RTU
ZINĀTNISKIE
RAKSTI

SCIENTIFIC
PROCEEDINGS
OF RIGA
TECHNICAL
UNIVERSITY

ARHITEKTŪRA UN BŪVZINĀTNE

ARCHITECTURE AND CONSTRUCTION SCIENCE

SĒRIJA 2
SĒJUMS 7

RĪGA 2006

SATURS

1. ARHITEKTŪRA UN PILSĒTPLANOŠANA

2. BŪVZINĀTNE

FINITE ELEMENT BUCKLING ANALYSIS OF STIFFENED COMPOSITE PANELS

RIBOTU KOMPOZĪTU PANEĻU NOTURĪBAS ANALĪZE AR GALĪGO ELEMENTU METODI

E. Eglitis, O. Ozolinsh, S. Gluhih and E. Barkanov

Finite element method, buckling, composites, thin-walled structure.

1. Introduction

Composite materials are desirable in lightweight structures due to their high specific stiffness and strength. Laminated composite materials provide the designer with freedom to tailor the properties and response of the structure for given loads to obtain the maximum weight efficiency. However, high modulus and strength characteristics of composites result in thin-walled structures that are often prone to buckling. Stiffened composite panels, subject of this study, are particular case of such thin-walled structures. Complexity of material and geometry of these panels makes it impossible to accurately predict their buckling behavior using analytical methods. Therefore, finite element analysis is widely used to do it. Numerous researches and studies have been performed over the years in fields of buckling of stiffened panels, mechanics of composite materials and buckling of stiffened composite panels. Finite element buckling analysis was studied since 60ties [1]. Modeling and analysis of stiffened panels is studied in source [2]. Buckling behavior of stiffened laminated plates using finite element method has been studied in [3]. Nowadays, many researches are performed on stiffened composite panels using finite element model to study buckling behavior of composite panels taking non-linear material characteristics, e.g. in [4]. Experimental studies are widely described in [5].

The aim of this study is to develop finite element model for buckling analysis of such plates that could be used in design procedure. Finite element modeling approaches, meshes and boundary conditions are analyzed to achieve best accuracy at acceptable calculation time. Several finite element models are developed and analyzed using ANSYS [6] and NASTRAN [7] finite element analysis software packages and results are presented in this paper.

2. Stiffened panel constructions

The panel consists of skin, ribs and T-stiffeners bonded to the skin (*Fig. 1*). Both skin and stiffeners are assumed to be of same material, uni-directional Intermediate Modulus (IM) carbon fibre tape. Thickness of one cured ply is 0.25 mm. Since fibre areal weight for selected material is 268 g/m^2, material density is 1.6 g/cm^3. Elastic modulus of material used for buckling analysis are shown in Table 1.

The distance between stiffeners (stiffener pitch) is 160 mm. The panel is attached to the complex structure by ribs perpendicular to stiffeners. Distance between ribs (rib pitch) is 800 mm.

Table 1. IM Fibre Tape Material properties

E_1 (N/mm2)	131000
E_2 (N/mm2)	7000
G_{12} (N/mm2)	3500
ν_{12}	0.35

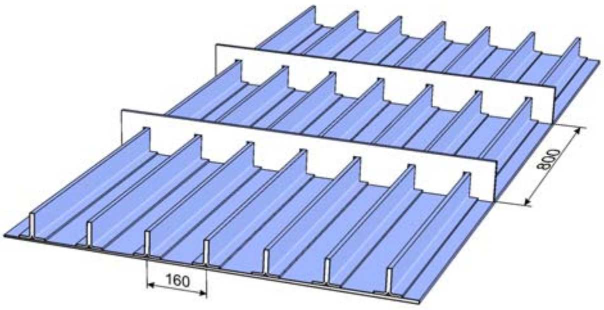


Fig. 1 Stiffened panel

3. Finite element models

Finite element (FE) models for three different panels presented in Table 2 were developed. Panels with 7 stiffener bays and 3 rib bays were used in this study. These panels differ in dimensions of stiffener (*Fig. 2*) and thickness and layer sequence (lay-up) of structure.

Table 2. Panel configurations

No.	H_W, mm	W_F, mm	Skin lay-up	Stiffener lay-up
1	18.5	40	$(\pm45;0_4;90;0)_S$	$(\pm45;0_3;90;0_4;90)_S$
2	45	75	$(\pm45;(0_4;90)_4;0)_S$	$(\pm45;0_3;(90;0_4)_3;90)_S$
3	70	110	$(\pm45_2;(0_4;90)_7;0)_S$	$((\pm45;0_3)_2;(90;0_4)_5;90)_S$

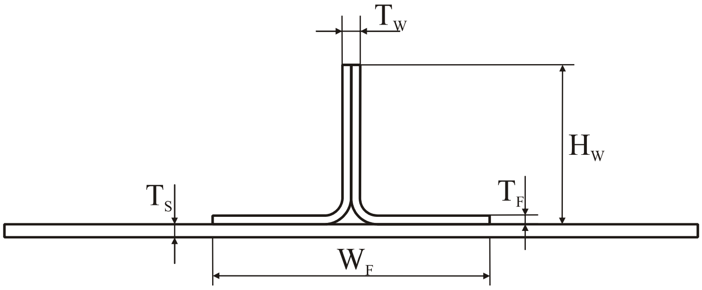


Fig. 2 Panel cross-section

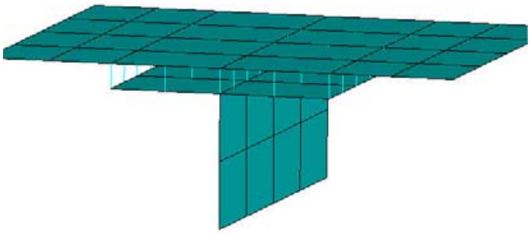


Fig. 3 ANSYS BEAM4 linked model

In ANSYS model an 8-node quadratic isoparametric layered shell element SHELL99 with six degrees of freedom per node is used to model both skin and stringer. Two basic approaches of the skin to stiffener attachment are compared in the preset study. In the first approach finite element model incorporates skin and stiffener actual off-set bonded together by BEAM4 3D elements with six DOF per node acting like rigid links (*Fig. 3*). In the second approach finite element model incorporates shared nodes between skin and stiffeners flanges (*Fig. 4*).

In NASTRAN model 4-node finite elements with six degrees of freedom per node were used to model both skin and stiffener. Model incorporates skin and stiffener actual off-set bonded with rigid elements (*Fig. 5*).

Two variations of rib incorporation are analyzed in this study. Actual rib model is compared with no rib model, where rib action is modeled as simply supported boundary conditions

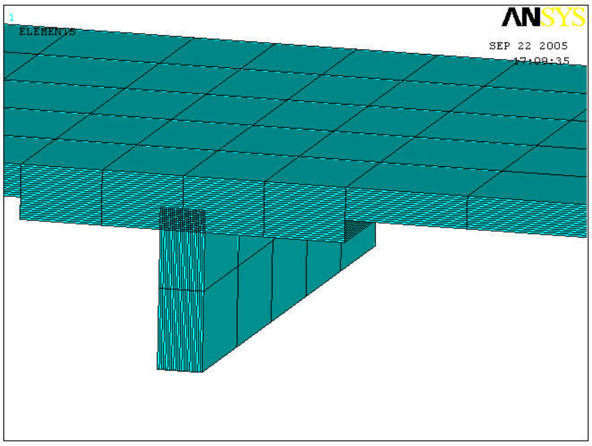


Fig. 4 ANSYS shared node model

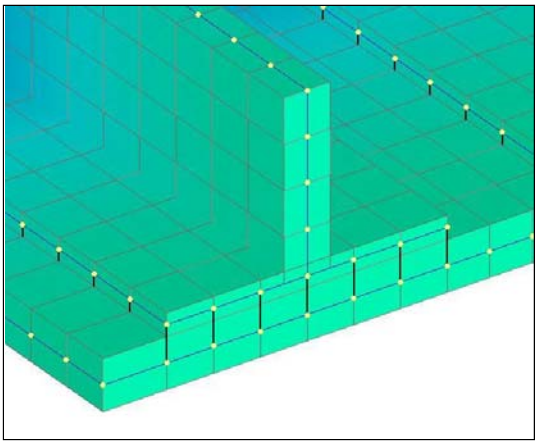


Fig. 5 NASTRAN rigid linked model

along the nodal line containing skin and flange nodes in shared node model and skin nodes in linked model. Rib attachment to the skin and flanges is made in the same manner for both models as attached stiffeners.

Three different boundary condition (B.C.) sets are compared in this study (*Fig. 6*). In the first set (*a*) ends of the panel are able to rotate around X-axis, and the load is applied to the interface of skin and stiffeners. This results an eccentricity of load. Second set (*b*) includes rotation constraint at the panel's ends. Therefore there is no load eccentricity, but side sections of the panel have more stiffness than middle one. Third set (*c*) is similar to first set, but there is additional bending moment applied to ends of panel to eliminate load's eccentricity. Calculation of panel's center of rigidity is needed to use the third set properly.

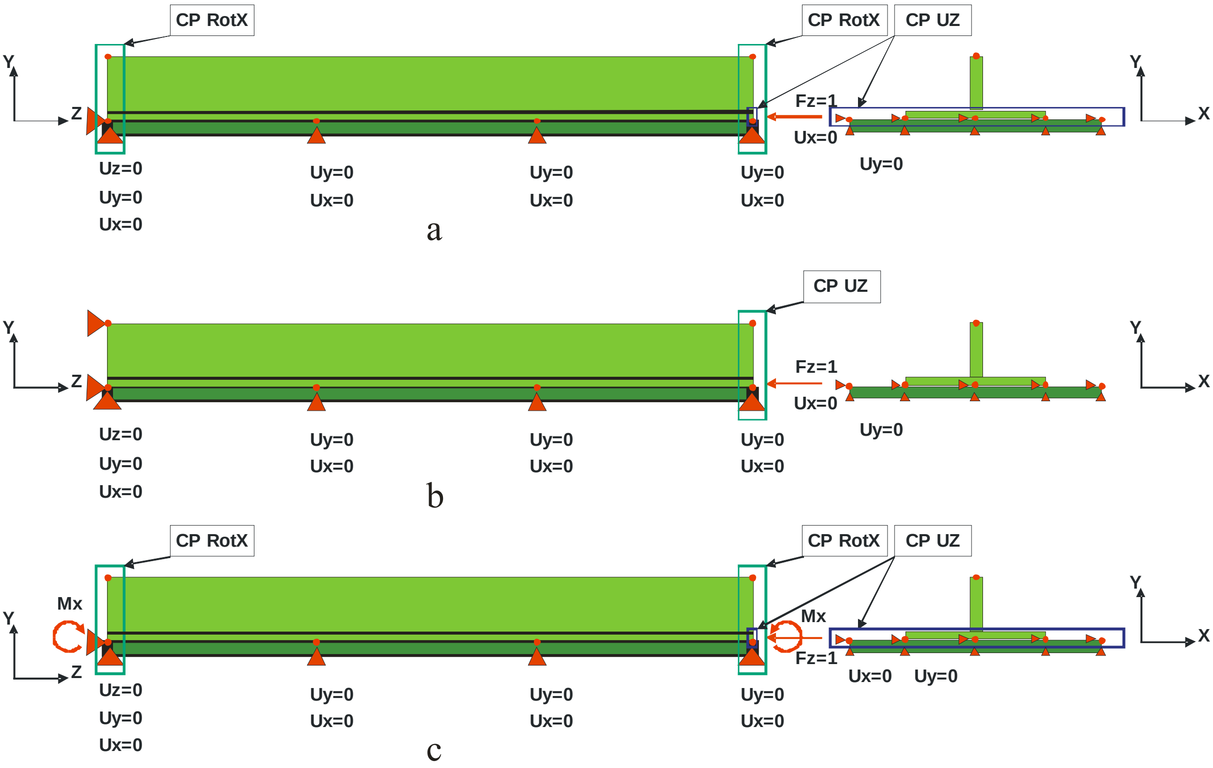


Fig. 6 Boundary conditions

a – simply supported at ends; b – clamped at ends; c – simply supported at ends, with additional bending moment applied.

4. Buckling analysis

Mesh density analyses were performed with ANSYS model for two possible options of the mesh partitions along the flange and skin span between flanges. Mesh partition along the stiffener flange having two elements and eight elements along the skin span and 150 elements along the three rib bays not showed any significant improvements in the accuracy. Twice coarser mesh with partitions of one element along the stiffener flange and four along the skin span and 75 elements along three rib bays are accurate enough. Stiffener web is two elements partition in both cases. 8-node elements produce acceptable results even when side ratio changes from 1 to 1 till 1 to 4 (greatest ratio of 1 to 3.45 is for stiffener web when its change its height from 18.5 to 70 mm with in-depth constant element size of 32 mm). Mesh density analysis was performed using both ANSYS models, using boundary condition set *a*. The results of mesh density analysis are presented in Table 3.

Table 3. Mesh density (ANSYS)

No.	Mesh	BEAM4 Pcr, kN	Difference Coarse/Fine, %	Shared Pcr, kN	Difference Coarse/Fine, %
1	Coarse	167	0.00	167	0.00
	Fine	167		167	
2	Coarse	3982	0.03	3982	0.23
	Fine	3983		3973	
3	Coarse	21002	0.29	20923	0.21
	Fine	20942		20880	

Table 4. Beam stiffness sensitivity

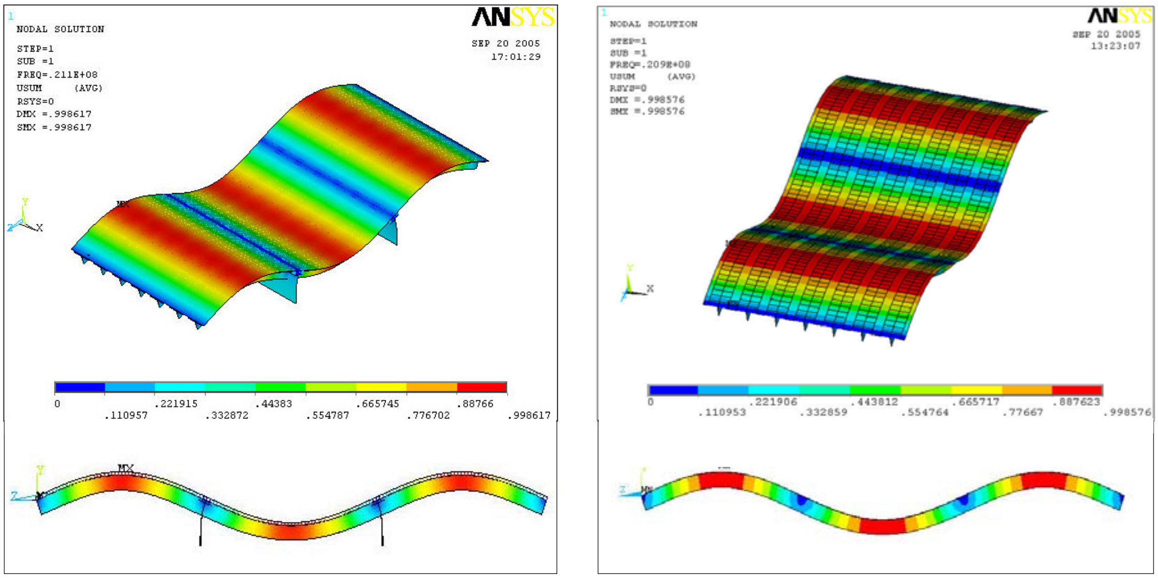
Beam Stfn., GPa	BEAM4 Pcr, kN	Difference of Pcr %
7	3979	0.046
35	3981	0.010
131	3982	0.003
655	3982	0.000

Stiffness of the BEAM4 elements used as rigid links between skin and stiffener flange did not show significant influence on the buckling load and can be freely chosen between material E_1 modulus and 25 times higher (higher than 25x cause solution failures, used 10 times higher E = 1310 GPa). The results of buckling analysis for Beam4 linked models with different beam stiffness are presented in Table 4.

Actual rib model and model with rib action modeled as simply supported boundary conditions were compared using ANSYS finite element models. No significant influence was observed (less than 1%) between actual rib model and simply supported boundary conditions model for both beam linked and shared node models. Results of buckling analysis performed to study the rib influence are presented in Table 5. The buckling mode shapes for panel configuration 3 are presented in Figure 7.

Table 5. Rib influence(ANSYS)

No.	Shared node model			Beam4 linked model		
	No rib Pcr, kN	Rib Pcr, kN	Differnce, %	No rib Pcr, kN	Rib Pcr, kN	Differnce, %
1	167	168	0.57	167	168	0.58
2	3982	3997	0.38	3982	4000	0.46
3	20923	20962	0.19	21002	21087	0.41



a b

Fig. 7 Rib influence
a – first buckling mode for model with actual ribs; b – first buckling mode for model with ribs substituted by boundary conditions.

NASTRAN model (similar to ANSYS beam linked model) with rigid elements linking skin and stiffener flanges showed good agreement with the both ANSYS models in terms of buckling load and mode shapes. The results of analysis with boundary condition set *a* are presented in Table 6.

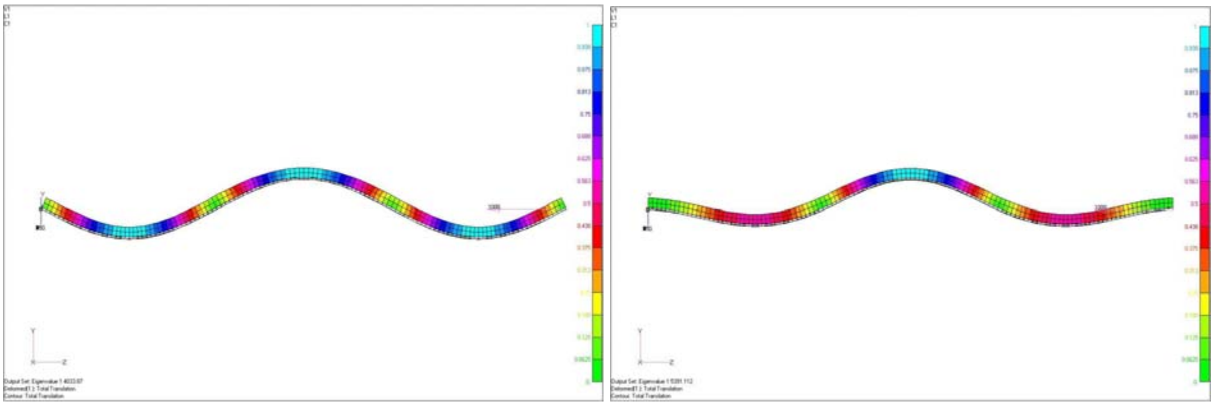
Table 6. Accordance between different models

	NASTRAN	ANSYS			
	Rigid linked	BEAM4 linked model		Shared node model	
No.	Pcr, kN	Pcr, kN	Diff., B-R,%	Pcr, kN	Diff., S-R, %
1	169	167	0.88	167	0.65
2	4034	3982	1.29	3982	1.29
3	21501	21002	2.32	20923	2.69

Three sets of boundary conditions (see *Fig. 6*) were compared in this study using NASTRAN finite element models. For first two panel configurations, results with boundary condition sets *a* and *c* were very similar differing less than 0.1% (*Table 7*). In case of third panel configuration, no eigenvectors could be found using Inverse power/Sturm and Lanczos methods for boundary condition set *c*. Boundary condition set *b* resulted in 29% to 44% higher critical buckling force, compared to buckling force calculated with boundary condition sets *a* and *c*. First buckling modes for panel configuration 2 and boundary condition sets *a* and *b* are presented in Figure 8.

Table 7. Comparison of B.C. sets

No.	B.C. set a	B.C. set b		B.C. set c	
	Pcr, kN	Pcr, kN	Diff., b-a, %	Pcr, kN	Diff., c-a, %
1	169	244	44.4	169	0.0
2	4034	5391	33.3	4037	0.0
3	21151	27192	28.7	n/a	n/a



a b

Fig. 8. Buckling mode shapes
a – buckling mode shape for B.C. set *a*; b – buckling mode shape for B.C. set *b*

5. Conclusions

For panels with configurations used in this study (see *Table 2* and *Fig. 2*) optimal mesh density for ANSYS model using 8-node elements SHELL99 is:

- 75 elements along three rib bays (25 elements per rib bay)
- two elements along stiffeners blade
- one element along stiffeners flange
- four elements along skin span between stiffeners flanges.

Use of twice finer mesh gives result differing only 0.3% in case of greatest difference (see *Table 3*). As observed during study, the finer mesh model needs about 7 times more calculation time than coarse mesh model.

It is possible to use BEAM4 elements to link stiffeners to the skin in FE models, and stiffness of their material can be freely chosen between material E_1 modulus and 25 times higher (see *Table 4*). However, use of such variation of attachment makes FE model more complicated without significant change in the accuracy of result (see *Table 3*).

There is no need to model ribs, perpendicular to loading direction. They can be substituted by simply supported boundary conditions without significant change in the result (there was less than 1% difference in all calculations, see *Table 5*).

NASTRAN finite element model can be used in calculation as well as ANSYS finite element model. Difference between results, achieved by ANSYS and NASRAN stays within 5% limit (see *Table 6*).

Boundary condition set *a* is best for calculation, if FE model represents only part of structure, as rotation at the ends of panel indicates panels continuity. There is no need to apply additional bending moment for elimination of loading eccentricity (B.C. set *c*) for buckling analysis, as it requires calculation of center of rigidity, but the changes of critical buckling force are below 0.1% (see *Table 7*). Clamped boundary conditions at the ends of plate (B.C. set *b*) may be used only if actual structure that is being modeled is also clamped at ends.

6. References

1. Gallagher, R.J. and Padlog, J. Discrete element approach to structural stability analysis // in AIAA Journal, No. 6, 1963, p. 1437-1439
2. Venkataraman, S. Modeling, analysis and optimization of cylindrical stiffened panels // dissertation, University of Florida, 1999, 210 p.
3. Guo, M., Harik, I.E. and Ren, W. Buckling behavior of stiffened laminated plates // International Journal of Solids and Structures 39, 2002, p. 3039-3055
4. Kalnins, K., Auzins, J. and Rikards, R. Fast simulation procedure for stiffened composite structures with material degradation // Riga Technical University, 2006, 8p.
5. Singer, J., Arbocz, J., Weller, T. Buckling experiments, Volume 2 // John Wiley & Sons, inc., 1998, 1732 p.
6. Moaveni, S. Finite Element Analysis: Theory and Applications with ANSYS // Prentice Hall, 2003, 840 p.
7. MSC.Software Corporation MSC.Nastran 2005 Quick Refrence Guide // Santa Ana, CA, 2004, 850 p.

Acknowledgement

The authors gratefully acknowledge support of this research work from the Latvian Coincil of Science under grant No. 06.29.8.

Edgars Eglitis, B.sc.ing.
Institute of Materials and Structures, Riga Technical University
Kalku 1, Riga, LV-1658
exec@inbox.lv

Olgerts Ozolinsh, M.sc.ing.
Institute of Materials and Structures, Riga Technical University
Kalku 1, Riga, LV-1658
olgerts@bf.rtu.lv

Sergey Gluhih, Dr.sc.ing., Doc.
Institute of Materials and Structures, Riga Technical University
Kalku 1, Riga, LV-1658
s_gluhih@mail.ru

Evgeny Barkanov, Dr.sc.ing., Asoc.Professor
Institute of Materials and Structures, Riga Technical University
Kalku 1, Riga, LV-1658
barkanov@latnet.lv

Eglītis E., Ozoliņš O., Gluhih S., Barkanov E. Ribotu kompozītu paneļu noturības analīze ar galīgo elementu metodi

Riboti paneļi tiek plaši izmantoti aviācijā, kuģubūvē un autoindustrijā, kā arī būvniecībā. Paneļi var tikt izgatavoti no dažādiem materiāliem, atkarībā no pielietošanas veida, un dažos gadījumos tie tiek izgatavoti no kompozītiem. Šādu paneļu analītisks aprēķins ir sarežģīts, un ir jāizdara vairāki pieņēmumi, kas samazina aprēķina precizitāti, lai šo aprēķinu būtu iespējams veikt. Tādēļ galīgo elementu metode tiek plaši pielietota projektēšanas gaitā šādu paneļu aprēķinam. Lai iegūtu precīzus aprēķinu rezultātus, ir jāizveido atbilstošs galīgo elementu modelis un robežnosacījumi. Šajā darbā ir parādīts modelēšanas pieejas, elementu tīkla sadalījuma un robežnosacījumu pētījums, kas veikts izmantojot trīs veidu ribotu paneļu galīgo elementu modeļus. Darba gaitā tika izmantotas divas galīgo elementu metodes aprēķinu programmas – ANSYS un NASTRAN. Šī darba rezultāti var tikt izmantoti veidojot galīgo elementu modeļus un veicot noturības aprēķinus sarežģītākām konstrukcijām, piemēram, lidaparātu spārnu virsmām.

Eglitis E., Ozolinsh, O., Gluhih S., Barkanov E. Finite element buckling analysis of stiffnened composite plate

Stiffened panels are widely used in aviation, shipbuilding and automotive industry, as well as in construction. The materials may vary depending on application and in particular cases layered composites are used to achieve maximum weight efficiency. Analytical analysis of such panels is complicated and several accuracy decreasing assumptions have to be made to perform it. Therefore finite element method is widely applied in analysis of such panels during design procedure. To achieve accurate results, development of appropriate model and boundary conditions is needed. This paper includes modeling approach, mesh density and boundary condition study, performed on three different finite element models of stiffened panels. Two finite element analysis software packages ANSYS and NASTRAN are used in this study. The results achieved in this study can be used for finite element modeling and buckling analysis of more complex structures, for example aircraft wing covers.

Эглитис Э., Озолыньш О., Глухих С., Барканов Е. Анализ устоичивости ребристых композитных панелеи методом конечных елементов

Ребристые панели широко применяются в авиации, судо- и автомобилестроении, а также в строительстве. Материал, из каторого производятся панели, зависит от их назначения. В отдельных случаях (самолетостроении) они производятся из композитов. Аналитический расчет таких паненеи сложный и не точный, так как приходиться делать некоторые упрощающие предподожения для получения решения задачи. Поэтому для расчета ребристых композитных панелеи часто обращаются к методу конечных элементов. Возникает необходимость создания соответствующих конечно-элементных моделей и граничных условиий для более правдоподобных результатов. В этои публикации представлены результаты сопоставления разных подходов моделирования, вариантов сетки конечных элементов и граничных условий. Расчеты выполнялись с помощью двух программ конечно элементного анализа - ANSYS и NASTRAN. Результаты, полученные в работе, могут быть использованы при разработке моделей расчета устойчивости более сложных структур, например, внешних панелей крыла самолета.