar{\varepsilon}^p$, the temperature T, the von Mises stress $\bar{\sigma}$ and the timeStep Δt for the different models for the element at the center of the impacting face (the red element in Figure III.3.1).

As reported in this figure and according to the results presented in Table III.3.2, a quite good agreement between the results is obtained.

code	L_f (mm)	D_f (mm)	T (°C)	$\bar{\varepsilon}^p$
DynELA	26.52	11.15	582.34	1.78
Abaqus	26.56	11.16	590.96	1.81

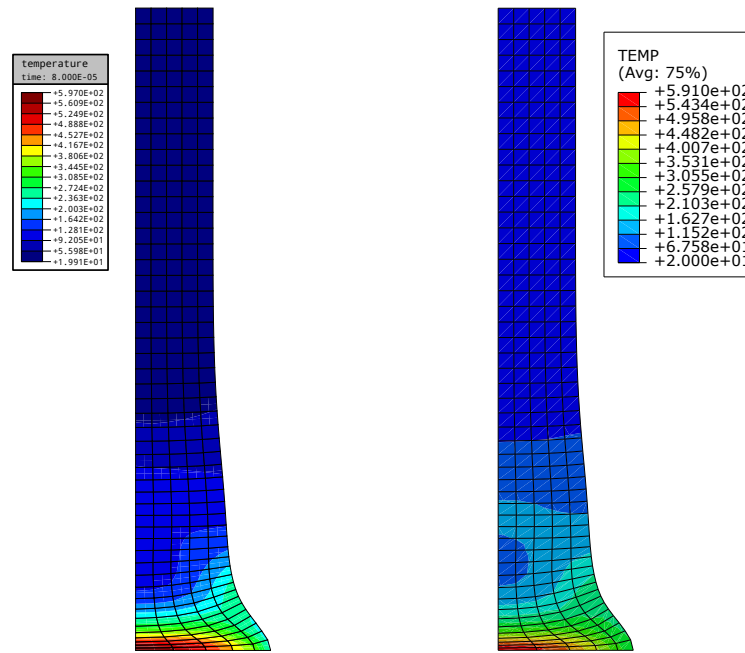**Table III.3.2 –** *Comparison of numerical results for the Axisymmetric Taylor impact test*

Figure III.3.2 – Temperature contourplots for Axisymmetric Taylor impact test (DynELA left and Abaqus right)

3.1.2   3D Taylor impact

The performance of the proposed code is validated under high deformation rate with the simulation of the Taylor impact test [18]. In the Taylor impact test, a cylindrical specimen is launched to impact a rigid target with a prescribed initial velocity. The numerical model, reported in Figure III.3.4 is established as axisymmetric. The height is $32.4\,mm$ and the radius is $3.2\,mm$. The axial displacement is restrained on the right side of the specimen while the radial displacement is free (to figure a perfect contact without friction of the projectile onto the target). A predefined velocity of $V_c = 287\,m/s$ is imposed on the specimen. The mesh consists of 4455 elements (55×81 elements). The total simulation time for the Taylor impact test is $t = 8.0\,10^{-5}s$.
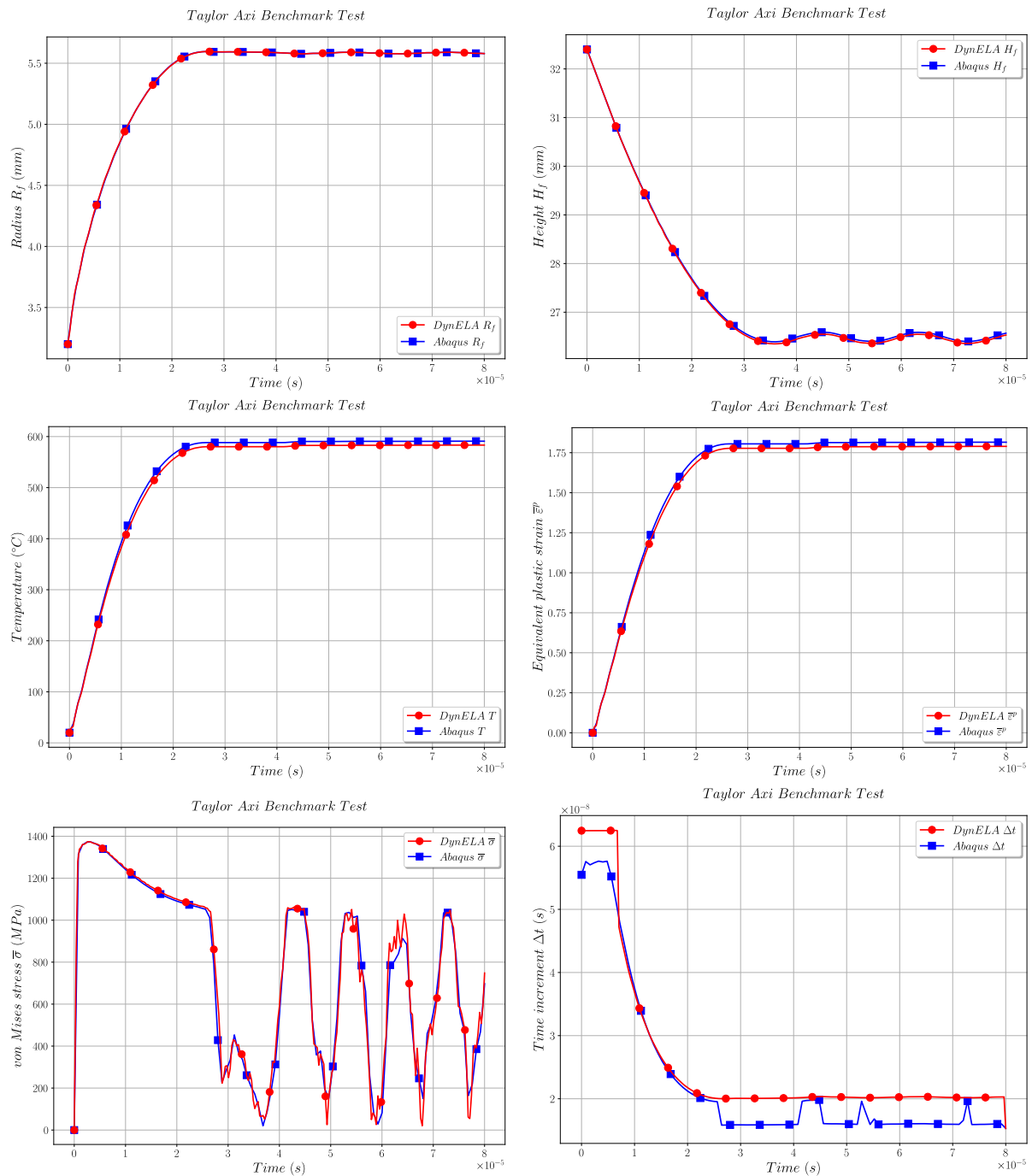
Figure III.3.5 shows the temperature contourplot of the deformed rod for both the DynELA Finite Element Code and Abaqus. The distributions of the temperatures are almost the same for both models. The maximum temperature T is located into the center element of the model (the red element in Figure III.3.4) and the models give quite the same results as reported in Table III.3.3 for $\overline{\varepsilon}^p$, T and the final dimensions of the specimen L_f (final length) and D_f (final diameter of the impacting face).

Figure III.3.6 shows the evolution of the the final dimensions of the specimen L_f (final length) and R_f (final radius of the impacting face), the equivalent plastic strain $\overline{\varepsilon}^p$, the temperature T, the von Mises stress $\overline{\sigma}$ and the timeStep Δt for the different models for the element at the center of the impacting face (the red element in Figure III.3.4).

As reported in this figure and according to the results presented in Table III.3.3, a quite good agreement between the results is obtained.
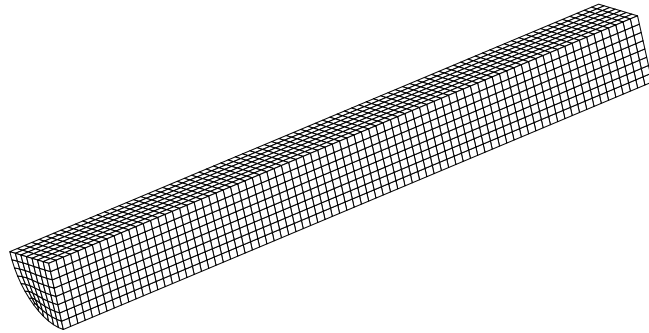
code	L_f (mm)	D_f (mm)	T (°C)	$\overline{\varepsilon}^p$
DynELA	26.52	11.18	597.10	1.84
Abaqus	26.55	11.22	597.01	1.84

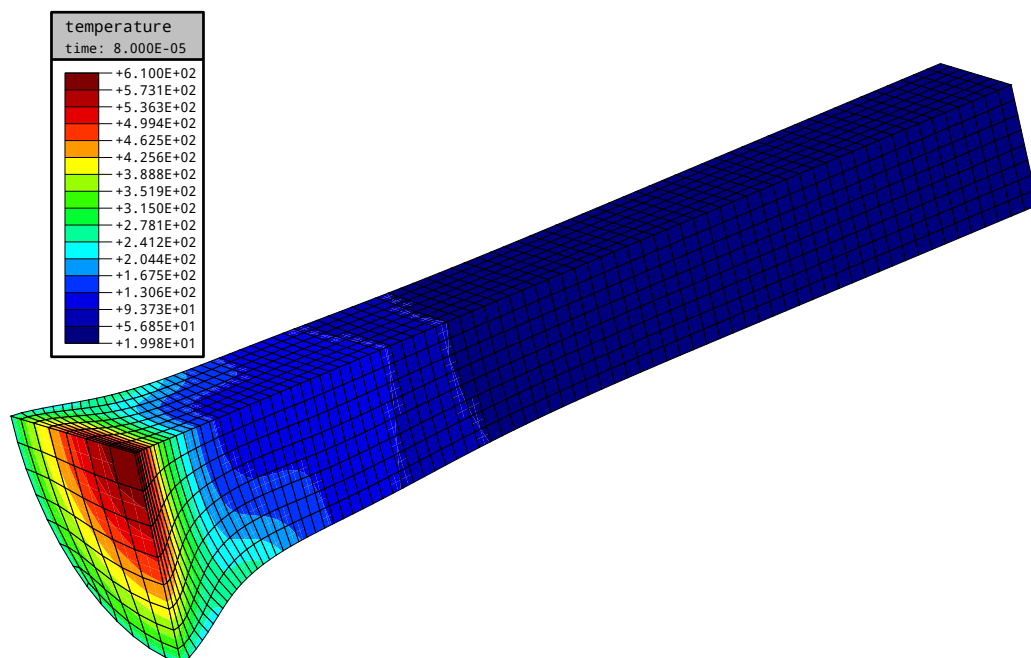*Table III.3.3 – Comparison of numerical results for the 3D Taylor impact test*

Figure III.3.3 – *Comparison of numerical and analytical results for the Axisymmetric Taylor impact test*

Figure III.3.4 – Numerical model for the 3D Taylor impact test

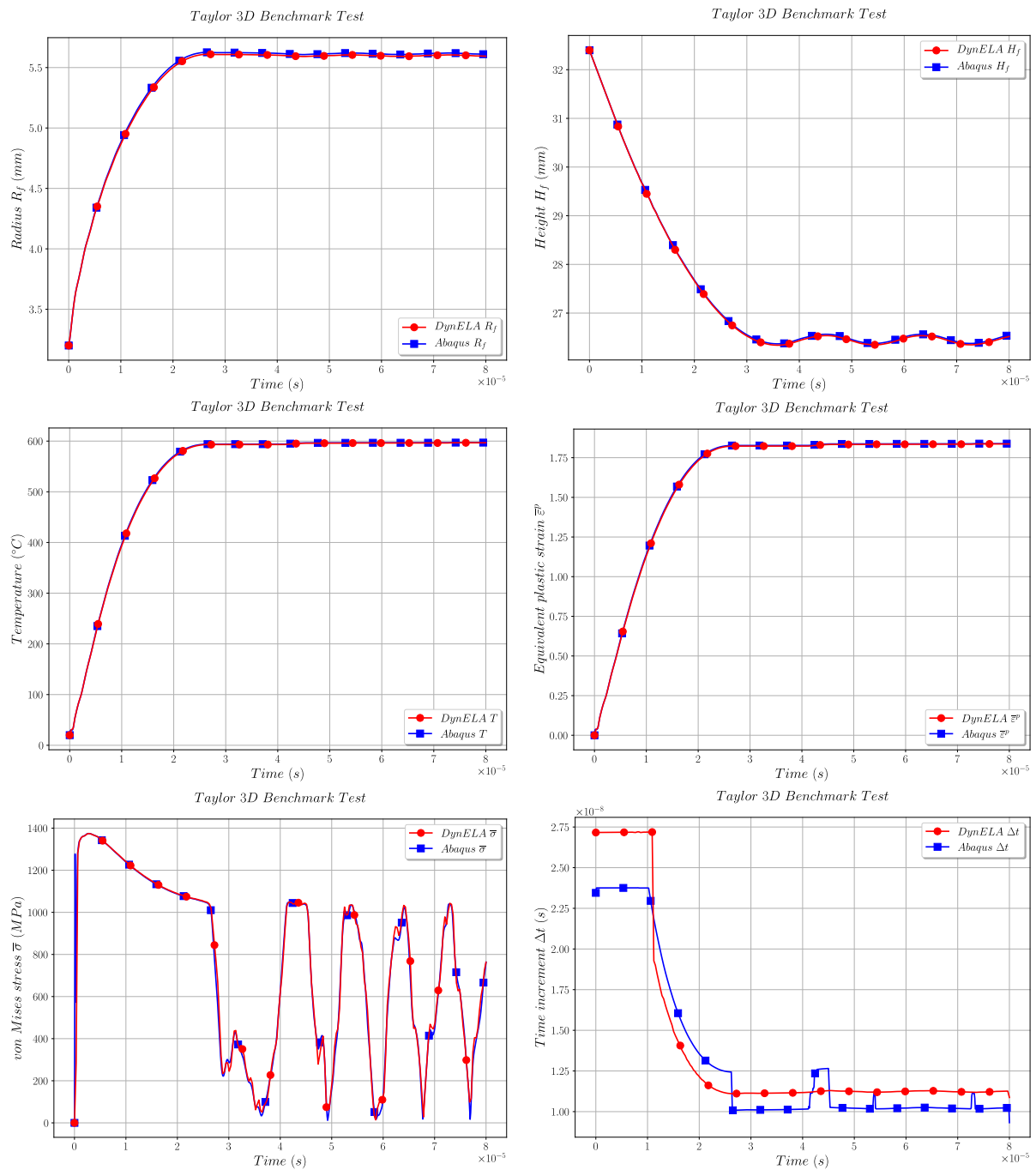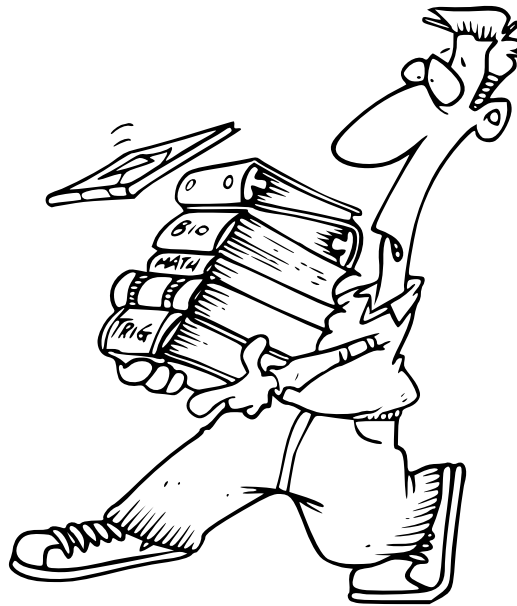*Figure III.3.5 – Temperature contourplot for the 3D Taylor impact test*

Figure III.3.6 – *Comparison of numerical and analytical results for the 3D Taylor impact test*

Bibliography

[1] Olivier Pantalé. An object-oriented programming of an explicit dynamics code: application to impact simulation. *Advances in Engineering Software*, 33(5):297–306, May 2002. 5

[2] Olivier Pantalé, Serge Caperaa, and Roger Rakotomalala. Development of an object-oriented finite element program: application to metal-forming and impact simulations. *Journal of Computational and Applied Mathematics*, 168(1):341–351, July 2004. 5

[3] Olivier Pantalé. Parallelization of an object-oriented FEM dynamics code: influence of the strategies on the Speedup. *Advances in Engineering Software*, 36(6):361–373, June 2005. 5

[4] Laurent Menanteau, Olivier Pantalé, and Serge Caperaa. A methodology for large scale finite element models, including multi-physic, multi-domain and multi-timestep aspects. *European Journal of Computational Mechanics*, 15(7-8):799–824, January 2006. 5

[5] Ionel Nistor, Olivier Pantalé, and Serge Caperaa. Numerical propagation of dynamic cracks using X-FEM. *European Journal of Computational Mechanics*, 16(2):183–198, January 2007. 5

[6] Ionel Nistor, Olivier Pantalé, and Serge Caperaa. Numerical implementation of the eXtended Finite Element Method for dynamic crack analysis. *Advances in Engineering Software*, 39(7):573–587, July 2008. 5

[7] Olivier Pantalé. [Rp] Parallelization of an object-oriented FEM dynamics code. *ReScience C*, 6(1):#8, June 2020. 5

[8] Laurent Menanteau, Olivier Pantalé, and Serge Caperaa. A coupled electro-thermo-mechanical FEM code for large scale problems including multi-domain and multiple time-step aspects. ISBN:84-95999-71-4, Santorini Island, May 2005. 5

[9] Ionel Nistor, Olivier Pantalé, and Serge Caperaa. On the modeling of the dynamic crack propagation by extended finite element method: numerical implementation in DynELA code. In CIMNE, editor, *VIII International Conference on Computational Plasticity*, pages 303–306, Barcelona, 2005. 5

[10] Olivier Pantalé and Serge Caperaa. Strategies for a parallel 3D FEM code: Application to impact and crash problems. In *Coupled Problems 2005 - Santorini Island*, 2005. 5

[11] Olivier Pantalé and Serge Caperaa. Développement d'un code de calcul explicite en grandes transformations: Application à la coupe des métaux. In *Séminaire Optimus*, 2004. 5

[12] Olivier Pantalé, Roger Rakotomalala, and Serge Caperaa. Développement d'un code de calcul explicite ALE : Application à la coupe des métaux. In *14 ème congrés francais de mécanique*, page 951, Toulouse, September 1999. 5

[13] Olivier Pantalé, Serge Caperaa, and Roger Rakotomalala. Development of an object-oriented finite element program: application to metal-forming and impact simulations. In *ACOMEN 2002 Second international conference on advanced computational methods in engineering*, ISBN 2-930322-39-X, Liège University (Belgium), May 2002. 5

[14] Laurent Francis Menanteau. *Développement d'un module de prototypage virtuel multiphysique, multidomaine et multitemps : application aux convertisseurs de puissance.* PhD Thesis, Toulouse, INPT, January 2004. 5

[15] Ionel Nistor. *Identification expérimentale et simulation numérique de l'endommagement en dynamique rapide : application aux structures aéronautiques.* PhD Thesis, Toulouse, INPT, November 2005. 5

[16] J. P. Ponthot. Unified stress update algorithms for the numerical simulation of large deformation elasto–plastic and elasto–viscoplastic processes. *International Journal of Plasticity*, 18(1):91–126, January 2002. 48

[17] J. C. Simo and T.J.R. Hughes. *Computational Inelasticity.* Springer, July 1998. /home/pantale/SynologyDrive/Bibliotheque/Livres Techniques/J. C. Simo/Computational Inelasticity (90)/Computational Inelasticity - J. C. Simo.pdf. 48

[18] G. I. Taylor. JAMES FORREST LECTURE 1946. THE TESTING OF MATERIALS AT HIGH RATES OF LOADING. *Journal of the Institution of Civil Engineers*, 26(8):486–519, October 1946. 52, 53