







Plate and shell structures

# NUMERICAL ANALYSIS OF A PANEL IN ROBOT SYSTEM .

Adam Wosatko and Anna Stankiewicz

v. 2.0, February 2013

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education



### Type of project and regional settings 1

Definition of example. Solve the panel shown in Fig. 1 using the ROBOT package. Geometry and material data are given below.



Figure 1: Geometry and material data.



(a) Window of project selection.

Figure 2: Type of project (selection).

Type of project. From the list of possible tasks choose the analysis of a panel structure (cf. Fig. 2). In the case of ROBOT 2012 the shell structure type should be selected (cf. Fig. 3) and as a next step, one has to choose  $GEOMETRY \rightarrow STRUCTURE TYPE$  and select *Plane stress structure design* as shown in Fig. 4.

**Regional settings.** Immediately after the project selection select the language and proper standards. From the top menu pick  $TOOLS \rightarrow PREFERENCES$  and change the following window options: Regional settings: Eurocode, Working language: English. The printout language self-adapts. Accept these settings (cf. Fig. 5).

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher







(a) Window of project selection.

(b) Shell structure.

Figure 3: Type of project (selection) – ROBOT 2012.



Figure 4: Type of task (selection) – ROBOT 2012.

## 2 Material and geometry settings

**Material.** In order to set material data choose  $TOOLS \rightarrow JOB \ PREFERENCES$ . In the window (cf. Fig. 6) select *Materials* from the list on the left-hand side. Next, click the button *Modification* and choose the *Other* option. Introduce the name of the material (PLSTREmat for example) and choose data for elasticity. Introduce the values of Young modulus E, Poisson coefficient  $\nu$  and find the value of Kirchhoff coefficient G according to the formula:  $G = \frac{E}{2(1+\nu)}$ . For the given data set G=10775.861 MPa. To finish the material definition, click *Add*, *OK* and close (click *OK*) the *Job Preferences* window.

**Contour definition.** To define the geometry of the panel choose  $GEOMETRY \rightarrow OBJECTS \rightarrow POLYLINE \rightarrow CONTOUR$ . In the window shown in Fig. 7 on the left, choose the *Definition Method* - *Contour* and introduce the coordinates of corners of the considered panel in the part *Geometry*. Confirm the coordinates of the points by clicking the *Add* button every time. To close the window select *Apply*. The window in Fig. 7 on the right shows the defined contour.

**Panel properties.** Choose *GEOMETRY*  $\rightarrow$  *PANELS*. In the window as in Fig. 8 on the left choose the *Contour Type - Panel*. Next, set the properties i.e. *Reinforcement - None*, *Model - Shell*. In order to assign the thickness, click the 'three dots' button and in the new window (cf. Fig. 8(b)) find material PLSTREmat in the list. Then, set the (constant) thickness Th=20 cm. Set the label TH20PS. Click *Add* and *Close* buttons. Now, in the window as in Fig. 8(a) choose *Creation with Internal point*, click anywhere in the area of the panel and close the window.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher







STAND	ARD		
Languages     General Parameters     View Parameters     Desktop Settings     Toolbark Menu     Printout Parameters     Authorization Key     Advanced	Regional settings: Working language: Printout language:	Eurocode   English  English	▲ 涨 涨

Figure 5: Regional settings.

#### 3 Definition of static and kinematic boundary conditions

**Kinematic boundary conditions.** In order to prevent the movement of the left edge, choose *GEOMETRY*  $\rightarrow$ SUPPORTS. In the window as shown in Fig. 9 select Linear support and choose Pinned or Fixed (in both boundary condition types the UX and UZ displacements are not allowed). Click the left edge of the panel to set the support and close the window.

Load types. Before the loading of the panel is defined, suitable types of loading have to be created. Choose  $LOADS \rightarrow LOAD$  TYPES to open the window shown in Fig. 10(a). Press the New button (or Add in the case of ROBOT 2012) to create a new case of loading. Change load nature into Live as shown in Fig. 10(b) and click the *New* button again (or *Add* in the case of ROBOT 2012) to create live load. In the case of ROBOT 2012 a loading subnature can be given (cf. Fig. 11). This load remains active, and the window can be closed now. The dead load case is necessary to take the self-weight of the structure into account. Loading of the top edge of the panel is going to be set in the LL1 case. To define the loading of the panel, choose  $LOADS \rightarrow LOAD$  DEFINITION. The window shown in Fig. 12(a) opens. Select Surface loading and introduce data according to the window in Fig. 12(b). Click Add and close the previous window.

In Fig. 13 the supported and loaded panel is shown.

#### 4 Mesh generation and FEM analysis

Mesh generation. Open the toolbar Options of FE Mesh Generation shown in Fig. 16. Select the area of the panel (by clicking) and the third icon from the left in the Options of FE Mesh Generation toolbar. In the window shown in

Fig. 15 select the *Meshing Method* and *Mesh Generation*. Press the *icon* to generate the finite element mesh. It is possible to click Advanced Options to set more details in a window as in Fig. 15(b).

Analysis. To perform calculations choose  $ANALYSIS \rightarrow CALCULATIONS$ .

#### 5 Results

**Results – contour plots.** Before reading the values of displacements (or stresses) one needs to set the units and formats. Select  $TOOLS \rightarrow JOB \ PREFERENCES \rightarrow UNITS \ AND \ FORMATS \rightarrow OTHER \rightarrow DISPLACEMENT.$ Compare the window in Fig. 17. To see the displacement values as a text, choose  $RESULTS \rightarrow DISPLACEMENT$ . The contour plots are available from *RESULTS*  $\rightarrow$  *MAPS*. In Fig. 18 the distribution of stress component  $\sigma_{xx}$  called SXX is shown.

Results – diagram for a cross-section. To draw a diagram of the dependence of one stress tensor component on the coordinate along a selected cross-section choose  $RESULTS \rightarrow PANEL CUTS$  and set options as shown in Figs. 19 to 21. Make sure that the Filling the interior option is not marked in the Display menu (available at the right mouse button)

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher











Name: plstremat	•	Descrip	otion: pss		
Elasticity			Resistance		
Young modulus, E:	25000	(MPa)	Design resistance:	0,00	(MPa
Poisson ratio, v:	0.16		Reduction factor for shear:	0	1
Shear modulus, G:	10775.861	(MPa)			
Force density (unit weight):	0,00	(kN/m3)			
Thermal expansion coefficient:	0,000000	(1/°C)			
Damping ratio:	0				

Figure 6: Definition of new material.









Figure 7: Definition of the panel contour.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher









Number:	1					
Contour type Panel	🔘 Opening			h		
Creation with				‡ <b>(</b>		
0,70; -1,03		(m)				
🔲 Only the	current selection		Label:	TH20PS	Color:	Auto 👻
🖱 Obiect list			00	Constant	Th = 20	,0 (cm)
		r II	O V	/ariable along a lin	e	
			©∨	ariable on a plane	) Concerce	
				Point co (r	ordinates ml	Thicknesses (cm)
🕑 Surface eler	nents (FE) list		P1:	0,00; 0,00	59.	0,0
Properties			P2:	0,00; 0,00		0,0
Reinforcement:	none	-	P3:	0,00; 0,00		0,0
Material:	PLSTREmat		F P	Reduction of the noment of inertia	1,00	*Ig >>
Thickness:	TH20PS					
initiations.		-		Parameters	of foundation	elasticity
Model:	Shell	COLUMN AND ADDRESS OF	and the second			WORKSHIT

Figure 8: Definition of the panel continued.

	]⊞ ð 🛩 ⊤
Nodal Linear Planar	
X Delete	
→ -> Pinned	
, PodUZ	
, UZ	
Current selection	
	~
	*

Figure 9: Kinematic boundary conditions.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education







Jase ues		1.12		Case acs			6
Nature:	dead 🔹	New		Nature:	Live1	•	New
Number:	1 Label: C	)L1		Number:	2	Label:	LL1
Name:	DL1			Name:	LL1		
list of de	fined cases:			List of del	ined cases		
No.	Case name	Nature	4	No.	Case nar	ne	Nature
▶1	DL1	dead	S	1 <b>→</b> 2	DL1 LL1		dead Live1
•	m		•		i	n -	
Modif	y Delete	Delete a		Modify		Delete	Delete all

(a) Dead load.

(b) Live load.



Case descri	ption			
Number:	3	Label:	LL2	
Nature:	live	<ul> <li>Subnature:</li> </ul>	Category A	•
Name:	LL2			
		Add	Modify	
List of define	d cases:			
No.	Case name		Nature	A
1	DL1		Structural	S
≁2	LL1		Category A	S
•	III			•
		Delete	Delete a	dl

Figure 11: Load subnature option, live load - ROBOT 2012.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher







oad Definition			H C		
se No: 2 : LL1 ected:			A		
Self-weight and mass	Val	lues P1	P2 (kN/m)	M1 (ki	M2 √*m/m )
Node Bar Sullace	X	0,00	0,00	0,00	0,00
	Y:	0,00	0,00	0,00	0,00
	Z:	0,00	-75	0,00	0,00
		Co	oordinates	Node	e number
	1	A: 0	,00; 1,50		
	ł	3: 🛛 🙎	.00: 1,50		
ply to	Coo	ord, syst	tem: 💿 Glo	bal 🤅	) Local
	Gar	nmə ən	de: 0.0	(Dec	-1

(a) Dead load.

(b) Live load.

Figure 12: Load definition.



Figure 13: Kinematic and static boundary conditions.

Option	ns of FE	Mesh	Gene	ration	ł.			×
2	围	92	-	PA A	X	٢		

Figure 14: Options of FE Mesh Generation.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education









feshing methods			
Available meshing methods			
Simple mesh generation (Coons)			
Simple mean generation (coons)			
💿 Complex mesh generation (Delaunay)			
Automatic selection of a meshing method			
Mesh generation	-		i and
C Automatic O User	Advanced Meshing Op	tions	8
C Element size	Available meshing methor	s	Finite elements
	Coons	often 🔫	3-node triangles
Division 1 : Division 2 :	C Delaunay	often 🔻	Type (volumetric):
1	Forcing ratio:	recommended •	4-node tetrahedrons   Forcing ratio:
	Mesh generation		recommended
Mesh of volumetric elements	C Automatic	<ul> <li>User</li> </ul>	Delaunay method parameters
	C Element size		Regular mesh     O Kana
	Unision 1	Division 2:	Delaunay and Kang
Fine Coarse	Mesh of volum	etric elements	H0 = 0.30 (m)
Additional meshing of solid surface			H max = 1000,00 Q = 1,2
	Fine Additional meshing of a	Coarse olid surface	Automatic emitters:
	Coons method parameter	C	At panel characteristic points     At support nodes
	Panel division type:		
Advanced options	<ul> <li>Triangles in triangular</li> <li>Triangles and squares</li> </ul>	ontour	Smoothing
	Triangles and squares	tri mangular contour Is in triangular contour	Triangularization near edges
	© Squares in rectangular	contour	Number of levels:
OK Cancel Help	<ul> <li>Triangles in rectangula</li> </ul>	r contour	Fine Coarse
	Enroing ratio	anu 🔻	



Meshing methods	
Available meshing meth	nods
🔘 Simple mesh genera	ation (Coons)
🔘 Complex mesh gene	eration (Delaunay)
Automatic selection	of a meshing method
Mesh generation	
Automatic	🔘 User
Element size	
Division 1 :	Division 2 ::
ru -	109.
Mesh of vol	umetric elements
Fine	Foorme
Additional meshing	of solid surface
[	6 X
Adva	nced options

Figure 16: Options of FE Mesh Generation.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education









📽 🖶 🗙 \star 🛛 P	EFAULTS				
<ul> <li>Units and Formats</li> <li>Dimensions</li> <li>Forces</li> </ul>	Displacement:	m Deg	•	0,654321e+001	
	Angle/Rotation (results):	Rad	-	0,321	
	Temperature: Weight	°C kG	•	0,21	IN E
Structure Analysis     Work Parameters	Mass:	kg	•	0,21	• • E
	Dimensionless quantity: Ruler:			0,21	- • • E
🔹 <u>O</u> pen defa	ult parameters				

Figure 17: Units and formats setting.



(a) Distribution of  $\sigma_{xx}$  without smoothing.

(b) Distribution of  $\sigma_{xx}$  with smoothing.

Figure 18: Contour plots for stress  $\sigma_{xx}$ .

efinition Cuts	Detailed	Principa	1   C
Definition of a (	out		
🔘 2 points	[		(m)
Parallel to a	axis	Z •	]
	1,07; 0,27		(m)
Point and c	direction		
	[		(m)
Cut name		( No	
A - A1			or
Normalize	global sm	oothing	
Open a new wi	ndow		
pply (	Close		He

Figure 19: Settings for a diagram.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher Education









Definition Cuts	Definition Cuts Detail	ed Principal Co	mple 🔹 🕨
Detailed			
Principal	24		
Complex	Direction X		
Diagrams	13	х уу ху	z
Reinforcement	Stresses - s	<b>v</b>	
SLS	Membrane forces - N		
	Moments - M		
	Shear stresses - t		
	Shear forces - Q		
	Displacements - u.w		
	Rotations - R		
	Soil reactions - K		
	Normalize global smoothing		

Figure 20: Settings for a diagram.



Figure 21: Diagram of  $\sigma_{xx}$  along a selected cross-section.

Project "The development of the didactic potential of Cracow University of Technology in the range of modern construction" is co-financed by the European Union within the confines of the European Social Fund and realized under surveillance of Ministry of Science and Higher