State of the art of rotor blade buckling tools

Inventory of the Dutch BLADKNIK project

C. Lindenburg (ECN)

G.D. de Winkel (WMC)

ECN-C--05-054

PREFACE

Within the Dutch research project BLADKNIK buckling load prediction tools were developed for application to wind turbine rotor blade design. This project can be considered as follow-up development of the European BUCKBLADE project, and the Dutch STARION project. Prior to the development of the buckling load prediction tools, an investigation is performed into the accuracy and the sources of uncertainties of the available tools for application to real rotor blade structures. Further development of the prediction methods will be done on basis of the results in this report.

The BLADKNIK project was supported by Senter*Novem*, project No: 2020-02-12-10-007.

ABSTRACT

Within the Dutch research project BLADKNIK buckling load prediction tools were developed for application to wind turbine rotor blade design. As basis for further investigations and validations within this project, Buckling load analyses were performed on existing rotor blade models. For many of these models experimental buckling loads are available. The goal for the BLADKNIK project is to implement the available tools in the wind turbine design package FOCUS. The buckling analyses reported here were performed as much as possible with FOCUS models of the rotor blades. For some of the blades that were analysed, special attention was paid to the modelling which is reported in detail, such that it reveals the (possible) weak points of building a blade model.

The buckling tools that were used are those that remain after the former BUCKBLADE and STARION research projects, on which some modifications were performed in order to eliminate some well-known discrepancies or bugs. The general conclusion is that if serious attention is paid to building a blade model, that the predicted buckling loads match to some extent to the experimental buckling loads. For some rotor blades however, large discrepancies appear that are mainly due to the fact that non-uniform material distributions are used (panel methods) or that the pre-buckling deformation is not included (FINSTRIP).

The result of these investigations are partly the basis for the detailed outset of further work within BLADKNIK.

Keywords

Buckling load, Composite material, Design tools, Rotor blade, Sandwich structure, Wind turbine.

CONTENTS

LIST OF SYMBOLS

A	[N/m]	In-plane or "membrane" stiffness matrix.
$A_{44},\ A_{55}$	[N/m]	Out-of-plane shear stiffnesses of a (sandwich-) panel: $A_{44} = G_{23}\, h^2/t_{\rm c}$; $A_{55} = G_{13}\, h^2/t_{\rm c}$.
B	[N]	Coupling matrix between in-plane and bending stiffnesses.
b	[m]	Width of the panel c.q. the half-waves or bulges.
C	[m/N]	Inverse of the membrane stiffness matrix $C = A^{-1}$.
c	[m]	Blade chord.
D	[Nm]	Out-of-plane bending stiffness matrix.
$\tilde{D}$	[Nm]	Reduced bending stiffness matrix $\tilde{D} = D - B \cdot A^{-1} \cdot B$
$D_{\rm s}$	[Nm]	Matrix with bending stiffnesses from compressive stiffness of **s**andwich-facings only.
$E_1,\ E_2$	[N/m^2]	Young's moduli in the direction of the largest- and smallest principal stiffness respectively.
G_{12}	[N/m^2]	In-plane shear modulus.
$G_{13},\ G_{23}$	[N/m^2]	Out-of-plane shear moduli in longitudinal- and transverse direction.
K	[m]	The matrix product $K = A^{-1} \cdot B$
$k_{\rm c},\ k_{\rm t},\ k_{\rm b},\ k_{\rm s}$		Dimensionless buckling factors $k_{\rm c} = -N_{\rm x}/[\,(\pi/b)^2 \sqrt{\tilde{D}_{11}\, \tilde{D}_{22}}\,]$, $k_{\rm t} = -N_{\rm y}/[\,(\pi/b)^2\, \tilde{D}_{22}\,]$, $k_{\rm b} = N_{\rm x\,bend}/[\,(\pi/b)^2 \sqrt{\tilde{D}_{11}\, \tilde{D}_{22}}\,]$ and $k_{\rm s} = N_{\rm xy}/[\,(\pi/b)^2 \sqrt{\tilde{D}_{22}\sqrt{\tilde{D}_{11}\, \tilde{D}_{22}}}\,]$.
L	[m]	Length of the panel c.q. of the half-waves or bulges. e.g. defined as $R_{\rm c} = N_{\rm x}/N_{\rm x\,cr}$.
$R_{\rm x},\ R_{\rm y}$	[m]	Radii of curvature in longitudinal- and transverse direction.
r	[m]	Radial co-ordinate in the rotor system, measured from rotor centre.
$r_1\ ,\ r_2$		Stiffness ratios for sandwich panels: $r_1 = (\pi/b)^2 \sqrt{\tilde{D}_{\rm s\,11}\, \tilde{D}_{\rm s\,22}}\,/A_{55}$; $r_2 = (\pi/b)^2\, \tilde{D}_{\rm s\,22}/A_{44}$.
$x,\ y,\ z$	[m]	Flapwise (downwind), lagwise, and spanwise co-ordinates which corresponds to the Germanischer Lloyd conventions.
Z		Curvature parameter $Z = b^2/[\,R_{\rm y} \sqrt{\tilde{D}_{22}\, C_{22}}\,]$; for isotropic panels $Z = b^2 \cdot \sqrt{12(1-\nu^2)}/(R_{\rm y}\, t)$; and for isotropic sandwich panels $Z = b^2 \cdot \sqrt{1-\nu^2}/(R_{\rm y}\, h/2)$.
β		Orthotropy parameter $\beta = (\tilde{D}_{12} + 2\, \tilde{D}_{66})/\sqrt{\tilde{D}_{11}\, \tilde{D}_{22}}$.
ν_{12}		Poisson's ratio; contraction in the "*2*" direction due to an elongation in the "*1*" direction.
ν_{21}		Contraction in the "*1*" direction due to an elongation in the "*2*" direction. These properties apply to $\nu_{12}\, E_2 = \nu_{21}\, E_1$.
π		Trigonometric constant: $\pi = 3.1415926536$.

TERMINOLOGY

In this document the following terms are used for programs, directions of loads and deformations, file formats, etcetera are described with the following terms:

Expression	Description
CROSTAB	Program for cross-sectional analysis, including panel-buckling.
FAROB	Program for structural analysis of a rotor blade, within FOCUS.
FINSTRIP	Program for linear buckling analysis of a blade cross section.
FOCUS	Design package for a wind turbines, including a detailed blade model.
MSC.MARC	Finite-element package with non-linear options for buckling.
`*.buc` file	File format that contains the geometry of a cross section, material definition, and the layup. This was introduced for **buc**kling analyses.
UD laminate	Laminate with all fibres in the same direction, '**U**ni **D**irectional'.
Bending-Torsion coupling	Modification of the blade structure such that a bending moment gives a torsional deformation. This can be realised e.g. with off-axis UD laminates (symmetric w.r.t. the blade axis) or with an aft-swept blade tip.
Structural Pitch	Modification of the blade structure that changes ('pitches') the principal stiffness direction and thus also the directions of blade vibration.
edge-wise	Along the local chord.
flap-wise	Perpendicular to the plane of rotation, positive downwind.
flat-wise	Perpendicular to the local chord, positive downwind.
lead-wise	In the plane of rotation, positive in rotational direction (opp. to 'lag-wise').

1 INTRODUCTION

The phenomena of buckling is failure of a load-carrying structure by strong deformation that distorts the geometrical topology and the associated load-carrying capabilities. Characteristic for buckling with regard to the analytical descriptions is that the energy used for deformation of the geometry (elastic or plastic) is equal or less than the release of strain energy (usually compression) in the loaded structure.

The phenomena of buckling has been investigated for decades in aerospace industry (for the design of e.g. launch rockets and the upper skin of aircraft wings). These investigations were mainly addressed to aluminium (isotropic material) structures, eventually stiffened with stringers. For buckling of fibre-reinforced composite structures a fundamental investigation has been performed within the European AGARD group, which has been reported by B. Geier (Stuttgart) [7]. The investigations reported by Geier dealt with curved orthotropic panels loaded by compression, which comes close to the problem of buckling of rotor blades loaded by bending. Partly based on the fundamentals of the work reported by Geier, the European research project BUCKBLADE was carried out, addressed to development and evaluation of different tools for the prediction of buckling of rotor blades. This project was followed by the Dutch STARION project which dealt with the application of the buckling load prediction tools to real wind turbine structures.

1.1 Previous work on Rotor Blade Buckling

BUCKBLADE

A European research project in which several buckling load analyses methods were investigated and/or developed. These methods included design rules based on graphs and formulae from handbooks, tools for buckling of panels, tools for buckling of cross-sections of prismatic structures (such as rotor blades), and finite element packages. Part of this research project was a set of tests on some real rotor blade structures at the University of Stuttgart. These rotor blade structures were designed by LM Glasfiber and built by LM AeroConstruct in the moulds of the former DEBRA blade.

STARION

Partly as follow-up of the European BUCKBLADE project the Dutch STARION project was addressed to the application of the tools from the BUCKBLADE project for buckling of real rotor blade structures. For this purpose 3 tests were carried out on the outer part of rotor blades. Also the panel-based method StaBlad by ECN, and the tool for cross-sections FINSTRIP by SPE were implemented in the design package FOCUS.

DOWEC

Within the Dutch offshore research and implementation project an investigation has been carried out into the state-of-the-art for buckling of rotor blades. This investigation summarised the results of the STARION project and of the BUCKBLADE project. In addition the buckling load prediction tools were also compared on basis of the RL48 blade and two large blades of LM Glasfiber.

1.2 Scope of the BLADKNIK Investigations

The previous investigations involved buckling load prediction tools for complete blade structures (Finite Element packages), for sections of long prismatic structures (FINSTRIP, CROSTAB), and for panels (different methods implemented in StaBlad and the so-called 'Design Rules').

The increasing dimensions of rotor blades introduces the need for higher material efficiency and more reliable structural design so that more effort may be spent on obtaining realistic and reliable buckling load predictions. This means that predictions for the complete blade structure have a preference over predictions for cross-sections, while predictions on basis of panels are of minor value. For this reason the BLADKNIK project focuses on buckling load analyses of complete blades that are modelled in FINSTRIP, using either finite element packages for the complete structure or the buckling tools FINSTRIP (WMC) and CROSTAB (ECN). The panel based methods that were developed in the BUCKBLADE project are not involved in much detail.

Prior to the investigations in the Dutch BLADKNIK project an inventory is made of the experimental data that are available for the evaluation of the buckling tools. The buckling tools are described briefly in chapter 2, while chapter 3 contains a short description of the reference problems. These problems are either defined by a single cross-section for which buckling was traced, or by a complete blade for which the buckling predictions have to be performed for the entire span. In the latter case the 'buckling load' is not in terms of a sectional moment but in terms of the applied force during the test.

For some of the test specimen, special effort was paid in building a realistic model in FAROB (structural blade model within FOCUS). This process is reported in Appendix A for the Double-Box beam and in Appendix B for the BUCKBLADE specimen.

The modifications that were issued on release "SEP-2002" of the program CROSTAB are listed in Appendix C.

2 BUCKLING TOOLS INVOLVED

2.1 Finite Element codes

For thorough buckling load analyses of rotor blades, the finite element codes are assumed to give the most detailed prediction. WMC has experience with the Finite Element code MSC.MARC, which will be used.

2.2 FINSTRIP

The program FINSTRIP was originally developed at Stork Product Engineering and was developed further by WMC. The program FINSTRIP can be used to calculate the buckling load of multi-cell thin-walled prismatic beams. The geometric non-linear effects such as the longitudinal curvature under loading, and the deformation from 'crushing loads' are not taken into account. After the former projects on buckling load prediction methods 'BUCKBLADE' and 'STARION' some details of FINSTRIP have been changed. The calculations for the State of the Art reported here were performed with version 1.5 of FINSTRIP.

2.3 CROSTAB

The program CROSTAB was developed at ECN Wind Energy by C. Lindenburg. Similar as FINSTRIP, CROSTAB can be used for buckling load calculations of thin-walled prismatic beams, including the geometric longitudinal curvature from bending and the effects of crushing loads. CROSTAB solves the buckling of a cross section by stability analyses of each of the individual panels, where FINSTRIP models the complete structural integrity within a cross-section. As for FINSTRIP, small modifications have been applied to the program since the STARION project. The calculations for the State of the Art reported here include CROSTAB release "SEP-2002" of which the definition of materials was made more robust in the winter of 2005.

2.4 Design Rules

The investigations within BLADKNIK were addressed to evaluation and further development of tools for the buckling load prediction of complete rotor blades. This implies that only finite-element codes and codes for buckling load prediction of complete cross-sections are investigated. In practise many engineers use formulae from some design handbooks for buckling load predictions of flat panels and strips, or for a fast prediction of the buckling strength in the pre-design. For this reason and also because a lot of knowledge on design rules has already been gained during the previous BUCKBLADE project, these 'Design Rules' are still involved in the BLADKNIK project, although they are only developed further for application to sandwich panels.

3 REFERENCE PROBLEMS AVAILABLE

The reference problems are descriptions of real structures of which experimental buckling loads are available. Following the scope of the BLADKNIK project only long thin-walled ('slender') structures are selected, preferably real rotor blades.

For all reference problems, a FAROB model is defined/composed from which so-called `.buc` files are generated as basis for the tools FINSTRIP and CROSTAB. For application of the 'Design Rules' the material properties, the overall bending stiffness of the cross-section, and the stiffness matrices of the laminate are given.

3.1 Double-Box Beam

Within the research into bending-torsion coupling to enhance the aeroelastic properties of rotor blades W.C. de Goeij tested two double-box carbon-fibre beams with asymmetric laminates. The tests showed an unforeseen buckling [9]. Although this box differs from a conventional rotor blade by its asymmetric laminate, the description of the geometry and material properties is very accurate and therefore useful as reference problem.

The test results of the double-box beams are reported in terms of displacement and torsional deformation of the tip as function of the applied force. The relation between the tip-force and tip displacement was linear up to a force of 300N and 350Nm for beam 1 and beam 2 respectively. The length of these beams from the root attachment to the load introduction point was 1.0m. This means that the experimental buckling load was roughly 300Nm to 350Nm. A more accurate buckling load would have been obtained with strain gauges on the panel where buckling occurs.

Comparison of experimental and theoretical buckling loads:

Measured	300Nm & 350Nm	
Design Rules (orthotropic)	421.6Nm	
CROSTAB (anisotropic)	311.8Nm	half-wave length 158mm
FINSTRIP (orthotropic)	551.4Nm	half-wave length 138mm

The finite element model was used for a geometric linear analysis so this did not show buckling. Except for some deviation of the FINSTRIP results the difference between the buckling load prediction tools shows to be relatively small.

Because of the off-axis carbon fibre UD laminates, the cross sectional properties are anisotropic for which reason the predictions with FINSTRIP depend heavily on the sectional bending stiffnesses. For buckling analyses with FINSTRIP these bending stiffnesses are used internally for the relation between sectional moments and local strains. Following is a comparison of the FINSTRIP buckling predictions with different bending stiffnesses:

Sectional stiffness	FINSTRIP	FAROB	CROSTAB
Tensile stiffness EA	24.398E+6	63.230E+6	40.151E+6
Edge-stiffness	47.058E+9	118.81E+9	44.976E+9
Flat-stiffness	20.867E+9	51.749E+9	19.991E+9
FINSTRIP crit. load	575Nm	1447Nm	551.4Nm

From this table it follows that the cross-sectional stiffnesses calculated for anisotropic beams depend strongly on the tool with which they are calculated. The critical load calculated with FINSTRIP appears to be proportional with the flatwise bending stiffnesses that are used as input.

The large differences between the stiffnesses from the different tools are caused by the anisotropic laminates with off-axis layers. In some of the tools these anisotropy terms in the stiffness matrices are excluded, while in other tools the panel shear deformation associated with axial loading is assumed zero. In the first case the beam-stiffnesses are under-estimated while in the latter case the beam-stiffnesses are over-estimated.

3.2 BUCKBLADE test specimen 1

Within the European BUCKBLADE project two buckling specimen were built by LM AeroConstruct in the moulds of the former DEBRA blade. The first of these specimen had 2 webs and was tested with a flapwise force at 6.0m from the root, in both 'upwind' and 'downwind' direction. The difference between those tests was that buckling occurs in the strong curved panel at the aerodynamic suction side, or in the weak curved panel at the aerodynamic pressure side. Both these tests showed buckling. By repeating one of these tests it appeared that the buckling load did not decrease much so that the buckling was nearly elastic. After the first comparison of the test results with the predictions it was found necessary to measure the contour of the specimen at some sections while a number of batches of material were cut-out for analysis of the basic laminate material properties. Predictions on basis of this 'as measured' description showed a better agreement with the measured buckling load than the predictions on basis of the 'as designed' description. For the purpose of evaluation of the buckling load prediction tools itself the 'as measured' description was chosen. The difference between the 'as designed' and 'as measured' properties is a problem of the design and manufacturing process. With the automised and vacuum production techniques used nowadays, it is expected that the difference between the 'as built' and the 'as designed' properties will reduce.

3.2.1 Results for lower central panel buckling

The test for buckling of the lower central panel was performed with a force applied at 6.0m from the blade root (r = 6.9m) that was perpendicular to the chord direction of the section at 2.1m from the root, which has a twist of 7.2°. During the test bulges appeared at 2.75m and at 3.7m from the root at an applied load of 11.3kN. Non-linear finite element calculations were performed by LASSO Ingenieursgesellschaft with the finite program LARSTRAN (**LAR**ge **STR**ain **AN**alysis) and with the linear finite element program NASTRAN.

For the buckling analyses with CROSTAB and with FINSTRIP, using the `.buc` files for 16 locations. The sectional loads used for these calculations were those for an applied force at 6.0m from the blade root as in the test. In addition to the dominating sectional bending moment, the sectional loading also includes the shear force in the cross section. This shear force was applied assuming that this will be included in later versions of CROSTAB and/or FINSTRIP.

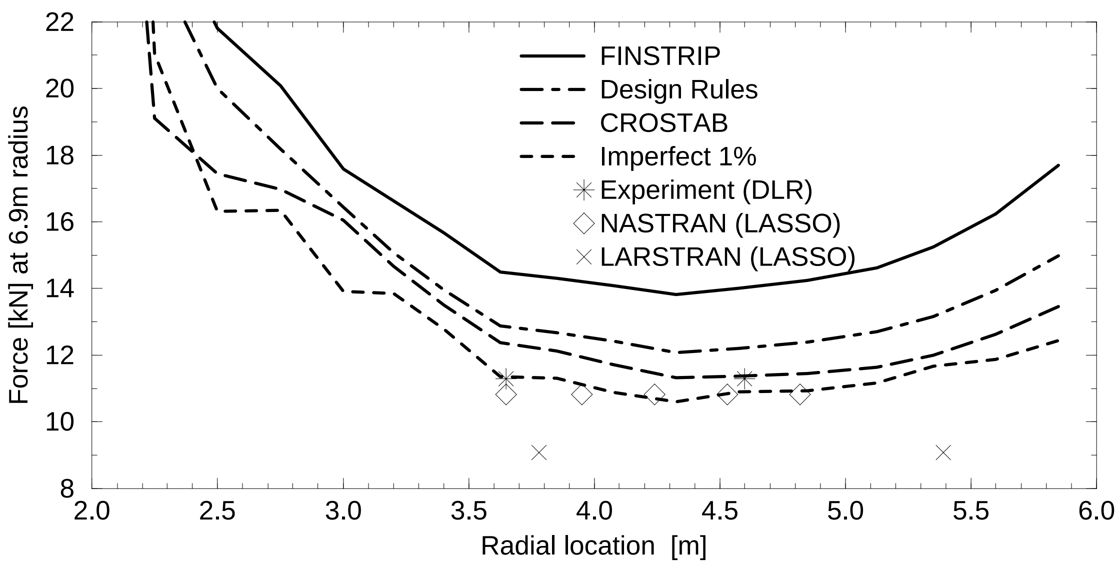

Figure 1: Experimental and predicted loads for lower central panel buckling

Figure 1 shows that the BUCKBLADE specimen is critical for buckling between r = 3.5m and r = 5.5m, from which it follows that the load introduction point at 6m from the root (r = 6.9m) was well chosen. The FAROB interpolation of the geometry in spanwise direction results in 'kinks' for the predicted buckling loads, at radii of 3.0m, 3.625m, and 4.325m.

The FINSTRIP predictions for the lower central panel appear to be the most unconservative.

Experimental and theoretical buckling loads including data from the BUCKBLADE project are shown in the table below:

	Force at 6.9m	Bulge location	Bulge size
FINSTRIP ('97) (BUCKBLADE)	9.805kN	r = 5.0m	
FINSTRIP version 1.5	13.82kN	r = 4.325m	160mm
StaBlad (BUCKBLADE)	6.654kN	r = 3.65m	1061mm
CROSTAB "SEP-2002"	11.291kN	r = 4.325m	745mm
CROSIMP 1% imperfection	10.574kN	r = 4.325m	216mm
Design Rules	11.861kN		
Experiment (DLR)	11.3kN	r = 3.65m & 4.60m	
NASTRAN (LASSO)	10.83kN	r = 3.65m & 4.82m	
LARSTRAN (LASSO)	9.08kN	r = 3.78m & 5.39m	

It is surprising to see that the critical loads calculated with FINSTRIP ('97) and with CROSTAB for the BUCKBLADE project are much smaller than the present predictions.

3.2.2 Results for upper central panel buckling

The test for buckling of the lower central panel was performed with a force applied at 6.0m from the blade root (r = 6.9m) that was perpendicular to the chord line of the section at 2.1m from the root, but in opposite direction as for buckling of the lower central panel. During the test bulges appeared at 1.15m and at 3.8m from the root for an applied load of 22.2kN. Non-linear finite element calculations were performed by LASSO Ingenieursgesellschaft (Leinfelden Echterdingen, DE) with the finite element program LARSTRAN (**LAR**ge **STR**ain **AN**alysis) which predicted buckling at 1.64m from the root for an applied load of 20.24kN.

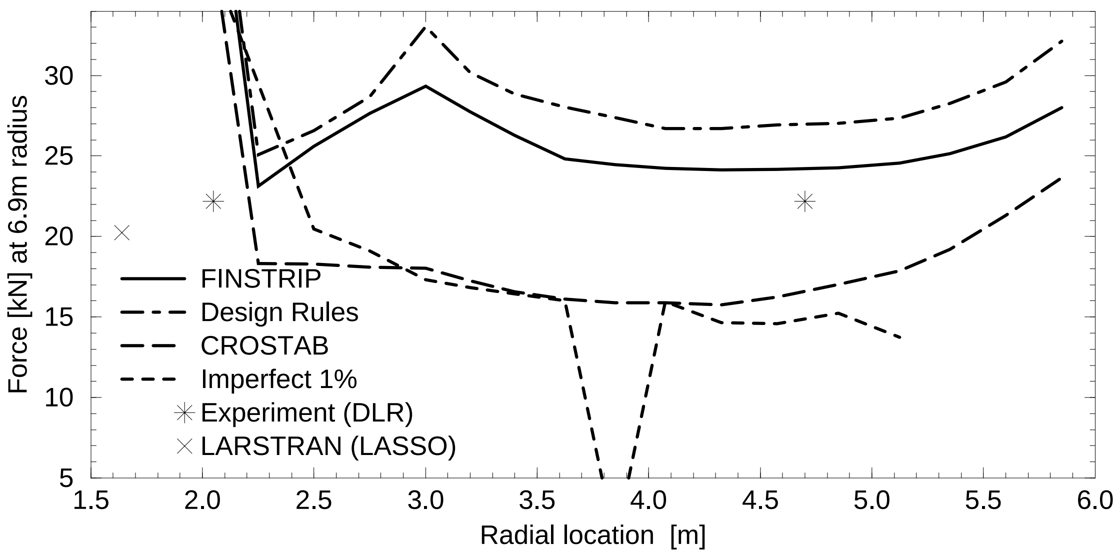

Figure 2: Experimental and predicted loads for upper central panel buckling

Experimental and theoretical buckling loads including data from the BUCKBLADE project:

	Force at 6.9m	Bulge location	Bulge size
FINSTRIP ('97) (BUCKBLADE)	18.806kN	r = 5.125m	
FINSTRIP version 1.5	26.69kN	r = 4.075m	112mm
StaBlad (BUCKBLADE)	11.28kN	r = 4.325m	
CROSTAB "SEP-2002"	15.762kN	r = 4.325m	193mm
CROSIMP 1% imperfection	13.747kN	r = 5.125m	long
Design Rules	24.138kN		
Experiment (DLR)	22.2kN	r = 2.05m & 4.70m & 4.60m	
NASTRAN (LASSO)	23.37kN	r = 3.45m ... 4.13m	
LARSTRAN (LASSO)	20.24kN	r = 1.64m	

As for the buckling load predictions of the lower central panel, we see also much lower buckling loads with the former FINSTRIP ('97) and the BUCKBLADE-Stablad versions.

3.3 BUCKBLADE test specimen 2

The second specimen built for the BUCKBLADE project had only one shear web. This specimen was to be tested with loading such that either the trailing edge panel (suction side) or the leading edge is critical for buckling. Due to a mis-communication the test on leading-edge buckling was performed with loading in a slightly different direction, for which the buckling strength of the blade was larger than the material strength in the leading-edge. As a result predicted buckling loads are investigated here for leading-edge buckling with loading in two load directions.

The test on trailing edge buckling showed local failure of the outer facing ('dimpling') of the sandwich tail.

3.3.1 Results for leading-edge buckling as analysed

In order to avoid damage in the trailing edge of specimen 2 during the test on leading edge buckling, a load-direction with an angle of 65^o with respect to the mould reference plane was applied. This 'mould reference plane' (y dir) is also used for the specification of the blade twist in the FAROB model. For this load direction FEM analyses were performed with NASTRAN, with MARC, and with LARSTRAN. The calculations with MARC showed buckling at r = 3.15m, at r = 3.65m, and at r = 2.57m for an applied load of 21.0kN, 29.2kN, and 30.8kN respectively. The linear calculations with the FEM package NASTRAN showed buckling at r = 3.57m and r = 4.93m for an applied load of 32.7kN. The non-linear calculations with the FEM package LARSTRAN showed buckling at r = 3.69m for an applied load of 24.3kN.

The results of these analyses are compared in Figure 3 together with the critical loads calculated with CROSTAB, with FINSTRIP, and with the Design Rules.

3.3.2 Results for leading-edge buckling as tested

The test on buckling of the leading edge was performed with a force applied under an angle of 25^o with respect to the mould reference plane, directed towards the pressure-side of the leading-edge. Figure 4 shows that for this load direction the experimental failure load is far smaller than the predicted buckling loads, which reflects the fact that the buckling load analysis tools applied here do not consider material failure. The linear FEM analysis with NASTRAN for this load direction shows buckling at r = 2.5m for an applied force of 41.2kN. The experimental failure load of 19.5kN in Figure 4 is pretty close to the buckling loads calculated with MARC, with FINSTRIP, and with LARSTRAN shown in Figure 3 which indicates that a load-direction of 65^o w.r.t. the mould reference plane would be more suitable to show leading-edge buckling.

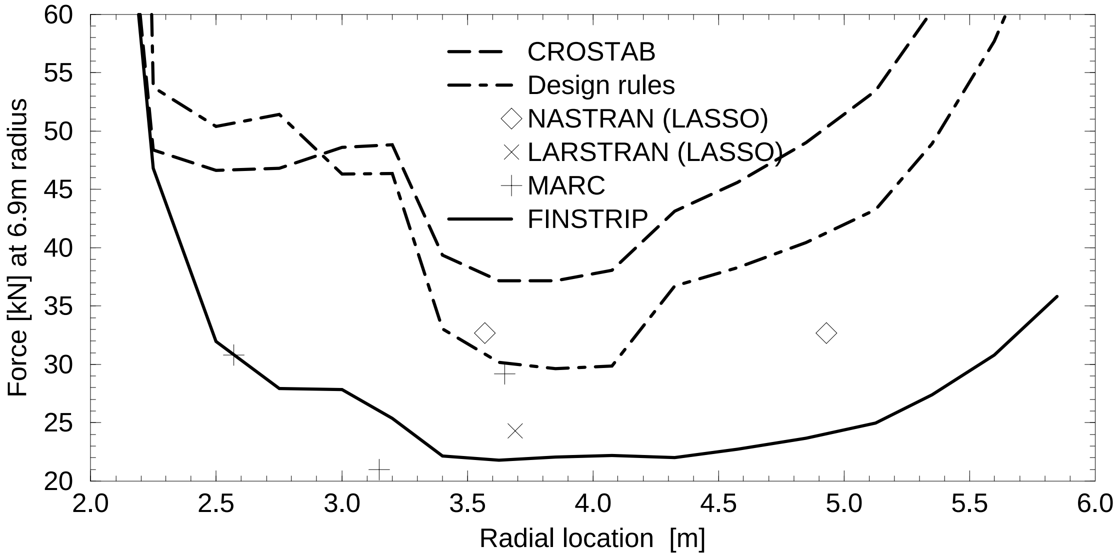

Figure 3: Predicted loads for leading edge buckling as analysed

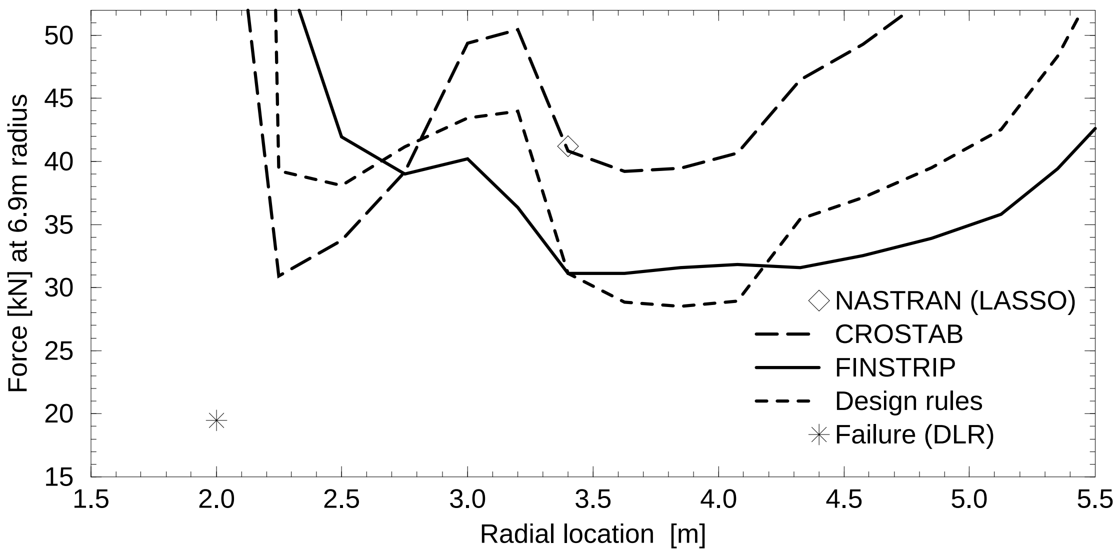

Figure 4: Experimental and predicted loads for leading edge buckling

3.3.3 Results for sandwich tail panel buckling

The test for buckling of the sandwich tail panel was performed with a force applied at an angle of 70^{o} with respect to the blade reference plane, directed towards the trailing-edge suction-side.

Experimental and theoretical buckling loads including those from the BUCKBLADE project:

	Force at 6.9m	Bulge location	Bulge size
FINSTRIP ('97) (BUCKBLADE)	28.31kN	r = 3.0m	
FINSTRIP version 1.5	33.01kN	r = 3.4m	440mm
StaBlad (BUCKBLADE)	32.06kN	r = 3.0m	434mm
CROSTAB "SEP-2002"	35.25kN	r = 3.0m	193mm
Design Rules	34.84kN	r = 3.0m	
Experiment (DLR)	26.2kN	r = 3.20m	
NASTRAN (LASSO)	40.73kN	r = 3.20m & 3.57m	
LARSTRAN (LASSO)	32.0kN	r = 2.69m & 3.11m	
MARC (ECN)	24.0kN	r = 3.37m	

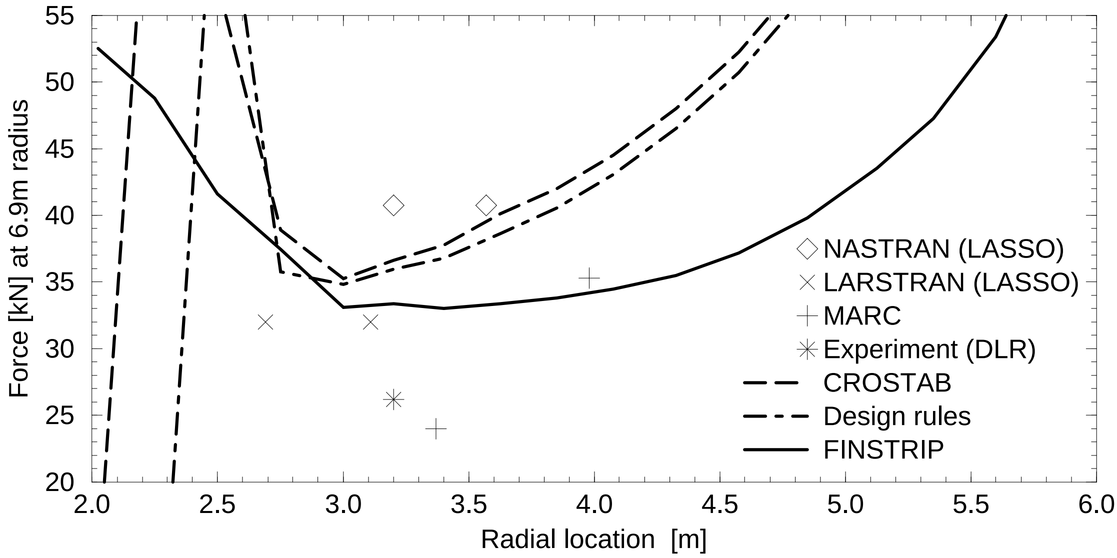

Figure 5: Experimental and predicted loads for sandwich tail panel buckling

Here the experimental failure load was small because of a surface imperfection. Taking this imperfection into account the buckling load predictions are quite reasonable.

3.4 STARION test specimen 1

In the framework of the STARION project, two full scale rotor blades were tested, both with a rotor radius of approximately 15 metres. Both blades were standard commercial blades, therefore no detailed information about geometry and layup were reported. The loading on the two blades was applied with a single hydraulic actuator at about one metre from the tip. Finite element analyses were carried out to determine the expected critical zone for possible failure due to buckling. The expected critical zone is between rotor radius r = 10 000mm and r = 13 000mm for both rotor blades. Based on the finite element analyses it was expected that the maximum displacement at failure for the STARION specimen 1 was larger than the maximum displacement of the hydraulic actuator. Therefore it was decided to add an additional horizontal support for the test of STARION specimen 1 at rotor radius r = 9000mm to limit the displacement at the loading point.

For a good comparison between the experimental results and the numerical results, it is important to know whether there are significant differences between the geometry of the numerical models and the actual geometry of the blade cross sections. On the rotor blades, seven cross sections between r = 10 000mm and r = 13 000mm with an increment of 500mm were marked. The cross sectional shapes at the marked locations were measured, digitised, and recorded. At fourteen locations at the compressive side of the rotor blades and two locations at the tension side, strain gauges were mounted. The loading rate during the test was 1000mm per 60 seconds.

Experimental results for specimen 1

The STARION specimen 1 failed by buckling of a skin section, within the marked region of the rotor blades. First the nose panel failed at r = 11750mm. After this local failure, total failure occurred at r = 12 000mm at a load of 23kN. At a load of approximately 13kN, the strain gauge signals start to deviate from the tangent lines, thus showing the nonlinear behaviour that can be expected in case of buckling. At a load of approximately 11.5kN, buckling waves were visually observed.

3.4.1 Results of panel based buckling analyses with Focus4

The critical load factors calculated with FAROB for the nose and tail panels in the region of the rotor blade between r = 9000mm and r = 13 000mm are shown in Figure 6. The critical load factor as shown here is the critical load calculated with FAROB divided by the maximum load of the test. The sawtooth shape of the curve for the tail panel is caused by discrete steps in the thickness in spanwise direction. The blade radius where failure occurred at the test (r = 12 000mm) is also marked in Figure 6. As can be seen in Figure 6 the blade failed at a lower load than the calculated load factors. However, the failure occurred at a load close to the full capacity of the cross section (the maximum compression stress was close to UCS). This means that this blade is not very sensitive for buckling. The critical load factor lines predict blade failure at r = 12 000mm and/or at r = 12 500mm. This agrees with the test.

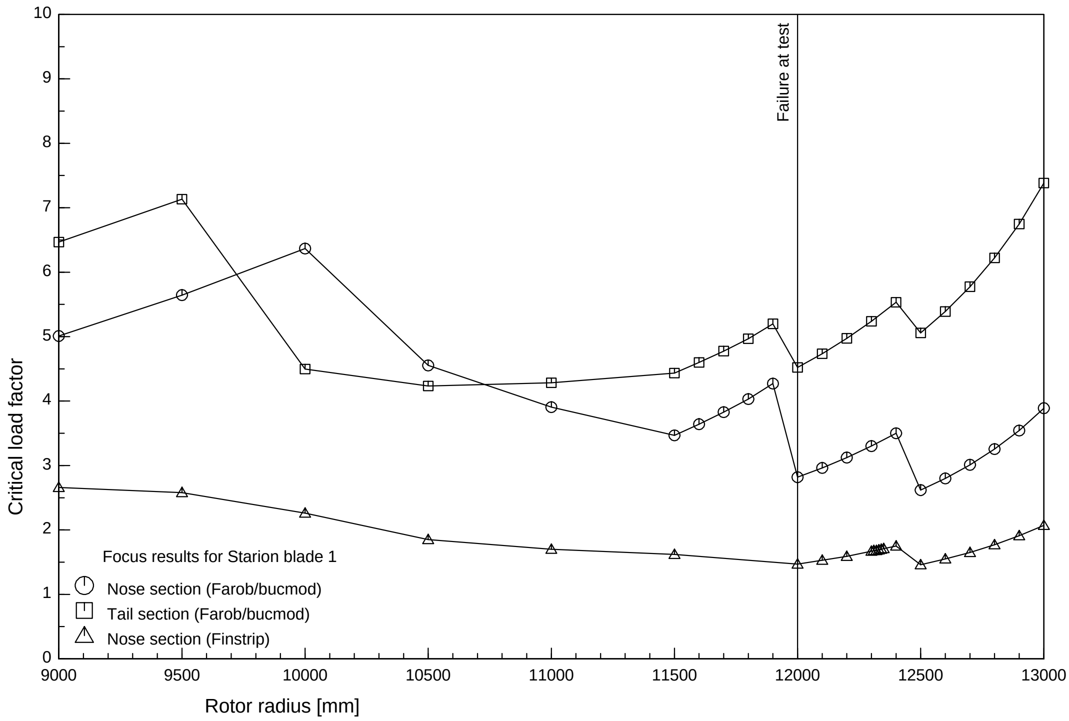

Figure 6: Critical load factors for STARION test specimen 1 calculated with Focus4

3.4.2 Results of cross section based buckling analyses with Finstrip

STARION test specimen 1 has been reanalysed with the FINSTRIP program (version 1.4). In the framework of the STARION project only the cross section that failed during the test was analysed with FINSTRIP. At the same locations where the panel based buckling analyses were performed 22 `.buc` files have been created using Focus4 between r = 9000mm and r = 13 000mm. The analyses were performed with default settings from Focus4 (maximum element size 50mm, minimum half-wave length 10mm, maximum half-wave length 1200mm, and 20 wavelength steps). In the critical area, these default settings resulted in a calculation accuracy of approx. 5%, which is acceptable. At all sections, the cross section failed by nose panel buckling under compression. This agrees with the test result. The analysis results are shown in Figure 6. The collapse mode at r = 12 000mm is shown in Figure 7. The calculated reserve factors are smaller than the panel based calculations, but higher than the test value. The difference could be caused by inaccurate material values (only the isotropic material data were known, no orthotropic data were available) and imperfections in the produced blade. The lowest critical load factors were found for r = 12 000mm and r = 12 500mm, which agrees with the panel based analyses results. These locations are also the expected locations for possible failure, because of a thickness change in the sandwich at these locations.

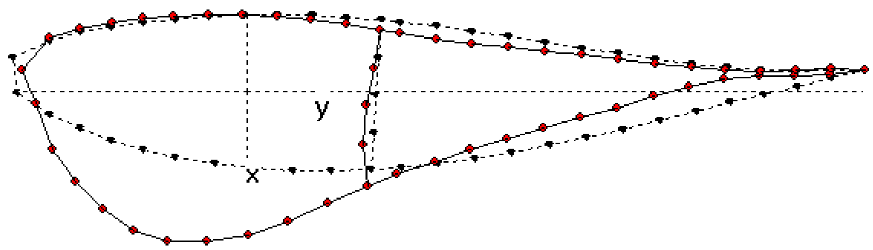

Figure 7: Failure mode of STARION test specimen 1 calculated with FINSTRIP

3.4.3 Buckling analyses with CROSTAB

The buckling analysis with CROSTAB were performed using the `.buc` files that were received from WMC for the same cross sections as used for the buckling analyses with FINSTRIP. For the buckling analyses with CROSTAB, the following modifications were issued:

panel_data, ON This input item has no effect on the buckling load calculation, but it gives additional output for each of the panels, including the buckling-load factors following the 'Design Rules'.

pre_buckling, ON With this option CROSTAB accounts for the effect of the initial curved geometric shape.

Shearweb material that is symmetric. In FINSTRIP all panels are defined symmetric w.r.t. their 'midplane' while in CROSTAB the material is located including their thickness. In the `.buc` files the material is symmetric if half of it is specified with negative thickness.

Shearweb with glue ('PASTA') in the tail. This is applied because CROSTAB assumes the tail not 'closed' if this is not explicitly specified.

Together with the `.buc` files also the `.ldc` files were provided by WMC, which contain the 'load combination'. These files contain only the sectional bending moments which correspond with a force of 2300Nm applied at r = 14 000mm. These shear forces are added to the `.ldc` files, although they are not used for the former versions of CROSTAB and FINSTRIP.

With the modified `.buc` files the critical loads for buckling were calculated with CROSTAB release "SEP-2002". Initially the buckling loads calculated with CROSTAB for the most critical panels were used. These loads were calculated for panel '4', which is the complete nose panel. This nose-panel appears to have a relatively thin lay-up at its panel-centre, which is probably the reason for the under-estimation. For comparison also the CROSTAB predictions for panel 4 are added which is the suction-side half of the leading-edge. The results of these analyses are compared with the results from the other tools in Figure 8.

Figure 8 shows that the predictions with FINSTRIP and with CROSTAB (for panel 4) agree reasonably well. The fact that the CROSTAB predictions for panel 3 are much smaller shows that it is necessary to include the detailed material distribution. The fact that both the FINSTRIP and CROSTAB (panel 4) predictions are too high compared with the experiments may indicate that both these prediction methods can be improved.

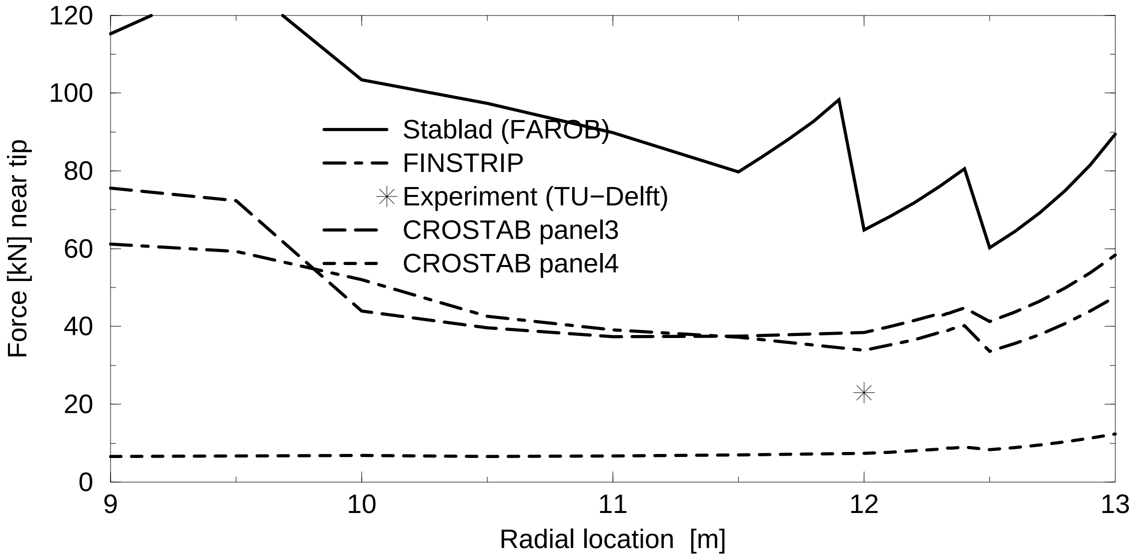

Figure 8: Experimental and predicted loads for STARION test specimen 1

3.5 STARION test specimen 2

The STARION test specimen 2 showed clearly several buckling waves before failure. Total blade failure occurred at r = 12 300mm at a load level of 21kN. At a load of approximately 15kN, the strain gauge measurements start to differ from the tangent lines, thus showing the nonlinear behaviour as can be expected in case of buckling. At a load of approximately 16kN, buckling waves were visually observed.

3.5.1 Results of panel based buckling analyses with Focus4

The critical load factors calculated with FAROB for the nose, middle, and tail panels for the part of the rotor blade between r = 5000mm and r = 14 000mm are shown in Figure 9. The critical load factor as shown here is the critical load calculated by FAROB divided by the maximum load of the test. For the tail panel two lines are given, one for the lay up at 65% and one for the lay up at 70%. Due to the difference in lay up and skin thickness at these two locations, completely different critical load factors were obtained at these two locations. The two lines can be seen as the upper and lower bound for the critical load factors for the tail panel. The sawtooth shape of the curve for the tail panel (lay-up at 70% chord) is caused by discrete steps in the skin thickness in axial direction. The blade radius where failure occurred at the test (r = 12 300mm) is also marked in the graph. According to the FAROB calculations, blade failure can be expected between r = 12 000mm and r = 12 800mm, which agrees with the test (failure at r = 12 300mm). At r = 12 300mm, the critical load factor is 0.45 for the lay-up at the 65% chord location and 3.45 for the lay-up at the 70% chord location. The actual critical load factor for this panel will be somewhere in between. Failure of the blade occurred at a load of approximately 60% of the UCS.

3.5.2 Results of cross section based buckling analyses with Finstrip

The FAROB model of the original STARION test specimen 2 has partly been rebuilt to make the model compatible with Focus4. The foam material definition in the model for the skin sandwich was changed from isotropic to core material. STARION test specimen 2 has been reanalysed with the FINSTRIP program (version 1.4). In the framework of the STARION project only the cross section that failed during the test was analysed with FINSTRIP. For the same locations where the panel based buckling analyses where performed, a set of 24 `.buc` files have been generated using

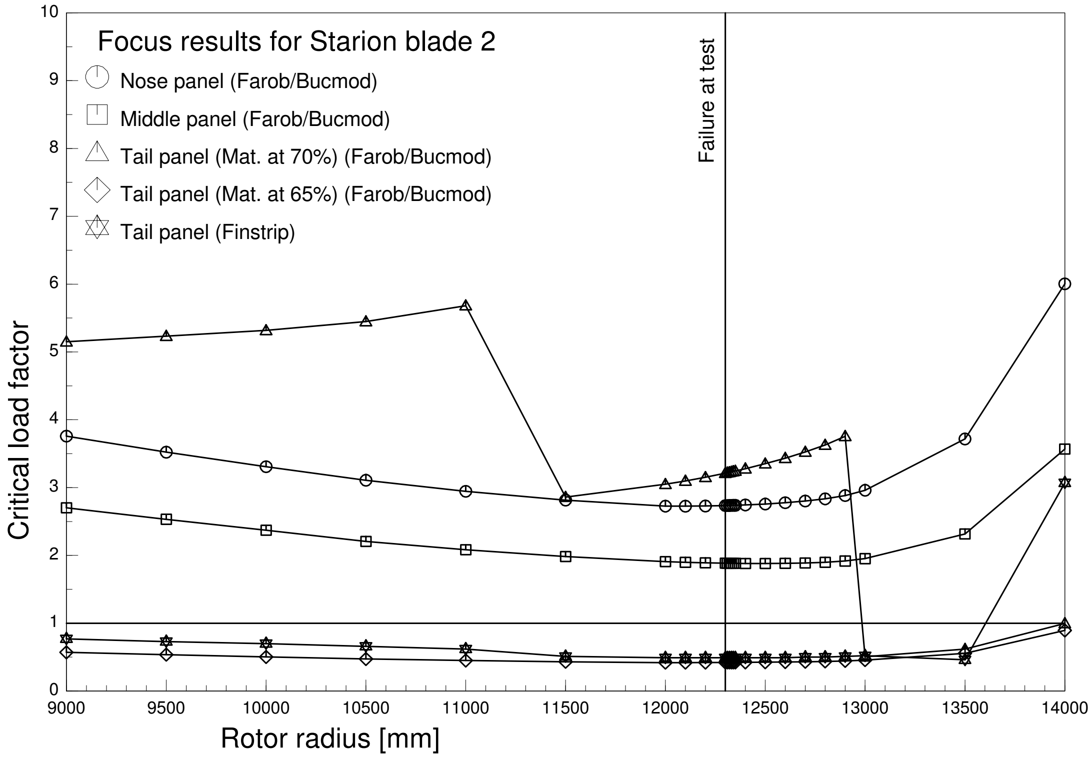

Figure 9: Critical load factors for panel based buckling analyses for STARION spec.2

Focus4. The analyses where performed with default settings from Focus4 (maximum element size 50mm, minimum half-wave length 10mm, maximum half-wave length 1200mm, and 20 wavelength steps). In the critical area, these default settings resulted in a calculation accuracy of approximately 2%, which is acceptable. For all sections tail panel buckling (under compression) was predicted. This agrees with the test result. The analyses results are shown in Figure 9. The buckling shape at $r = 12\,300$mm is shown in Figure 10. Interestingly, the FINSTRIP results follow approximately the 65% line of the panel based buckling analyses and not the 70% line. The difference is that the panel of the 70% line includes the layup of the sandwich. The sandwich covers only partly the tail panel, and is probably not placed optimal –for buckling– in the tail panel. According to the FINSTRIP calculations, the buckling strength of the tail panel is not significantly improved by the sandwich.

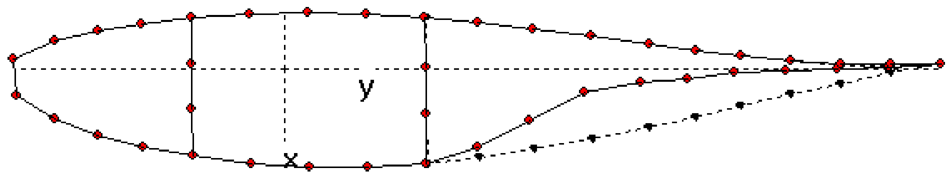

Figure 10: Failure mode of STARION test specimen 2 calculated with FINSTRIP (41 nodes)

3.5.3 Buckling analyses with CROSTAB

The buckling analyses with CROSTAB were performed using the `.buc` files that were received from WMC, similar as for STARION specimen 1. For the buckling analyses with CROSTAB, the following modifications were issued on these files:

panel_data, ON This input item does not have an effect on the buckling load calculation, but it gives additional output for each of the panels, including the buckling-load factors following the 'Design Rules'.

pre_buckling, ON With this option CROSTAB accounts for the effect of the initial curved geometric shape.

imperfection, 0.01 This is only used for the version CROSIMP and specifies a geometric imperfection of 1% of the wall thickness.

PVC core material The PVC material used in the `.buc` files used for the analyses was initially defined as isotropic. Because the PVC foam served as core it was finally modelled as 'core'.

Shearweb material that is symmetric. In FINSTRIP all panels are defined symmetric w.r.t. their 'midplane' while in CROSTAB the material is located including their thickness. In the `.buc` files the material is symmetric if half of it is specified with negative thickness.

Shearweb with glue ('PASTA') in the tail. This is applied because CROSTAB assumes the tail not 'closed' if this is not explicitly specified.

With the `.buc` files also the `.ldc` files were provided by WMC, which contain the 'load combination'. These files contain only the sectional bending moments which correspond with a force of 2100Nm applied at r = 14 000mm. These shear forces are added to the `.ldc` files, although they are not used for the former versions of CROSTAB and FINSTRIP.

For the modified `.buc` files the critical buckling loads were calculated with CROSTAB release "SEP-2002". The results of these analyses are compared with the results from the other tools in Figure 11.

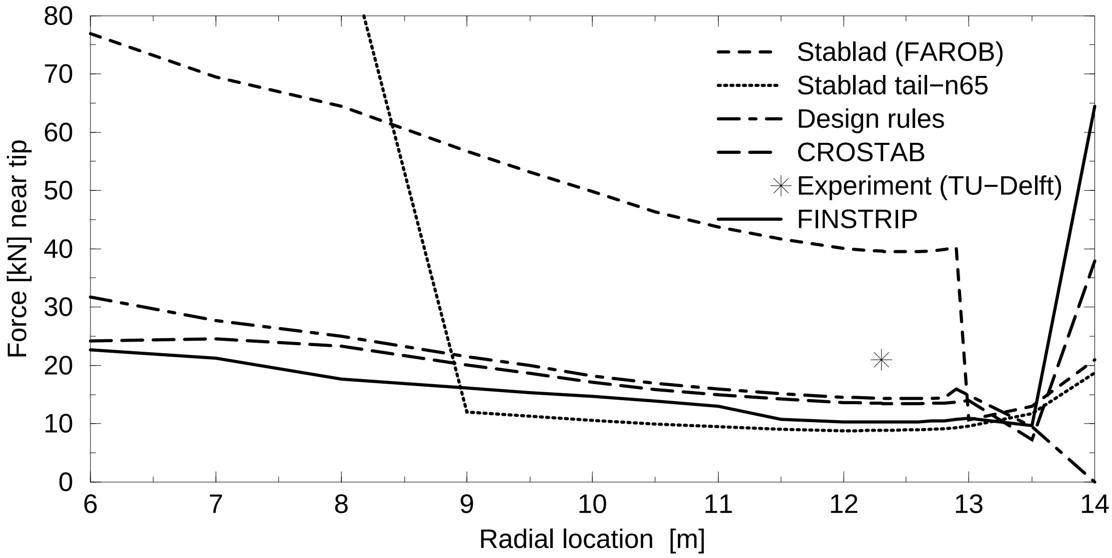

Figure 11: Experimental and predicted loads for STARION specimen 2

3.6 STARION test specimen 3

Within the Dutch STARION project several rotor blades were tested on buckling, among which was one 9m long rotor blade, the 3-rd specimen.

On this blade a complete test programme was performed including fatigue testing, static testing, and material testing. During the static testing at a high load, buckling was detected. Since the predicted buckling load was substantially higher than the design load, no detailed measurements were taken to determine the exact buckling load (force/strain curves). Neither was the geometry of the cross-sectional contour measured.

3.6.1 Experimental Results

After the first test three different locations were marked where buckling had occurred at a load of 7500N. These locations were s = 3.28m, s = 3.85m and s = 4.11m. The location of the bulges in the cross section has not been reported.

After unloading, another test was done to determine the load at which buckling started. During this second test the depth and the dimension of each "buckle" was measured. At a load of 5761N the first signs of buckling were detected with a depth of approximately 1mm. Further loading up to 7500N showed the same buckle depth as in the first test from which it was concluded that there is no plastic deformation nor material failure.

The third specimen has not been tested up to failure.

3.6.2 Comparison of Experimental and Calculated Results

For the 3-rd blade the critical bending moments were calculated with CROSTAB and with FINSTRIP. For this purpose a `.buc` file of a relevant cross section has been compiled by the manufacturer.

All calculations show buckling of the central panel between both webs, which does not have a sandwich layup. The critical moments for the other panels in the blade cross section are far higher. For confidentiality and for evaluation of the calculated results the critical moments are made dimensionless by dividing through the experimental buckling load. Since buckling is seen over a certain span-wise distance the experimental buckling load is the bending moment in the middle of the area where bulges occur. The dimensionless critical moments are:

		Central panel		Tail panel	
method	condition	moment	1/2-wave	moment	1/2-wave
FINSTRIP	(version 1.0)	0.9747	0.157m		
Design rules	load at centre	0.686		8.93	
CROSTAB	pre-buckling	0.721	0.222m	10.12	0.161m
CROSTAB	no pre-buckling	0.781	0.192m	10.11	0.161m
CROSCLAMP	Clamped	1.045	0.135m	54.69	0.248m
CROSIMP	1% imperfect	96.9%	0.257m	100%	0.340m

The predictions on the last 3 lines are added to show the sensitivity for some related aspects.

The calculations with FINSTRIP are performed by ECN for which the maximum element length used is 15mm. From the fact that the FINSTRIP results and the "clamped" results are both close to the experimental load it can be concluded that the connections of the panel-edges are close to clamped and that the modelling of this cross section looks correct in both models.
The values calculated with CROSTAB are smaller because the edges are assumed to be simply supported.

The values following the 'Design rules' are calculated with the loading in the centre of the panel for flat-wise bending.

All values from CROSTAB include the solution of the 'pre-buckling deformation'. This deformation includes the longitudinal curvature from bending of the blade and the out-of-plane bending deformation of the panels due to the "crushing loads".

The "imperfections" are variations on the geometric blade contour that are periodic in longitudinal (spanwise) direction with a shape that is similar to the collapse mode and an amplitude that is expressed as fraction of the panel wall thickness. The values listed for the "imperfect" panels are divided by the values calculated with the same algorithm as for the perfect geometry. Although the "imperfect" buckling loads are smaller, the reduction in critical load is not alarming.

4 COMPILATION OF RESULTS

Following is a compilation of the measured and predicted forces for which bucking occurs:

Method:	Experim.	NL-FEM	FINSTRIP	StaBlad	CROSTAB	Design rules
Double box 1	300N		551.4Nm		311.8Nm	416.8Nm
Double box 2	350N		551.4Nm		311.8Nm	416.8Nm
BB lower panel	11.3kN	9.08kN	13.82kN		11.32kN	12.07kN
BB upper panel	22.2kN	20.24kN	25.07kN		15.87kN	23.15kN
BB nose panel	26.0kN	41.2kN	31.13kN		30.90kN	28.50kN
BB sandw. tail	26.2kN	32.0kN	33.01kN		35.25kN	34.84kN
Starion spec. 1	23kN		33.58kN	60.22kN	6.51 / 37.38	
Starion spec. 2	21kN		10.29kN	39.39kN	13.45kN	14.32kN
Starion spec. 3	1.0		0.9747		0.721	0.686

The 'double' values for the CROSTAB predictions for STARION test specimen 1 are for the complete leading-edge panel (6.51kN) and for the suction-side half of the leading-edge panel (37.38kN). The strong under-estimation for the complete leading-edge panel is caused by the fact that release "SEP-2002" of crostab uses the lay-up at the centre of the panel and it expresses the curved panel geometry with only a few parameters.

The results for STARION test specimen 3 are made dimensionless with the experimental load.

Not all test blades were analysed with all tools. The compilation of all results still shows that serious differences appear between the predictions with each of the tools, and between the predictions and the tests.

Reasonable agreement is still found for the BUCKBLADE test specimen of which the FAROB model was built with serious attention on the geometry definition and on the material specifications.

For some of the other cases it was already noticed that the material definition was not very detailed, see the STARION Specimen 1.

5 CONCLUDING REMARKS

In general the discrepancy between the calculated and experimental buckling load, and the discrepancy between the predictions with each of the tools appears to be larger if less attention is paid to the structural model. A structural model meant for buckling analyses requires better modelling of e.g. the material data than a model meant for simply obtaining the cross-sectional bending stiffnesses. Despite the discrepancies in predicted buckling loads for some of the models, the predictions all show roughly the same critical location which indicates that the discrepancies may be caused by inaccurate material specification.

With regard to future investigations within BLADKNIK, the conclusions drawn here are addressed to a large extent to building a blade model, and to issues related to the different buckling load analysis tools.

5.1 Remarks on the program FOCUS

Following are some modifications/improvements that were found necessary or at least save a lot of 'hand-correction' work afterwards. For the latter, see **Remarks on generating .buc files**.

Orthotropic material properties Release 5.0b of FOCUS-5 does only use the longitudinal stiffness (Young's modulus) and the shear modulus for the cross sectional properties of FAROB. The programs CROSTAB and FINSTRIP are based on laminate theory which means that all stiffness properties have to be valid, including the Poisson's ratio.

Analyse and eliminate the difference in bending stiffness The difference in sectional bending stiffnesses between FAROB and CROSTAB has to be eliminated. Initially this removes the necessity of adding these stiffnesses in the `.buc` files for FINSTRIP analyses. It is expected that allowing web-joints and including all orthotropic material properties in FOCUS will eliminate these differences to a large extent. After all, elimination of the differences in bending stiffnesses improves the reliability of both FAROB and CROSTAB.

5.2 Remarks on building a FOCUS model

Remarks on building a FOCUS model of a blade are given in Appendix B which is based on the experience with modelling the DEBRA blade. In fact most of these remarks are related to the program FAROB, which generates the structural sectional model used for analyses with CROSTAB and with FINSTRIP.

Modelling of the blade-axis through the airfoils For most airfoils the contour is described with respect to the chord-line, which usually runs from the leading edge to the trailing edge. Due to the camber of the airfoil the geometry is asymmetric such that the thickness on the suction side is larger than the thickness on the pressure side. When modelling the blade axis through e.g. the 25% or 30% chord locations of the airfoils the elastic axis and also the mass centreline are not smooth in spanwise direction. With regard to secondary stresses in the structure, the girder laminates should have a smooth continuous geometry in spanwise direction. To this end the designer not only has to provide a smooth blade thickness distribution but also design the blade axis in the middle of the suction and the pressure side of the airfoils, assuming that the stiffness laminates ('girders') are in the region of the largest airfoil thickness.

Account for details such as glue and lap joints The lap-joints contribute to the panel stiffness as well as the overall bending stiffness of the cross section, which both result in an under-prediction of the buckling load. Glue joints are used to connect the blade halves and are located in the trailing edge, in the leading edge, and at least in the lap joints at one side of the shear web. If these glue-joints are bridging a gap in the leading and trailing edge this will give a reduction in torsional stiffness. A glue layer between lap-joints will increase the laminate thickness and correspondingly the panel bending stiffness. For a conservative structural analysis one should model the smallest glue thickness in the lap-joints and the largest leading- and trailing edge gap that may appear in the production process.

Specify many cross sections near blade-geometry changes The program FAROB performs linear interpolation of the geometry between the cross sections for which the airfoil contour is specified. For the non-linear spanwise distribution of the blade geometry in the area of the largest chord this 'linear interpolation' may lead to inaccuracies of the structural analyses. These inaccuracies can be reduced by specifying the blade area near the largest chord with more `.PRF` files with contour coordinates.

5.3 Remarks on generating .buc files

The buckling load predictions with both CROSTAB and/or FINSTRIP are performed using the `.buc` files that are generated with FAROB.

Add lap-joints on shear web afterwards The lap-joints on the shear web can not be modelled in release 5.0b of FOCUS5. For this reason these joints were added afterwards when modelling the DEBRA blade for buckling analysis. The governing version of FOCUS does only use the longitudinal stiffness (Young's modulus) and the shear modulus for the cross sectional properties of FAROB. The programs CROSTAB and FINSTRIP are based on laminate theory which means that all stiffness properties have to be valid, including the Poisson's ratio.

Add sectional properties from CROSTAB The sectional properties that are written at the end of the `.buc` files may be an under-estimation because: **1)** the FAROB model does not allow lap-joints on the shear webs, and **2)** the orthotropic material properties are not fully accounted for. Too low sectional bending stiffnesses give an under-prediction of the buckling load because the strain in the panels is larger and because the longitudinal curvature (from elastic bending) is stronger. Note that the latter effect was not included in FINSTRIP version 1.5.

5.4 Suggestions for the buckling load prediction tools

In the BLADKNIK proposal a number of improvements have been suggested for both CROSTAB and FINSTRIP. Based on the a-priori knowledge of CROSTAB and FINSTRIP and based on the findings from performing the work for this 'state-of-the-art' report the necessities of a number of improvements are listed here.

Include transverse out-of-plane shear flexibility Following common engineering practise, the out-of-plane shear deformation is only included for sandwich panels in a structure. The combination of large laminate thicknesses and the fact that the resin in a fibre-reinforced material gives an out-of-plane shear stiffness that is smaller that the fibre-stiffness the out-of-plane shear flexibility will give a reduction of the buckling strength in the blade root region.

Shear webs with symmetric material The current `.buc` files do not always describe the shear web material symmetric w.r.t. the web mid-surface. In CROSTAB the location of the material is taken into account. For the investigations reported here, the material layup definition of all shear webs was modified. An improvement would be to let FOCUS perform this task. The difference in results however is not large.

Eigenvalue solver in FINSTRIP The eigenvalue solver that is used in FINSTRIP returns all eigen-modes, while for buckling only the lowest mode is of interest. An acceleration of the analyses with FINSTRIP is envisaged by looking for a more dedicated and efficient eigenvalue solver. With a faster solver, it is also possible to use more elements in the FINSTRIP model. Realise that the models of the BUCKBLADE test specimen have about 80 to 90 nodes, the model in Figure 7 has 53 nodes, and the model in Figure 10 has 43 nodes. For about 100 nodes the influence of the element size is reduced to an insignificant property.

5.5 More sophisticated improvements

Shear web buckling Shear web buckling requires **1)** a model in e.g. CROSTAB and/or FINSTRIP for the transverse shear loading in the cross section and **2)** a solution method for buckling of panels with shear stress (combined with in-plane bending and transverse compression). This has already been foreseen in the BLADKNIK proposal.

Geometric longitudinal panel curvature The blade root area has a non-linear thickness distribution such that the longitudinal curvature of the shell panels is opposite to the transverse curvature (in chordwise direction). The combination of a positive and negative curvature in the perpendicular directions gives a smaller buckling load than curvatures of the same sign. This means that omission of the geometric longitudinal curvature gives an over-prediction of the buckling load, which is not conservative. Including the geometric longitudinal curvature of the blade outer shells requires that this curvature also has to be modelled in the FAROB model and written in the `.buc` files or that this has to be added afterwards ('by hand'). The FAROB model in the current FOCUS version does still apply a linear interpolation of the geometry between neighbouring cross-sections.

5.6 Aspects of minor importance

For the majority of the current (and future) generation of rotor blades the following aspects or items of a buckling load analysis are not of serious importance.

Geometric twist of the blade outer contour Mainly as a result of the aerodynamic twist of the rotor blade but also due to the spanwise variation of the airfoil shape, the 'panels' of the outer contour also have some amount of twist. To a less extent as for the longitudinal curvature, this twist can give a reduction of the predicted buckling load. Accounting for this twist requires that this is also modelled in FAROB, and that CROSTAB and/or FINSTRIP has to use this 'twist' information.

Anisotropic material properties Most rotor blades have orthotropic laminates, viz that do not have coupling between shear loading and the other load components in the panels.

REFERENCES

[1] Bert, C.W. and Crisman, W.C. and Nordby, G.M. ;
'*Fabrication and Full-Scale Structural Evaluation of Glass-Fabric Reinforced Plastic Shells.*'. Journal of Aircraft, Vol.5 No.1, Jan-Feb 1968,
University of Oklahoma, Norman, Oklahoma.

[2] Böhnish, H., Hald, H., Kensche, Ch., Kussmann, A., Molly, J.P., and Seifert, H. ;
'*Entwicklung, Bau und Betrieb einer 30/100 kW Windkraftanlage*'.
DFVLR-Mitteilung 88-06, Stuttgart, Germany, 1988.

[3] Bruhn, E.F. ;
'*Analysis and Design of Flight Vehicle Structures*'.
Tri-state Offset Company, USA, June 1973.

[4] Brush, Don O., and Almroth, Bo O. ;
'*Buckling of Bars, Plates and Shells*'.
McGraw Hill Kogakusha Ltd, 1975, ISBN 0-07-032758-0.

[5] Budiansky, B. ;
'*Buckling of Structures*'. IUTAM Symposium, Cambridge/USA, 1974, Springer Verlag Berlin Heidelberg New York 1976.

[6] Geier, B. ;
'*Buckling and Postbuckling Behaviour of Composite Panels*'.
DFVLR/IB 131-87/04, also GARTEUR/TP 029,
DFVLR Institut für Strukturmechanik, Braunschweig, April 1987.

[7] Geier, B. ;
'*Buckling and Postbuckling Behaviour of Composite Panels*'.
DFVLR/IB 131-87/04, also GARTEUR/TP 029,
DFVLR Institut für Strukturmechanik, Braunschweig, April 1987.

[8] Geiger, T. (DLR) ;
'*Report about Buckling Tests*'.
Deutsches Zentrum für Luft- und Raumfahrt (DLR),
Institut für Bauweisen- un Konstruktionsforschung, May 26, 1998.

[9] Goeij, W.C. ;
'*Implementation of Bending-Torsion coupling in the design of a wind turbine rotor blade*'.
ECN-I--98-047, Petten, The Netherlands, October 1998.

[10] '*HANDBUCH STRUKTUR BERECHNUNG, Beulwerte k_{c} in Abhängigkeit van Belastungsart, Randbedingung und Seitenverhältnis.*'.
VFW 45111-01, Ausgabe A 1969.

[11] Kooijman, H.J.T. ;
'*BENDING TORSION COUPLING OF A WIND TURBINE ROTOR BLADE*'.
ECN-I--96-060, ECN, Petten, December 1996.

[12] Kooijman, H.J.T. ;
'*ULTIMATE WIND TURBINE*'.
ECN-CX--98-024 (confidential), ECN Petten, March 1998.

[13] Lee, R.L. and Lu, S.Y. ;
'*General Instability of Included-Stiffened Cylinders Under Bending*'.
Journal of Applied Mechanics, September 1969 pp.403-407.

[14] Lindenburg, C., Joosse, P.A. (SPE), Geiger T. (DLR) and Weisser, B. (LASSO) ;
'*BUCKBLADE Part 1, Model Descriptions and Design Guidelines*'.
ECN-CX--99-025 (confidential), ECN Petten, June 1999.

[15] Lindenburg, C., Geiger, T. (DLR), Joosse, P.A. (SPE) and Weisser, B. (LASSO) ;
'*BUCKBLADE Part2, Validation of Tools on Rotor Blades* '.
ECN-CX--99-026 (confidential), ECN Petten, June 1999.

[16] Lindenburg, C., Bax, T., Joosse, P.A., Pronk, P., Vergouwen, P., and De Winkel, G.D. ;
'*STARION, Model Description and Evaluation of Stability Analysis Tools*'.
ECN-CX--99-030 (confidential), ECN Petten, June 1999.

[17] Lindenburg, C. ;
'*STABLAD, Stability analysis Tools for Anisotropic rotor BLADe panels*'.
ECN-CX--99-031 (confidential), Petten, June 1999.

[18] '*Buckling of Thin-Walled Circular Cylinders, Space Vehicle Design Criteria (Structures)*'.
NASA SP-8007, August 1968.

[19] Nemeth, M.P. ;
'*Buckling Behaviour of Long Anisotropic Plates Subjected to Combined Loads*'.
NASA Technical Paper 3568, Langley Research Center, Hampton Virginia, 1995.

[20] Roark, R.J. and Young, W.C. ;
'*Formulas for Stress and Strain, fifth edition*'.
McGraw-Hill Kogakusha Ltd., 1975, ISBN 0-07-053031-9.

[21] Sun, G. ;
'*Buckling and Initial Post-Buckling Behaviour of Laminated Oval Cylindrical Shells Under Axial Compression*'.
ASME Journal of Applied Mechanics, Vol.58, pp.848-851, September 1991.

[22] Tennyson, R.C. (University of Toronto, Ontario) and Muggeridge, D.B. (Fleet Manufacturing Ltd, Fort Erie, Ontario);
'*Buckling of Laminated Anisotropic Imperfect Circular Cylinders under Axial Compression*'. Journal of Spacecraft, Vol.10, No.2, February 1973.

[23] Varik, M.J. van ;
'*FINSTRIP : Theoretical Reference*'.
BU-TN-003, STORK Product Engineering B.V., June 30, 1997.

A MODELLING OF THE DOUBLE BOX BEAM IN FOCUS

W.C. De Goeij investigated the applicability of bending-torsion coupling on wind turbine rotor blades. For this purpose 2 'Double-Box' beams of 1m length were built and tested at the Aerospace laboratory of the Technical University of Delft, see [9].

Although the Double-Box Beam investigated by De Goeij is not very representative for wind turbine rotor blades, it is included as reference beam because of its simple rectangular geometrical shape which allows analytical verification of some properties if needed.

A.1 Specification of the 'Profile'

The FAROB model of the Double-Box Beam was built as if the box is a blade with one of the diagonals equal to the chord. The profile contour was given the coordinates of the outer dimensions of the box, which has a width of 123.3mm and a 'height' or 'thickness' of 86.5mm. The 'chord length' is thus 150.62mm. Giving this 'box-blade' a twist of 35.05° finally sets it in an upright position.

The geometry of the cross section has to be defined in FAROB with a '*Profile*' file. The coordinates that were written in this file were the coordinates of the edges and also the coordinates of the points that are about 9mm on each side of the edges. The latter coordinates were added to avoid that CROSTAB will interpolate a curved 'spline' around the edges.

A.2 Specification of the Lines for the shear web

The panels of the Double-Box that were bonded did form a shear web. This shear web was modelled between 'Lines' that were defined in terms of coordinates along the chord line. Here it was found that defining the chord line in terms of coordinates along the chord and on other radial stations in terms of coordinates along the contour may not give a straight line because the 'chord coordinate' and the 'contour coordinate' are interpolated independent of each other.

A.3 Specification of materials

The walls of the Double-Box beam contain (off-axis) UD layers of carbon fibre and 0°/90° glass fibre crossply. The material properties of these layers are:

	E_1 [MPa]	E_2 [MPa]	ν_{12}	G_{12} [MPa]	density [kg/m^3]
Carbon fibre UD	95000	5000	0.3	4300	1550
Glass fibre 0/90	13000	13000	0.0	4300	2060

A.4 Generating .buc files

For the Double-Box beam the so-called `.buc` files were generated for the locations 0.0m, 0.2m, 0.4m, and 0.6m from the root. The cross-sections for these locations are identical. The files are still generated for different sections, to verify whether the different options to specify the *Lines* are consistent.

A.5 Modifications to the .buc files

Shear web material

The `.buc` files generated with FAROB have shear web specifications of which all layers have 0°orientation w.r.t. the blade axis (beam axis). For the Double-Box beam the shear web also contains off-axis layers of UD carbon fibre sheets. In the `.buc` files these fibre-sheets were given the correct orientation of 25°.

Specific items for CROSTAB

For using the program CROSTAB the following input items were modified or added.

Shearweb for closing the trailing edge.
In the program CROSTAB the begin and the end of the 'Profile' are only structurally connected if a 'Shearweb' is specified explicitly between these points. For the `.buc` files of the Double-Box beam this was done with a 0.8mm thick layer of 0°/90° material between 0.1% and 99.9% of the Profile contour. By giving this layer an orientation of 45° it will not add much torsional flexibility.

Shearweb material that is symmetric.
In FINSTRIP all panels are defined symmetric w.r.t. their 'midplane' while in CROSTAB the material is located including their thickness. In the `.buc` files the material is symmetric if half of it is specified with negative thickness. This was done by hand in the files for the Double-Box beam.

panel_data Flag (LOGICAL) that indicates whether panel related properties (geometry and A, B, D matrices) are written.

pre_buckling Flag (LOGICAL) that indicates whether the elastic pre-buckling deformation of the loaded blade structure, such as longitudinal curvature under bending, is to be solved or not.

nodes at the edges.
The program CROSTAB performs buckling analyses for panels within a cross-section. The 'nodes' of these panels are modelled at the web-profile intersections and at the begin and end of the profile. For the rectangular shape of the Double-Box beam, the edges do not allow much deformation, and can therefore be modelled as additional nodes. This is realised by adding some explicit 'node' statements in the `.buc` files for 3 of the edges, knowing that the 'trailing edge' is already modelled as a 'node'.

It should be noted that after future developments of CROSTAB and of FOCUS these hand-modifications of the `.buc` files are to be avoided as much as possible.

Specific items for FINSTRIP

For using the program FINSTRIP the following input items were modified or added.

farob Input item written by FAROB with the sectional bending stiffnesses.
This input item is used to compensate for the fact that in FINSTRIP the wall thickness is not modelled in detail. The lines with sectional stiffnesses were copied from the output of CROSTAB calculations, which are quite different because CROSTAB accounts for the correct laminate theory, in particular for the off-axis UD layers of the Double-Box beam.

lambda_max The maximum spanwise half-wave length of the bulges (or 'buckles') for which CROSTAB and FINSTRIP search for the smallest buckling load. This was set to 200mm.

lambda_min The minimum spanwise half-wave length of the bulges (or 'buckles') for which FINSTRIP searches for the smallest buckling load. This was set to 30mm.

lambda_steps The number of increments of the spanwise half-wave length between **lambda_min** and **lambda_max** for which FINSTRIP searches for the smallest buckling load. The value of **lambda_steps** was set to 20, for which the subsequent 'lambda' values differ less than 10%.

maxlength_element The maximum length over the contour of the 'elements' used for buckling analyses with FINSTRIP. The FAROB output contains a value 50.0mm. For the relatively small Double-Box beam the maximum length of an element was set to 9mm. The cross section of the Double-Box beam thus contains 61 nodes.

A.6 Comparison of bending stiffnesses

Initially the sectional bending stiffnesses are written by FAROB at the end of the `.buc` files. In the current version 5.0d of FOCUS5 this does not yet include the lap-joints on the shear webs while the panel stiffnesses are not completely based on laminate theory. To compensate this deficit the sectional bending stiffness properties were calculated with CROSTAB release "SEP-2002" and added later.

Comparison of sectional properties from FAROB and CROSTAB:
Zx_E, Zy_E, Area*E, Ixx_Z*E, Iyy_Z*E, Ixy_Z*E (x_farob = x_finst; y_farob = z_finst)

Farob, 0.10580E+01, 0.60430E+00, 0.63006E+08, 0.11652E+12, 0.50630E+11, -0.16324E+10
Crostab, -5.4751E-01, 1.0762E-01, 2.4050E+07, 4.4974E+10, 2.0008E+10, 1.0597E+08

It appears that the stiffnesses calculated with FAROB are about 2.5 times as high from which it follows that anisotropic (especially 'unbalanced') laminates should not be treated as orthotropic.

B MODELLING OF THE DEBRA 25 BLADE IN FOCUS5

For verification of the buckling load predictions within the BLADKNIK project, the BUCK-BLADE test specimen that were built in the DEBRA moulds were modelled in FOCUS5. It was realised that the process of modelling may have some effect on the calculated buckling loads, for which reason this process is reported here. This description is based on the properties supplied by DLR, from: 'Entwicklung, Bau und Betrieb einer 30/100 kW Windkraftanlage', [2]. The Debra25 wind turbine was developed in 1988 within a cooperation between the German institute DLR (formerly DFVLR) and the Brazilian company CTA, 'Centro Tecnico Aeroespacial'. The turbine had a 3-bladed downwind 25m diameter rotor with a 22.7m hub height. The turbine had 2 asynchronous generators with a nominal power varying between 30kW and 100kW, at a nominal rotor speed of 51.29rpm. The downwind rotor had a -10^o tilt angle and a -7^o cone angle. The distance between the rotor centre and tower axis was -0.9m. The tower had an outer diameter of 0.66m and was supported by guy wires connected at 16.3m height. The lightweight structure of the turbine and the tower aimed for easy assembly in remote areas.

The FAROB blade model was defined for the entire length of 11.6m (12.5m radius) and also the global turbine properties were defined in FOCUS5 for completeness of the model.

B.1 Blade geometry

In May 1996 DLR did send a mail with the dimensionless geometric coordinates for 11 blade cross sections. For 4 of these cross sections Andreas Röhm (LM AeroConstruct) did send the measured contour coordinates, that were finally used for the evaluation in the BUCKBLADE project, and also for the 'state of the art' reported here. The files with cross-sectional coordinates were processed/interpolated to tables with 161 points for each section, of which the first and last point are identical.

From [2] the blade root is defined at 0.9m and the chord distribution is read from the root until the tip (being 0.27m). The location of the largest chord is read from a graph and is at about r = 3.0m which is 2.1m from the rotor centre. As response to a question on the chord and twist distribution Christoph Kensche did sent a fax (Febr. 6, 1997) with the table:

Radius [m]	Blade Chord [m]	Twist [^{o}]
1.0	0.500	20.6
1.6	0.545	16.6
2.025	0.682	11.8
2.5	0.830	8.5
3.0	0.898	7.2
3.625	0.858	6.16
4.325	0.812	5.1
5.125	0.759	3.8
5.85	0.711	2.7
6.388	0.676	2.0
6.7	0.655	1.7
7.525	0.601	0.9
8.3	0.550	0.3
9.25	0.487	-0.28
10.25	0.421	-0.71
11.125	0.364	-0.96
12.25	0.290	-1.00
12.50	0.200	-1.00

The last record in this table was added by ECN and described the 'rounded' shape of the tip. From graphs in [2] it looked as if the tip chord of the mould is nearly horizontal. From Figure 49 of [2]

the aft web at 4.325m looks to have an angle of about 5.3^o w.r.t. the perpendicular to the chord. It is thus concluded that the chord at r = 8.75m (7.85m from the root) is horizontal (zero twist).
On Febr. 10, 1997, C. Kensche did send an e-mail saying that in a 'diplom thesis' at DEWI the largest chord of 0.9m is located at r = 3.04m (2.14m from root). With this information the chord for locations more than 2.14m from the root was described with the linear relation:

$$chord = 0.2725\mathrm{m} + 0.627\mathrm{m} \cdot (12.5\mathrm{m} - r)/9.5\mathrm{m} = 1.0974\mathrm{m} - 0.066 \cdot r\,.$$

Using this information the following geometry properties were derived:

r	from root	chord	twist	leading edge	thickness	factor
1.0m	0.1m	0.5m	20.6^o	0.25m	100.0%	1.0
1.25m	0.35m	0.5m	20.6^o	0.25m	100.0%	1.0
1.6m	0.7m	0.545m	16.6^o	0.246m	86.0%	1.1
2.025m	1.125m	0.682m	11.8^o	0.240m	67.6%	1.2
2.5m	1.6m	0.830m	8.5^o	0.2326m	51.3%	1.4
3.0m	2.1m	0.897m	7.2^o	0.2247m	42.7%	1.5
3.625m	2.725m	0.85825m	6.16^o	0.21456m	35.78%	1.7
4.325m	3.425m	0.81205m	5.1^o	0.20301m	31.93%	1.8
5.125m	4.225m	0.75925m	3.8^o	0.18981m	28.13%	1.9
5.85m	4.95m	0.7114m	2.7^o	0.17785m	24.84%	2.0
6.388m	5.488m	0.67589m	2.0^o	0.16897m	22.84%	2.1
6.7m	5.8m	0.6553m	1.7^o	0.16382m	21.78%	2.14

Note that the chord at L = 2.1m from the root was estimated because at that location the chord distribution is non-linear. With these properties and the dimensionless contour coordinates the following tables with contour coordinates in the mould reference plane were calculated. With these tables the contours are plotted in the graph which is also enclosed. Note that the webs are drawn at L = 2.1m from the root.

The geometric description for locations outboard of L = 5.8m is modelled for simplicity with the sectional shape as for L = 5.8m.

B.2 Preparing the FAROB geometry files

The geometry is interpolated with the tool CROSTAB, see [17]. As most interpolation tools CROSTAB give smooth interpolations for uniform curvatures, and nearly 'perfect' results for cylinders. For the strong local curvature variation at the leading edge, the interpolation may not be as smooth as expected. For this reason the 'thickness coordinate' in the geometry files was scaled with roughly the inverse of the square root of the relative thickness, see the rightmost column in the table above. This scaling 'blows up' the shape. Next the interpolation is done in steps:

1 Interpolate 2 points near the leading edge and add some points in regions where a few coordinates are given.

2 Split the geometry in 2 parts and interpolate further until each part has 81 data points.

3 Merge the upper and lower part.

The 'splitting' of the contour aims at the same number of data points on the suction side and pressure side such that the spanwise interpolation between profiles in FAROB gives a smooth (approximately straight) leading edge.

The files with contour data for each section are finally written in the format of FOCUS5, where the aerodynamic suction side has a negative! Y-coordinate.

B.3 Definition of Materials

The first calculations within the BUCKBLADE project were performed for the material properties 'As Specified' that were provided by LM AeroConstruct GmbH. After the tests on the first specimen, six coupons have been cut from each buckling area of the first specimen of which the wall thickness, the fibre-resin contents and the longitudinal stiffness have been measured, see also BUCKBLADE '*Report about Buckling Tests*' by [8] DLR. These measurements give a fibre-resin fraction of 52% a thickness of 4.6mm on the lower central panel and a thickness of 4.8mm on the upper central panel. The tensile tests give a Young's modulus of 28.1GPa on the lower central panel and 29.9GPa on the upper central panel. After some investigations of the measurements by T. Geiger of LM AeroConstruct (Nov. 4, 1997) the engineering constants are those listed in the table below.

The blade shells of the BUCKBLADE specimen were cured under a vacuum bag. The assembly of these shells was done with some $\pm 45^o$ layers which have a lower fibre-resin fraction. From measurements on coupons of specimen 1 LM AeroConstruct has derived an estimate! for the $\pm 45^o$ layers as cured with a vacuum bag. The layer thicknesses were 0.46mm and 0.31mm for the 600gr/m^2 and for the 400gr/m^2 layers. The material properties derived by LM AeroConstruct are listed in the table below as 'vacuum bag'. For the 600gr/m^2 layers cured without vacuum bag a 0.54mm thickness was used while the material properties are scaled down with 0.46/0.54. These properties are listed in the table as 'no bag'.

The material properties for the 'As Measured' configuration are thus:

Material	E_1 [MPa]	E_2 [MPa]	ν_{12}	G_{12} [MPa]	density [kg/m^3]
UD fibres	33844	11657	0.211	3231	1160
$\pm$ 45^o vacuum bag	10434	10686	0.581	10210	1150
$\pm$ 45^o no bag	8888	9103	0.581	8697.4	1110
PVC60 foam (core!)	60	60	0.4	21.4	60
Glue 'Joint bonding'	3250	3250	0.3	1250	1100

In this table the 'Glue' is the adhesive with which the shear web is connected to the upper half of the specimen, and with which the trailing edge was connected. The properties for the "Joint bonding" are considered isotropic while the stiffness is a conservative estimate by LM AeroConstruct.

Little effort was paid to obtain a realistic density since this is not relevant for buckling load predictions.

B.4 Definition of Lines

Within the design package FOCUS5 the blade structure is modelled with the program FAROB. The definition of shear-webs and material-layers in FAROB is done using '`Lines`'. These '`Lines`' can be defined for a series of radial locations, using a combination of chordwise locations and a coordinate over the contour. The basic locations within the sections are the leading edge and trailing edge (for the definition of the joint laminates and the 'gaps') and the intersections of the shear webs with the outer contour. The latter intersections also determine the locations of the 'girder' laminates and (naturally) the web-joints. In the chord-related axis system the leading edge and trailing edge are simply the 0% and the 100% locations.

Web-shell intersections

The 'DEF LINE' command in the FAROB program also has an option to specify a '`Line`' as intersection of a flat plane with the contour. The specification of a flat plate is done by adding the

`/P` option to the '`DEF LINE`' command, followed by two '`point`' specifications.

For specimen 1 the webs are located at 107mm and 432mm aft of the leading edge. The first of the two 'point' specifications for the web is given by the angle of the shear web with respect to the chord line. Later (July 2005) a third '`point`' specification was added to avoid that towards the tip the 'lines' run out of the contour.

For specimen 2 the central web is located at 5% chord aft of the blade axis for the part outboard of 3.625m. For the more inboard part of the blade the central web was straight. In the FAROB model, the central web of specimen 2 was described with `point` specifications for r = 3.625m and r = 6.7m.

Leading-edge gap

The leading-edge gap of both BUCKBLADE specimen from the Debra moulds was modelled with a width of 6mm; 3mm on both the suction and pressure side. This gap was described between the `Lines` named NGAP_S and NGAP_P on the **S**uction- and **P**ressure side respectively.

Trailing-edge gap

The layers in the outer shell were modelled from the 'leading-edge gap' to the 'trailing-edge gap'. The trailing-edge gap was described with `Lines` 'TGAP_S' and 'TGAP_P' on the **S**uction- and **P**ressure side respectively.

Leading-edge lap joints

The leading edge was closed with a 80mm wide lap joint, glued symmetric with respect to the leading edge. This lap joint was described between the `Lines` named NLAP_S and NLAP_P on the **S**uction- and **P**ressure side respectively.

Aft end of the sandwich panel

The tail sections of the specimen contained a sandwich panel. The trailing-edge side of the sandwich panels was described with the `Lines` SEND_S and SEND_P which were limited by the sandwich thickness and the airfoil thickness.

Glue joint in trailing edge

In the format of the `.buc` files the contour is described starting on one side of the trailing edge and ending on the other side. The program CROSTAB assumes that the trailing edge is not connected unless this is done explicitly with a 'Shearweb'. In practise the shell halves of a rotor blade are connected with glue. At the trailing edge this 'glue joint' was modelled in the FAROB model of the Debra blade with a thickness of 10.0mm between the `Lines` GLAP_S and GLAP_P. Near the blade root these `Lines` nearly coincide with the trailing edge. In the outer part of the blade the airfoil is more thin for which the `Lines` GLAP_S and GLAP_P have a larger distance (a few mm) from the trailing edge.

UD girder edges

For specimen 1 the girders were 450mm wide and symmetric with respect to the shearwebs, so with 225mm on either side. Similar as for the web-shell intersections the edges of the girders were defined with the /P option of the `DEF LINES` command, where +225mm or -225mm was added as 'distance along the contour'.

For specimen 2 the girders were 208mm wide and symmetric with respect to the central web, so 104mm on either side of the web. Also for this specimen the girder edges were defined with the /P option while adding +104mm or -104mm as 'distance along the contour'.

Lap joints on the outer shell

For specimen 1 the shearweb was connected to the outer shells with two lap joints on either side of the web. Each of these lap-joints has a width of 50mm over the outer contour. The edges of these lap-joints were defined with the /P option while adding -50mm or +50mm. This specification does not take the thickness of the shear web into account but gives a correct representation of the amount of material.

For specimen 2 the lap joints are 80mm wide and defined similar as for specimen 1.

Lap joints on the shear web

The width of the lap-joints on the shear webs was not given by DLR and thus assumed to have a width similar as for the part on the outer contour, which was 50mm for specimen 1 and 80mm for specimen 2. For FAROB these lap-joints are 'Sections' of material over a part of the shear web, which could not be defined in former FAROB versions.

FAROB versions under Focus5.1d and later, allow definition of lines on shear webs and likewise 'Sections' of material over part of the web area using the `/P` option. This option allows modelling of lines that are located in a plane. This plane can be defined through 2 'points' on 2 different spanwise sections, and a direction perpendicular to the Profile-chord of the first 'point' specification.

This definition is in terms of a relative location on the shear-web width. For the lap-joints on the webs of specimen 1 and specimen 2, the absolute width of the shear webs are:
(following table is based on CROSTAB output of January 2005)

Radius	Spec 1 fore-web	Spec 1 aft-web	Spec 2
2025mm	364.36mm	412.33mm	457.87mm
2250mm	345.63mm	398.90mm	435.02mm
2500mm	327.20mm	382.71mm	414.98mm
2750mm	308.57mm	361.10mm	394.73mm
3000mm	291.92mm	340.49mm	375.82mm
3200mm	274.61mm	315.18mm	352.09mm
3400mm	257.45mm	289.91mm	328.56mm
3625mm	238.50mm	261.55mm	303.14mm
3850mm	228.10mm	244.19mm	287.92mm
4075mm	217.66mm	226.85mm	272.91mm
4325mm	206.02mm	207.68mm	256.41mm
4575mm	196.19mm	191.58mm	242.46mm
4850mm	185.22mm	173.93mm	227.02mm
5125mm	174.09mm	156.40mm	211.48mm
5350mm	166.97mm	144.99mm	200.72mm
5600mm	158.93mm	132.52mm	188.70mm
5850mm	150.77mm	120.29mm	176.62mm

B.5 Sandwich panels

Specimen 1 of the BUCKBLADE project was designed for buckling in the panels between the shear webs. The sandwich tail panels and also the shear webs of this specimen had a 15mm thick PVC foam 'core' with a density of 60kg/m^3. The sandwich shear webs were modelled symmetrical with respect to their centre-lines, which means that first one half of the web is specified with negative thickness (including the 'facing') and next the other half was specified with positive thickness.

Specimen 2 of the BUCKBLADE project was designed for buckling in the leading edge and also in the 'upper' tail panel. On the aerodynamic suction side the tail panel of this specimen had a 10mm thick core while on the aerodynamic pressure side the tail panel had a 15mm thick core. For this specimen the foam was also assumed 60kg/m^3 while DLR mentioned that the density and likewise the elastic properties of the foam may show variations.

B.6 Generating .buc files

For buckling load analyses with the programs CROSTAB and FINSTRIP a set of so-called `.buc` files was generated, which contain a description of the material properties for a cross section. The format of the `.buc` files was originally defined by FINSTRIP. These files were generated for some of the cross sections for which the geometric contour was specified, and also for a number of cross sections on intermediate locations which is a more thorough approach as in the former BUCKBLADE project. The locations for which the `.buc` files were generated are:

No	Radius	Loc.	L.I.P. distance
1	**2500mm**	**1.6m**	4.40m
2	2750mm	1.85m	4.15m
3	**3000mm**	**2.1m**	3.90m
4	3200mm	2.3m	3.70m
5	3400mm	2.5m	3.50m
6	**3625mm**	**2.725m**	3.275m
7	3850mm	2.95m	3.05m
8	4075mm	3.175m	2.825m
9	**4325mm**	**3.425m**	2.575m
10	4575mm	3.675m	2.325m
11	4850mm	3.95m	2.05m
12	**5125mm**	**4.225m**	1.775m
13	5350mm	4.45m	1.55m
14	5600mm	4.70m	1.30m
15	**5850mm**	**4.95m**	1.05m

The bold typed locations are the sections for which the contour coordinates were specified. The last column in this table contains the distance from the **L**oad **I** ntroduction **P**oint, which is at L = 6.0m from the root for specimen 1 and specimen 2.

B.7 Modifications on the .buc files

Definition of shear web material

In the `.buc` files the layup of the shear webs is specified with 'positive-thickness' layers only. In contrast with the existing FINSTRIP version the program CROSTAB accounts for the wall thicknesses and the asymmetry of the laminates, which means that the shear-web layup has to be changed to half the amount of layers on each side of the web-centre line (with negative and positive sign for the thickness).

Specific items for CROSTAB

For using the program CROSTAB the following input items were modified or added.

panel_data Flag (LOGICAL) that indicates whether panel related properties (geometry and A, B, D matrices) are written.

pre_buckling Flag (LOGICAL) that indicates whether the elastic pre-buckling deformation of the loaded blade structure, such as longitudinal curvature under bending, is to be solved or not.

node In the current CROSTAB release the buckling load analysis is performed for a number of individual panels. These panels are defined between the intersections of the outer shell and the shear webs. From the structure of a rotor blade cross section such as the BUCKBLADE specimen, the stiffness laminates (UD) have a much larger bending stiffness than the leading edge or trailing edge panels which implies that the weakest failure modes of the leading- and trailing edges probably do not involve deformation of these stiffness laminates. To meet these failure modes the edges of the stiffness laminates are specified in the `.buc` files with the **node** input item, followed by the corresponding contour coordinate.
It should be noted that after future developments of FOCUS the amount of hand-modifications of the `.buc` files are to be reduced as much as possible.

Specific items for FINSTRIP

For using the program FINSTRIP the following input items were modified or added.

farob Input item written by FAROB with the sectional bending stiffnesses
The fact that FINSTRIP reads the sectional bending stiffnesses from the `.buc` files (given by the keyname `farob`) implies that the sectional stiffnesses have to be corrected after adding the lap-joints on the shear webs. This was done with the sectional stiffnesses from a second CROSTAB calculation with lap-joints on the shear webs. For this purpose the output of the program CROSTAB contains a line starting with `farob,` followed by 6 sectional properties. These lines with sectional stiffnesses were copied from the output of CROSTAB calculations after the lap-joints on the shear webs were added, as described above.

lambda_max The maximum spanwise half-wave length of the bulges (or 'buckles') for which CROSTAB and FINSTRIP search for the smallest buckling load. For specimen 1 this was 400mm and for specimen 2 this was 600mm for most of the cross sections.

lambda_min The minimum spanwise half-wave length of the bulges (or 'buckles') for which FINSTRIP searches for the smallest buckling load. For specimen 1 this was 80mm and for specimen 2 this was 90mm for most of the cross sections.

lambda_steps The number of increments of the spanwise half-wave length between **lambda_min** and **lambda_max** for which FINSTRIP searches for the smallest buckling load. For specimen 1 this was 400mm and for specimen 2 this was 600mm for most of the cross sections. The value of **lambda_steps** was chosen such that the subsequent 'lambda' values differ less than 10%.

maxlength_element The maximum length over the contour of the 'elements' used for buckling analyses with FINSTRIP. The FAROB output contains a value 50mm. For the somewhat small length (nowadays) of the Debra blade the value for **maxlength_element** was set to 35mm, although the buckling loads calculated with **maxlength_element 40.0** are not more than 1% larger.

With **maxlength_element 35.0** he number of nodes of the sectional models in FINSTRIP varies between 70 and 90 for specimen 1 and between 60 and 80 for specimen 2.

The finite element model used for the calculations with MSC.MARC for the BUCKBLADE project had 74 elements in the critical cross-section, of which 4 elements for the shear web, which is of the same order as the FAROB models used for the FINSTRIP calculations.

For modern (large) rotor blades the 'default' value of 50mm gives a model with fairly sufficient degrees of freedom.

The values for **lambda_min** and **lambda_max** were obtained from the analyses with CROSTAB, which require only **lambda_max** that was set to the large value of 1000mm.

B.8 Comparison of bending stiffnesses

Initially the sectional bending stiffnesses are written by FAROB at the end of the `.buc` files. In the current version 5.0b of FOCUS5 this does not yet include the lap-joints on the shear webs while the panel stiffnesses are not completely based on laminate theory. To compensate this deficit the sectional bending stiffness properties were calculated with CROSTAB and added later. The ratio between the flatwise bending stiffnesses from FAROB and from CROSTAB is plotted in Figure 12 and 13 for BUCKBLADE specimen 1 and 2 respectively. For specimen 2 also the cross sectional stiffnesses of the model used for the BUCKBLADE project are included.

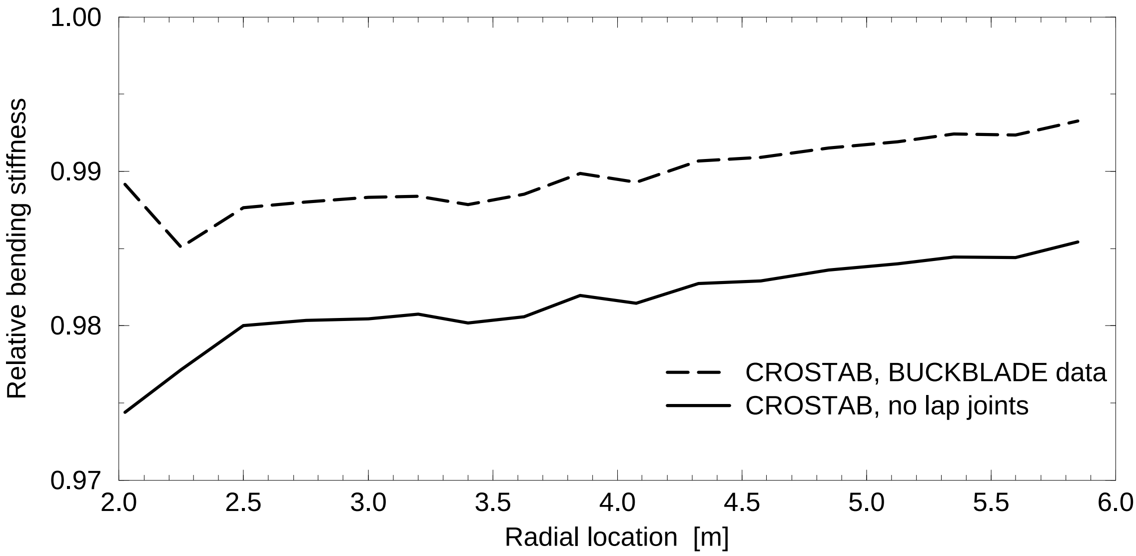

Figure 12: Ratio between flapwise stiffnesses for specimen 1

As can be seen from Figure 12 and 13 the flapwise bending stiffnesses of both specimen increase with about 1% to 1.5% if the shear-web lap-joints are modelled. Here one should bear in mind that these test specimen were designed for buckling and are thus relatively light and likewise sensitive for omission of small structural elements. In addition to this it follows that the FAROB model gives an under-prediction of the bending stiffness of about 0.8%, which is mainly due to not accounting for the shell theory. This under-prediction is not related to the specimen from the BUCKBLADE project, but may also occur for any layered structure.

Comparison of the 'coupling stiffnesses' from FAROB and CROSTAB for these specimen shows differences that are even too small to present in a graph, so assumed negligible.

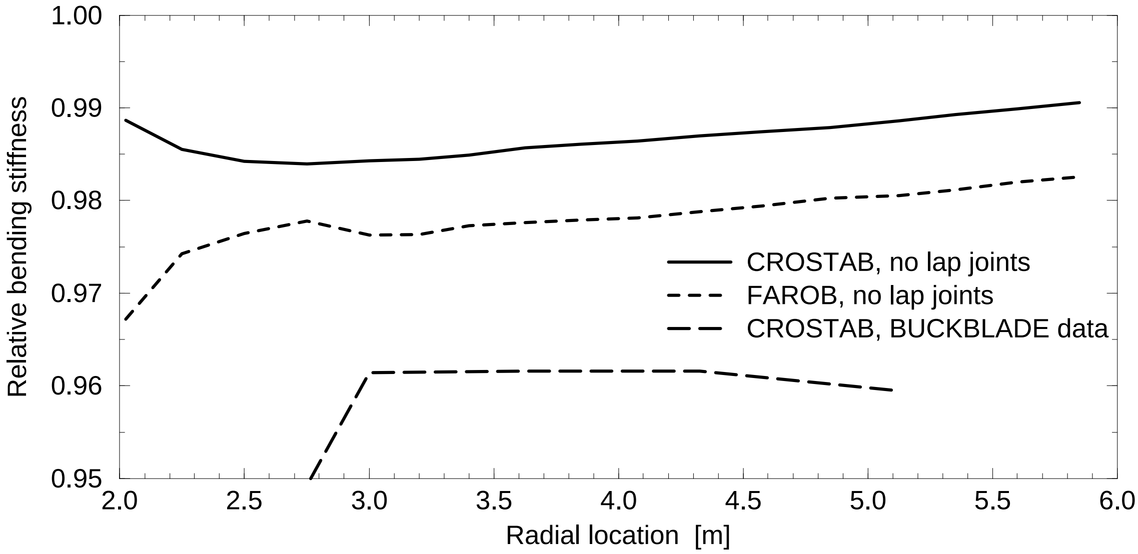

Figure 13: Ratio between flapwise stiffnesses for specimen 2

B.9 Points of Attention when Modelling a Blade for Buckling

Based on the experiences during modelling of the BUCKBLADE specimen in FOCUS5 the following recommendations are drawn.

Use about 200 airfoil geometry coordinates For the specimen from the Debra25 moulds 161 data points were used.

Equal number of geometry coordinates on both sides The spanwise interpolation of the blade contour is done by linear splining between the specified coordinate points on the contour. For this reason one should use the same number of geometry coordinates for each radial section. It is also recommended to use the same number of geometry coordinates on the suction side and on the pressure side so that the interpolated leading edge is smooth, viz without twisted wrinkles.

Use CHORD specification To reduce numerical errors from a badly defined leading edge (and thus the chord direction) one should add in the `.PRF` file the line (before 'POINTS'):

```
CHORD            0.0      0.0      1.0      0.0
```

which fixes the orientation of the chord.

Mirror airfoil Compared to the more common (XFOIL) convention for airfoils, the contour has to be specified with the aerodynamic suction side to the negative Y-axis.

Define lines The lines are used to specify the location of the layers as '**Sections**'. The lines used for the edges of the shear web should be straight because the shear web is usually flat, which is specified using `SURF-\R` or `SURF+\R` .

Define materials The program FAROB does not use the full orthotropic material properties for the calculation of the elastic properties of a cross section. However, the orthotropic material properties are used by CROSTAB and by FINSTRIP for which reason it is recommended to specify most material properties as orthotropic in FAROB. This means that in addition to `E-modulus` also `E11` and `E22` etcetera are to be specified.

Model glue in the trailing edge The trailing edge of real rotor blades is not only bonded with layers but also with glue (eventually mixed with chopped fibres). This complies with the real blade structure while it increases the geometric stability of the trailing edge. In addition it satisfies the requirement of CROSTAB that the trailing edge should be closed explicitly.

Model structural details The details are for example the lap-joints between the shear web and the outer contour. Each additional layer increases the stiffness, in particular in case of a +/-45^o layer on a UD girder laminate.

C MODIFICATIONS ON CROSTAB

Although the CROSTAB calculations at the beginning of the BLADKNIK project were addressed to describing this 'State of the Art', a few modifications were issued to the program CROSTAB. These modifications deal with bugs and small omissions of which the presence was already known at the end of the BUCKBLADE project or that were found during the calculations reported here. These corrections were issued to release "SEP-2002" of the program CROSTAB such that the results in the underlying report reflects the state of the art of the buckling load prediction methods with minimum errors from implementation in the design programs.

C.1 Output record with stiffnesses for FINSTRIP

In the program FINSTRIP the material is always modelled symmetric with respect to the outer contour which gives an over-prediction of the sectional stiffnesses. For application under FOCUS this over-prediction was eliminated by writing a record in the `.buc` files with the sectional stiffnesses. This record starts with the key-name **farob** and is read by FINSTRIP to correct the buckling load factor.

However, version 5.0b of FOCUS5 does not (yet) account for the complete orthotropic material properties which gives an under-estimation of the stiffness properties of laminates consisting of different layers, and finally a small under-estimation of the bending stiffness of a complete cross-section. It was decided to compensate for this omission and also for the fact that the lap-joints on the shear webs were not included in the sectional stiffness properties calculated with CROSTAB. For this purpose the CROSTAB output contains a record with sectional stiffnesses similar as written by FAROB, so also starting with the key-name **farob**.

C.2 Format of the Load-Combination files

In version 1.5 of FINSTRIP the format of the load-combination files (with extension `.ldc`) was modified such that it contains the sectional loads F_{x}, F_{y}, F_{z}, M_{x}, M_{y}, and M_{z} in the columns 3 through 8, with *x*, *y*, and *z* following the G.L. recommendations. The CROSTAB read-statements for the sectional loads were adapted to these FINSTRIP conventions.

C.3 Unambiguous definition of materials

It was mentioned earlier by WMC that CROSTAB may merge two materials if the name of one material equals the name of another material to which some characters are added. In the former CROSTAB versions it also appeared possible to use **ortho** materials and **core** materials with the same name which easily can lead to erroneous results from CROSTAB. These mis-interpretations were eliminated by more-rigorous checking during reading of material names and using the same arrays with material properties for **ortho** and for **core** materials. The last modification led to a reduction of the amount of source code.

C.4 Writing the average panel strains

CROSTAB has an option **panel_data** which results in output of information for each panel, that includes the strain level. The panel strains for each load combination are used together with the critical values of the panel loading for buckling load predictions using the 'Design Rules'. For curved panels the 'Design Rules' recommend to use the 6.5% + 87% + 6.5% of the strains in the first edge, the centre, and the last edge of curved panels. For this purpose, the program CROSTAB was modified to also write these 6.5% + 87% + 6.5% average panel strains.