UTILIZATION OF LARGE COHESIVE INTERFACE ELEMENTS FOR DELAMINATION SIMULATION

B. Bak[1*2], E. Lund[2]

[1]*Siemens Wind Power A/S, Denmark*
[2]*Department of Mechanical and Manufacturing Engineering, Aalborg University, Denmark*
**brian.bak@siemens.com*

Keywords: cohesive zone model, interface elements, numerical integration, composite structures.

Abstract
This paper describes the difficulties of utilizing large interface elements in delamination simulation. Solutions to increase the size of applicable interface elements are described and cover numerical integration of the element and modifications of the cohesive law.

1 Introduction

This work is conducted by Siemens Wind Power A/S in collaboration with Aalborg University. Siemens Wind Power A/S is one of the leading manufacturers of wind turbines. The blades of the wind turbine are made as a laminated glass-epoxy-balsa sandwich structure. The main failure mechanism of such structures is delamination damage from a single load or cyclic loading. Thus, it is important to be able to predict the onset of delamination damage and its development. The work presented here is conducted within the framework of cohesive zone modeling that was first introduced in [1,2] and is an indirect way of applying classical fracture mechanics where the critical energy release rate is represented by the work of tractions applied on the crack faces. There are several research contributions within the field of implementing the cohesive zone model into the finite element method. The element treated in this paper is the bilinear, 8-noded, zero thickness interface element for 3D models with capability of simulating mixed mode delamination using a bilinear traction-separation law [3,4].

Currently, the implementations of the cohesive zone model are not suitable for simulating delamination damage in large structures in the magnitude of meters when the cohesive zone is in the magnitude of a few millimeters, because of the high resolution of elements needed with a high computational effort as a consequence. Furthermore, the perspectives of using the cohesive elements with a stress-cycle criterion for modeling fatigue driven delamination calls for a better prediction of the in-situ stresses in the cohesive zone, which traditionally is solved using smaller elements.

So far most of the work on increasing the solution speed of delamination simulations by utilization of larger elements has focused on decreasing the onset stress of the cohesive traction-separation law [5]. This is a way to enlarge the cohesive zone and thereby having a higher resolution of elements in the zone without decreasing the element size. However, the downside of relaxing the onset stress is that it has a direct influence on the ability to predict the onset of delamination damage.

2 Reasons for Simulation Difficulties Using Large Cohesive Elements

In 2D FE models of DCB specimens an oscillating response curve is observed when using large elements [5]. This is an indication of that the critical energy release rate is varying along the interface during simulation due to the element discretization. The mechanisms governing this behavior are the discretization error in relation to the displacement field and integration of the element stiffness and stress. The discretization error of the displacement field introduces significant difference between the "true" model and the finite element model while the integration error is an error introduced in relation to the finite element model. In this paper focus is given to pure mode I only as this mode often governs the applicable element size in simulations of delamination in laminated composite structures.

2.1 Integration Error

The integration of cohesive elements is typically done using first order Newton-Cotes integration with one integration point at each pair of nodes, 2 (2D) or 4 (3D). This means that stresses and stiffness are estimated by a linear interpolation between the values at the integration points. This means that higher values between integration points are neglected. In Figure 1 an example of this is illustrated for the stresses in a 2D element where one node pair is undamaged and opened Δ_1, and the other node pair is damaged and opened Δ_2.

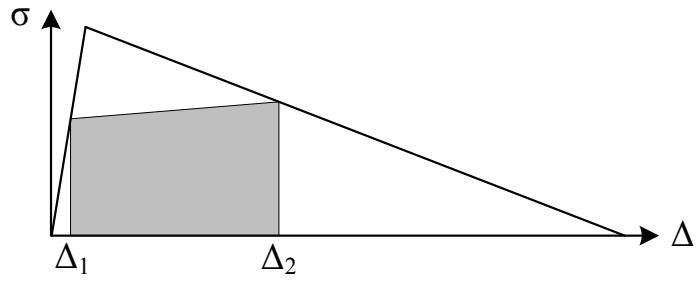


Figure 1. Illustration of integration error using first order Newton-Cotes integration with two integration points.

In order to investigate the integration error introduced in large elements the element formulation has been programmed in Maple 13 and evaluated in the entire domain for different opening configurations. Material properties used for the element are for UD glass-epoxy laminate, cf. table 2. The nodal displacement is based on the element deformation extracted from a pure mode I DCB model simulation with dimensions h×l×w = 5mm×110mm×23mm, a precrack length of 46mm and 3 mm large elements. The size of the cohesive zone is approximately 5mm. The element opening is shown in Figure 2 a) and the corresponding stiffness and stress development is seen in Figure 2 b) and c), respectively.

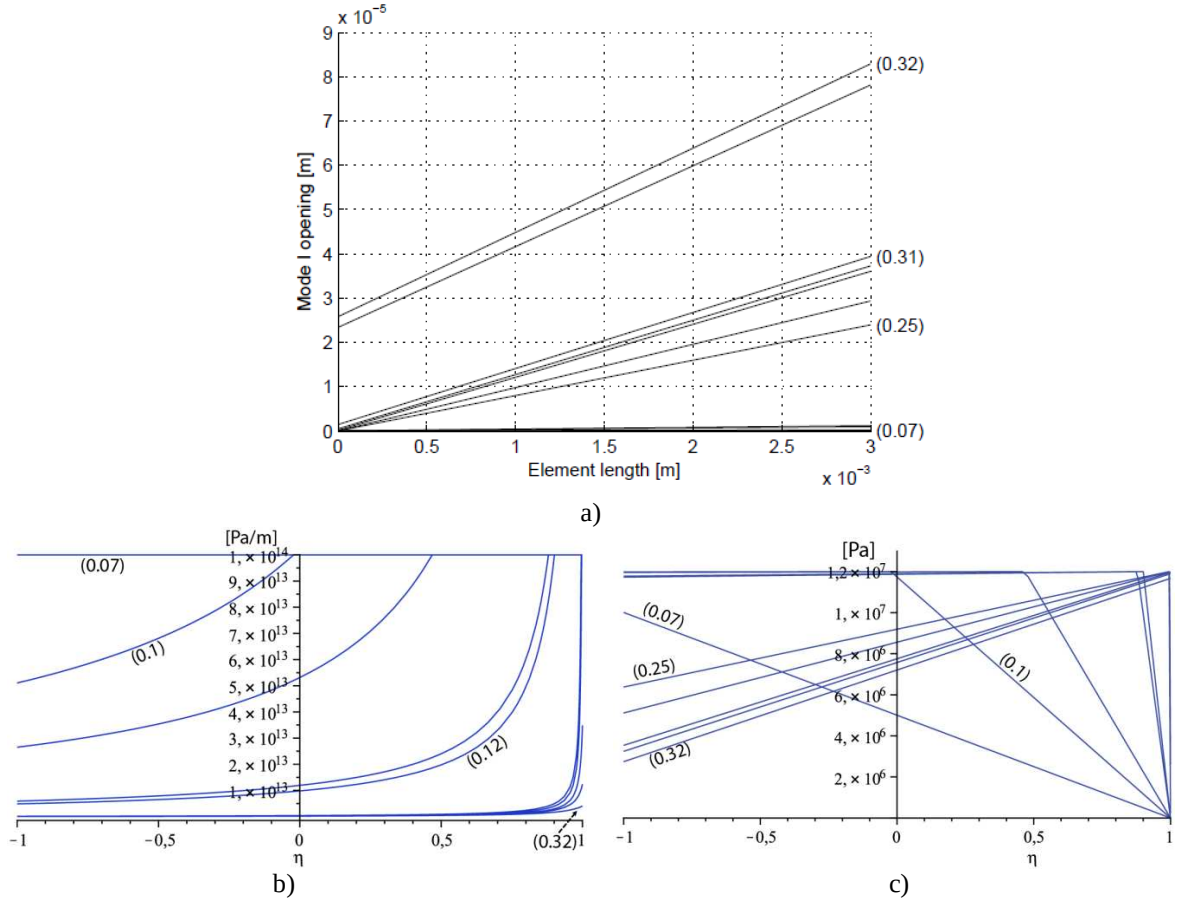


Figure 2. a) Element opening displacement and corresponding variation of b) stiffness and c) stress in the element. The numbers in the brackets are referring to the solution pseudo time.

Based on the opening displacement in Figure 2 a) the error of integrated stress and stiffness are calculated and shown in Figure 3. EDP is an abbreviation for evenly distributed integration points and the number indicates the number of integration points in each natural coordinate of the element. For more than 2 points a composite first order Newton-Cotes integration rule is used. It is seen that the error of the integrated stiffness is very high, up to 3700%, and the integrated stress error is down to -64%, which seems reasonable when considering the shape of the stiffness and stress across the element, cf. Figure 2 b) and c).

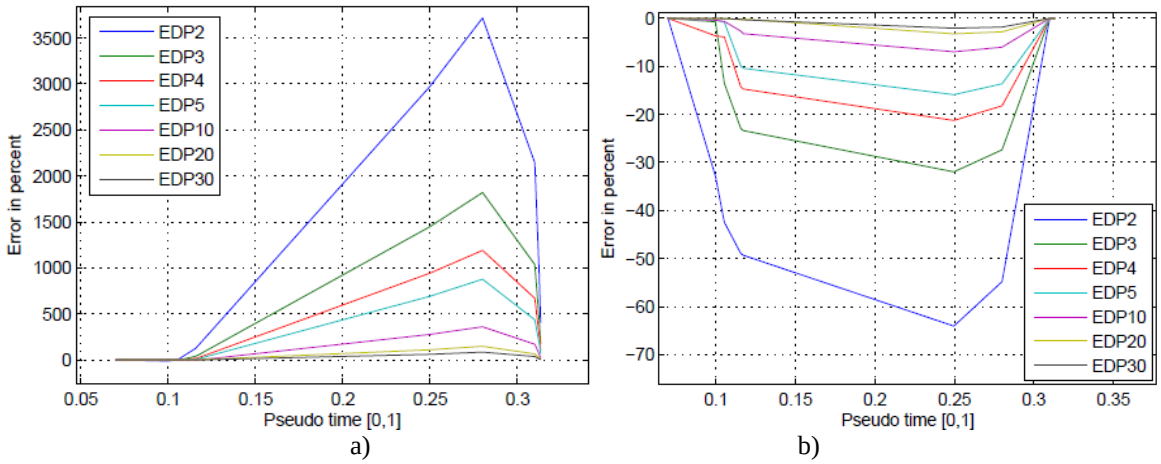


Figure 3. Error in a) integrated stiffness and b) stress over the element with the opening displacement referring to figure 2 a).

With EDP30 the integrated stiffness error is reduced from 3700% to 80% and the integrated stress is reduced from -64% to -2%.

2.2 Displacement Field Error

The curvature of the displacement field near the crack tip is large which means that it is difficult to approximate the displacement field using large linear elements. In Figure 4 a) a true displacement field, based on a simulation with a very fine mesh, is sketched and on that the element approximation going through this displacement is shown. It is seen that especially in opening 2) to 4) there is a large difference between the two displacement fields. Here the work needed to open the crack with the approximated displacement field is larger than needed by the true displacement field which in the simulation results in that the element is not opened as much as it is supposed to. In order to reach the onset opening of damage in the element there is a buildup of energy in the model because the resulting critical energy release rate of the element increases. When the opening of the element corresponds to a certain damage state, the critical energy release rate decreases which implies unstable crack growth and creates the oscillating structural response. This is illustrated in Figure 4 b) where the element to the left opens in a snap from 4 to 5.

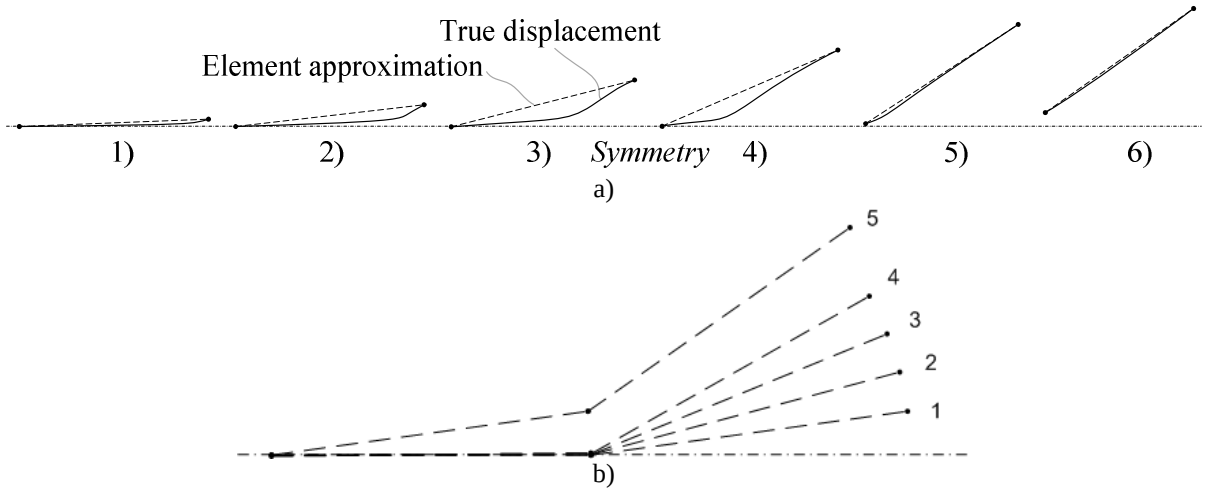


Figure 4. a) Comparison between the "true" opening displacement of the crack faces and a linear element approximation. b) Snap like behavior of large elements in simulation due to variation of the critical energy release rate as the element opens up.

The response of the before mentioned DCB specimen with 3mm large elements is shown in Figure 5. Integration with EDP30 has been used in order to remove the oscillating response related to the integration error. The model is solved using Abaqus 6.11. It is not possible to get converged solutions using the arc-length solver. Instead a line search solver with displacement control is used. Hence, the entire force displacement curve cannot be traced and therefore only the maximum critical energy release rate can be calculated and not the minimum. The relative overshoot in critical energy release rate is calculated at the peaks using an analytical beam solution and shown in the graph.

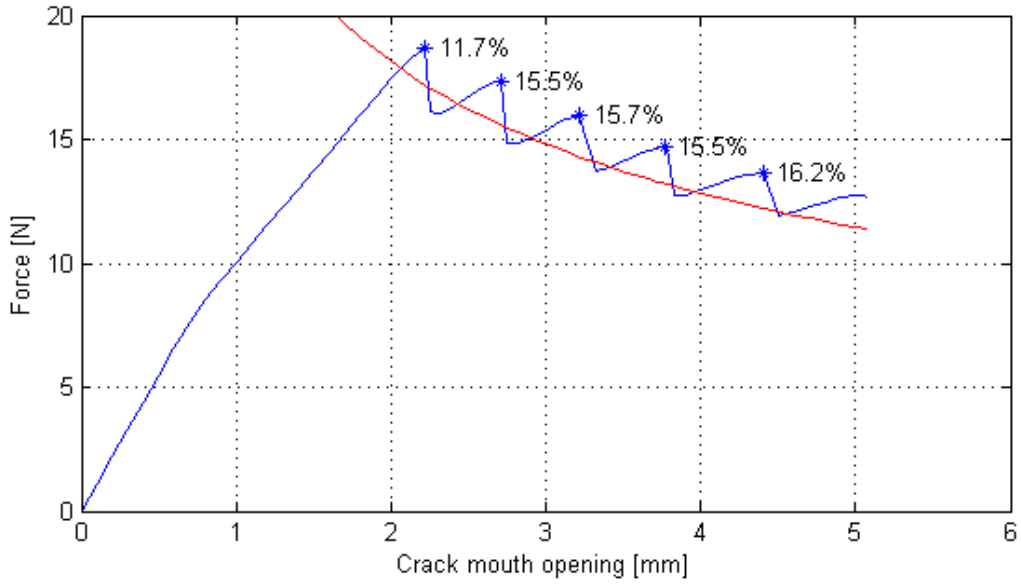


Figure 5. Response curve of DCB model using 3mm large elements (blue). LEFM Beam theory solution (red).

In order to reduce the variation of the critical energy release rate following from the error in the displacement field the most elegant solution would be to introduce a better approximation of the opening displacement than the linear used here. Different ways of doing this while still maintaining the number of nodes could be to interpolate using cubic splines between nodes with the constraint that the slope should be the same at the nodes between elements or use a function taking nodal displacements and do a curve fit to the true displacement field. Another way and the one shown here is to change the traction separation law, which does not change the solution as long as the cohesive zone is small. The change suggested to make on the traction-separation law is to change the opening stiffness according to Figure 6 while maintaining a high stiffness when the crack faces overlap. In this way the interface strength and the final opening are maintained resulting in a small change in the size in the cohesive zone (from 4.4mm to 4.7mm) compared to changing the interface strength.

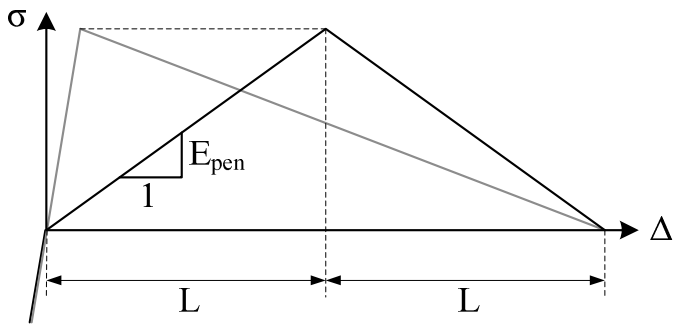


Figure 6. Proposed traction-separation law.

Based on the displacement field from a DCB model simulation with 0.01mm large elements the shape of the displacement field is shown for interface stiffness equal 10^8 MPa/m and $2.35 \cdot 10^5$ MPa/m, respectively.

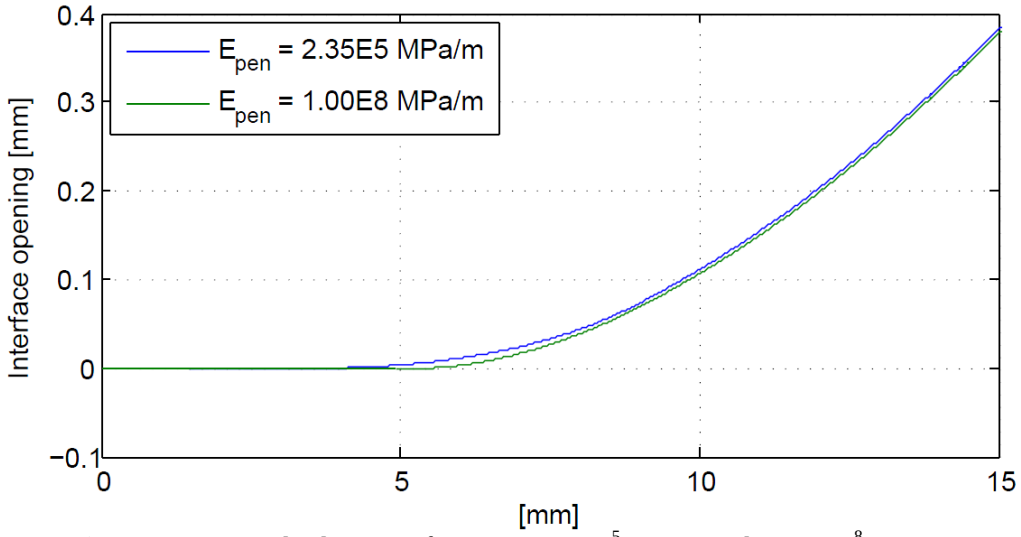


Figure 7. Opening displacement for $E_{pen} = 2.35 \cdot 10^5$ MPa/m and $E_{pen} = 10^8$ MPa/m.

3 Simulation Results

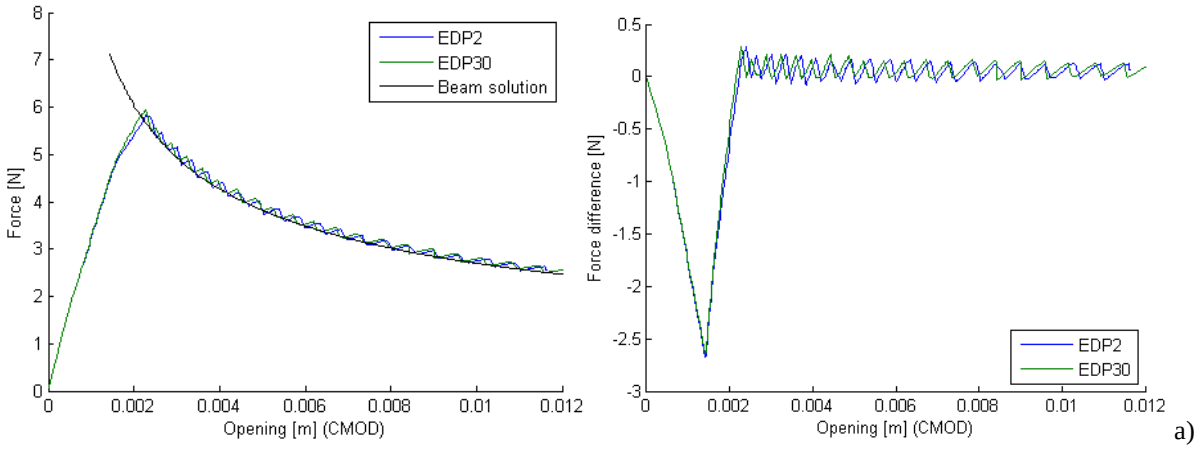
The improved integration and the proposed traction-separation law has been implemented in a user written element for Abaqus based on the formulation in [4] The computational effort for the internal force vector and stiffness matrix is approximately 200 times larger with EDP30 compared to EDP2 for an 8-noded element. Because of this an adaptive scheme is used that utilizes that undamaged and fully damaged elements are integrated correctly using EDP2. When damage is developing in an element, the integration is changed to EDP30. This reduces the computational effort for simulation with EDP30 to approximately the same as EDP2. The properties used in the following results are shown in Table 1.

E_1	40 GPa	G_I	613 J/m^2
E_2	10 GPa	G_{II}	2252 J/m^2
E_3	10 GPa	σ_1	12 Mpa
ν_{12}	0.29	σ_2	22 Mpa
ν_{13}	0.29	E_{pen}	10^8 MPa/m
ν_{23}	0.07	η	1.4
G_{12}	4 GPa		
G_{13}	4 GPa		
G_{23}	2.5 GPa		

Table 1. Material properties used in the simulations if nothing else stated.

3.1 Simulation Results for Improved Integration

Simulation results for the improved integration compared to the commonly used EDP2 for an initial stiffness of 10^8 MPa/m are shown in Figure 9 for 2mm large elements. The simulations for element size 1, 2, 3 and 4 mm using EDP2 did not complete. With EDP30 all these simulations completed indicating that simulations using EDP30 are more robust. It is seen in the figure that the oscillations are slightly smaller in the case of EDP30 but the main difference between EDP2 and EDP30 is the robustness.



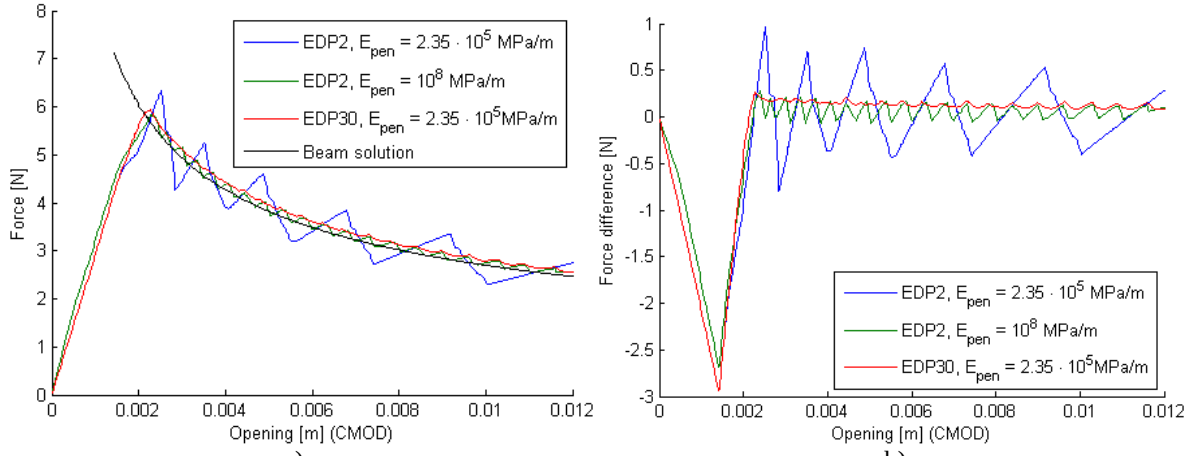
a)

b)

Figure 8. a) Response curves for simulations with 2mm large elements using EDP2 and EDP30. b) Difference in force between simulations and beam solution.

3.2 Simulation Results for Proposed Cohesive Law

Simulation results for EDP30 compared to the commonly used EDP2 using the proposed cohesive law are shown in Figure 9. Less of the simulations fail to complete for both EDP2 and EDP30. The oscillating behavior is reduced significantly for simulations with EDP30 for 1, 2 and 3 mm, but for larger elements there is no difference. For the simulations using EDP2 the oscillating response increases with the proposed cohesive law, which indicates that the integration error is the governing error and that the integration error increases when using this proposed cohesive law.



a) b)

Figure 9. a) Response curves for simulations with 2mm large elements using the proposed cohesive law. b) Difference in force between simulations and beam solution.

4 Discussion and Conclusion

The results show that the commonly used integration of cohesive elements introduces large errors on the calculation of stiffness and stress when using large elements. The integration error is believed to be one of the governing reasons for low robustness of simulations using large elements as the simulations using 1,2,3,4 and 5 mm large elements were more robust using EDP30 over EDP2. The cohesive law has been changed leading to a smooth displacement field in the cohesive zone without increasing the size of the cohesive zone significantly. This results in less oscillation of the structural response when using EDP30 while the oscillations increase when using EDP2.

5 Acknowledgements

The work presented was conducted as part of an Industrial Ph.D. project carried out in collaboration between Siemens Wind Power A/S, Denmark and the Department of Mechanical and Manufacturing Engineering, Aalborg University, Denmark. The project has received partial sponsorship from the Danish Agency for Science, Technology and Innovation. The support received is gratefully acknowledged.

References

[1] Dugdale D. S. Yielding of steel sheets containing slits. *Journal of the Mechanics and Physics of Solids* **8(2)** pp.100–104 (1960).

[2] Barenblatt G. The Mathematical Theory of Equilibrium Cracks in Brittle Fracture. *Advances in Applied Mechanics* **7**, pp. 55–129 (1962).

[3] Camanho P. P., Dávila C. G., de Moura M. F. Numerical Simulation of Mixed-Mode Progressive Delamination in Composite Materials. *Journal of Composite Materials* **37(2)** pp. 1415–1438 (2003).

[4] Turon A., Camanho P.P., Costa J., Dávila C. G. A damage model for the simulation of delamination in advanced composites under variable-mode loading. *Mechanics of Materials* **38(11)** pp.1072–1089 (2006).

[5] Turon A., Dávila C. G., Camanho P.P., Costa J. An Engineering Solution for using Coarse Meshes in the Simulation of Delamination with Cohesive Zone Models. *NASA Tech. rep.* (2005).