Optimisation of Thick Tapered Composite Structures using Isight

D. Smith

AgustaWestland, Lysander Road, Yeovil, Somerset, BA21 2YB

Abstract: Thick composite laminate structures are now widely adopted within the aerospace industry as they offer significant benefits such as saving weight, damage tolerance and the potential for the tailoring of mechanical properties. An inherent problem in laminate plates under load is the generation of interlaminar (or through-thickness) stresses in regions of thickness change which can lead to splitting (or delamination) between plies. It is therefore important to ensure that interlaminar stresses are minimised at the design stage in order to prevent premature failure.

A layup optimisation study is presented which reviews the effectiveness of adopting the Simulia Isight 5.6-1 software package for automating the iterative process of adjusting ply layup orientations with the aim of minimising interlaminar stresses. By running a finite element model (FEM) through a Design of Experiments (DOE) sim-flow, the method offers the potential to significantly reduce iterative cycle times and generate a thorough evaluation of the sensitivity of layup changes on interlaminar effects.

Keywords: Tapered composites, Ply-drop, Interlaminar Stress, Optimisation

1. Introduction

The paper presents a study exploring the use of Simulia Isight 5.6-1 as a tool for automating and evaluating the effects of layup changes on the interlaminar stresses generated in the tapered region of a unidirectional (UD) glass-fibre composite laminate plate.

The thick composite plate represents a potential replacement for a titanium flexure-link on a helicopter rotor hub. By nature, the flexure-link will be exposed to very high centrifugal and bending loads generated by the rotor blades.

To meet dynamic obligations, the inboard portion of the plate is very stiff whereas the mid-section is designed to be flexible to allow for the desired hinging requirement of the rotor. To achieve the correct stiffness distribution, it is necessary to vary the thickness of the laminate from thick to thin by terminating (or dropping-off) plies to create a tapered region (see Figure 1). Plies are dropped off internally rather than on the external surface, as this is found to reduce the risk of delaminations (Cairns, 1999).

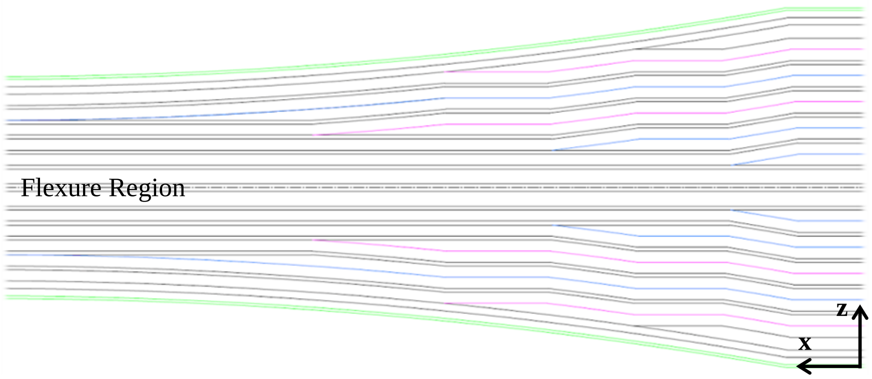

Figure 1. Internal ply drop-off scheme.

The interlaminar stresses in the laminate structure are evaluated using the finite element method. The ply layup in the thin flexure region is fixed to meet the dynamic stiffness requirement; therefore it is only possible to tailor the orientations of the terminating plies. Even so, the possible number of layup combinations demands for evaluation of a large number of iterations in order to make a reasonable sensitivity assessment. A 3D solid element FEM would be the preferred approach but would be costly in terms of resource and time, therefore, a 2D plane stress approach is adopted. To provide confidence in the 2D method, a 2D section FEM is compared with a 3D FEM of equivalent layup.

The Isight framework software package is used as a tool for constructing a simulation work flow (or sim-flow) to automate the iterative process and make a sensitivity assessment of layup changes in terms of interlaminar stress.

2. Sim-Flow

A DOE component links together a collection of tasks within a process-flow loop (see Figure 2). The simulation parameters and goals are defined within the DOE such that when executed, data is passed from one task to another in the necessary order to achieve the desired output.

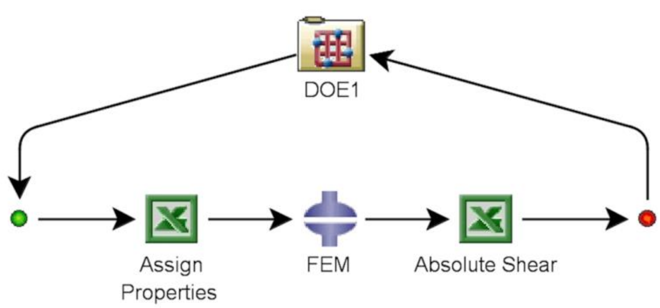

Figure 2. Simulation work flow.

2.1 Design of Experiments

The DOE component assesses the impact of lay-up orientation changes on the magnitude of interlaminar stress. This is done by generating a design space matrix of layup iterations which are passed one-by-one through the sim-flow loop (see Figure 2). On completion of each cycle, the effects on interlaminar stress are collated into the design space. An approximation model is used to fully populate the design space around which optimisation techniques are employed to find an ideal solution (see section 2.4).

Only the plies terminating in the taper region are available for layup tailoring. These are grouped into six laminate modules as labelled in Figure 3.

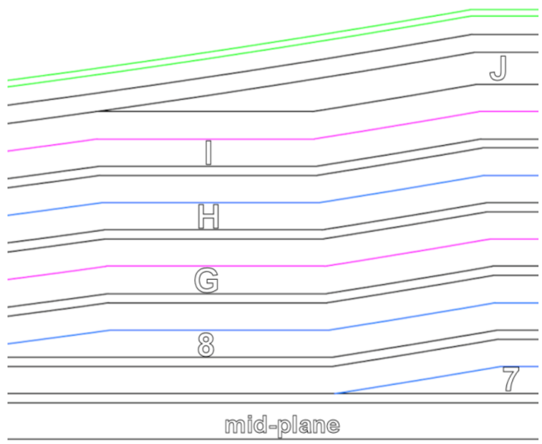

Figure 3. Interchangeable laminate modules.

Each laminate module contains either six or seven plies. In order to define the laminate layups, the design space matrix contains four numeric values for each module. These values represent either bias or single plies as defined in Table 1, e.g. for a $[0,90,\pm45_2]$ laminate, the numeric quantities passed into the dataflow shall be [5,7,7,0]. Note that all laminate combinations are considered symmetric since they will be mirrored about the mid-plane.

It would be impractical to assess all of the potential layup iterations; hence the need to create a design space in which approximations can be made between known results. The DOE component has various options for generating a design space that will produce an acceptable blanket of results. For this study, the Optimal Latin Hypercube technique was selected to create a design space for 220 iterations. This technique is described by (Dassault Systèmes, 2011) as being effective for sampling large design spaces.

Numeric ID.	Ply Orientation
0	*Empty*
1	0°
2	90°
3	[0°,00°]
4	[90°,90°]
5	[0°,90°]
6	[±30°]
7	[±45°]
8	[±60°]

Table 1. Laminate definition identifiers.

2.2 Assigning Material Properties

The composite plate is made from UD glass-fibre with an average cured ply thickness of 0.236mm. The UD elastic constants are reported in Table 2.

E_1	46500 MPa
$E_2 = E_3$	13430 MPa
G_{12}	5010 MPa
G_{23}	4500 MPa
G_{31}	5010 MPa
υ_{xy}	0.3
υ_{23}	0.4
υ_{31}	0.08

Table 2. UD elastic constants.

The dataflow from the DOE feeds the layup identifiers (see Table 1) for each module into an Excel spreadsheet. Based on the lamina properties given in Table 2, the spreadsheet uses classical laminate theory (CLT) in order to calculate the equivalent elastic constants to be substituted into the FEM (Mohite, 2014). The method is summarised as follows.

For an orthotropic lamina, the principle stress-strain relations are:

$$\{\sigma\}_{1,2,3} = [Q]\{\varepsilon\}_{1,2,3} \quad \textbf{(2-1)}$$

Where the reduced stiffness matrix $[Q]$ is defined as:

$$[Q] = \begin{bmatrix} \frac{1-\upsilon_{23}\upsilon_{32}}{E_2E_3\Delta} & \frac{\upsilon_{21}+\upsilon_{23}\upsilon_{31}}{E_2E_3\Delta} & \frac{\upsilon_{31}+\upsilon_{21}\upsilon_{32}}{E_2E_3\Delta} & 0 & 0 & 0 \\ & \frac{1-\upsilon_{13}\upsilon_{31}}{E_1E_3\Delta} & \frac{\upsilon_{32}+\upsilon_{12}\upsilon_{31}}{E_1E_3\Delta} & 0 & 0 & 0 \\ & & \frac{1-\upsilon_{12}\upsilon_{21}}{E_1E_2\Delta} & 0 & 0 & 0 \\ & symmetric & & G_{23} & 0 & 0 \\ & & & & G_{13} & 0 \\ & & & & & G_{12} \end{bmatrix} \quad \text{(2-2)}$$

Where,

$$\Delta = \frac{1-\upsilon_{12}\upsilon_{21}-\upsilon_{23}\upsilon_{32}-\upsilon_{13}\upsilon_{31}-2\upsilon_{21}\upsilon_{32}\upsilon_{13}}{E_1E_2E_3}$$

A composite laminate stack is constructed of layers of various fibre orientations. To obtain the overall laminate stiffnesses, it is useful to define the stress-strain relationships of the constituent layers in terms of a single coordinate system.

In the case of a cross ply laminate where the fibres are inclined at an angle ±θ to the global x-axis, the principle lamina stresses can be described by the relation:

$$\{\sigma\}_{1,2,3} = [T]\{\sigma\}_{x,y,z} \quad \text{(2-3)}$$

Where $[T]$ is the transformation matrix:

$$[T] = \begin{bmatrix} \cos^2\theta & \sin^2\theta & 0 & 0 & 0 & 2\sin\theta\cos\theta \\ \sin^2\theta & \cos^2\theta & 0 & 0 & 0 & -2\sin\theta\cos\theta \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\theta & -\sin\theta & 0 \\ 0 & 0 & 0 & \sin\theta & \cos\theta & 0 \\ -\sin\theta\cos\theta & \sin\theta\cos\theta & 0 & 0 & 0 & \cos^2\theta-\sin^2\theta \end{bmatrix}$$

For matrix $[T]$ to be common to both stress and strain, the principle material engineering strain is described as:

$$\{\varepsilon\}_{1,2,3} = [R][T][R]^{-1}\{\varepsilon\}_{x,y,z} \quad \text{(2-4)}$$

Where $[R]$ is the *Reuters* matrix: $\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2 & 0 & 0 \\ 0 & 0 & 0 & 0 & 2 & 0 \\ 0 & 0 & 0 & 0 & 0 & 2 \end{bmatrix}$

By rearranging (2-1) and substituting in (2-3) and (2-4), the stress-strain relations of an inclined orthotropic lamina are defined as:

$$\{\sigma\}_{x,y,z} = [T]^{-1}[Q][R][T][R]^{-1}\{\varepsilon\}_{x,y,z} \quad \textbf{(2-5)}$$

Here, the matrix product $[T]^{-1}[Q][R][T][R]^{-1}$ is termed the transformed reduced stiffness matrix $[\bar{Q}]$, such that:

$$\{\sigma\}_{x,y,z} = [\bar{Q}]\{\varepsilon\}_{x,y,z} \quad \textbf{(2-6)}$$

The strain variation through the thickness of a laminate is written as:

$$\{\varepsilon\}_{x,y,z} = \{\varepsilon^0\}_{x,y,z} + z\{\kappa\}_{x,y,z} \quad \textbf{(2-7)}$$

Where,

ε^0 represents the mid-plane strains

κ represents the mid-plane curvatures

z represents the distance from the mid-plane in the thickness direction

Substituting (2-7) into (2-6) gives the stress-strain relations in the k^{th} layer in terms of the mid-surfaces strains and curvatures:

$$\{\sigma\}^k_{x,y,z} = [\bar{Q}]^k\left[\{\varepsilon^0\}_{x,y,z} + z\{\kappa\}_{x,y,z}\right] \quad \textbf{(2-8)}$$

The forces and moments acting on a laminate can be determined by integration of the stresses in each lamina, such that:

$$\{N\}_{x,y,z}b = \int_{-h/2}^{h/2}\{\sigma\}_{x,y,z}b\,dz = \sum_{k=1}^{N}\int_{z_{k-1}}^{z_k}\{\sigma\}^k_{x,y,z}b\,dz \quad \textbf{(2-9)}$$

$$\{M\}_{x,y,z}b = \int_{-h/2}^{h/2}\{\sigma\}_{x,y,z}b\,z\,dz = \sum_{k=1}^{N}\int_{z_{k-1}}^{z_k}\{\sigma\}^k_{x,y,z}b\,z\,dz \quad \textbf{(2-10)}$$

Where *N* and *M* are the forces and moments per unit width. Because the laminate shall be exposed to very high in-plane radial load, the strain variation due to curvature through the thickness is expected to be slight. Therefore, for the scope of this work, we need only focus on the terms relating to the in-plane forces, i.e., (2-10) shall be neglected.

Removing the width measurement b and substituting (2-8) into (2-9) gives:

$$\{N\}_{x,y,z} = \sum_{k=1}^{N}[\bar{Q}]^k\int_{z_{k-1}}^{z_k}\left[\{\varepsilon^0\}_{x,y,z} + z\{\kappa\}_{x,y,z}\right]dz \quad \textbf{(2-11)}$$

Because $\{\varepsilon^0\}_{xyz}$ are not functions of z, their terms can be removed from the summation. Therefore, (2-11) can be rewritten as:

$$\{N\}_{x,y,z} = \begin{bmatrix} A_{11} & \cdots & A_{16} \\ \vdots & \ddots & \vdots \\ A_{16} & \cdots & A_{66} \end{bmatrix} [\varepsilon^0]_{x,y,z} + \begin{bmatrix} B_{11} & \cdots & B_{16} \\ \vdots & \ddots & \vdots \\ B_{16} & \cdots & B_{66} \end{bmatrix} [\kappa]_{x,y,z} \tag{2-12}$$

Where,

$$A_{ij} = \sum_{k=1}^{N} \left(\bar{Q}_{ij}\right)_k (z_k - z_{k-1})$$
$$B_{ij} = \frac{1}{2} \sum_{k=1}^{N} \left(\bar{Q}_{ij}\right)_k (z_k^2 - z_{k-1}^2) \tag{2-13}$$

For symmetric laminates, the terms of the $[B]$ matrix will always equate to zero. For the layup optimisation procedure, all of the interchangeable laminate modules will be mirrored about the neutral axis, and as such they are assumed symmetric. Therefore, (2-12) is rewritten as:

$$\{N\}_{x,y,z} = \begin{bmatrix} A_{11} & A_{12} & A_{13} & \dots & \dots & A_{16} \\ A_{12} & A_{22} & A_{23} & \dots & \dots & A_{26} \\ A_{13} & A_{23} & A_{33} & \dots & \dots & A_{36} \\ \vdots & \vdots & \vdots & A_{44} & \dots & A_{46} \\ \vdots & \vdots & \vdots & \vdots & A_{55} & A_{56} \\ A_{16} & A_{26} & A_{36} & A_{46} & A_{56} & A_{66} \end{bmatrix} \begin{bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \varepsilon_z^0 \\ \varepsilon_{yz}^0 \\ \varepsilon_{xz}^0 \\ \varepsilon_{xy}^0 \end{bmatrix} \tag{2-14}$$

The mid-plane strains in terms of in-plane forces are found by inverting (2-14):

$$\{\varepsilon^0\}_{x,y,z} = [A]^{-1}\{N\}_{x,y,z} = [a]\{N\}_{x,y,z} \tag{2-15}$$

Where, $[a]$ is the extensional laminate compliance matrix. Combining (2-15) with the knowledge that the average laminate stresses and Poison's ratio are:

$$\{\bar{\sigma}\}_{x,y,z} = \frac{1}{h}\{N\}_{x,y,z} \tag{2-16}$$

$$\upsilon = -{}^{\varepsilon_{trans}}/{}_{\varepsilon_{axial}} \tag{2-17}$$

The six engineering constants required for the plane stress assumption (see section 2.3) are:

$$E_x = \frac{\sigma_x}{\varepsilon_x} = \frac{1}{a_{11}t} \qquad E_z = \frac{\sigma_z}{\varepsilon_z} = \frac{1}{a_{33}t} \qquad G_{yz} = \frac{\tau_{yz}}{\gamma_{yz}} = \frac{1}{a_{44}t}$$
$$G_{xz} = \frac{\tau_{xz}}{\gamma_{xz}} = \frac{1}{a_{55}t} \qquad G_{xy} = \frac{\tau_{xy}}{\gamma_{xy}} = \frac{1}{a_{66}t} \qquad \upsilon_{xz} = -\frac{\varepsilon_z}{\varepsilon_x} = -\frac{a_{13}}{a_{11}} \tag{2-18}$$

2.3 Finite Element Model

The tapered composite plate is modelled as a 2D section in ABAQUS CAE using linear quadrilateral S4 shell elements (see Figure 4). To ensure a practical element size, equivalent material properties are assigned to discrete modules containing multiple plies (calculated using CLT as described in section 2.2). Linear elastic behaviour is defined by specifying lamina elasticity under the condition of plane stress, i.e. there is zero stress acting perpendicular to the x-z plane ($\sigma_y = 0$).

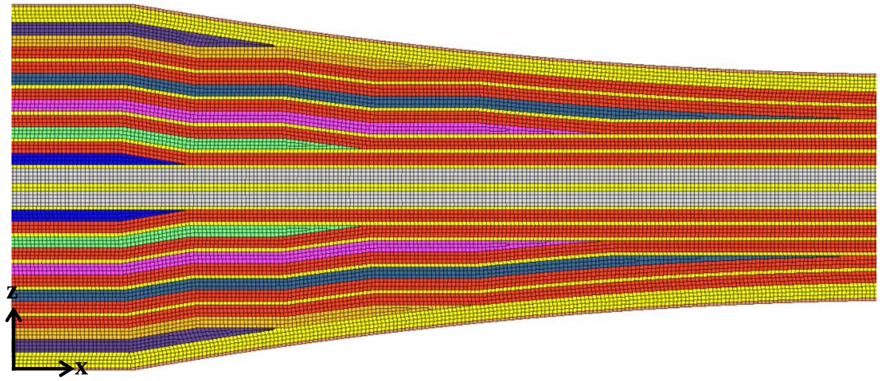

Figure 4. 2D FEM of tapered section.

The plane stress assumption does not account for constraint effects in the width direction; however, there is concern that step changes in the laminate thickness will generate stress through the width of the plate due to changes in Poisson's ratio. To provide confidence in the approach, a 2D plane stress FEM is compared to a 3D FEM of equivalent layup and dimension. The 3D FEM is constructed in MSC Laminate Modeller which builds solid CHEXA (Hex8) elements onto a shell element surface (see Figure 5).

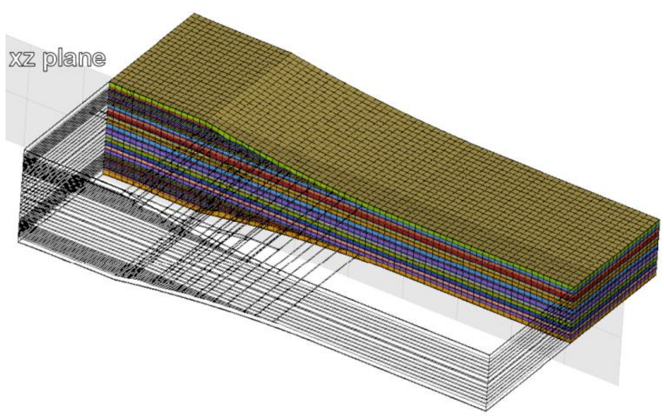

Figure 5. 3D Laminate Modeller FEM.

Both models are fully constrained at the thick end with radial shear and in-plane bending shear applied at the thin end. In order to provide a sensible comparison, results from the 3D FEM are taken on the x-z plane (see Figure 5) so as to minimise the influence of edge effects.

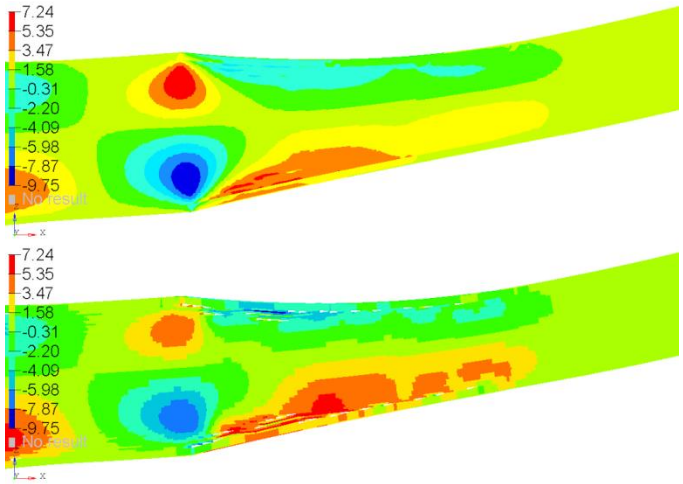

Figure 6. Interlaminar direct stress comparison between 2D FEM (upper) and 3D FEM (lower).

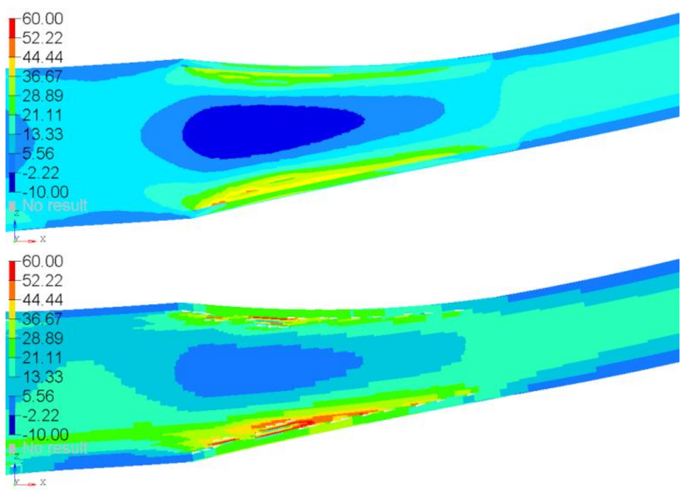

Figure 7. Interlaminar shear stress comparison between 2D FEM (upper) and 3D FEM (lower).

Under the same load condition, both models demonstrate similar levels of deflection. Interlaminar stress plots are presented in Figure 6 and Figure 7. These show comparisons between the two methods for interlaminar direct stress and interlaminar shear stress respectively. Similar stress levels are shown across both models; however, the 3D model exhibits higher local peaks at ply discontinuities which are to be expected. The 2D model gives slightly higher max/min values away from the edges which is most likely due to having a finer mesh density. Due to the good degree of similarity, the 2D plane stress method is considered to give accurate through thickness results.

For the optimisation process, a non-linear solution is requested in order to allow the proper interaction between the radial extension and in-plane flap bending. An additional limitation of the plane stress modelling approach is that we are only able to apply in-plane loads; therefore lead/lag shears are ignored. This is considered an acceptable compromise for the optimisation process as interlaminar stress in the flexure region of the plate is known to be predominately dictated by radial extension and flap bending.

The dataflow into the ABAQUS FEM component (see Figure 2) contains the elastic constants for each of the six laminate modules (see Figure 3) as generated by the Excel spreadsheet (described in section 2.2). During execution of the DOE loop, the ABAQUS component operates by opening the ABAQUS .cae file and substituting the elastic constants for the relevant modules with those from the incoming dataflow. It then generates a unique .inp file for submission to the ABAQUS solver. Once a solution is reached, outputs are required for the maximum interlaminar tension and maximum/minimum interlaminar shears at the element centroid, as well as the associated element identification numbers. Generally, outputs can be requested from the ABAQUS component GUI, however a problem is found with Isight 5.6-1 wherein the ABAQUS component does not allow the option to extract the desired S22 and S12 stress components (see Figure 8) from the .odb file. It is therefore necessary to extract the information using a custom python script:

```
from odbAccess import *
odb = openOdb(path='C:\ISIGHT_RunTime\Flexure_Section.odb')
Ifile = open("C:\ISIGHT_RunTime\user_params.txt",'w')

taper=odb.rootAssembly.instances['PART-1-1'].elementSets['TAPER']
lastFrame=odb.steps['Step-3'].frames[-1]
interlamtension = lastFrame.fieldOutputs['S']
field1=interlamtension.getSubset(region=taper, position = CENTROID)
fieldvalues1=field1.values

maxS22=-0.1
maxS12=-0.1
minS12=0.1
maxElem1=0
maxElem2=0
maxElem3=0
```

```
for v in fieldvalues1:
        if (v.data[1] > maxS22):
                maxS22=v.data[1]
                maxElem1=v.elementLabel
        if (v.data[3] > maxS12):
                maxS12=v.data[3]
                maxElem2=v.elementLabel
        if (v.data[3] < minS12):
                minS12=v.data[3]
                maxElem3=v.elementLabel

Ifile.write('Step_3__taper__Max_InterLaminarTension' + "\t" + '%6.4f' %
(maxS22) + "\n")
Ifile.write('Step_3__taper__Max_InterLaminarShear' + "\t" + '%6.4f' %
(maxS12) + "\n")
Ifile.write('Step_3__taper__Min_InterLaminarShear' + "\t" + '%6.4f' %
(minS12) + "\n")

paramsfile.close()
Ifile.close()
odb.close()
```

Custom scripts are initiated in the sim-flow by selecting the appropriate tick box within the ABAQUS component GUI. To avoid any erroneous results that may be present (such as at boundary constraint locations), the search area for results is focused on the elements in the taper region.

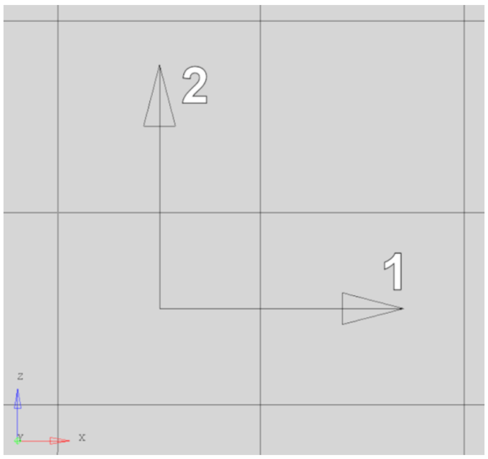

Figure 8. Material coordinate system.

The outgoing dataflow from the ABAQUS component passes the interlaminar tensile stress result directly back to the DOE. The maximum and minimum interlaminar shears are first passed to a second Excel spreadsheet, which determines the absolute maximum shear (see Figure 2), before being passed to the DOE.

2.4 Optimum Design Approximation

With the DOE design space populated, an approximation model is generated. A Radial Basis Function (RBF) approximation algorithm is used to generate a simplified mathematical model around the output parameters obtained by the DOE (Dassault Systèmes, 2011). The objective is set to search the design to find the ideal elastic constants for the six laminate modules (see Figure 3) such that the interlaminar stresses in the taper region shall be minimised.

Isight offers an extensive range of optimisation techniques for finding an ideal solution based on the approximation model (Van der Velden, 2010). For this study, the large-scale generalised reduced gradient (LSGRG) optimisation technique is used to find the optimum design point. The technique was selected based on trial and error by comparing the predicted minimal interlaminar stress with those seen by assessing the FEM using the optimum elastic constants.

3. Results

The evaluated design space gives the following ranges of interlaminar direct stress and shear stress:

$$\sigma_z = 4.2MPa - 11.5MPa$$

$$|\tau_{xz}| = 21.4MPa - 35MPa$$

The design search of the approximation model predicts the optimum achievable through thickness stresses to be:

$$\sigma_{z\,max} = 4.2MPa$$

$$|\tau_{xz}|_{max} = 20.8MPa$$

The elastic constants associated to the optimum design are reported inTable 3. It is found that through thickness direct and shear moduli are not significantly affected by layup change; hence we are to focus only on the x-direction elastic modulus.

The values given in Table 3 shall be implemented as guidelines for generating layups of comparable (or best-fit) stiffness using CLT (described in 2.2) by incorporating only the ply orientations outlined in Table 1.

The derived best-fit layups for each module are as reported in Table 4. Re-evaluation of the FEM using these values predicts the maximum interlaminar stresses to be:

$$\sigma_{z\,max} = 3.9MPa$$

$$|\tau_{xz}|_{max} = 22.1MPa$$

Although some degree of error would be expected due to the need to 'best-fit' the recommended elastic constants, a close correlation is observed both in terms of interlaminar tension and interlaminar shear.

Module	E_x
7 (6-ply)	33184
8 (6-ply)	25231
G (6-ply)	32830
H (6-ply)	31272
I (6-ply)	31704
J (7-ply)	20257

Table 3. Optimum elastic constants.

Module	Layup	E_x	E_z	ν_{xz}	G_{xz}
7 (6-ply)	[0,0,±30,±30]	33439	13860	0.18	4925
8 (6-ply)	$[\pm30]_3$	26331	14040	0.14	4883
G (6-ply)	[0,0,±30,90,90]	30029	14417	0.32	4798
H (6-ply)	[0,0,±30,90,90]	30029	14417	0.32	4798
I (6-ply)	[0,0,±30,90,90]	30029	14417	0.32	4798
J (7-ply)	[±45, ±45, ±30,90]	19553	14441	0.24	4755

Table 4. Best-fit layups.

4. Conclusion

The work assesses the use of Simulia Isight 5.6-1 for automation and optimisation of a thick tapered composite laminate in order to minimise interlaminar stresses. The software offers an easy to understand working environment for constructing sim-flows within a DOE and contains many useful tools suitable for the manipulation and post processing of finite element models.

The Isight sim-flow has been found to be very useful for automating a procedure that requires many iterative steps by eliminating the need to manually change simulation design variables and re-submit work.

Previous to this study, similar automation tasks have been undertaken by writing bespoke code to run numerous iterations. These codes have been similarly successful in terms of process automation; however, Isight 5.6-1 offers additional benefits such as the definition and ranking of analyses in terms of design targets.

A significant advantage of using Isight for iterative design tasks is in its ability to create approximation models around a known set of results. This means it is not necessary to analyse every possible design combination (or in this case layup) in order to accurately estimate the optimum design point. To demonstrate this, the work contained herein would require over 500,000 design points to fully assess all possible layup iterations. For a FEM that takes approximately ten minutes to run (using a single processor), this number of iterations is completely impractical. Instead, the Isight 5.6-1 optimisation tool estimates the ideal design point to achieve minimal interlaminar stresses based on an approximation model generated from a more manageable 220 iterations. The Isight estimation generated from these results has been shown to be accurate by incorporating the optimised layups into the FEM.

Isight 5.6-1 has been found to offer huge benefits for iterative design tasks. Due to a simple means of process automation, it can lead to significant cut-backs on labour time. In addition, it is capable of making design optimisation estimates to a good degree of accuracy.

5. References

1. Cairns, D.S., J.F. Mandell, and J.Z. Maccagnano, "Design and manufacturing considerations for ply drops in composite structures", Composites: Part B 30, 1999
2. Dassault Systèmes: Isight Component Online Help , 2011
3. Dassault Systèmes: Abaqus 6.13 Documentation , 2013
4. Van der Velden, A., P. Koch, "Isight Design Optimisation Methodologies", ASM Handbook Volume 22B Application of Metal Processing Simulations, 2010
5. Mohite, P.M., "Composite Materials and Structures, 3D Constitutive Equations", Available from: http://nptel.ac.in/courses/101104010., 2014