Using the existing capability in Abaqus to model the draping & consolidation of composite preforms.

Colmar Wocke, Ali Siddiqui

The Dow Chemical Company, Bachtobelstrasse 3, CH-8810 Horgen, Switzerland

Abstract: Composite preforms are loosely-held assemblies of fibres. There is often a desire to be able to predict how preforms will drape under the action of gravity, say, or what level of force is required to press the preform into a predefined mould shape.

User routines in Abaqus/Standard or Abaqus/Explicit may be developed to adequately describe the preform kinematic/mechanical behaviour. Such routines need to be developed theoretically and then implemented and verified on the computer. However, Abaqus has a built-in capability that may be profitably used for solving this problem. The internal bookkeeping and consistent theory already exists as a skeleton in Abaqus; to solve this specific problem one needs to suitably build a model that captures the major variables of interest for ultimately solving the real engineering problem i.e. the draping or consolidation analysis of the preform.

The work is concerned with the development of such a predictive model using a rebar-approach in Abaqus. In this approach, the fibres are held in a "shell" by some other means – perhaps stitched loosely or held by uncured prepreg resin. Any forming deformation rearranges the fibres and needs to do this against the frictional or other restraint offered internally by the "shell". The anisotropic properties of the preform are naturally represented by the model. The model is used to solve some verification problems from the literature, as well as performing draping/forming analysis in a particular mould using a calibrated preform kinematic/mechanical model.

Keywords: *Abaqus, Preforms, Composite, Anisotropic.*

1. Introduction

Composite materials are made from more than one phase of material in such a way that there is synergy of their individual properties; modern fibrous composites have relatively-stiff and/or strong aligned-fibres embedded in a matrix material. In order for such fibrous composites to become more widely-accepted, the production processes need to be tailored with mass-production in mind. One promising technique for economically manufacturing composite parts, is to start with a preform of fibres and then infuse this preform with a thermosetting resin, typically a fast-curing epoxy. The infusion step promises to improve the variability in part performance, minimize wastage and ensure very quick cycle times. Work has been conducted at Dow in this regard, as evidenced by VORAFORCE™ resins that have been optimized for quick cycle resin transfer moulding (RTM).

The anisotropic permeability, typical of such preforms, may affect the ability to easily-infuse the part and may necessitate the introduction of alternative injection gates and/or injection strategies. Dow's investigation (Siddiqui, 2012) into the RTM production process of fibrous preforms has highlighted the desirability of being able to make predictions of how such preforms will conform to, or drape onto, mould surfaces. Thus, there is a need for a predictive model which is able to track the rearrangement of fibres and use this information to set up an anisotropic permeability model as part of a second infusion analysis step. The setting up of such a fibrous preform kinematic/mechanical model is the subject of the present paper.

We follow a previous approach (Jauffrès, 2010), hereafter referred to as the reference paper, in which the behaviour of the said preform is represented by a finite element model. However, instead of developing a special routine for this in the form of a **`UMAT`** or **`VUMAT`**, as has been done in this referenced work, we are able to model the problem by using a built-in capability within Abaqus viz. the rebar reinforcing of shell elements. The fibres are held in a preform "shell" by some other means – typically stitched loosely as is shown in Figure 1. Subsequent deformation rearranges the fibres against the restraint offered internally by the "shell" – this internal restraint can in principle be quite general.

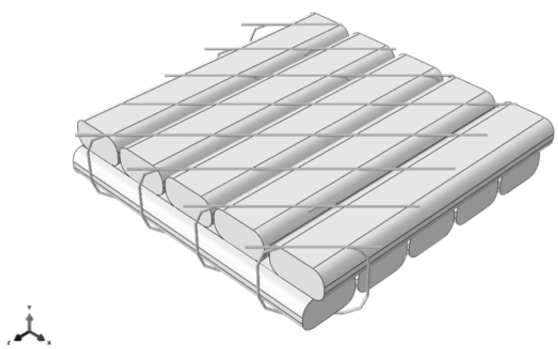

Figure 1. Example of a loosely-stitched representative element of preform.

We do not study a true verification problem with an accepted theoretical solution but have used the work in the reference paper to act as a gauge for the suitability of our approach. We solve the bias extension test from this reference, in which a preform is extended along a loading axis oriented at ±45° from the dominant warp and weft directions respectively. In addition, we solve a reverse draping problem, very similar to the "Stretching of a thin sheet with a hemispherical punch", documented as Problem 1.3.3 in the Abaqus Example Problems Manual (SIMULIA, 2013). Enough information is listed in the reference paper for us to duplicate their results in character and with quite close numerical correspondence for both problems. Lastly, we have used the method on solving some problems of interest to us viz. the draping of a double dome and a complex tool draping/consolidation analysis.

2. Theoretical background on modeling the preform deformation

Abaqus embeds a discrete rebar in the underlying shell element and replaces it with an equivalent orthotropic smeared layer, suitably oriented. This orientation may be w.r.t. any global coordinate system, however in Figure 2 it is shown as an angle θ, referencing the isoparametric coordinates in the shell element midplane. Deformation stretch of the rebar neighbourhood is obtained from the underlying shell element deformation gradient, F - along the rebar axis and perpendicular/normal to the rebar. The stretch vector component, λ_r along the rebar direction $\boldsymbol{T}$, together with the rebar material determines the internal force generated whilst the stretch λ_p along direction $\boldsymbol{P}$ affects the rebar spacing. Finite strain shells also have a thickness stretch, λ_t along direction $\boldsymbol{N}$ which is determined by an effective section Poisson's ratio. The rebar itself only resists deformation through a 1-dimensional axial stress, σ_{11} - all other components of stress only exist in the underlying shell.

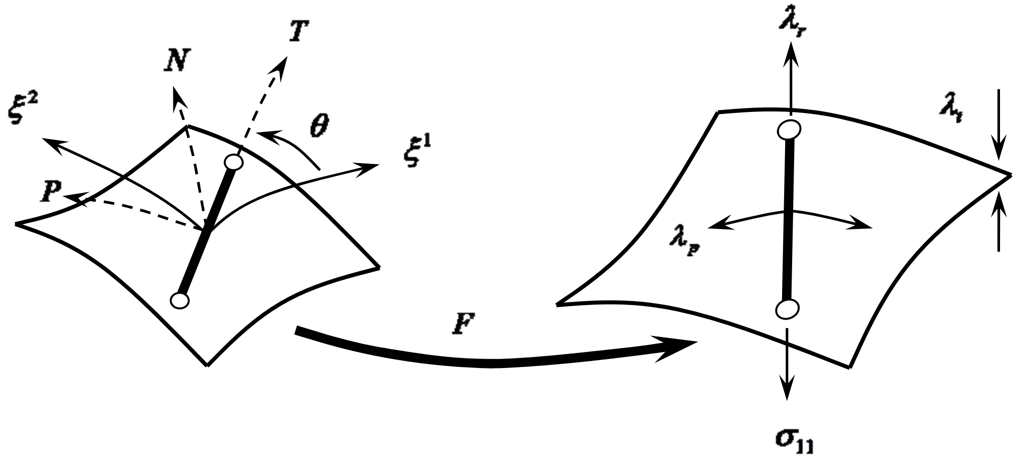

Figure 2. Deforming rebar embedded in the deforming shell surface.

The full Abaqus treatment of the non-linear theory is obviously complex, but a linear-elastic small-strain explanation for the case of a discrete rebar embedded in an isotropic shell, highlights how the load-apportioning is done. Consider a thin lamina as in Figure 3, with equilibrating forces per unit width, R referenced along the global x-y direction; $\boldsymbol{z}$ is the thickness coordinate. In this figure, the smeared rebar layer is embedded in the shell lamina and shown as the included dotted square – the full lamina response consists of a shell contribution and a rebar contribution, labeled ***shell*** and ***rebar*** respectively.

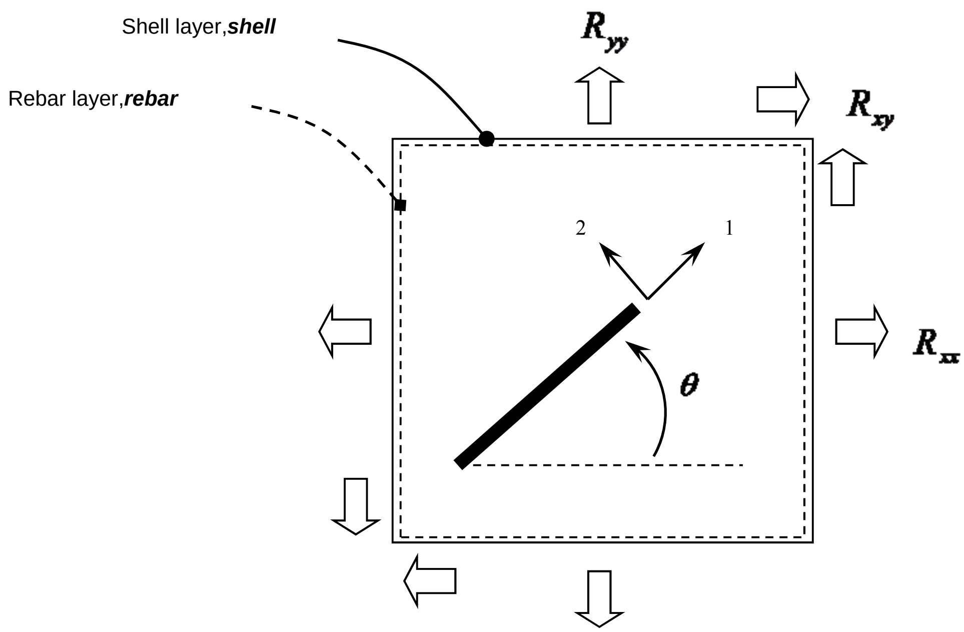

Figure 3. Shell lamina surface, with embedded rebar layer.

Considering a force balance and following notation commonly-used for laminated composites (Herakovitch,1998):

$$\{R\} = \int \{\sigma\}_{xy} dz$$

Equation 1

The stress state, $\{\sigma\}$ of any such lamina, or contributing layer, is given by:

$$\{\sigma\} = [Q]\{\varepsilon\}_m$$

Equation 2

where $[Q]$ is the relevant plane stress stiffness matrix (symmetric) and $\{\varepsilon\}_m$ are the membrane strains .

Since the membrane strains (stretches, in the non-linear case) are the same for the shell and rebar contributing layers and noting that the rebar only resists an axial force, we get:

$$\{R\} = \left\{ \begin{bmatrix} m^4Q_{11} & n^2m^2Q_{11} & nm^3Q_{11} \\ n^2m^2Q_{11} & n^4Q_{11} & n^3mQ_{11} \\ nm^3Q_{11} & n^3mQ_{11} & n^2m^2Q_{11} \end{bmatrix}^{rebar} t_{rebar} + \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{bmatrix}^{shell} t_{shell} \right\} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix}$$

Equation 3

with $m = cos\theta$, $n = sin\theta$ and the positive direction of rotation is as shown in Figure 3.

If the rebars come in equally-thin pairs lying at ±45° to the loading direction initially (such as may approximately be the case in a plain weave with equal numbers of warp and weft reinforcement in the tensile bias-test) then:

$$\begin{Bmatrix} R_{xx} \\ 0 \\ 0 \end{Bmatrix} = \left\{ \begin{bmatrix} \frac{1}{4}Q_{11} & \frac{1}{4}Q_{11} & 0 \\ \frac{1}{4}Q_{11} & \frac{1}{4}Q_{11} & 0 \\ 0 & 0 & \frac{1}{4}Q_{11} \end{bmatrix}^{rebar} 2t_{rebar} + \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{bmatrix}^{shell} t_{shell} \right\} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix}$$

Equation 4

For cases where the warp and weft are not equal, the equation is similar, but weighted differently with the individual thicknesses. The bias-test is a complex loading case and the internal resistance is due to both fibres and the matrix phase. However, in practical cases the matrix is ***much more flexible*** than the fibrous phase, so in the bias-test the fibres are dominant. This is more true at higher and higher extensions when the evolving angular alignment adds ever more stiffness to the rebar contribution in Equation 4; the rebar tends to align with the force. A simple load-apportioning equation can be similarly set up for other cases.

Equations 2 to 4 are clearly simplified versions of the full truth. However, they do enable a feel to be gained for the relative apportioning of the loading into the fibre phase of the preform and the matrix phase of the preform and for identifying tests to be used for determining suitable data for the model.

3. A suitable FEM model

In the Abaqus implementation, the internal resistance to fibre rearrangement is specified in the shell description and the fibres added as reinforcement; in all cases S4R elements were used to allow easy transfer of models between Abaqus/Standard and Abaqus/Explicit The rebar layer is summed into the stiffness contribution of the matrix phase, into the consolidated response of the whole shell – the requisite volumetric space need not be available, in reality.

In terms of the keyword-input, the matrix is described by the following template for a homogenous shell:

```
*SHELL SECTION, ELSET = SHELL,MATERIAL = MATRIX
<shell_thickness>,<number_of_integration_pts>
```

The fibrous phase is described by the following template:

```
*REBAR LAYER
Identifying_name,<Area_of_Rebar>,<Spacing>,<Position_of_Rebar>,
<Material_name>,<Orientation>,η
```

As written in the above keyword-template, the orientation angle is specified w.r.t the local η-direction as shown in Figure 4. This is usually the default directions of the element which is dependent on the numbering sequence of the nodes in the element connectivity list. This direction can be re-oriented using the `*ORIENTATION` keyword.

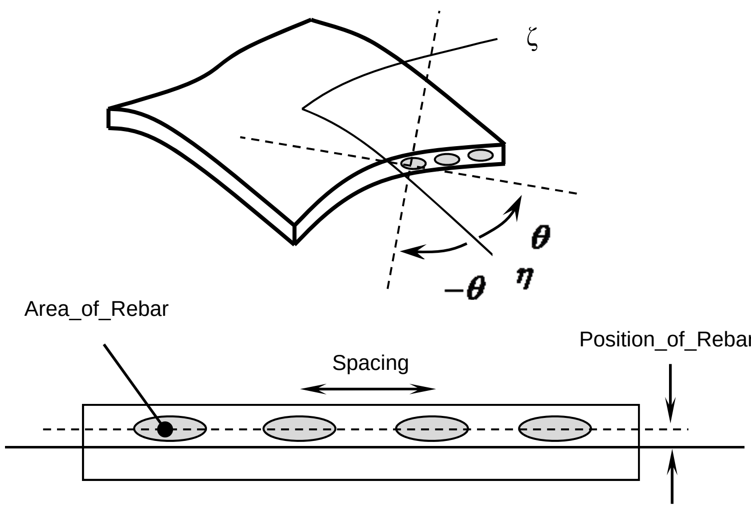

Figure 4. Discrete reinforcement in shell, at orientation to isoparametric directions.

In order to remove this coupling from the problem – in plane direct/shear coupling and membrane/bending coupling in general, the following form is used:

```
*REBAR LAYER
Plus_rebar,<HalfArea_of_Rebar>,<Spacing>,0.0,<Material_name>,<Plus_angle>,η
Minus_rebar,<HalfArea_of_Rebar>,<Spacing>,0.0,<Material_name>,<Minus_angle>,η
```

3.1 Axially-loaded rebar material model for fibrous phase

In such woven fabrics, there may even be a small amount of initial slack that needs to be modeled; this is a less-stiff zone in the deformation history of the fibre, before it is fully-taut. Dealing with such internal slack, can also be easily accomplished using the rebar-technique and treating the fibre as hypoelastic as shown in Figure 5.

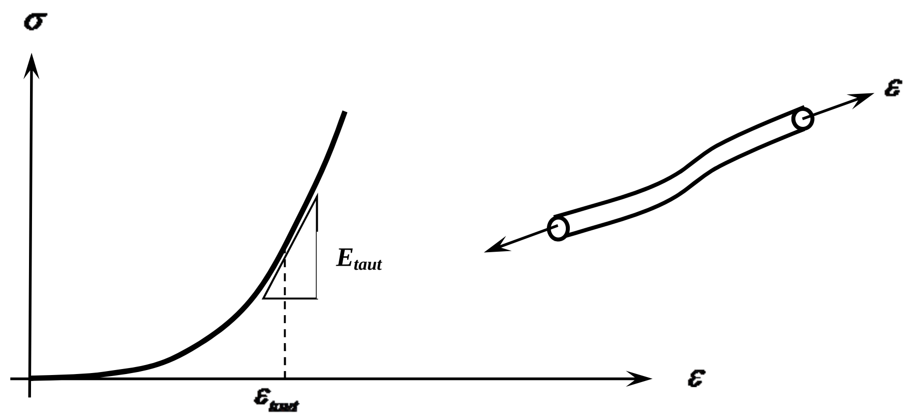

Figure 5. Initial slack in fibre, before reaching the fully-taut modulus.

The keyword-format for specifying such slack in the fibre as a function of the strain, via strain invariants, is:

```
*MATERIAL, NAME = REBAR_MATERIAL_SLACK
*HYPOELASTIC
<Tangent modulus>,<Poisson's ratio>,<I1>,<I2>,<I3>
.
```

Of course, the rebar can also be considered as being fully stiff from the beginning, as is normally-done. The hypoelastic material model is only available in Abaqus/Standard.

3.2 Shell implementation of material model for the internal resistance

As the basic shell, acting invisibly behind-the-scenes in which the fibres are embedded is deformed, the fibres slide over one another and experience internal ***shearing*** resistance as shown in Figure 6.

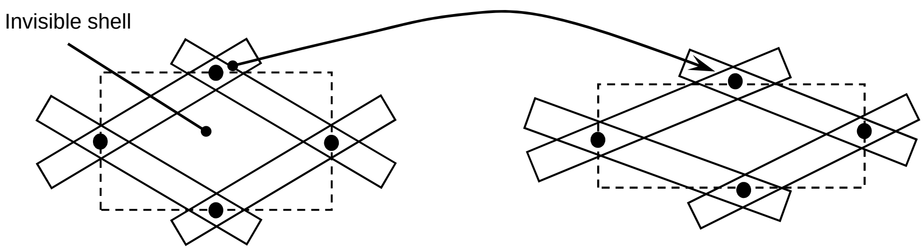

Figure 6. Shearing resistance at junctions modeled via invisible shell.

Thus, the most natural description of this would be a J_2-plasticity model, as is found in Abaqus using the following keyword structure:

```
*MATERIAL, NAME = MATRIX
*ELASTIC
.
*PLASTIC
.
```

We have the yield surface (writing the stresses w.r.t. the principal axes) defined in terms of the ***shear stress*** able to be supported by the material point, before yielding or slippage occurs, as:

$$3\tau^2 = \frac{1}{2}\left[\left(\sigma_{11} - \sigma_{22}\right)^2 + \left(\sigma_{22} - \sigma_{33}\right)^2 + \left(\sigma_{11} - \sigma_{33}\right)^2 + 6\left(\sigma^2{}_{23} + \sigma^2{}_{31} + \sigma^2{}_{12}\right)\right]$$

Equation 8

An elastic-perfectly plastic description, would model sliding after a particular value of shear allowable has been exceeded at the junction between fibres themselves and also perhaps interacting with stitching fibres. Tension dependency of the internal friction, can also be relatively-easily implemented as shown in Figure 7. In the most severe case, an internal binding i.e. locking can also be included. As the overlaying fibres become tensioned, they bind more strongly onto their supporting fibres, which then increases the local frictional force that needs to be overcome for slippage or rearrangement to occur.

A convenient form, fairly-easy to calibrate, for modeling the internal friction is the Drucker-Prager ideal plasticity model:

```
*MATERIAL, NAME = MATRIX
*ELASTIC
<matrix_e>,<matrix_poisson>
*DRUCKER PRAGER
<friction_angle>,<dilation_angle>
*DRUCKER PRAGER HARDENING
<drucker_matrix_yield>,0.0
```

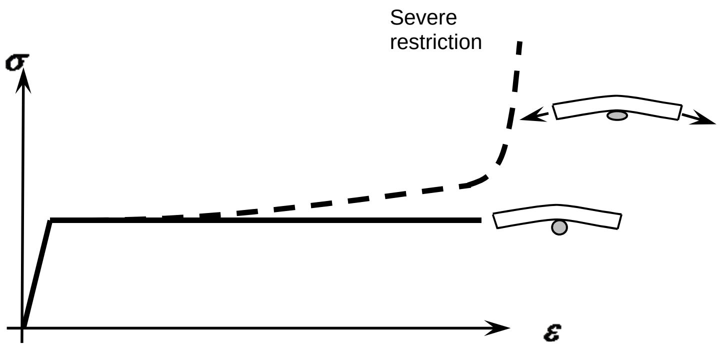

Figure 7. Modeling internal frictional tightening in the invisible shell.

4. Test problems

4.1 Bias tension test

In the bias tension test, a strip of fabric typically twice as long as it is wide, is loaded in tension. The problem analysed in Abaqus/Standard – for both the plain weave (PW) case or twill weave (TW) case was a rectangular strip 115mm wide by 230mm long. This 2:1 aspect ratio leads to an internal shearing deformation field which is divided into 3 clear regions (Jauffrès, 2010); it is used quite often to characterize such fabrics.

Table 1. Glass fabric data (Jauffrès, 2010).

Material Property	PW	TW (Warp/weft)
Fibre volume fraction	0.35	0.35
Consolidated thickness (mm)	0.5	1.0
Linear density – tex (g/1000m)	1870	1870/1870 x 2
Yarn effective cross-section (mm)	1.25	1.25/1.25 x 2
Yarn interval (mm)	4.92	2.44/5.1
Crimp ratio (%)	2	4.4/0.8

The individual fibre stiffness is given i.t.o. the tangent modulus with the strain variable being true strain. This is also how Abaqus interprets strain for the hypoelastic model, thus the reported results are directly useable.

The fibre tangent stiffness varies as a function of strain, as is shown schematically in Figure 5, as:

$$E = C\varepsilon_{true}^{2}$$

Equation 9

Table 2. Fibre stiffness parameters for different weaves (Jauffrès, 2010).

Yarn	C (MPa)	E_{taut} (MPa)	ε_{taut}
PW	1.079E9	19800	0.0264
TW-warp	1.06E8	20080	0.0574
TW-weft	1.68E10	20120	0.0106

Since we are modeling directly via the rebar-approach, we are able to use the reported diameter of the fibres, their spacing and the consolidated thicknesses in our data input to the problem.

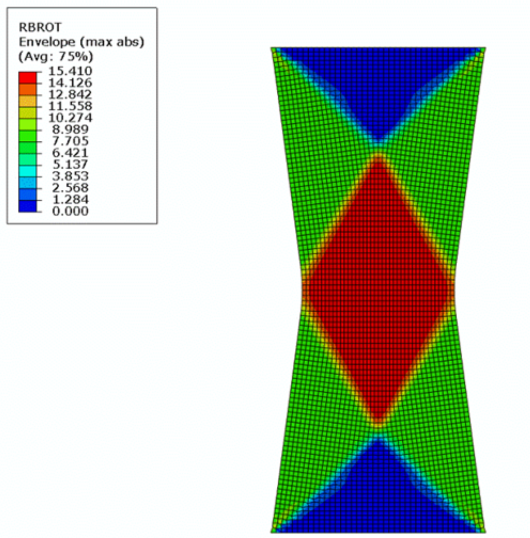

Figure 8. Bias test simulation for the case of PW-fabric, showing three clear zones of shear deformation.

The simulation results from the Abaqus/Standard model, for both the PW and the TW cases, show excellent visual agreement with the reported results in the reference paper. The Abaqus variable **RBROT** has been contoured in Figure 8 – this gives a direct measure of the rotation of the rebar. In this particular case, it represents half the total included shear angle – both the warp and the weft undergo a similar amount of rotation for their respective rebars. The crosshead reaction force vs. crosshead displacement results, as presented in Figure 9, also show excellent agreement with the experimental data from Figures 15 and 16 in the reference paper – some additional fine tuning of the matrix properties will even allow better agreement to be obtained. To obtain these data easily from the Abaqus output, two single nodes (not shown) were tied via respective **`*MPC`** keywords to the edges of the strip; the one node was held fixed, the other was given a prescribed displacement.

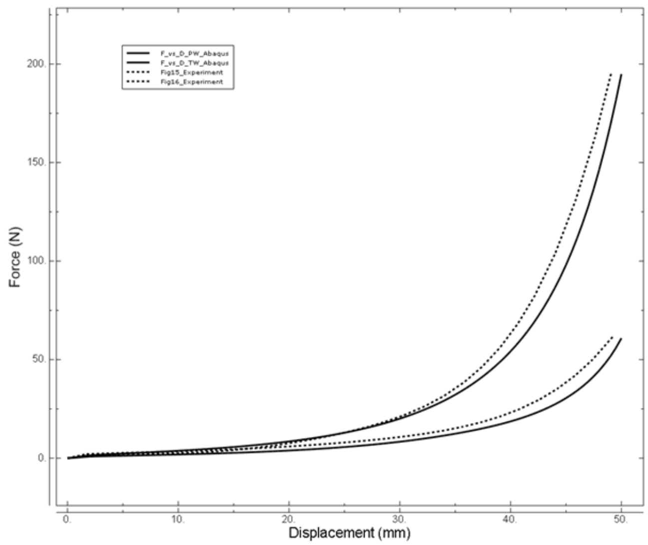

Figure 9. Bias test force vs. displacement comparison - PW and TW fabrics.

4.2 Inverse draping test

In this test, a square piece of preform fabric – a sheet - is pushed into a die using a round-nosed punch. The fabric is laid flat onto the flat peripheral surface of the die and held in place by a binder. The areal size of the deforming square weave is not given in the reference paper in their "hemisphere forming simulation"; this has been estimated from the figures in the paper as being 500mm square. The pertinent geometric dimensions for the analytical rigid surfaces in the problem, are shown in Figure 10. The binding force is held at 650N once contact has been established between preform and the die periphery. Analytical rigid surfaces represent the rigid surfaces with the preform modeled using S4R elements. Friction is the same all over, using 0.3 as the static friction coefficient. The analysis is performed in 3 steps, similar to what was done in Problem 1.3.3 in the Abaqus Example Problems Manual (SIMULIA, 2013):

Step 1 – Establish contact, using softened exponential contact to avoid "numerical chattering" .
Step 2 – Apply the preload onto the binder, via its reference node.
Step 3 – Move the punch reference node by the required displacement.

The punch is moved 100mm downwards in a **`*STATIC`** step in Abaqus/Standard and of course, a **`*DYNAMIC,EXPLICIT`** step in Abaqus/Explicit.

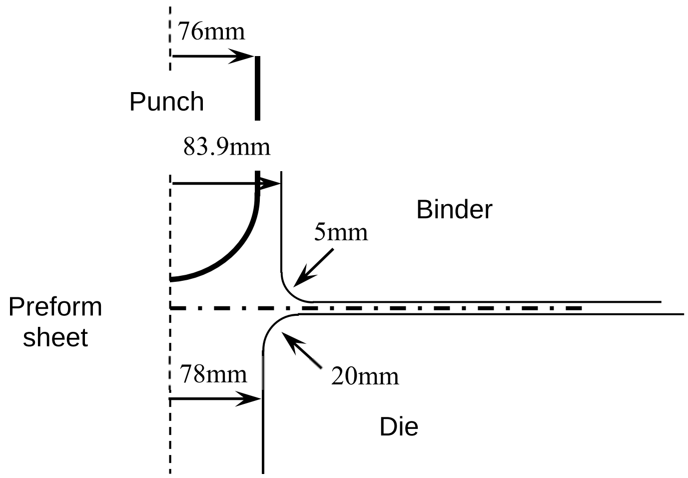

Figure 10. Geometric details of the analytical rigid surfaces.

The deformed shape as shown in Figure 11, shows excellent agreement with Figure 18 in the reference paper. This analysis represents a weave in which the warp and weft directions are aligned with the global x- and y-directions, respectively.

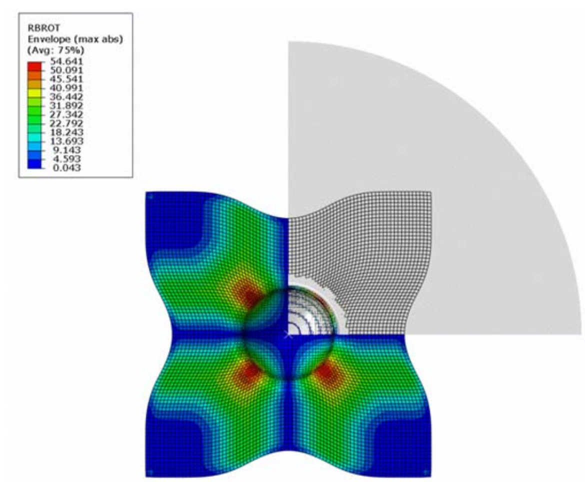

Figure 11. PW fabric deformed `RBROT` contours at end of draping from the Abaqus/Standard analysis.

In the Abaqus/Standard solution of the problem, the contact surfaces and pairing were explicitly defined. When solving the problem in Abaqus/Explicit the contact definitions were replaced with the keywords:

```
*CONTACT
*CONTACT INCLUSIONS, ALL EXTERIOR
```

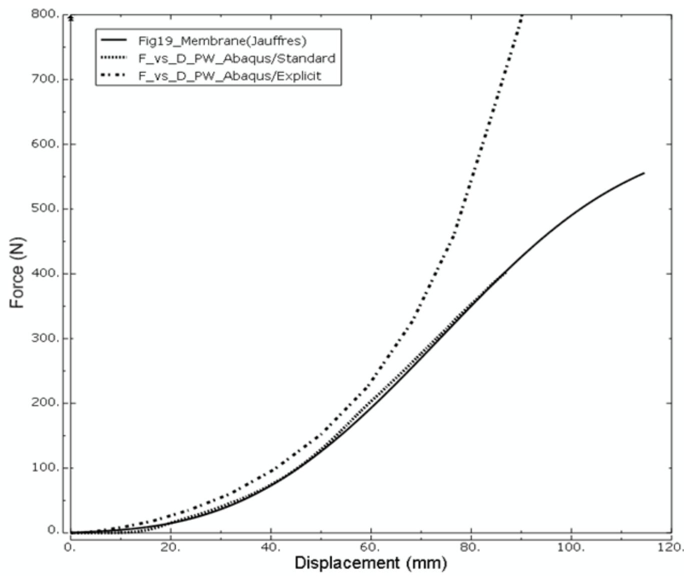

Figure 12. Inverse draping force vs. displacement comparison - PW fabric.

The force versus displacement results are shown in Figure 12. Acceptable agreement is observed between the results obtained from both Abaqus/Standard and Abaqus/Explicit and the results shown in the reference paper for membrane elements. However, there is an over-estimation of the predicted force from the Abaqus/Explicit analysis. Since the hypoelastic material definition is not available in Abaqus/Explicit, rebars are considered fully stiff from the beginning; the over-estimation of the forces is in part, but not wholly, due to this assumption. The force vs. displacement results from the Abaqus/Standard analysis show excellent agreement with Figure 19 in the reference paper and also the deformed shape obtained.

4.3 Draping of complex forms

As an initial test of the suitability of the method on complex tools, the double dome geometry was used for a plain weave (PW) simulation. This is a well-known test (Willems et al.) for investigating the draping of fabrics and is not discussed in further detail here. The geometry is composed of two circular domes connected with a flatter bridging portion, having slanting surfaces, leaving a doubly-symmetric final shape of tool. The solved for variation in the amount of inward sliding of the fabric during draping, is consistent with the differing arc-lengths from point-to-point across the tool surface. The shear angles across the tool surface are symmetric, as indeed they are expected to be.

Dow has developed a split-leg complex tool, designed to manufacture complex parts for the RTM process. The main motivation in developing such a complex tool is to study the effects of geometric features on the ability to drape fabric during preforming as well as the influence on the permeability of the fabric for a subsequent resin filling process. The split-leg complex tool is shown in Figure 13.

Figure 13. Split-leg complex tool.

The forming simulations for the complex tool are performed using Abaqus/Explicit. A plain weave (PW) fabric preform blank, measuring 920mm x 515mm, is modeled with S4R elements of size 10mm. The blank has a thickness of 2mm. The deformed shape at the end of 100mm of tool movement, is shown in Figure 15. The circular part of the split-leg tool is analogous to the double dome geometry; there is a similar, symmetric-like behaviour of the fabric. At the other end of the tool where the geometry is asymmetrical, there is some wrinkling and overlapping of the fabric. The orientation of the fabric in the region where the two legs of the geometry meet is different, since the local slopes are different there.

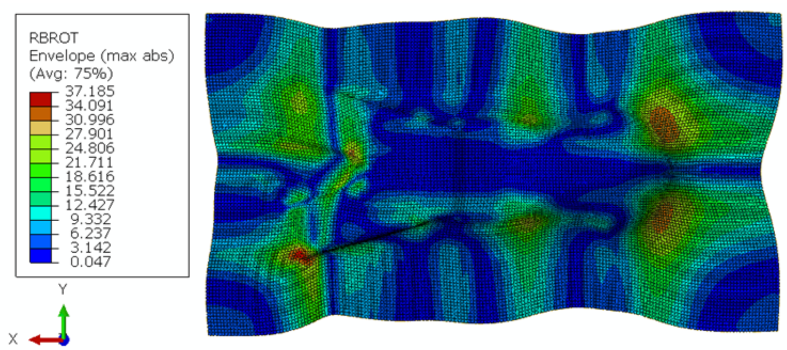

Figure 15. PW fabric deformed RBROT contours at end of draping of complex tool.

5. CONCLUSIONS

The technique of solving the preform draping/consolidation problem using rebars in shells, works acceptably well. We can summarize the advantages as follows:

- No `UMAT` or `VUMAT` routines for Abaqus needed to be developed.
- The fibrous weave need not align with the mesh. This is a great advantage when performing analyses on complex geometries.
- Complicated weave architectures, say triaxial weaves, can be relatively-easily accommodated.
- The shell behaviour may, in the general case, be a prepreg resin with some viscous behaviour.
- Abaqus reports the rotation of the fibrous-rebar, which is directly useful in setting up models for the preform permeability for performing the RTM modeling, to follow as a separate analysis.

Tuning the different properties may be fairly-easily accomplished, as has been shown in "**Theoretical background on modeling the preform deformation**". In all cases, we have solved the membrane version of the problem, since the rebars were always placed on the neutral axis of the shell. It is possible to adjust this and to introduce more bending resistance. However, initial trials with this for the PW-case showed that this was not so straightforward. It is suspected that this may be due to the shell being extremely flexible since it is so thin (0.5mm consolidated thickness).

6. References

1. Siddiqui, M. Ali, "High Pressure RTM Process modeling of Carbon Fibre Composites with Ultra-Fast-Cure resins", Kunststoffe + SIMULATION 2012, 23 & 24 April 2012, München, Germany.
2. Jauffrès, David, Sherwood, James A., Morris, Corey D., Chen Julie, "Discrete mesoscopic modeling for the simulation of woven-fabric reinforcement forming", International Journal of Material Forming (2010) 3 (Suppl. 2):S1205–S1216.
3. Abaqus 6.11 Documentation Collection, Abaqus Example Problems, Keywords & Theory Manuals, Dassault Systèmes Simulia Corp.
4. Herakovitch, Carl T. Mechanics of Fibrous Composites, John Wiley & Sons, ISBN 0-471-10636-4.
5. Willems A., Lomov S. V., Vandepitte D. and Verpoest I., Double dome forming simulation of woven textile composites, 7th International Conference ESAFORM 2006, Glasgow, UK, April 26-28,2006, S747-S750.