Finite Element Verification of Non-Homogeneous Strain and Stress Fields during Composite Material Testing

Lars P. Mikkelsen

Composite and Materials Mechanics, DTU Wind Energy,
Technical University of Denmark, 4000 Roskilde, Denmark

Abstract*: Uni-directional glass fiber reinforced polymers play a central role in the task increasing the length of wind turbines blades and thereby lowering the cost of energy from wind turbine installations. During this, optimizing the mechanical performance regarding material stiffness, compression strength and fatigue performance is essential. Nevertheless, testing composites includes some challenges regarding stiffness determination using conventional strain gauges and achieving correct material failure unaffected by the gripping region during fatigue testing. Challenges, which in the present study, has been addressed using the finite element method. During this, a verification of experimental observations, a deeper understanding on the test coupon loading and thereby improved test methods has been achieved.*

Keywords*: Composites, Experimental Verification, Fatigue, Scripting*

1. Introduction

Glass fiber reinforced polymers are a lightweight material which compared to its weight and price is a stiff and a strong material in the fiber direction. Glass fiber composites is a material technology, which has made it possible for the wind turbine industry designing wind turbine blades with a length of a little more than 80 m at present. A development making it possible making onshore wind turbines, which regarding the cost of energy, can compete with conventional energy sources. On the other hand, offshore wind turbines is still too expensive due to larger installations and maintenance cost regarding a direct competition with conventional energy sources and will properly need to grow with at least 50% in diameter in order to be competitive. Therefore, there are intensive research activities focusing on making better but still cheap composites materials. Those activities involve large amount of experimental material characterizations where the main focus is on improving the materials stiffness, compression strength and fatigue life.

One of the challenges testing composite materials are the highly anisotropic properties for the uni-directional glass fiber reinforced polymers used for the load carrying parts in a wind turbine blade. Anisotropy which are resulting in stiffness and strength properties more than 10 times larger in the fiber direction compared with loading in the transverse direction and under shear. In addition, even in the fiber direction the material is rather compliant compared with metallic materials. During mechanical testing, both the anisotropy and the compliant material behavior causes some challenges regarding load introductions during testing (Hojo et al., 1994; Mikkelsen and Bech, 2013; Xie and Adams, 1995). Challenges which based on numerical predictions are found to results in highly non-homogenous stress field which subsequently is found to influence the measured statically and fatigue strength significantly. An alternative to measuring the macroscopic strength is by a micro-mechanical determination and understanding of failure mechanisms during mechanical testing. Studies requiring exporting actually micromechanical structures determined e.g. by micro x-ray CT scans (Emerson et al., 2015) to micromechanically based finite element models.

Not only the strength properties are influenced by the non-homogeneous strain and stress fields. Determining the material stiffness require a precise strain measurement. Due to the price and ease of using, strain gauges are often used as the deformation measurement testing composites. Strain gauges are measuring the deformation of the substrate through an electrical resistance change along an approximately 5 µm thick and 30 µm wide conductive wire made of a 180 GPa stiff constantan alloy printed as a measurement grid on a 45 µm thick backing polymeric film. Despite the small thickness of the measuring grid, finite element simulations verify that the stiff grid material introduce some strain inhomogeneity (Zike and Mikkelsen, 2014). A verification which explains a difference of 2-5% on the measured composite material stiffness comparing measurements based on strain gauges with clip-on extensometers or optical methods. Subsequently, based on an intensive parametric study facilitated through the python scripting language in the Abaqus CAE, an improved grid design has been proposed (Mikkelsen and Zike, 2014).

2. Strain gauges introduced non-homogeneity

An important material property, designing wind turbine blade with a length of over 80 m, is a high material stiffness (Brøndsted et al., 2005). Designing this; a precise strain measurement is essential. Strain gauges are considered one of the most reliable strain measurements devices and is therefore, do to its low price and ease to use, by far the most common device for strain measurements. Nevertheless, applied on a moderate stiff composite material with a stiffness in the range of $E \in [3;35]\mathrm{GPa}$, strain gauges is observed to overestimate the material stiffness with 1-10% error depending on the stiffness and strain gauge grid length (Zike and Mikkelsen, 2014).

In the present study, this is studied on the cases, where a 1.5, 3 or 10 mm strain gauges is mounted on both side of a 30mm thick, 30 mm wide and 100 mm long specimen. Due to symmetry, only one quarter of the specimen is modelled. The chosen width and thickness values of the test coupons are found sufficiently large such that the overall stiffness of the test sample is not affected by the strain gauges. During the analysis, the elongation of the specimen is prescribed by a prescribed elongation corresponding to an overall axial strain of 0.25%. Nevertheless, due to the length, a prescribed force could equally have been chosen. The prescribed elongation, is in agreement with the test standard, (ISO-527-1, 1997), where the material stiffness of a polymer or a fiber reinforced polymer should be determined by the slope of the stress versus strain curve in the strain range $\varepsilon \in [0.05;0.25]\%$. From the finite element model, all the materials are found deforming essentially elastically why a linear elastic finite element simulation solved using only one deformation increment has been found sufficiently.

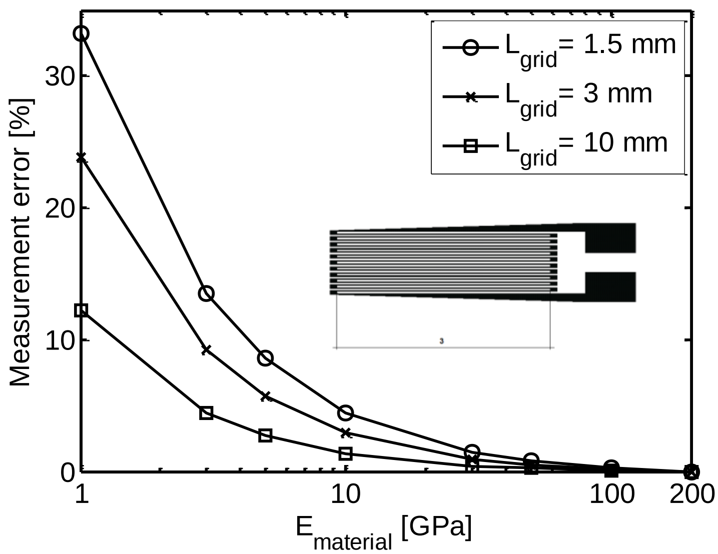


Figure 1. Predicted measurements error on the material stiffness using a 1.5, 3 and 10 mm strain gauges, respectively.

Figure 1, show the finite element predictions of the erroneous of the strain gauges determined material stiffness as a function of the actually stiffness. Physical strain gauges measure the strain through a resistance change in the measurement grid. This resistance change are correlated to the strain in the substrate through a gauge factor found by calibrating the strain gauge mounted on a steel material. In the finite element model, the strain gauges output is defined by the elongation of the measurement wire and virtually calibrated on a 200GPa stiff material. Similar to the experimentally observations in (Zike and Mikkelsen, 2014), the strain is found to significantly overestimate the material stiffness in the moderate material stiffness range $\varepsilon \in [0.05; 0.25]\%$.

Figure 2, show the strain field in the case of a pure epoxy at an overall straining of the material of 0.25% using a 3 mm strain gauge. Despite a thickness of the measuring grid on only 5 µm, the non-uniform strain state is found to spread significantly into the measuring grid of the strain gauges and thereby causing a under prediction of the strains. As a consequence of the underestimated strains, an overestimate of the material stiffness is found as shown in figure 1. Thus, the only 5 µm thick but 180 GPa conductive wire in the strain gauges measurement grid is found to introduce significant strain inhomogeneity in the substrate material resulting in an erroneous of the experimentally determined material stiffness. An erroneous validated in the present work through a linear elastic finite element model.

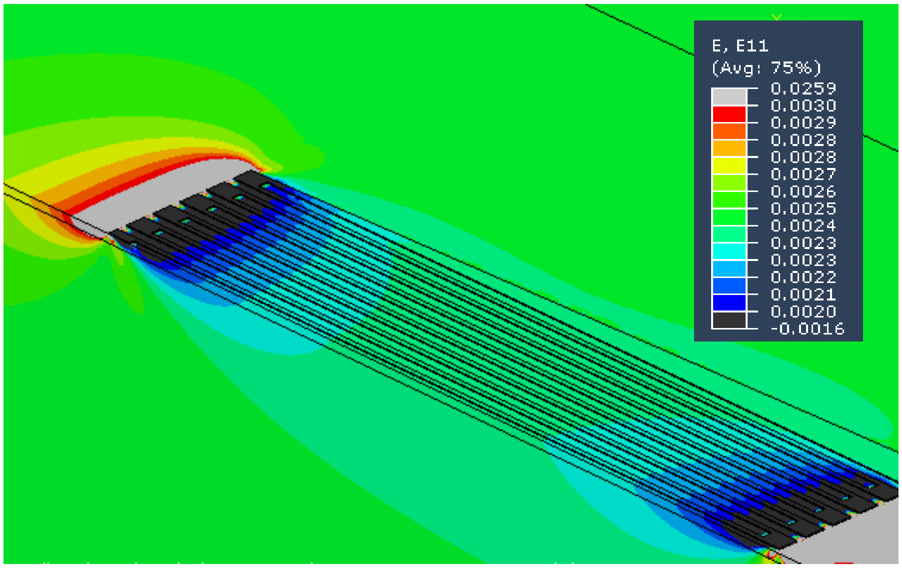


Figure 2. Non-homogeneous strain field applying a 3 mm strain gauges on a 3 GPa stiff polymer matrix.

3. Gripping introduced non-homogeneity

Uni-directional glass fiber reinforced composites has tensile and compression strength in the range of 1-2 GPa while the shear and transverse tensile strength in the range of 50-100 MPa. In addition, the axial stiffness of a uni-directional glass fiber composite material is around 40 GPa while the clamping material has a stiffness of 200 GPa. Therefore, it is a challenge loading material test coupons in a way such that the resulting material failure occurs in the gauge section. Especially during fatigue loading, failure in the gripping regions is often observed. Therefore, the measured strength and fatigue properties can often only be considered as conservative values making it difficult comparing different materials and thereby optimizing the material components. Figure 3 show a typical gripping fatigue failure together with a numerical simulation of the corresponding stress state during loading. The numerical finite element simulation is performed using a 3D linear elastic model where the interaction between the grip and tabs are simulated by a fictional contact with the friction coefficient of μ=0.4. Due to symmetry, only one eight of the model has been modelled. The material test is performed using hydraulic grips, where it is possible independently to prescribe the clamping and tensile force. Similar to the experimental setup, the clamping force are just prescribed to be high enough avoiding the test coupons slipping out of the grips. Similar to the real case, the test sample is loaded up though the grips until the tensile stress in the gauge section are equal to 1 GPa. At this load-level the finite element simulation show high shear stresses in the gripping region of the test coupons. Shears stresses approaching 60 MPa and locally penetrating into the test material and thereby properly causing tab shear failure as the failure shown on the picture in Figure 3.

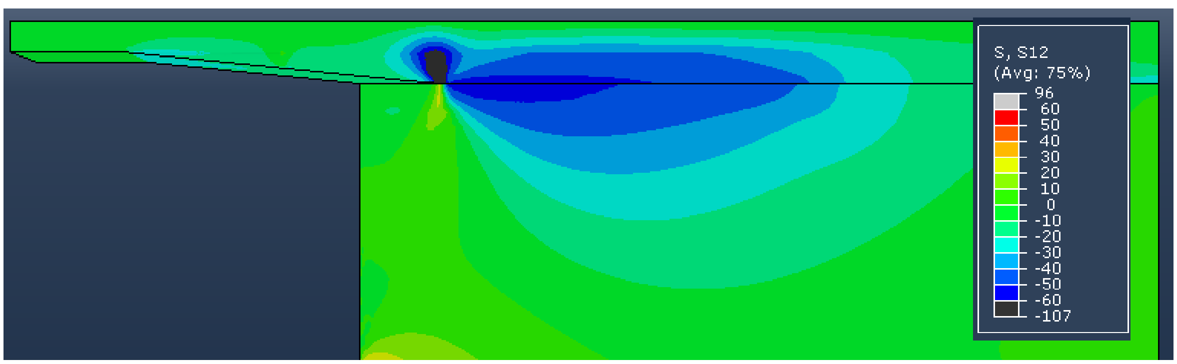


Figure 3. Non- homogeneous shear strain state introduced by the gripping of a tension-compression fatigue test specimen shown together with the observed failure mechanism in the test coupon.

4. Scripting using the Python interface in Abaqus/CAE

In order to perform a parametric study, the Python scripting language interface to the Abaqus/CAE pre- and post-processor gives great advances. In the flowing, a generic example is shown including two Python scripts used by the author performing the parametric studies regarding pre- and post-processing. Using the scripts it is possible to modify a large variety of parameters defining geometry, materials, meshing etc. Eventually, the two scripts can be built together resulting in only one rpt-file for the chosen output values from each studied case.

4.1 Pre-processing

```
## Default ABAQUS inputs
from part import *
from material import *
from section import *
from assembly import *
from step import *
from interaction import *
from load import *
from mesh import *
from optimization import *
from job import *
from sketch import *
from visualization import *
from connectorBehavior import *

NameModelBase='ModelName'
E1s=(1000,3000,10000,30000,100000,200000)

for i in range(0,6):
    ## Model in names
    NameCAE='Model.cae'
    NameModelin=NameModelBase
    NameModel=NameModelBase+'-E1-'+str(E1s[i])
    print NameModelin
    Ncpus=20  # in the case of parallel processes

    ## Import model from other cae-file
    mdb.openAuxMdb(pathName=NameCAE)
    mdb.copyAuxMdbModel(fromName=NameModelin, toName=NameModel)
    mdb.closeAuxMdb()

    ## Eventually Modify model
    mdb.models[NameModel].materials['Tab'].elastic.setValues(table=((E1s[i],
            0.3333), ))

    ## Generate job
    modelJob=mdb.Job(atTime=None, contactPrint=OFF, description='', echoPrint=OFF,
            explicitPrecision=SINGLE, getMemoryFromAnalysis=True, historyPrint=OFF,
            memory=90, memoryUnits=PERCENTAGE, model=NameModel, modelPrint=OFF,
            multiprocessingMode=MPI, name=NameModel, nodalOutputPrecision=SINGLE,
            numCpus=Ncpus, numDomains=Ncpus, numGPUs=0, queue=None, scratch='',
            type=ANALYSIS, userSubroutine='', waitHours=0, waitMinutes=0)

    ## The following lines submit the job, and wait for completion before continue
    modelJob.submit(consistencyChecking=OFF)
    modelJob.waitForCompletion()
```

4.2 Post-processing

```
## Default ABAQUS inputs
from part import *
from material import *
from section import *
from assembly import *
from step import *
from interaction import *
from load import *
from mesh import *
from optimization import *
from job import *
from sketch import *
from visualization import *
from connectorBehavior import *
from abaqusConstants import *
from viewerModules import *
from driverUtils import executeOnCaeStartup

# Model in names
NameModelBase='ModelName'
E1s=(1000,3000,10000,30000,100000,200000)
EpsX=0.3

for i in range(0,6):
    NameModel=NameModelBase+'-E1-'+str(E1s[i])
    ODBfile=NameModel+'.odb'
    RPTfile=NameModel+'.rpt'

    ## Save parameters from simulation
    session.viewports['Viewport: 1'].setValues(displayedObject=
            session.openOdb(name=ODBfile))
    odb = session.odbs[ODBfile]

    # Make path
    session.Path(name='Path-1', type=POINT_LIST, expression=
            ((0.0+EpsX, 0.0, 0.0), (1.0+EpsX,-1.0, 0.0)))
    session.Path(name='Path-2', type=POINT_LIST, expression=
            ((0.0+EpsX, 0.0, 0.0), (0.0+EpsX,-1.0, 0.0)))

    # Extracting data from path
    session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
            variableLabel='S', outputPosition=INTEGRATION_POINT, refinement=(COMPONENT,
            'S11'))
    pth = session.paths['Path-1']
    session.XYDataFromPath(name='XYData-1', path=pth, includeIntersections=True,
            projectOntoMesh=False, pathStyle=PATH_POINTS, numIntervals=10,
            projectionTolerance=0, shape=UNDEFORMED, labelType=TRUE_DISTANCE)
    pth = session.paths['Path-2']
    session.XYDataFromPath(name='XYData-2', path=pth, includeIntersections=True,
            projectOntoMesh=False, pathStyle=PATH_POINTS, numIntervals=10,
            projectionTolerance=0, shape=UNDEFORMED, labelType=TRUE_DISTANCE)
    x0 = session.xyDataObjects['XYData-1']
    x1 = session.xyDataObjects['XYData-2']
    session.writeXYReport(fileName=RPTfile+'1', appendMode=OFF, xyData=(x0))
    session.writeXYReport(fileName=RPTfile+'2', appendMode=OFF, xyData=(x1))
    del session.xyDataObjects['XYData-1']
    del session.xyDataObjects['XYData-2']
    del session.paths['Path-1']
    del session.paths['Path-2']
```

5. Improved experimental procedures extracted from finite elements studies

After a finite element model has been validated against experimental measurements, the model can be used suggesting new improved measurements strategies. Based on experimental validation of the finite element model used for the predictions presented in chapter 2 and 3, an extensive numerical study has been performed using the Python interface shown in chapter 4. Based on those simulations, improved experimental strategies have been suggested lowering the effect of the non-homogeneity of the strain and stress fields on the experimental measurements. Those improved test strategies will be presented below.

5.1 Strain-gauges

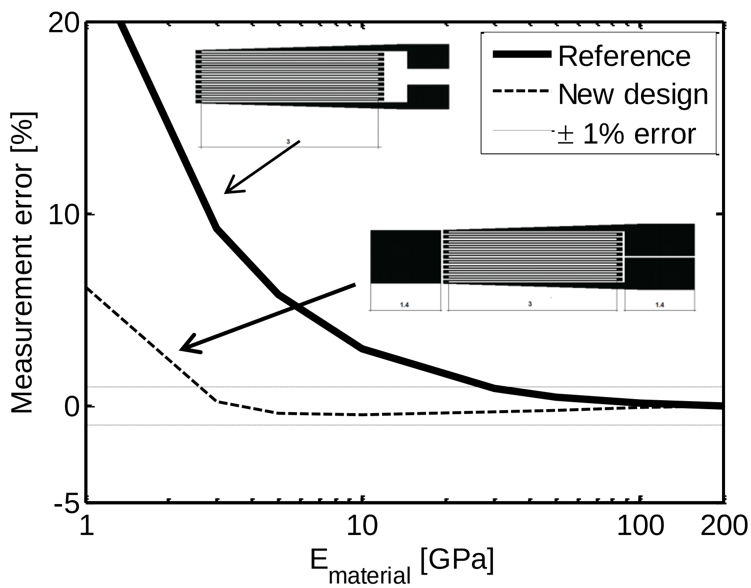


Figure 4. Improved strain gauges design of the 3mm strain gauge case

The lower of the two grid pattern in Figure 4, show an improved strain gauge design resulting in a significant lower erroneous of the stiffness prediction for a large range of material stiffness. A strain gauges design which has been presented in the patent application (Mikkelsen and Zike, 2014). Using a strain gauge grid pattern which includes reinforcement parts at the end of the strain gauges it is possible to move the non-homogeneous strain field outside the measuring part of the strain gauges as illustrated in Figure 5. The improved strain gauges design has not yet been validated experimentally.

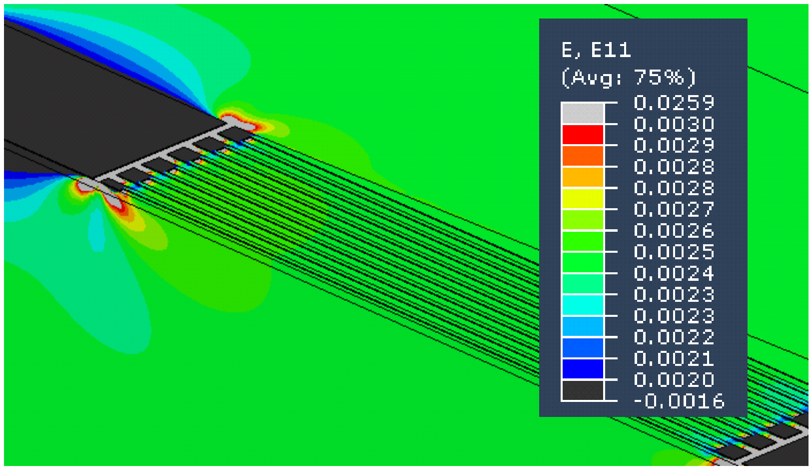
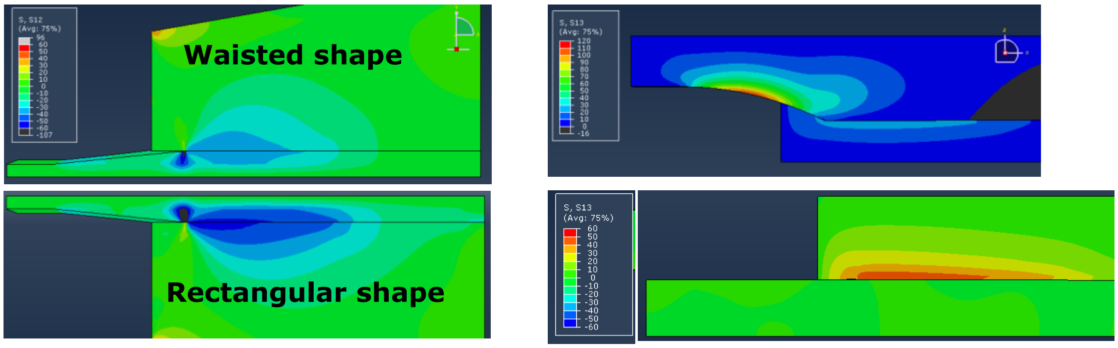


Figure 5. Moving the non-homogenous strain field outside the measuring grid.

5.2 Tensile testing

A way to decrease the shear stress level in the gripping region in a tensile test coupon is by using a waisted shape as illustrated on the upper right figure in figure in An improved waisted shape of the tension-compression test coupon is presented in the two top figure in Figure 6. In figure 6, a comparison of tab shear stresses (left figures) and the splitting shear stresses (right figures) are shown for the waisted (upper figures) and the rectangular (lower figures) test coupons geometries. Comparing the stress states, it can be seen that the tab shear stresses are lowered significant but with the drawback that there will be some splitting shear stresses. The optimal design will make a good compromise between these two shear stresses depending on the actually shear strength of the test coupons in the two loading directions. No matter what, the waisted test geometry has experimentally been found to give improved measured material strength compared with the rectangular ISO specimen.



(a) Tab shear side view **(b) Splitting shear top view**

Figure 6. Comparing the shear stresses in waisted and a rectangular tensile-compression test coupon.

6. Conclusion

Finite element simulations have been used validating and improving the material test procedure regarding stiffness and strength/fatigue determination of composite materials. The work is related testing composite materials for wind turbine blade. An application where a low material price and weight together with high material stiffness and fatigue properties are essential designing larger wind turbines with an resulting lower cost of energy. During the study, an improved strain gauge grid design and a test coupon shape has been proposed.

7. Acknowledgement

This research was supported by the Danish Centre for Composite Structure and Materials for Wind Turbines (DCCSM), grant no. 09-067212, from the Danish Strategic Research Council (DSF).

8. References

1. Brøndsted, P., Lilholt, H., Lystrup, A., 2005. Composite materials for wind power turbine blades. Annu. Rev. Mater. Res. 35, 505–538.
2. Emerson, M.J., Jespersen, K.M., Jørgensen, P.S., Larsen, R., Dahl, A.B., 2015. Dictionary based segmentation in volumes, in: 19th Scandinavian Conference on Image Analysis,. pp. 1–12.

3. Hojo, M., Sawada, Y., Miyairi, H., 1994. Influence of clamping method on tensile properties of unidirectional CFRP in 0° and 90° directions — round robin activity for international standardization in Japan. Composites 25, 786–796.
4. ISO-527-1, 1997. 527-1: Plastics—Determination of tensile properties—Part 1: General principles.
5. Mikkelsen, L.P., Bech, J.I., 2013. Secondary stress effects during load introduction into unidirectional composite test coupons, in: Thomsen, O.T., Sørensen, B.F., Berggreen, C. (Eds.), 6th Int. Conf. on Composites Testing and Model Identification. Aalborg.
6. Mikkelsen, L.P., Zike, S., 2014. A strain gauge. Patent Application: EP14180022.7.
7. Xie, M., Adams, D.F., 1995. Effect of Specimen Tab Configuration on Compression Testing of Composite Materials. Compos. Technol. Res. 17, 77–83.
8. Zike, S., Mikkelsen, L.P., 2014. Correction of Gauge Factor for Strain Gauges Used in Polymer Composite Testing. Exp. Mech. 54, 393–403.