NATIONAL TECHNICAL UNIVERSITY OF ATHENS (NTUA)

SCHOOL OF CIVIL ENGINEERING

INSTITUTE OF STRUCTURAL ANALYSIS AND ANTISEISMIC RESEARCH

E. J. SAPOUNTZAKIS, Professor

R.P. 62318200

Verification Examples for

SCADA Pro™
Eurocodes

Athens 2014

Copyright

CONTENTS

INTRODUCTION

This publication presents a series of structures that have been analysed using program ScadaPro. The purpose of these analyses is to demonstrate not only the range of applications of the aforementioned program but its adequacy and accuracy in all studied cases as well. Therefore, representative cases of engineering applications are examined and a comparison of key results with theoretical solutions from literature or results from other reliable software packages is presented.

This publication constitutes a collaboration product between the Institute of Structural Analysis & Antiseismic Research (ISAAR) of National Technical University of Athens (NTUA) and ACE-Hellas S.A.

	Institute of Structural Analysis & Antiseismic Research of NTUA	**ACE-Hellas S.A.**
Design	**Vasilios G. Mokos** Civil Engineer, M.Sc., PhD **John A. Dourakopoulos** Civil Engineer, M.Sc., PhD **Angeliki D. Koroneou** Civil Engineer	**Christos P. Kostopanagiotis** Civil Engineer, M.Sc. **Chara C. Mitropoulou** Civil Engineer, M.Sc., PhD **Amalia Bagourdi Degkleri** Civil Engineer, M.Sc.
Passed	**Evangelos J. Sapountzakis** Civil Engineer, M.Sc., DIC, PhD Professor NTUA	**Dionysios S. Ioakim** Civil Engineer, M.Sc., PhD CEO Ace-Hellas SA

Part A:
Static & Dynamic Analysis
of Frame, Plate and Disc Structures

A1. Introduction

In "Part A" single members, plane frames, three dimensional frames, plates, discs and shell structures are addressed employing elastic linear static and dynamic analysis. The analysis is based on the finite element method (FEM), using beam elements (according to Euler–Bernoulli theory and considering elastic stiffness for the concrete members) and/or shell elements (plate & disc action simultaneously) with six degrees of freedom at each node.

A series of test problems (examples) have been analysed to test the various features of the program and demonstrate its adequacy in several applications. For each example, a short description of the problem and the tested technical features of the program are presented in addition to the comparison of results obtained from ScadaPro software and from other computer programs.

To validate and verify ScadaPro, the test problems were run on a PC platform that was a HP machine with Intel Core i5 Processor and 6 GB of RAM operating on a Windows 7 operating system.

Concerning acceptance criteria, the comparison of ScadaPro results with results obtained from other computer programs is represented by the percentage difference evaluated by the

formula $Difference\left(\%\right) = 100\left(\dfrac{Scada\,\text{Pro Results} - \text{Other Software Result}}{\text{Other Software Result}}\right)$. This difference is

typically characterized in one of the following three ways:

- **Exact:** There is no difference between ScadaPro results and results obtained from other computer programs and bibliography.

- **Acceptable:** Concerning displacements, base reactions and dynamic characteristics of structures, the difference between ScadaPro results and results obtained from other computer programs and bibliography does not exceed five percent (5%). For internal stress values and resultant forces/moments the difference does not exceed ten percent (10%). Concerning comparison with experimental results, the difference between ScadaPro and other programs and bibliography does not exceed twenty five percent (25%).

- **Unacceptable:** Concerning displacement, reactions and dynamic characteristics of structures, the difference between ScadaPro results and results obtained from other computer programs and bibliography exceeds five percent (5%). For internal stress values and resultant forces/moments, the difference exceeds ten percent (10%).

Concerning comparison with experimental results, the difference between ScadaPro and other programs and bibliography exceeds twenty five percent (25%).

CONCLUSION

ScadaPro is the latest release of the ScadaPro series of computer programs. Since development, ScadaPro has been used widely for structural analysis. Additionally, the verification process conducted as described in this document demonstrates that the program features tested are operating reliably and with accuracy consistent with current computer technology capabilities.

A2. Static / Dynamic Analysis of Frame Structures

A2.1. Steel beam under torsional loading

Problem Description

A cantilever beam, subjected to concentrated torsional moment at its free end is examined (Figure A2.1.1).

Geometry, Properties and Loading

The cantilever beam of length L=4.0m has a cross section of IPE-100. Modulus of elasticity of the material is 210GPa and Poisson ratio is 0.30. The applied torsional moment at the beam's free end is Mt=10kNm.
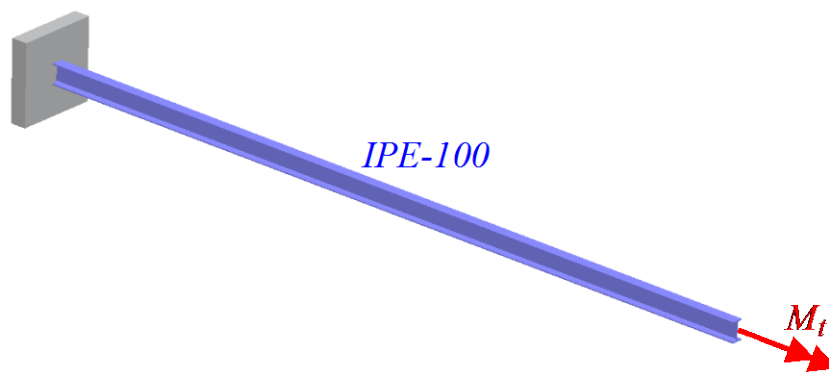
Figure A2.1.1. Steel beam under torsional loading.

Assumptions

Section properties are automatically recovered from cross section library of ScadaPro and are presented in Table A2.1.1. Concerning analysis with Sap2000 and Abaqus software, section properties of ScadaPro library are used for comparison reasons.
For the analysis, Saint Venant theory is employed.

Table A2.1.1. Cross section properties of IPE-100.

$A = 1.032x10^{-3}\ m^2$	$I_y = 1.710x10^{-6}\ m^4$
$I_x = 1.202x10^{-8}\ m^4$	$I_z = 1.592x10^{-7}\ m^4$

Technical Features in ScadaPro Tested

➢ One – dimensional analysis of beam elements

7

➢ Beam under torsional loading

➢ Fixed boundary conditions

Results Comparison

Theoretical results for angle of twist (*Pilkey, W.D., Analysis and Design of Elastic Beams: Computational Methods, Wiley, New York, 2002*) are compared with the ones obtained from ScadaPro analysis in Table A2.1.2. In the same table, results from Sap2000 and Abaqus are also shown.

Table A2.1.2. Comparison of Results for Angle of Twist.

Angle of Twist (rad) at	Theoretical	ScadaPro	Sap2000	Abaqus
x=L	41.20	41.20	41.20	41.20
x=L/2	20.60	20.60	20.60	20.60

Computer Files

The project file for this example is "FF1". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison shows an exact match between ScadaPro software, theoretical values and results obtained from Abaqus and Sap2000.

A2.2. Plane Steel Frame

Problem Description

A one-story plane frame subjected to vertical or horizontal loading is examined (Figure A2.2.1).

Geometry, Properties and Loads

The span opening is 7.0m and height of vertical members (columns) is 5.0m. The cross sections of columns and horizontal members are IPE-450 and IPE-330, respectively. Modulus of elasticity of the material is 210GPa and Poisson ratio is 0.30. Footing's dimensions are 150cmx150cmx50cm for each column and ground stiffness is 100kPa/cm. Two cases of loading are studied. According to the first one, self-weight is taken into account (ε_s=78.5kN/m^3) and horizontal member is subjected to distributed vertical load of 10kN/m (Load Case 1). In the second case, self weight is ignored and column AB is subjected to a distributed horizontal load of 5kN/m (Load Case 2).
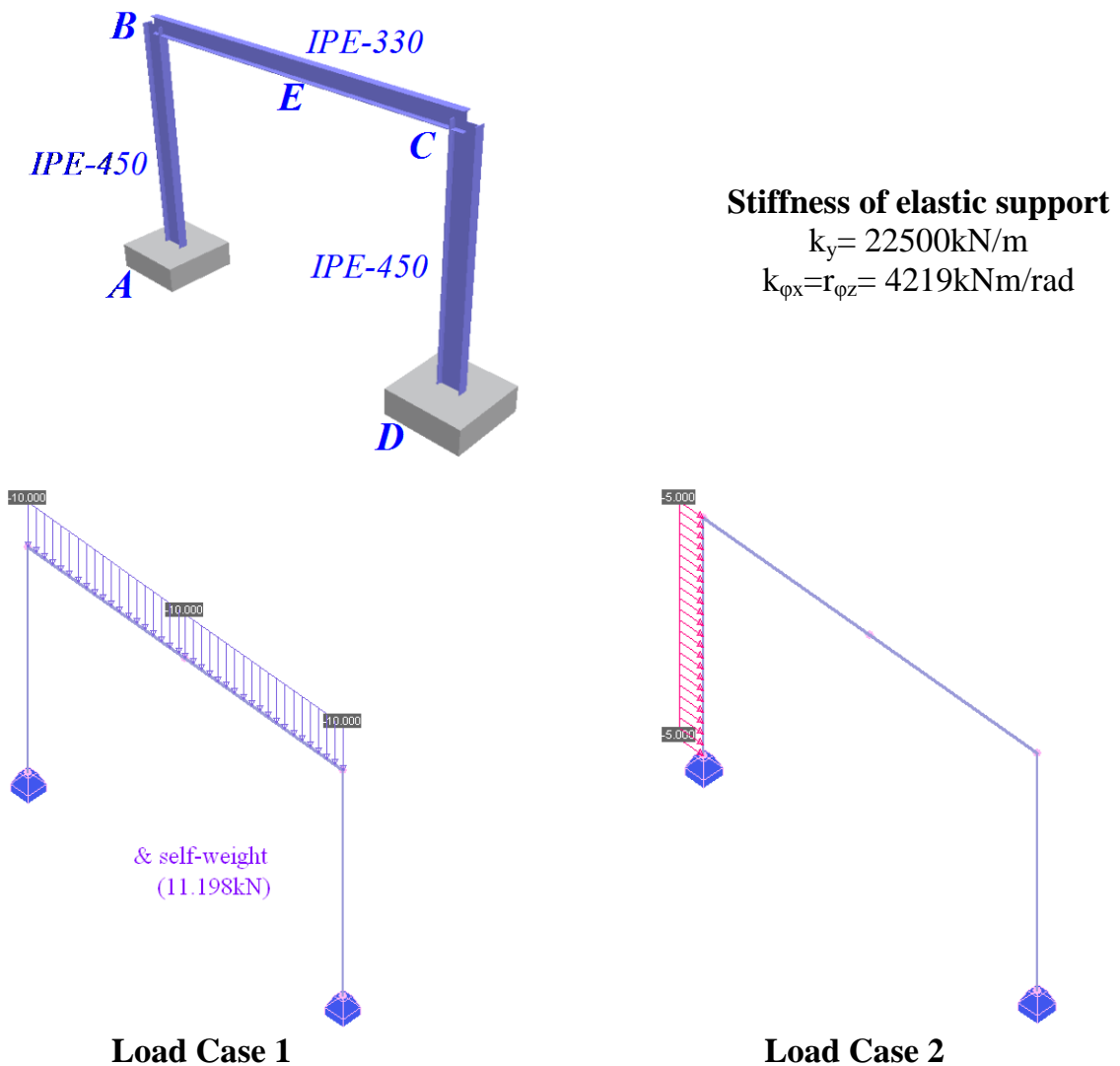
Stiffness of elastic support
k_y= 22500kN/m
$k_{\varphi x}$=$r_{\varphi z}$= 4219kNm/rad

Load Case 1   **Load Case 2**

Figure A2.2.1. Plane Steel Frame subjected to vertical or horizontal loading.

Assumptions

During analysis, shear deformations are neglected. Section properties are automatically recovered from cross section library of ScadaPro and are presented in Tables A2.2.1 and A2.2.2.

Table A2.2.1. Section Properties of IPE330.

$A = 6.261x10^{-3}\ m^2$	$I_y = 1.177x10^{-4}\ m^4$
$I_t = 2.815x10^{-7}\ m^4$	$I_z = 7.881x10^{-6}\ m^4$

Table A2.2.2. Section Properties of IPE450.

$A = 9.882x10^{-3}\ m^2$	$I_y = 3.374x10^{-4}\ m^4$
$I_x = 6.687x10^{-7}\ m^4$	$I_z = 1.676x10^{-5}\ m^4$

Technical Features in ScadaPro Tested

➢ Two – dimensional analysis of beam elements

➢ Beam under vertical loading

➢ Flexible boundary conditions

Results Comparison

Maximum values of vertical, horizontal displacements and bending moments at different positions for each load case obtained from ScadaPro analysis are presented in Table A2.2.3, A2.2.4 and Table A2.2.5, A2.2.6, respectively and compared with those obtained from Sap2000 and Abaqus software. Moreover, in Tables 2.9 and 2.10, base reactions are presented for each load case.

Table A2.2.3. Comparison of Results for Vertical Displacements (Load Case 1).

Vertical Displacement (mm) at	ScadaPro	Sap2000	Abaqus
middle of horizontal beam (node E)	6.04	6.04	6.10

Table A2.2.4. Comparison of Results for Horizontal Displacements (Load Case 2).

Horizontal Displacement (mm) at	ScadaPro	Sap2000	Abaqus
top of column (node B)	8.90	8.90	8.95

Table A2.2.5. Comparison of Results for Bending Moments (Load Case 1).

Bending Moment (kNm) at	ScadaPro	Sap2000	Abaqus
middle of horizontal beam (node E)	27.43	27.43	27.47
ends of horizontal beam (nodes B,C)	36.83	36.83	36.76

Table A2.2.6. Comparison of Results for Bending Moments (Load Case 2).

Bending Moment (kNm) at	ScadaPro	Sap2000	Abaqus
middle of horizontal beam (node E)	1.04	1.04	1.04
ends of horizontal beam (nodes B,C)	21.66/23.75	21.66/23.75	21.63/23.71

Table A2.2.7. Comparison of Base Reactions (Load Case 1).

Reactions at	ScadaPro	Sap2000	Abaqus
Vertical/Horizontal Force (node A) (kN)	40.60/7.70	40.60/7.70	40.60/7.68
Vertical/Horizontal Force (node D) (kN)	40.60/7.70	40.60/7.70	40.60/7.68
Reaction Moment (node A) (kNm)	1.65	1.65	1.62
Reaction Moment (node D) (kNm)	1.65	1.65	1.62

Table A2.2.8. Comparison of Base Reactions (Load Case 2).

Reactions at	ScadaPro	Sap2000	Abaqus
Vertical/Horizontal Force (node A) (kN)	6.49/18.68	6.49/18.68	6.48/18.68
Vertical/Horizontal Force (node D) (kN)	6.49/6.33	6.49/6.32	6.48/6.32
Reaction Moment (node A) (kNm)	9.22	9.22	9.25
Reaction Moment (node D) (kNm)	7.88	7.88	7.91

11

Computer Files

The project file for this example is "FF2". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Sap2000 and Abaqus.

A2.3. Three Dimensional Steel Frame Structure

Problem Description

A three dimensional steel frame subjected to vertical loading is examined (Figure A2.3.1).

Geometry, Properties and Loads

The cross section of columns is IPE-450 and of inclined and horizontal members is IPE-330. Footing's dimensions are 150cmx150cmx50cm for each vertical member (column) and ground stiffness is 100kPa/cm. Self-weight is taken into account (ε_s=78.5kN/m^3) and inclined members are subjected to distributed vertical load of 10kN/m. Modulus of elasticity of the material is 210GPa and Poisson ratio is 0.30.

Figure A2.3.1. Three Dimensional Steel Frame Structure.
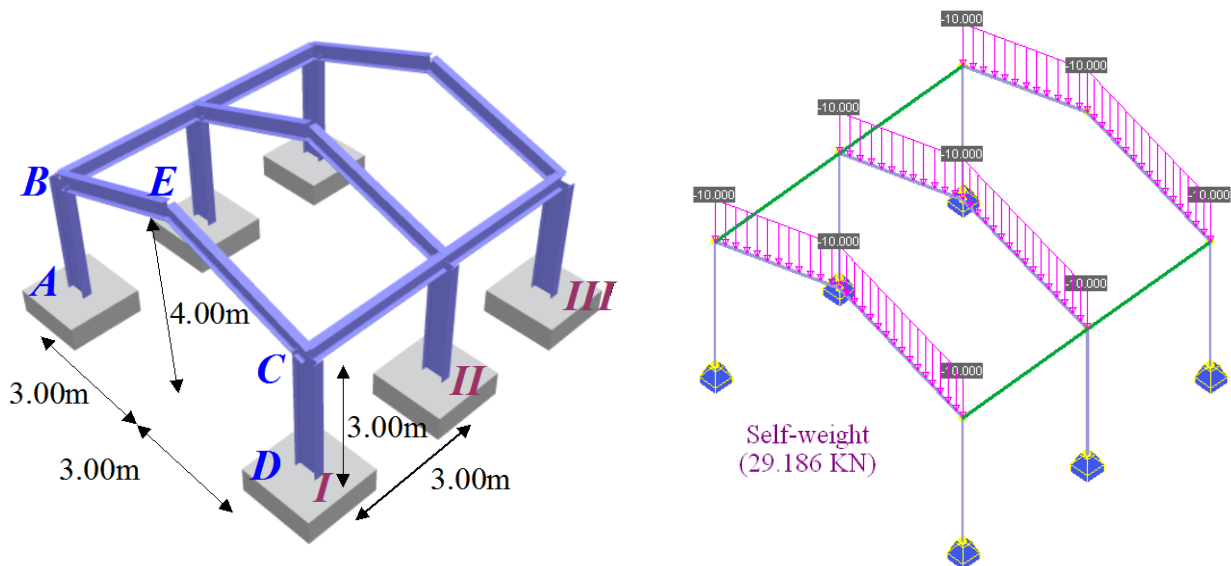
Assumptions

During analysis, shear deformations are neglected. Section properties are automatically recovered from the cross section library of ScadaPro and have been presented in previous example.

Technical Features in ScadaPro Tested

- ➢ Three – dimensional analysis of beam elements
- ➢ Inclined beams under vertical loading
- ➢ Flexible boundary conditions

13

Results Comparison

Vertical Displacements and Bending Moments along principal axis of cross section, for each load case, obtained from ScadaPro analysis, are presented in Table A2.3.1 and A2.3.2 respectively as compared with those obtained from Sap2000 and Abaqus software.

Table A2.3.1. Comparison of Results for Vertical Displacements of Frame I & III.

Vertical Displacement (mm) at	ScadaPro	Sap2000	Abaqus
highest point of inclined beam (node E)	3.27	3.28	3.34

Table A2.3.2. Comparison of Results for Bending Moments of Frame I & III.

Bending Moment (kNm) at	ScadaPro	Sap2000	Abaqus
highest point of inlined beams (node E)	13.46	13.46	13.56
ends of inclined beams (nodes B, C)	26.88	26.88	26.77
base of columns (nodes A, D)	1.42	1.42	1.40

Computer Files

The project file for this example is "FF3". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Sap2000 and Abaqus.

A2.4. Three Dimensional Composite (Steel & Concrete) Structure

Problem Description

A three dimensional composite structure forming an octagon shape and subjected to vertical loading is examined (Figure A2.4.1).

Geometry, Properties and Loads

The cross section of internal horizontal steel members is IPE-500 (Group A, B) and the dimensions of horizontal concrete beams' rectangular cross-section are 30cmx60cm. Every column (Group C) has a Concrete Circular Cross Section of 60cm diameter with ends fixed within ground.

All steel beams of Group A forming a square shape are subjected to vertical uniformly distributed load of 10kN/m. Modulus of elasticity and Poisson ratio of the material for steel and concrete members are 210GPa, 0.30 and 29GPa, ν=0.20, respectively.

Assumptions

During analysis, shear deformations are neglected. Section properties are recovered from the cross section library of ScadaPro and are presented in Tables A2.4.1, A2.4.2 and A2.4.3.

Table A2.4.1. Section Properties of Concrete Rectangular Cross Section bxh=30cmx60cm.

$A = 0.18\,m^2$	$I_y = 0.54x10^{-2}\,m^4$
$I_x = 0.37x10^{-2}\,m^4$	$I_z = 0.135x10^{-2}\,m^4$

Table A2.4.2. Section Properties of Concrete Circular Cross-Section d=60cm.

$A = 0.2827\,m^2$	$I_y = 0.6362x10^{-2}\,m^4$
$I_x = 0.01272\,m^4$	$I_z = 0.6362x10^{-2}\,m^4$

Table A2.4.3. Section Properties of IPE-500 Steel Cross-Section.

$A = 0.01155\,m^2$	$I_y = 0.482x10^{-3}\,m^4$
$I_x = 0.8929 \times 10^{-6}\,m^4$	$I_z = 0.2142x10^{-4}\,m^4$

Group A

Group B

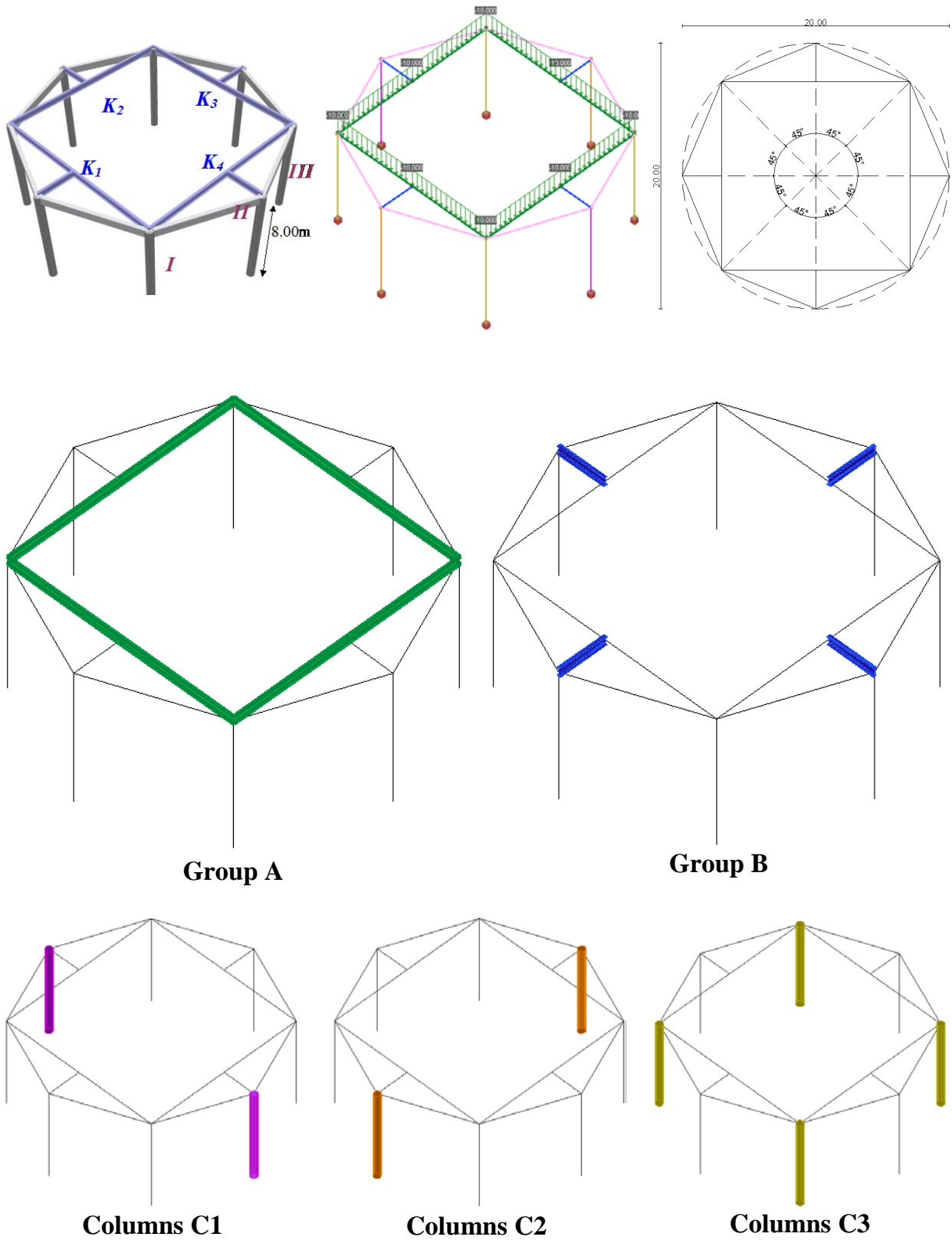**Columns C1**

Columns C2

Columns C3

Figure A2.4.1. Three-Dimensional Composite (Steel & Concrete) Structure.

Technical Features in ScadaPro Tested

➢ Three – dimensional analysis of beam elements

16

> ➤ Horizontal members under vertical loading

> ➤ Fixed boundary conditions

> ➤ Two different materials

Results Comparison

Vertical displacements and bending moments along horizontal principal axis obtained from ScadaPro analysis are presented in Table A2.4.4 and Table A2.4.5 respectively and compared with those obtained from Sap2000 and Abaqus software.

Table A2.4.4. Comparison of Results for Vertical Displacements.

Vertical Displacements (mm) at	ScadaPro	Sap2000	Abaqus
nodes K1, K2, K3, K4	7.02	7.02	7.16

Table A2.4.5. Comparison of Results for Bending Moments.

Bending Moment (kNm) at	ScadaPro	Sap2000	Abaqus
ends of steel beams forming square shape (Group A)	80.87/19.97	80.69/19.97	80.91/20.25
ends of steel beams outside square shape (Group B)	123.52/0.04	123.53//0.04	123.25/0.04
ends of columns (C1, C2, C3)	C1:108.12/53.18 C2:0.00/0.00 C3:72.30/36.16	C1:108.12/53.18 C2:0.00/0.00 C3:72.30/36.16	107.83/52.92 0.00/0.00 72.23/35.83

Computer Files

The project file for this example is "FF4". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between the ScadaPro software and results obtained from Sap2000 and Abaqus.

A2.5. Three Dimensional Concrete Structure

Problem Description

A dynamic analysis of a three dimensional concrete structure is presented in this example (Figure 2.5.1).

Geometry, Properties and Loads

The cross section of vertical beam members (columns) is square 50cmx50cm and of horizontal members is rectangular 30cmx60cm.The foundation of the structure is strip footing with dimensions 30cmx100cmx100cmx50cm. Modulus of elasticity and Poisson ratio of the material is 29GPa and v=0.20, respectively, while the ground stiffness is 300kPa/cm. Two cases of dynamic analysis have been employed, taking into account the self-weight of structure (ε_c=25.0kN/m^3). According to the first one, no diaphragms are considered in any floor while on the second case diaphragms in every floor are employed.
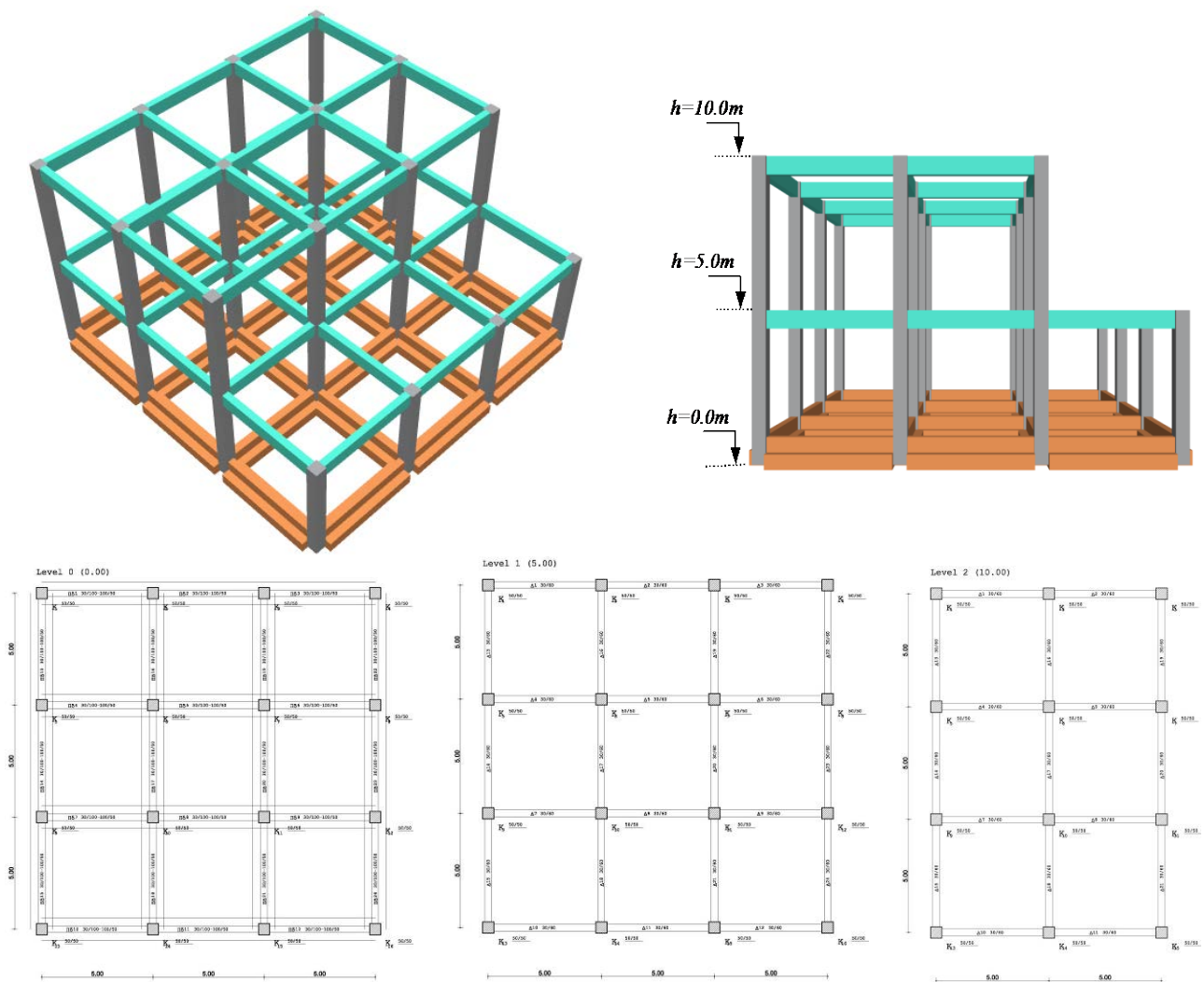

Figure 2.5.1. Three Dimensional Concrete Structure.

Assumptions

During analysis, shear deformations are neglected. Section properties are recovered from the cross section library of ScadaPro and are presented in Tables 2.5.1, 2.5.2 and 2.5.3.

Table 2.5.1. Section Properties of Concrete Square Cross Section bxh=50cmx50cm.

$A = 0.25\,m^2$	$I_y = 0.5208x10^{-2}\,m^4$
$I_x = 0.8787x10^{-2}\,m^4$	$I_z = 0.5208x10^{-2}\,m^4$

Table 2.5.2. Section Properties of Rectangular Cross-Section bxh=30cmx60cm.

$A = 0.18\,m^2$	$I_y = 0.5400x10^{-2}\,m^4$
$I_x = 0.3708x10^{-2}\,m^4$	$I_z = 0.1350x10^{-2}\,m^4$

Table 2.5.3. Section Properties of Strip Footing (30cmx100cmx100cmx50cm).

$A = 0.65\,m^2$	$I_y = 0.04279\,m^4$
$I_x = 0.03143\,m^4$	$I_z = 0.04239\,m^4$

Technical Features in ScadaPro Tested

➢ Three – dimensional dynamic analysis of beam elements

➢ Rigid Length Offsets

➢ Eigenmodes – Eigenvalues

➢ Elastic Support

➢ Rigid diaphragms

Results Comparison

Modal Periods and Frequencies are compared in the following Table 2.5.4 and Figures 2.5.2- 2.5.5.

Table 2.5.4. Comparison of Results for Vertical Displacements.

Eigenperiods (sec)	ScadaPro		Sap2000		Etabs	
	without diaphragm	with diaphragm	without diaphragm	with diaphragm	without diaphragm	with diaphragm
1st Mode	0.244	0.238	0.248	0.243	0.240	0.235
2st Mode	0.230	0.230	0.234	0.234.	0.226	0.225
3st Mode	0.204	0.189	0.209	0.199	0.203	0.189
4st Mode	0.164	0.092	0.167	0.095	0.161	0.092

Figure 2.5.2. 1st Eigenmode without and with diaphragms according to ScadaPro.

Figure 2.5.3. 2nd Eigenmode without and with diaphragms according to ScadaPro.
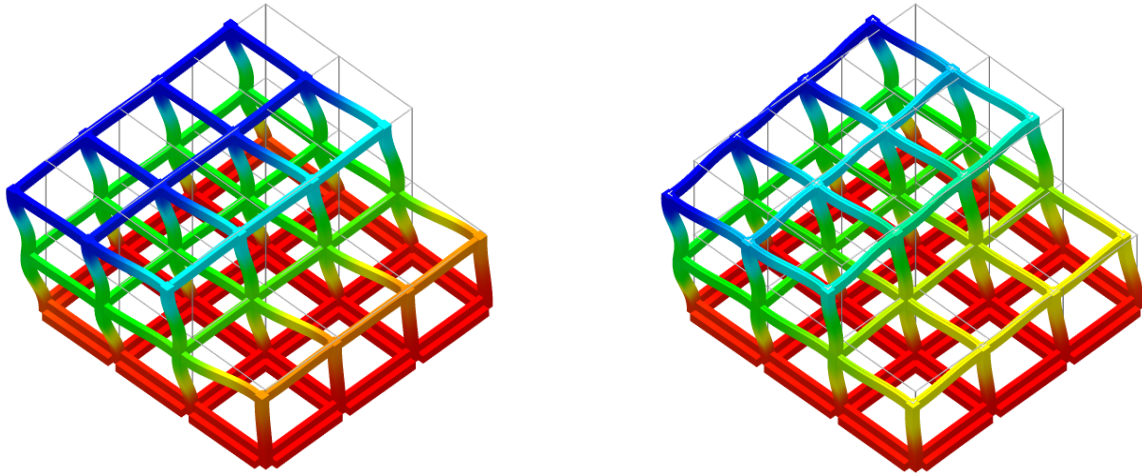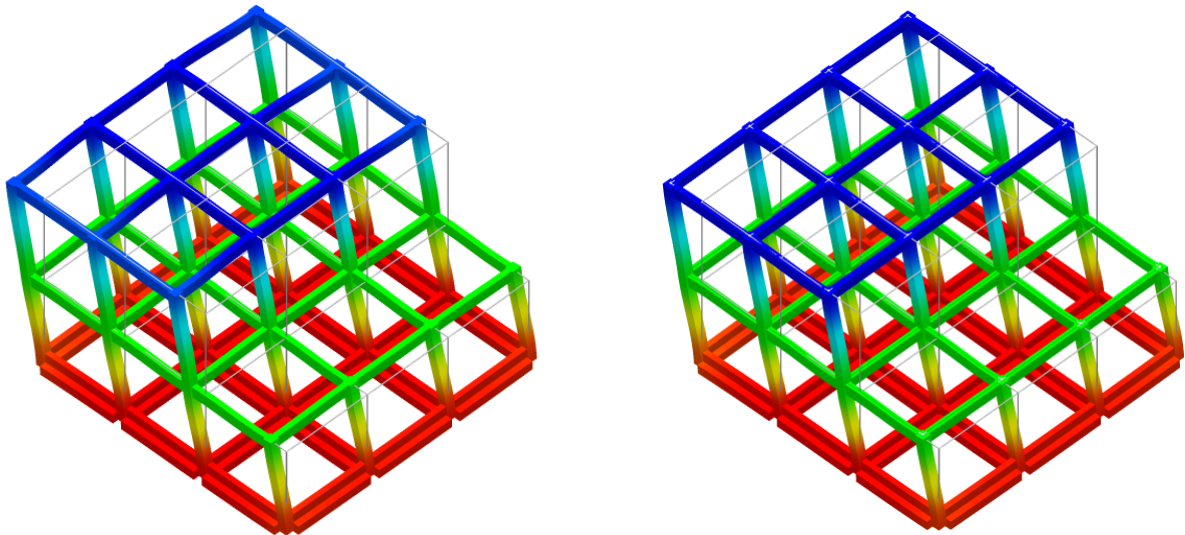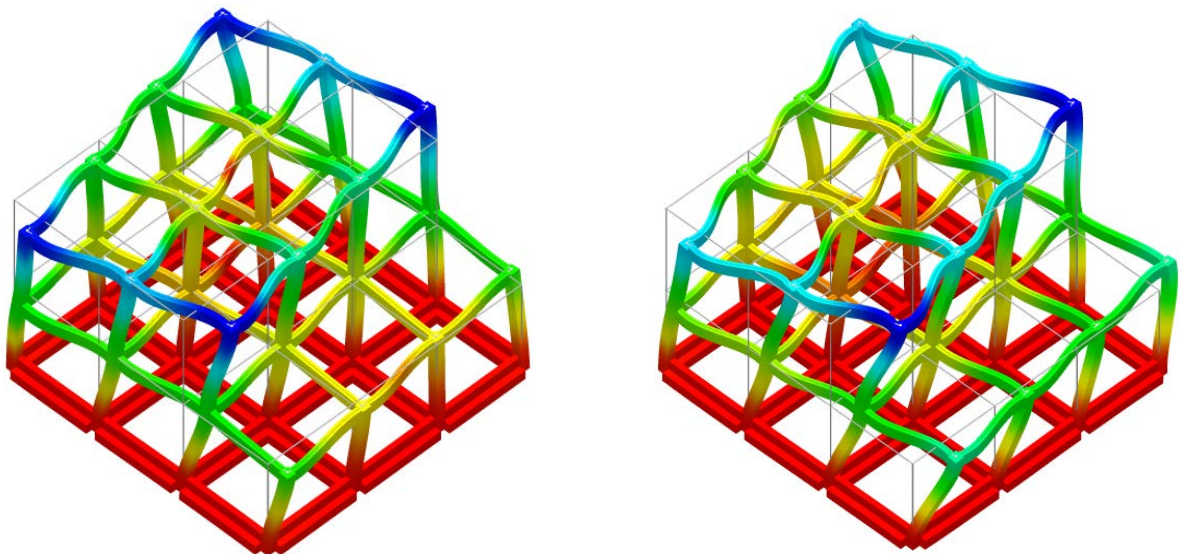

Figure 2.5.4. 3rd Eigenmode without and with diaphragms according to ScadaPro.

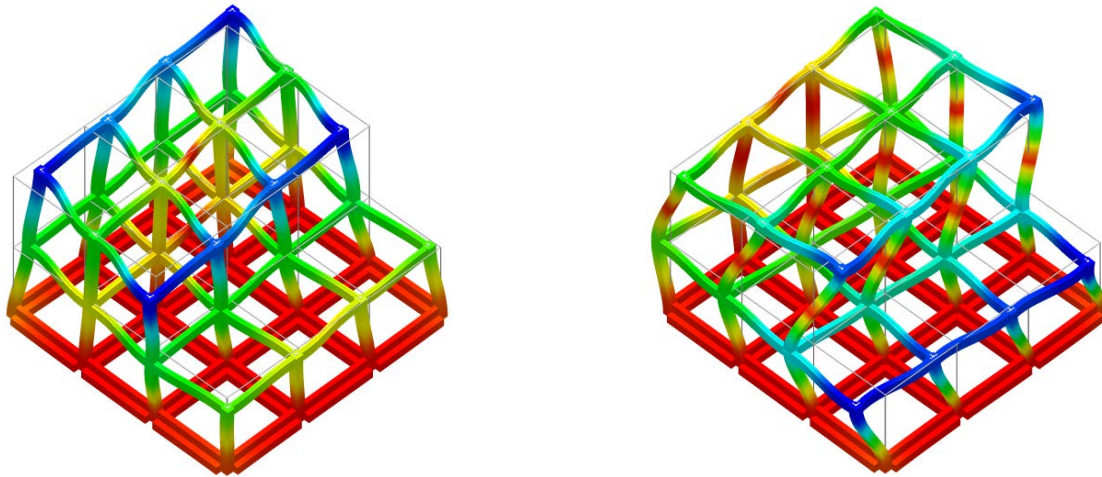*Figure 2.5.5. 4th Eigenmode without and with diaphragms according to ScadaPro.*

Computer Files

The project files for this example are "FF5_withD" & "FF5_withoutD". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Sap2000 and Etabs.

A3. Static Analysis of Plate Structures under Transverse Loading

A3.1. Rectangular Plate

Problem Description

A two dimensional square plate, subjected to vertical loading is examined (Figure A3.1.1).

Geometry, Properties and Loads

The plate of dimensions 5.00m x 5.00m has 20cm thickness. Material is concrete with modulus of Elasticity E=29 GPa and Poisson Ratio v=0.20. Three cases of loading and boundary conditions are examined. According to the first one (Load Case 1), plate is subjected to its self-weight (ε_c=25.0kN/m^3) plus 2kN/m^2 uniformly distributed load and its boundary is fixed. In the second case (Load Case 2), plate is subjected to its self-weight (ε_c=25.0kN/m^3) plus 2kN/m^2 uniformly distributed load and its boundary is simply supported. Finally, according to the third case (Load Case 3), plate is subjected to its self-weight (ε_c=25.0kN/m^3) plus a concentrated load of 100kN applied at the middle point of the plate, while all plate is elastically supported on ground with stiffness 5000kN/m^3.
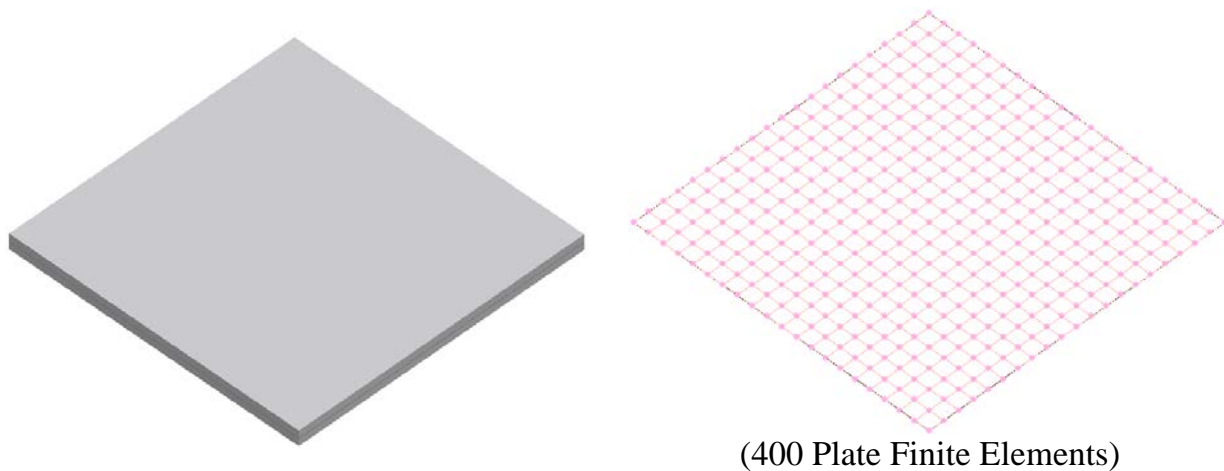

(400 Plate Finite Elements)

Figure A3.1.1. Rectangular Plate examined for three cases of boundary conditions and loading.

Assumptions

During analysis, Kirchhoff theory of plates has been adopted.

Technical Features in ScadaPro Tested

➢ Analysis of plate elements

➢ Plate elements under transverse loading

➢ Fixed and Simply supported boundary conditions

➢ Flexible boundary conditions

Results Comparison

Vertical displacements and bending, twisting moments obtained from ScadaPro analysis are presented in Tables A3.1.1 and Table A3.1.2 respectively and are compared with those obtained from analytical solution (*Timoshenko, S. and Woinowsky-Krieger, S., 1959. Theory of Plates and Shells, 2nd ed., McGraw- Hill Book Company, New York*), Sap2000 and Abaqus results. Moreover, in Figures A3.1.2, A3.1.3 contourlines of bending moment Mxx and twisting moment Mxy, respectively, for Load Case 1 (fixed boundary conditions) are presented.
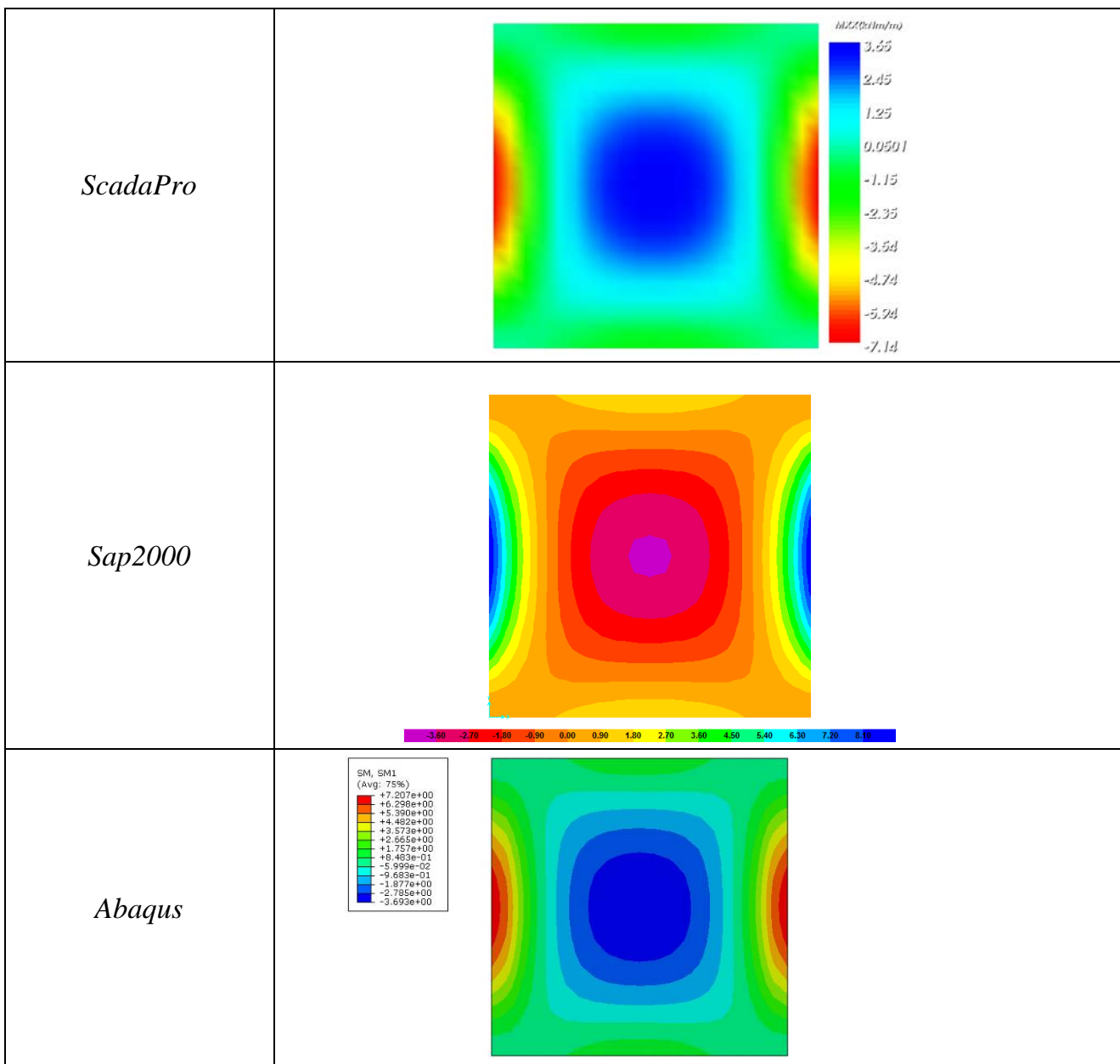
ScadaPro	
Sap2000	
Abaqus	

Figure A3.1.2. Bending Moment Mxx (kNm/m) for Load Case 1 according to ScadaPro, Sap2000 and Abaqus Analysis.

ScadaPro	
Sap2000	
Abaqus	
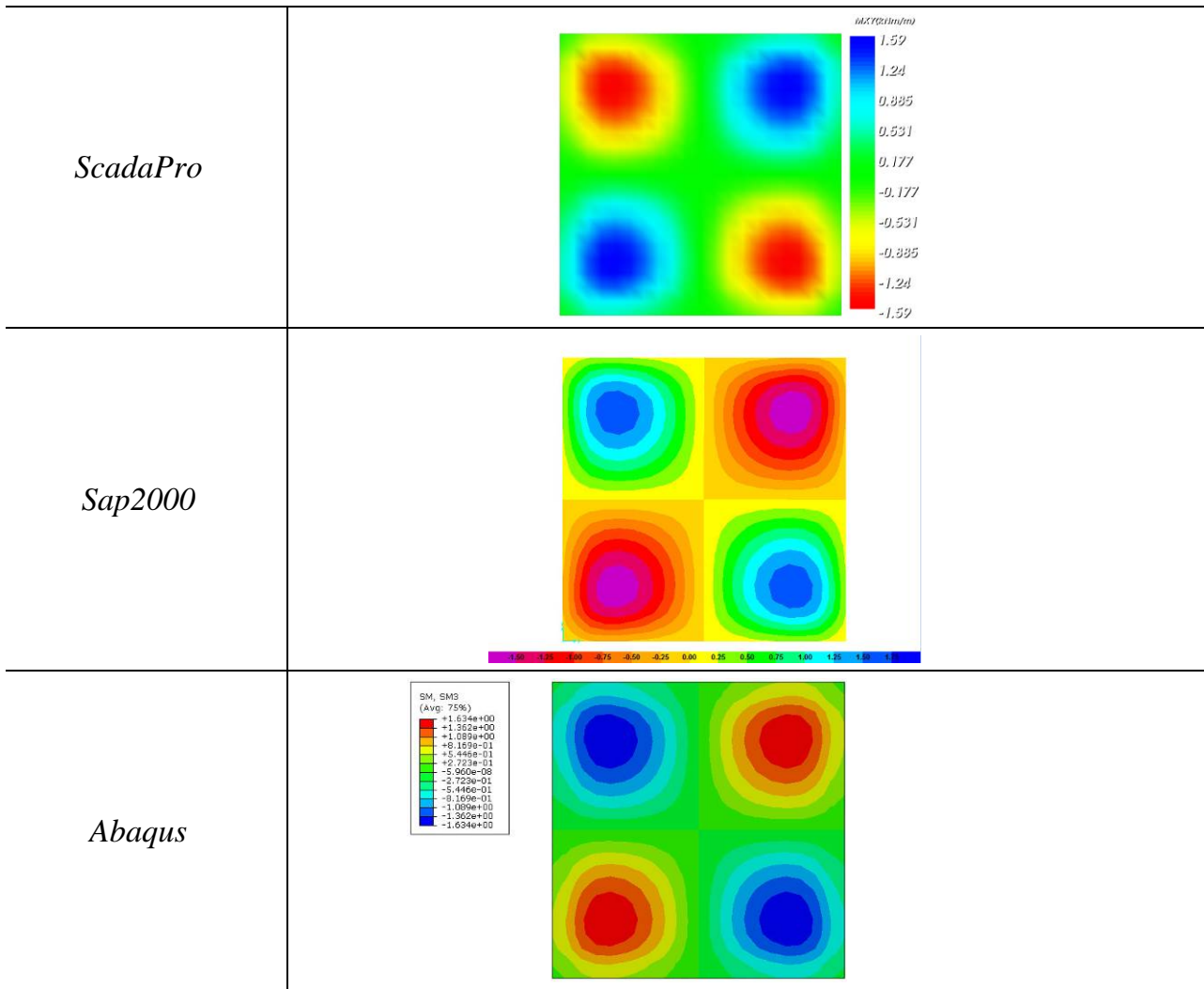
Figure A3.1.3. Twisting moment Mxy (kNm/m) for Load Case 1 according to ScadaPro, Sap2000 and Abaqus Analysis.

Table A3.1.1. Comparison of Results for Vertical Displacements.

Vertical Displacement (mm) at	ScadaPro	Sap2000	Abaqus	Exact Solution
center point of plate (Load Case 1)	0.274	0.277	0.282	0.274
center point of plate (Load Case 2)	0.878	0.882	0.919	0.882
center point of plate (Load Case 3)	2.495	2.500	2.570	-

Table A3.1.2. Comparison of Results for Bending Moments (Load Case 1).

Bending Moment	ScadaPro	Sap2000	Abaqus	Exact Solution
Mxx (kNm/m) at center of plate (=Myy)	3.65	3.67	3.69	4.04
max value of Mxy (kNm/m)	1.69	1.72	1.63	-

Computer Files

The project files for this example are "FF6_Fix", "FF6_SS" & "FF6_EF". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Sap2000 and Abaqus.

A3.2. Rectangular Plate with Hollows

Problem Description

A two dimensional plate, subjected to transverse loading is examined (Figure A3.2.1).

Geometry, Properties and Loads

The plate of polygon shape has 20cm thickness and its external boundary is fixed while the internal boundary is free. Material is concrete C20/25 (Modulus of Elasticity =29GPa and Poisson ratio 0.2). The plate is subjected to its self-weight (ε_c=25.0kN/m^3) plus 2kN/m^2 uniformly distributed load normal to its plane.
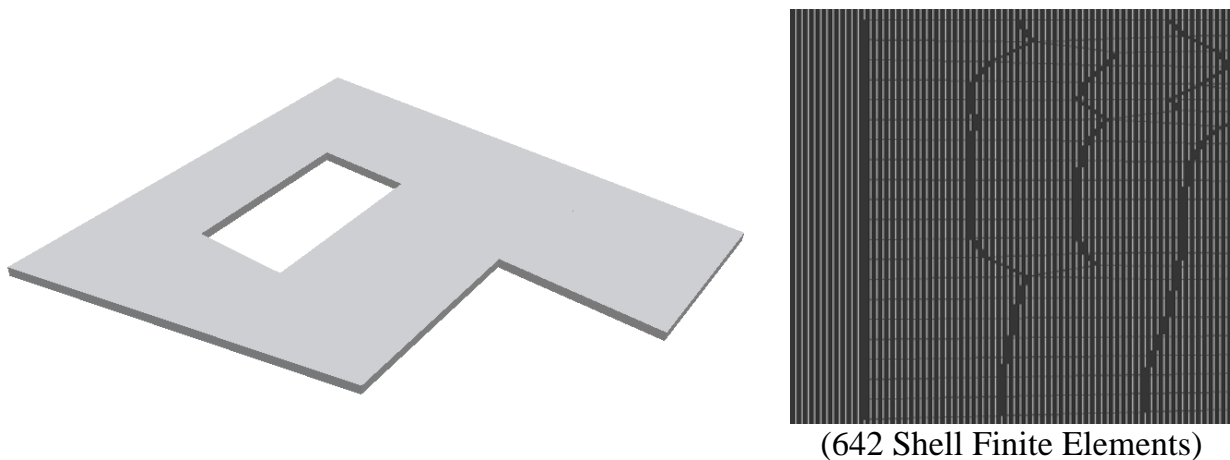

(642 Shell Finite Elements)

Figure A3.2.1. Rectangular plate with hollows.

Assumptions

During analysis, Kirchhoff theory of plates has been adopted.

Technical Features in ScadaPro Tested

- ➢ Analysis of plate elements under transverse loading
- ➢ Fixed boundary conditions

Results Comparison

Maximum values of vertical displacements obtained from ScadaPro analysis are presented in Table A3.2.1 respectively and compared with those obtained from Sap2000 and Abaqus software. Moreover, in Figures A3.2.2- A3.2.3 contourlines of bending moments Mxx and Myy are presented.

Table A3.2.1. Comparison of Results for Displacements.

Vertical Displacement (mm)	**ScadaPro**	**Sap2000**	**Abaqus**
max values	1.62	1.78	1.80

ScadaPro	
Sap2000	
Abaqus	
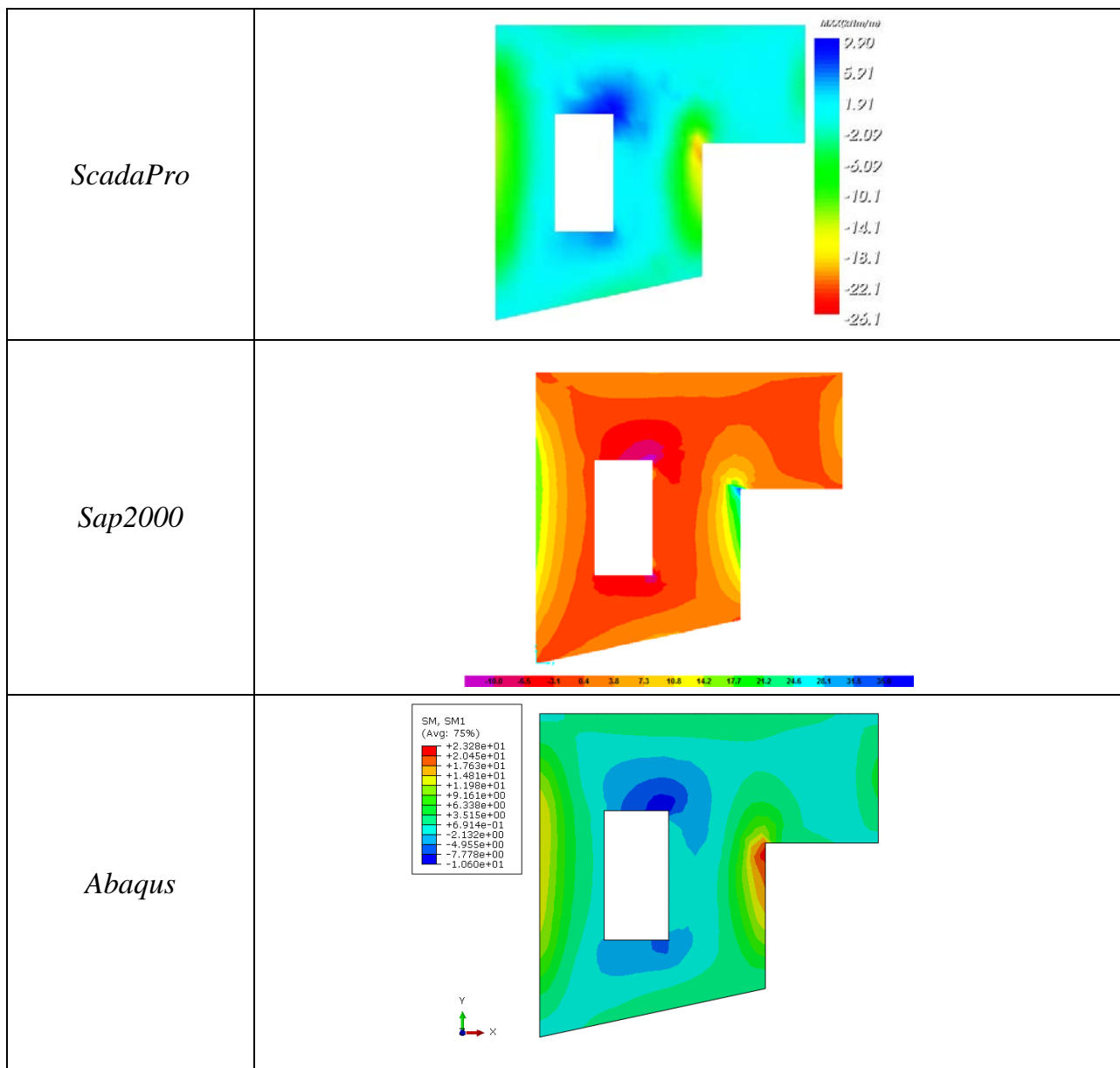
Figure A3.2.2. Bending Moment Mxx (kNm/m) according to ScadaPro, Sap2000 and Abaqus Analysis.

27

ScadaPro	
Sap2000	
Abaqus	

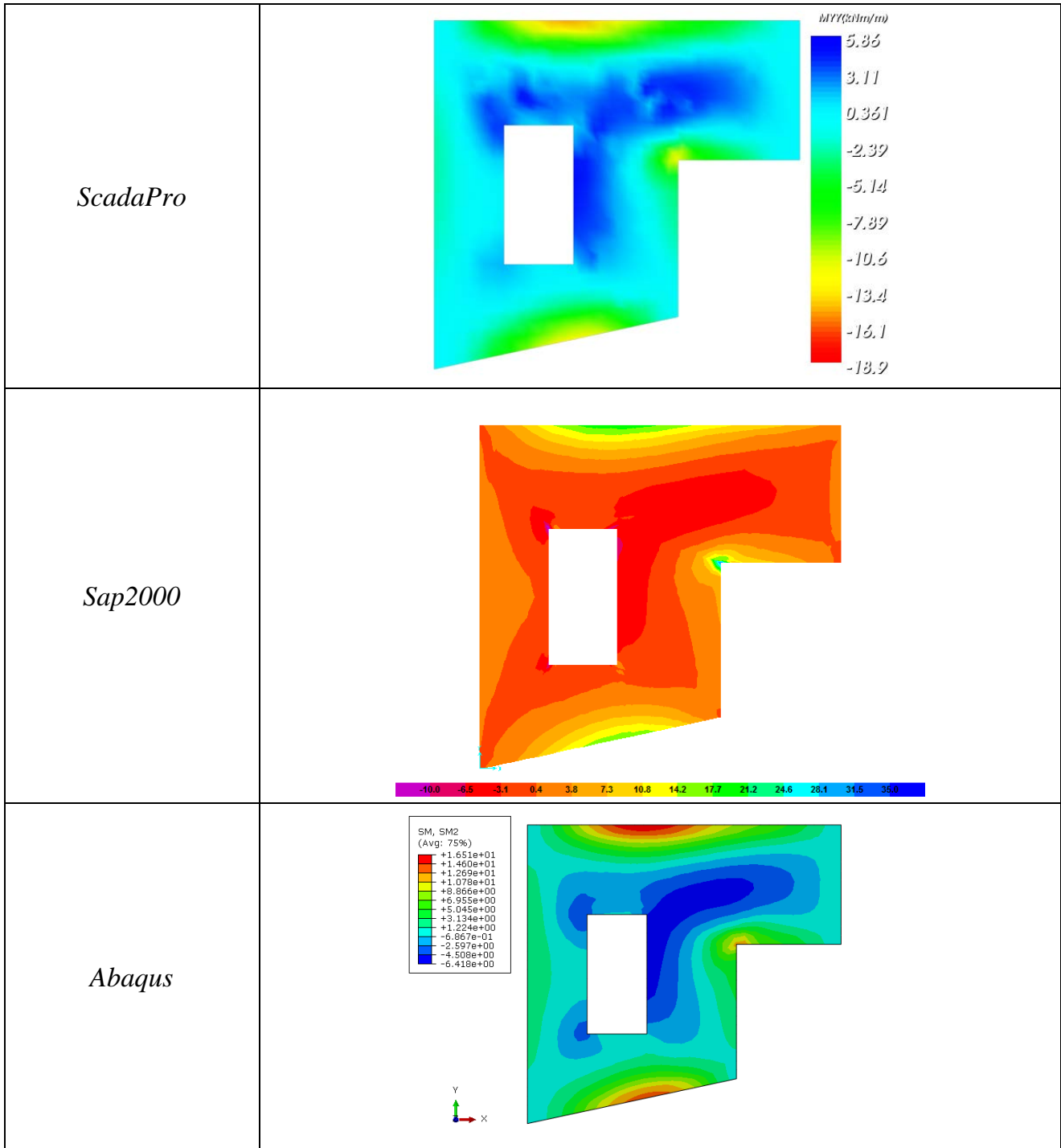*Figure A3.2.3   Bending Moment Myy (kNm/m) according to ScadaPro, Sap2000 and Abaqus Analysis.*

28

Computer Files

The project file for this example is "FF7". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Sap2000 and Abaqus.

A4. Static Analysis of Plate Structures under Inplane Loading (Discs)

A4.1. Wall

Problem Description

A two dimensional disk, subjected to coplanar loading is examined (Figure A4.1.1).

Geometry, Properties and Loads

The plate of dimension 10mx5m has 20cm thickness and is subjected to an inplane concentrated load of 50kN. Modulus of Elasticity is 29GPa and Poisson ratio ν=0.20. Plate is fixed along its smaller sides and free along the others.
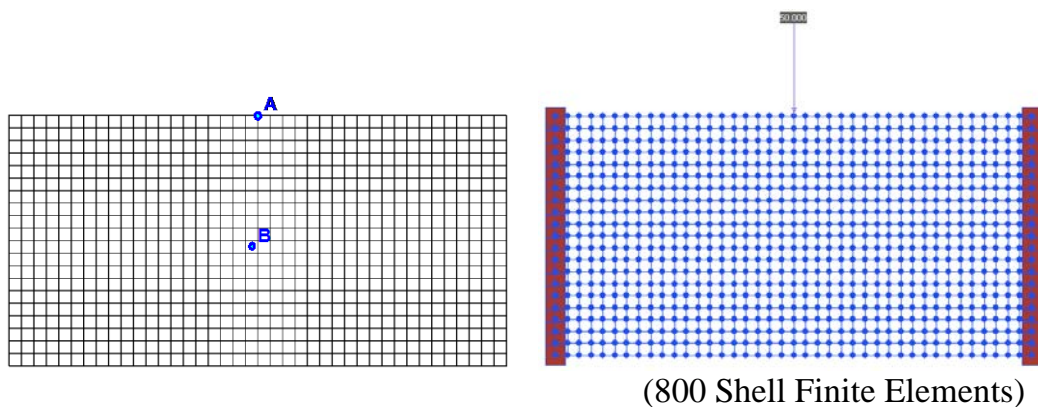

(800 Shell Finite Elements)

Figure A4.1.1. Disk under inplane loading.

Assumptions

During analysis, theory of discs (plates under inplane loading) has been adopted.

Technical Features in ScadaPro Tested

 ➢ Plate elements under inplane loading

 ➢ Fixed boundary conditions

Results Comparison

Maximum values of inplane displacements along load direction obtained from ScadaPro analysis and stresses at point B (Figure 4.1) are presented in Table A4.1.1 and A4.1.2 respectively and are compared with those obtained from Sap2000 and Abaqus software. Moreover, in Figures A4.1.2- A4.1.3 contourlines of stresses σxx and σyy are presented.

Table A4.1.1. Comparison of Results for Displacements.

	ScadaPro	**Sap2000**	**Abaqus**
maximum inplane displacements (mm)	0.0335	0.0347	0.0348

Table A4.1.2. Comparison of Stresses at point B.

	ScadaPro	**Sap2000**	**Abaqus**
σxx (kN/m^2)	10.38	10.35	10.37
σyy (kN/m^2)	40.97	41.08	40.93
σxy (kN/m^2)	4.12	4.35	4.09
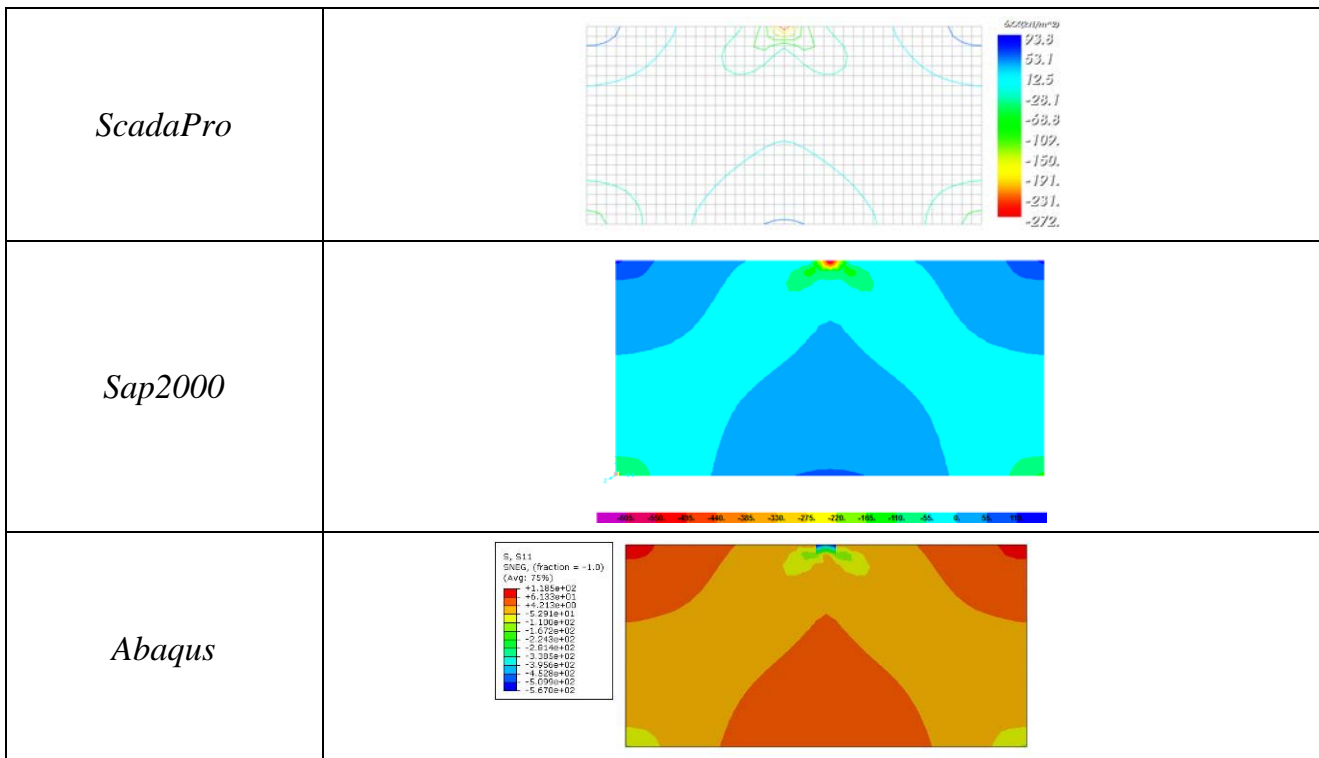
ScadaPro	
Sap2000	
Abaqus	

Figure A4.1.2. Stresses σxx (kN/m^2) according to ScadaPro, Sap2000 and Abaqus Analysis.

ScadaPro	
Sap2000	
Abaqus	

Figure A4.1.3. Stresses σyy (kN/m^2) according to ScadaPro, Sap2000 and Abaqus Analysis.

Computer Files

The project file for this example is "FF8". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Sap2000 and Abaqus.

A4.2. Wall with Hollows
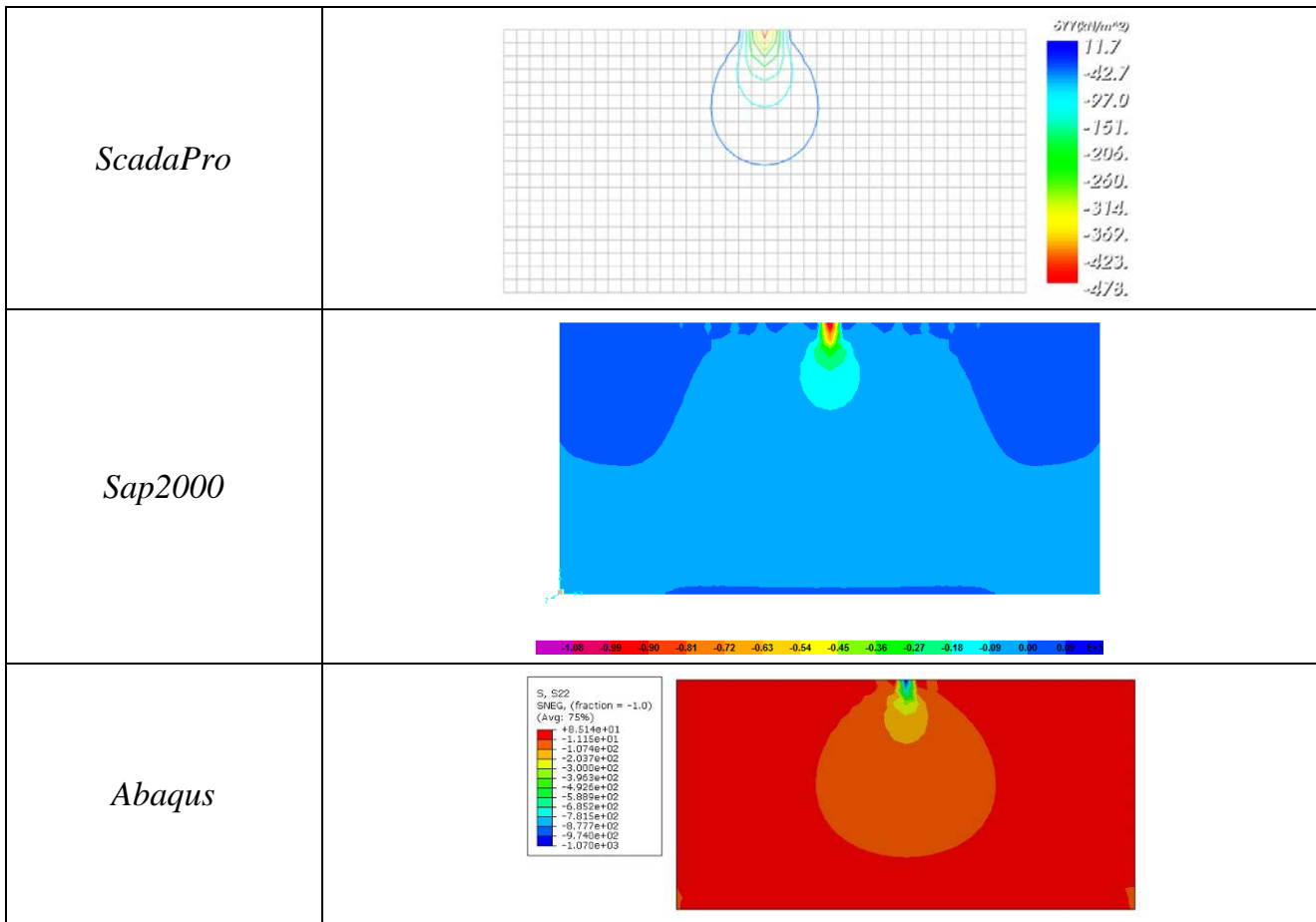
Problem Description

A two dimensional disk, subjected to inplane loading is examined (Figure A4.2.1).

Geometry, Properties and Loads

The plate of polygon shape has 25cm thickness and is subjected to a horizontal concentrated load of 50kN. Modulus of elasticity is 29GPa and Poisson Ratio is 0.20. Finally, the boundary at the base of the disk is fixed.
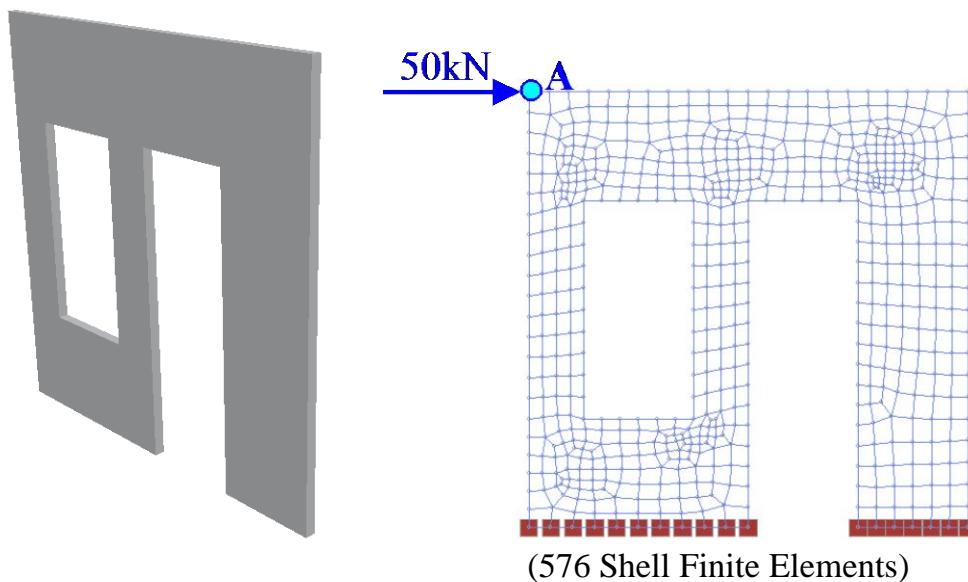

(576 Shell Finite Elements)

Figure A4.2.1. Wall with hollows.

Assumptions

During analysis, theory of discs (plates under inplane loading) has been adopted.

Technical Features in ScadaPro Tested

➢ Plate elements under inplane loading

➢ Fixed boundary conditions

Results Comparison

Horizontal displacements obtained from ScadaPro analysis are presented in Table A4.2.1 and compared with those obtained from Sap2000 and Abaqus software. Moreover, in Figures A4.2.2- A4.2.4 contourlines of stresses σxx, σyy and σxy are presented.

Table A4.2.1. Comparison of Results for Displacements.

	ScadaPro	**Sap2000**	**Abaqus**
Horizontal Displacement (mm) at node A	0.283	0.281	0.286
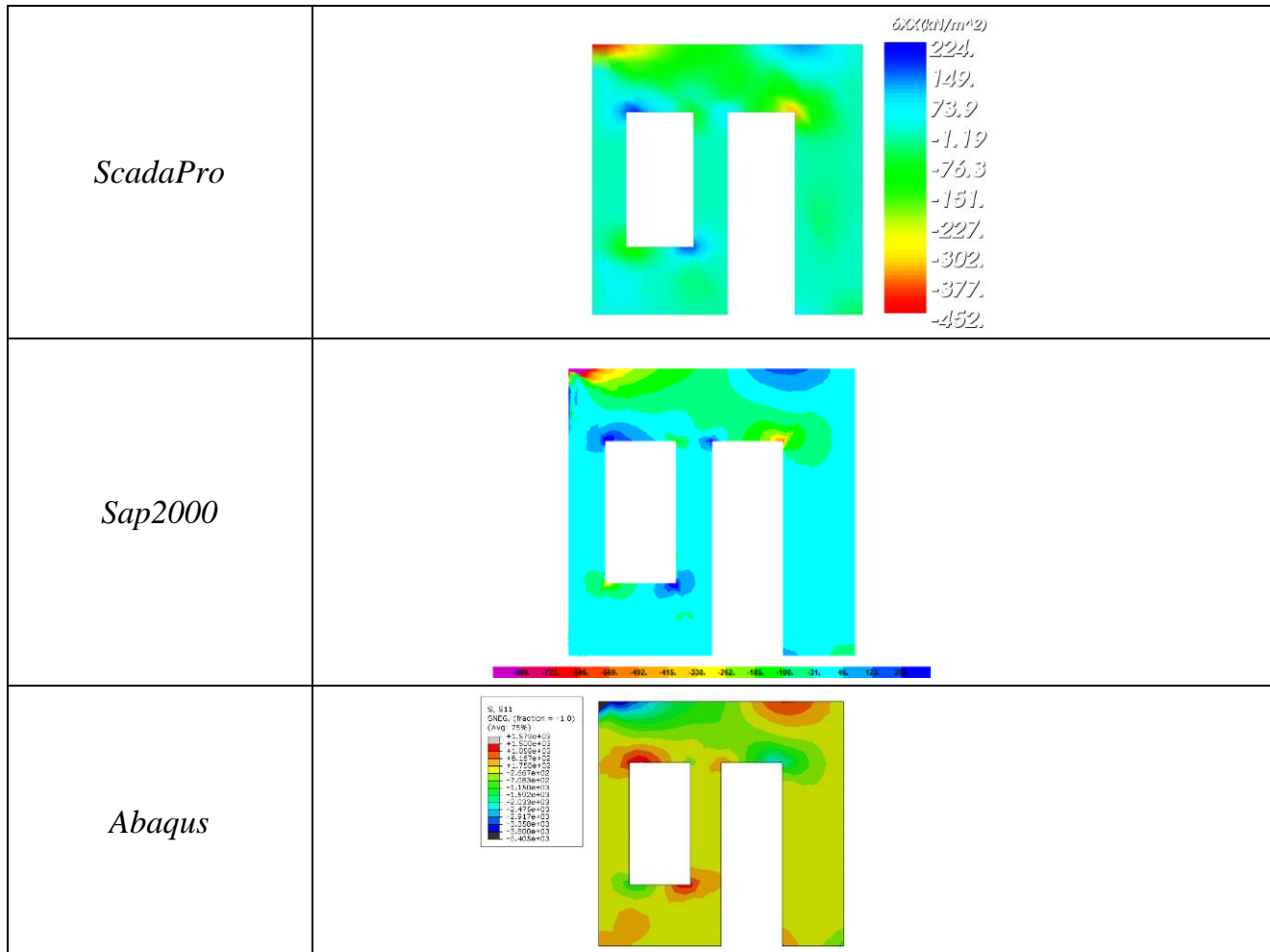
ScadaPro	
Sap2000	
Abaqus	

Figure A4.2.2. Stresses σxx (kN/m^2) according to ScadaPro, Sap2000 and Abaqus Analysis.

ScadaPro	
Sap2000	
Abaqus	

Figure A4.2.3. Stresses σyy (kN/m²) according to ScadaPro, Sap2000 and Abaqus Analysis.
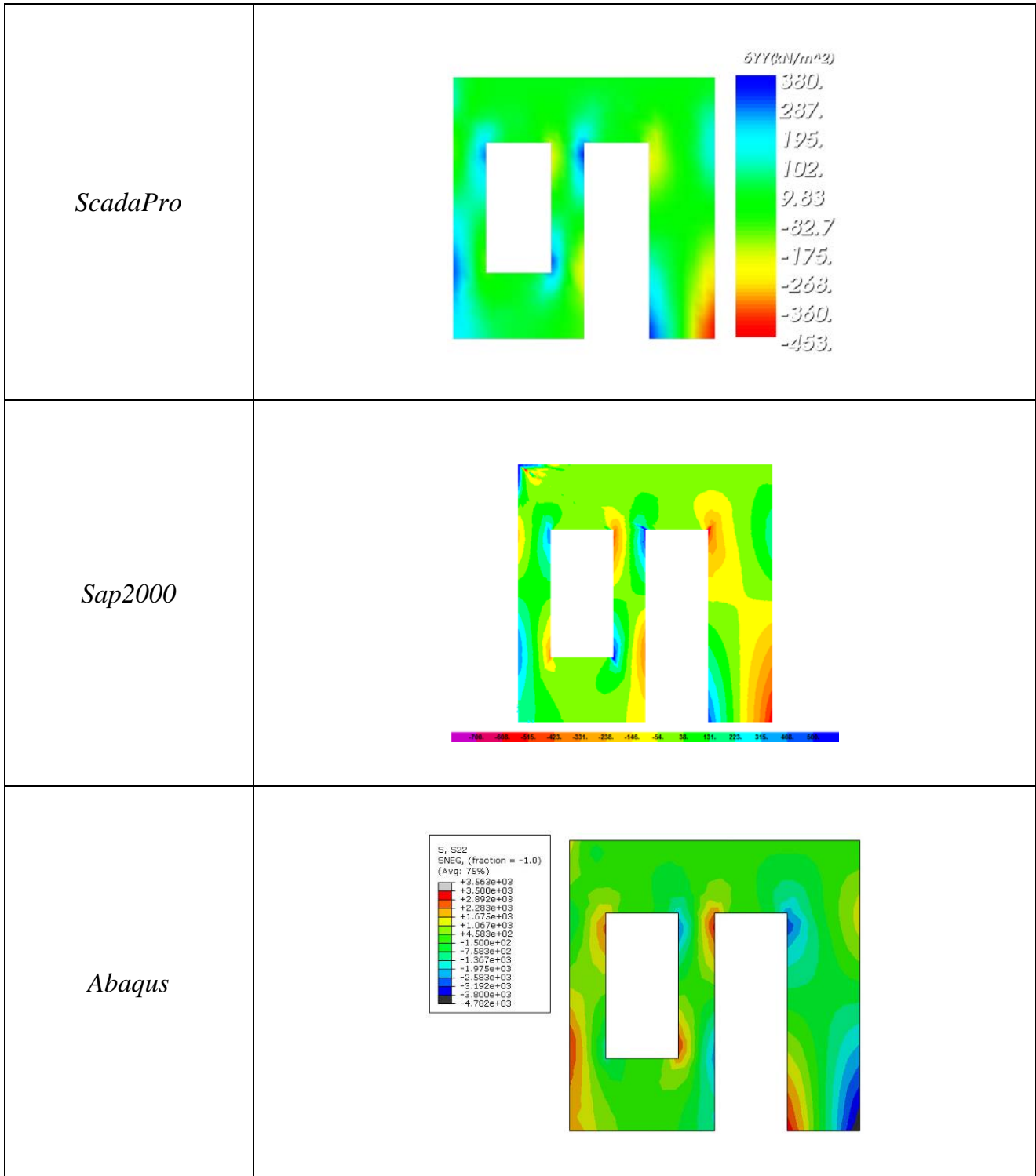
ScadaPro	
Sap2000	
Abaqus	

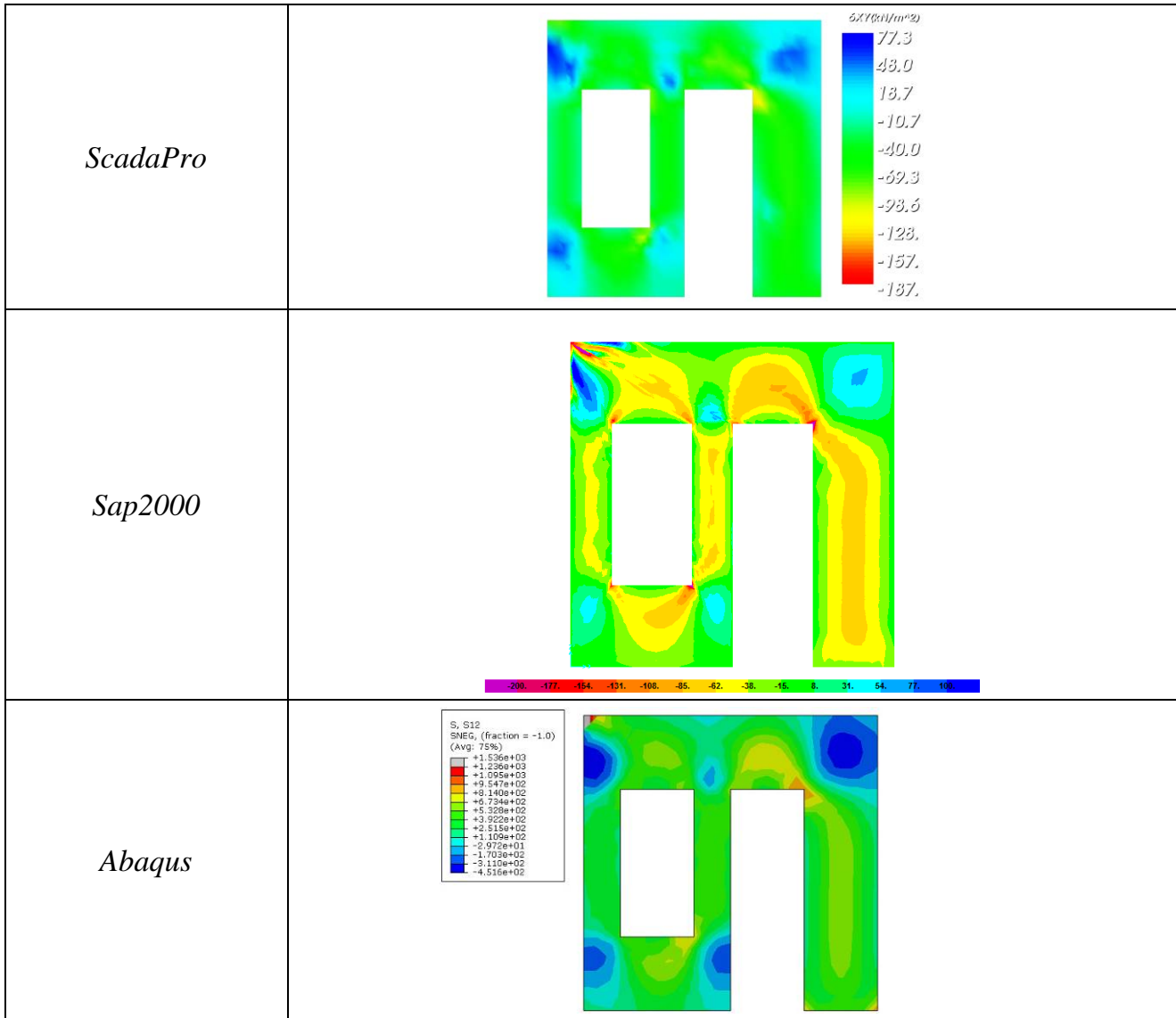*Figure A4.2.4. Stresses σxy (kN/m^2) according to ScadaPro, Sap2000 and Abaqus Analysis.*

Computer Files

The project file for this example is "FF9". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Sap2000 and Abaqus.

A5. References

[1].   *ABAQUS/CAE 6.11: User's Manual, Online Documentation Help*, Dassault Systèmes USA (2011).

[2].   Argyris, J. and Mlejnek, H.P., *Die Methode der Finiten Elemente*, Vol. I, II, III, Vieweg,Braunschweig (1986, 1987, 1988).

[3].   Bathe K.J., *Numerical methods in finite element analysis*, Prentice-Hall (1976).

[4].   *CSI Analysis Reference Manual for SAP2000, ETABS, SAFE and CSiBridge*, Computers and Structures, Inc., Berkeley, USA (1995).

[5].   *ETABS: Integrated Building Design Software, User Interface Reference Manual (version 8)*, Computers and Structures, Inc, (2002).

[6].   *ETABS: Version 8: Integrated building design software, Structural and earthquake engineering software*, Computers and Structures, Inc, (2003).

[7].   Gautam Puri, *Python Scripts for Abaqus (Learn by Example)*, Gautam Puri, USA (2011).

[8].   Hibbitt, Karlsson and Sorensen, Inc. *ABAQUS Analysis user's manual (v 6.7)*, Pawtucket, Rhode Island (2007).

[9].   Pilkey W.D. *Analysis and Design of Elastic Beams -Computational Methods*, Wiley, New York (2002).

[10].   *SAP2000: Integrated finite element analysis and design of structures: analysis reference*, Computers and Structures, Inc., Berkeley, (1998).

[11].   *SAP2000: Integrated Finite Element Analysis and Design of Structures: Concrete Design Manual; Steel Design Manual*, Computers and Structures, Inc., (1998).

[12].   *SAP2000: Software Verification Manual*, Computers and Structures, Inc., (2006).

[13].   *SAP2000: Three Dimensional Static and Dynamic Finite Element Analysis and Design of Structures*, Computers and Structures, Inc, (1998).

[14].   Sapountzakis E.J. and Dourakopoulos J.A. Shear Deformation Effect in Flexural-Torsional Vibrations of Composite Beams by BEM, *Journal of Vibration and Control*, 16(12), 1763-1789 (2010).

[15].   Sapountzakis E.J. and Mokos V.G. A BEM Solution to Transverse Shear Loading of Beams, *Computational Mechanics,* 36, 384-397 (2005).

[16].   Sapountzakis, E.J. and Mokos, V.G., Nonuniform torsion of composite bars by boundary element method, *Journal of Engineering Mechanics* (ASCE), 127(9), 945-953 (2001).

[17].   Schueller, W., *Building Support Structures: Analysis and Design Using SAP2000 Software*, Computer and Structures, Inc., (2008).

[18].   Timoshenko, S. and Woinowsky-Krieger, S., *Theory of Plates and Shells,* 2nd ed., McGraw-Hill Book Company, New York (1959).

[19]. Timoshenko, S. and Young, D. H., *Vibration Problems in Engineering*, Van Nostrand, New Jersey (1955).

[20]. Timoshenko, S. and Young, D.H., *Vibration Problems in Engineering*, New Jersey, Van Nostrand (1955).

[21]. Timoshenko, S.P. and Goodier, J.N., *Theory of Elasticity*, 3rd edn, McGraw-Hill, New York (1984).

[22]. Vlasov, V.Z.: *Thin-walled elastic beams*, Israel Program for Scientific Translations, Jerusalem (1961).

[23]. Wilson E.L., Hollings J. P. and Dovey H. H., *ETABS: Three Dimensional Analysis of Building Systems*, National Information Service for Earthquake Engineering /Computer Applications, Earthquake Engineering Research Center, University of California (1979).

[24]. Wilson E.L., *SAP: A General Structural Analysis Program*, Report No. UC SESM 70- 20, Structural Engineering Laboratory, University of California, Berkeley (1970).

[25]. Wilson E.L., *SOLID SAP: A Static Analysis Program for Three Dimensional Solid Structures*, Report No. UC SESM 71-19, Structural Engineering Laboratory, University of California, Berkeley (1972).

Part B:
Analysis & Design
of Concrete and Steel Structures

B1. Introduction

In "Part B" single members, plates and three-dimensional frames are addressed employing elastic linear static analysis. The analysis is based on the finite element method (FEM), using beam elements (according to Euler–Bernoulli theory and considering elastic stiffness for the concrete members) and/or shell elements (plate & disc action simultaneously) with six degrees of freedom at each node.

A series of test problems (examples) have been analysed to test the various features of the program and demonstrate its adequacy in several applications. For each example, a short description of the problem and the tested technical features of the program are presented in addition to the comparison of results obtained from ScadaPro software and from other computer programs.

To validate and verify ScadaPro, the test problems were run on a PC platform that was a HP machine with Intel Core i5 Processor and 6 GB of RAM operating on a Windows 7 operating system.

Concerning acceptance criteria, the comparison of ScadaPro results with results obtained from other computer programs is represented by the percentage difference evaluated by the

formula $Difference\,(\%) = 100\left(\dfrac{Scada\,\text{Pro Results} - \text{Other Software Result}}{\text{Other Software Result}}\right)$. This difference is

typically characterized in one of the following three ways:

- **Exact:** There is no difference between ScadaPro results and results obtained from other computer programs and bibliography.

- **Acceptable:** Concerning displacements of structures, the difference between ScadaPro results and results obtained from other computer programs and bibliography does not exceed five percent (5%). For internal forces values, required reinforcements and maximum degree of utilization of steel sections, the difference does not exceed ten percent (10%). Concerning comparison with experimental results, the difference between ScadaPro and other programs and bibliography does not exceed twenty five percent (25%).

- **Unacceptable:** Concerning displacements of structures, the difference between ScadaPro results and results obtained from other computer programs and bibliography exceeds five percent (5%). For internal forces values, required reinforcements and maximum degree of utilization of steel sections,, the difference exceeds ten percent (10%). Concerning comparison with experimental results, the

difference between ScadaPro and other programs and bibliography exceeds twenty five percent (25%).

CONCLUSION

ScadaPro 14 is the latest release of the ScadaPro series of computer programs. Since development, ScadaPro has been used widely for structural analysis. Additionally, the verification process conducted as described in this document demonstrates that the program features tested are operating reliably and with accuracy consistent with current computer technology capabilities.

B2. Analysis & Design of Concrete Structures

B2.1. Rectangular Plate under Vertical Loading

Problem Description

A two dimensional rectangular plate, subjected to vertical loading is examined (Figure B2.1.1).

Geometry, Properties, Loads and Loading Combinations

The plate of dimensions 9.75m x 4.75m has 20cm thickness. Material is reinforced concrete of modulus of elasticity 29GPa and Poisson ratio ν=0.20. The concrete strength class is C20/25 while the class of reinforcing steel is B500C. The centre distance of the reinforcement from the plate edge is 20mm.

The plate is subjected to its self-weight (ε_c=25.0kN/m^3), to a uniformly distributed permanent load 2kN/m^2 and to a uniformly distributed variable load 5kN/m^2.

Two cases of loading combinations are examined. In the first one (Comb1) the value of the participation factor of all loads is equal to 1.0. In the second case (Comb2) the participation factor of permanent loads is equal to 1.35 while the corresponding one of the variable load is equal to 1.50.

Two cases of boundary conditions are studied. According to the first one (Support Case 1) the plate is simply supported along the long sides while the short sides are free. In the second case (Support Case 2) the plate is simply supported along all sides. The analysis is based on the finite element method, using plate elements.
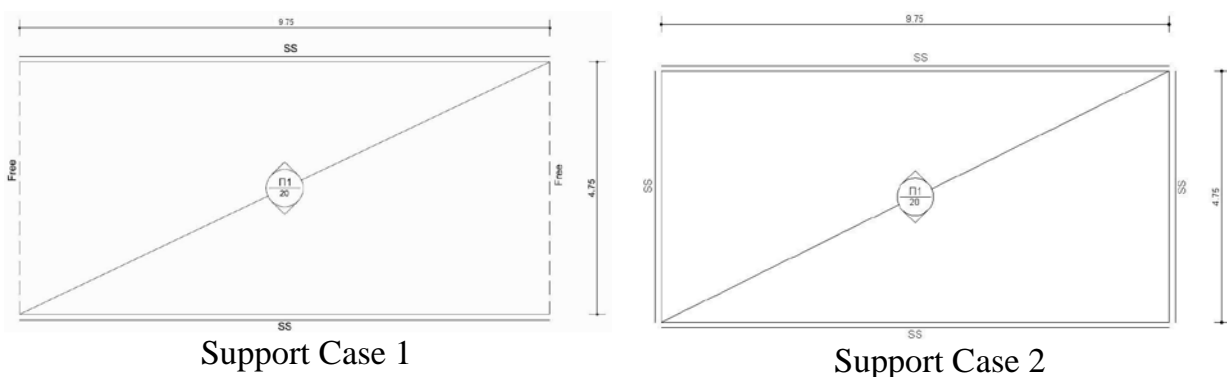

Support Case 1   Support Case 2

Figure B2.1.1. Rectangular Plate examined for two cases of boundary conditions

Assumptions

During analysis, Kirchhoff theory of plates has been adopted.

Technical Features in ScadaPro Tested

➢ Analysis of plate elements under vertical loading

➢ Simply supported boundary conditions

➢ Plate elements design

Results Comparison

Vertical displacements due to combination Comb1 and bending moments, bottom reinforcement due to combination Comb2 obtained from ScadaPro analysis are presented in Tables B2.1.1 ÷ B2.1.3, respectively and compared with those obtained from Sofistik and Cedrus-5 software. Moreover, in Figures B2.1.2 ÷ B2.1.4 contour lines of bottom transverse reinforcement, for support cases Sup1 and Sup2 respectively, are presented for all of the aforementioned software.
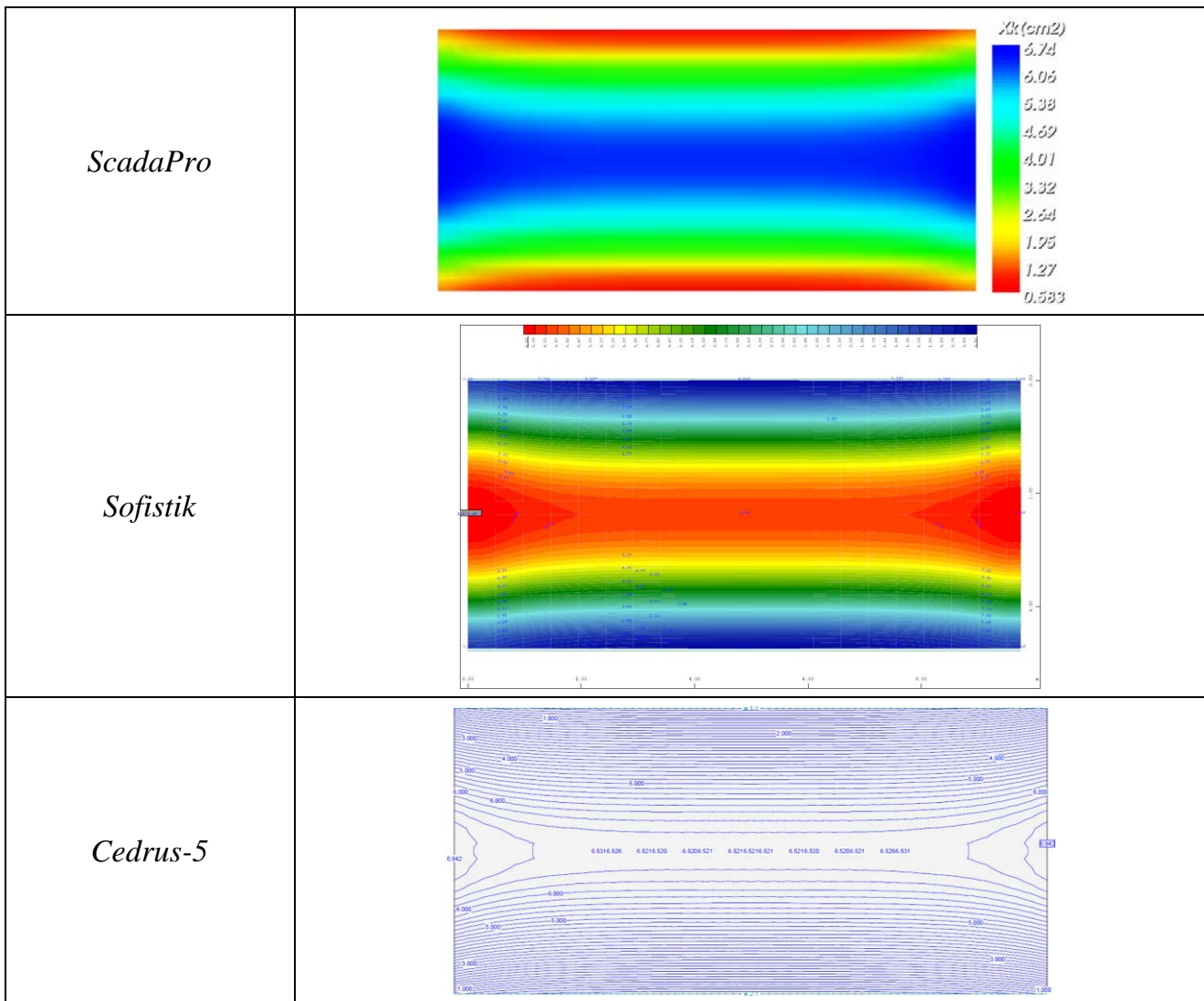
ScadaPro	
Sofistik	
Cedrus-5	

Figure B2.1.2. Bottom transverse reinforcement Xk (cm^2/m) for Support Case Sup1 according to ScadaPro, Sofistik and Cedrus-5 software (Comb2).

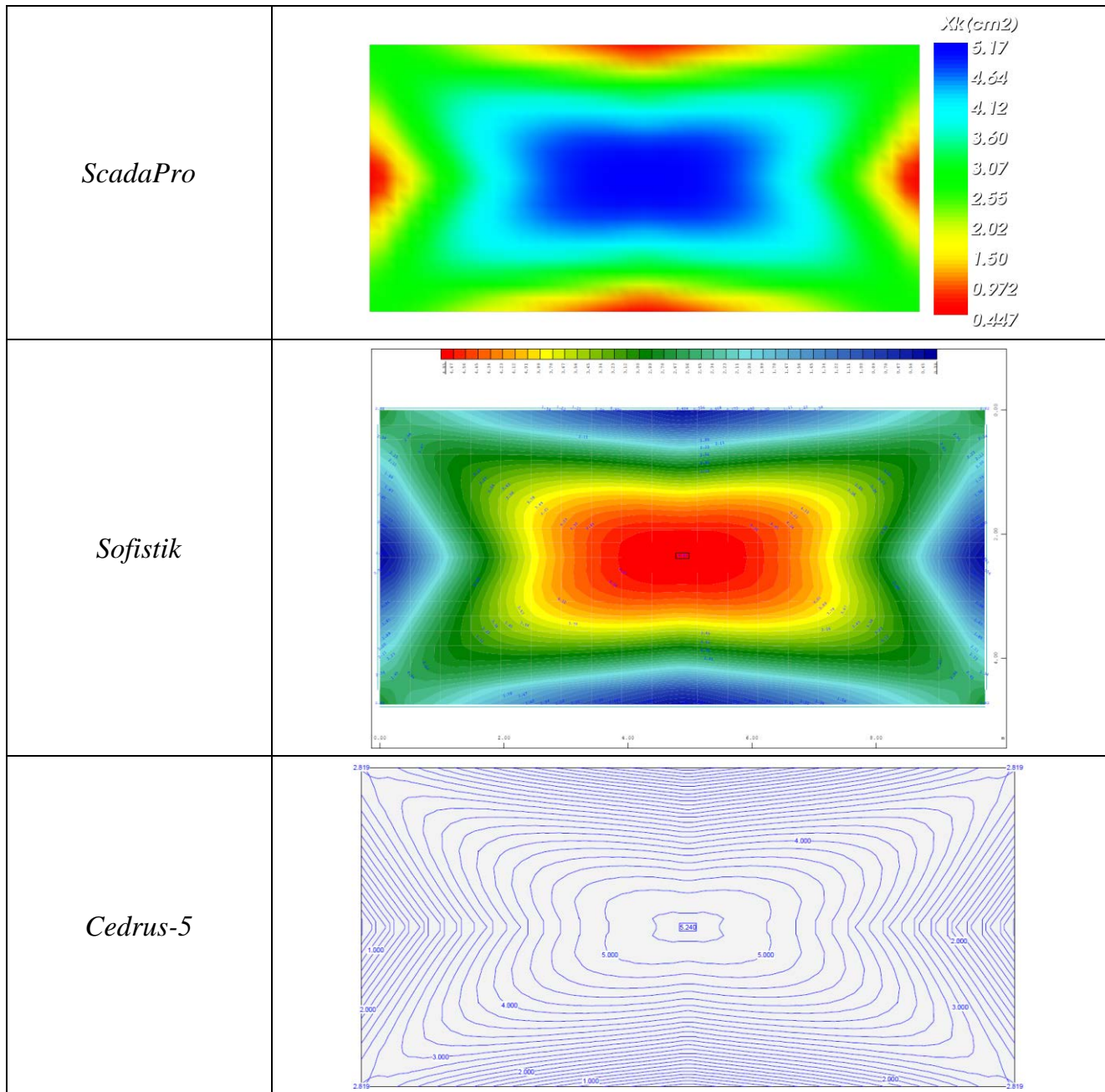|	
ScadaPro	
Sofistik	
Cedrus-5	

Figure B2.1.3. Bottom transverse reinforcement Xk (cm²/m) for Support Case Sup2 according to ScadaPro, Sofistik and Cedrus-5 software (Comb2).

Table B2.1.1. Comparison of Results for Vertical Displacements (Comb1).

Vertical Displacement (mm) at	ScadaPro	Sofistik	Cedrus-5
center point of plate (Support Case 1)	3.91	3.80	3.92
center point of plate (Support Case 2)	3.12	3.04	3.13

Table B2.1.2. Comparison of Results for Bending Moments (Comb2).

Bending Moment	ScadaPro	Sofistik	Cedrus-5
Mxx (kNm/m) at center of plate (Support Case 1)	9.36	9.36	8.00
Myy (kNm/m) at center of plate (Support Case 1)	47.00	47.11	47.40
Mxx (kNm/m) at center of plate (Support Case 2)	13.70	13.73	12.70
Myy (kNm/m) at center of plate (Support Case 2)	38.60	38.70	38.60

Table B2.1.3. Comparison of Results for Bottom Transverse Reinforcement (Comb2).

Reinforcement	ScadaPro	Sofistik	Cedrus-5
Zk (cm^2/m) at center of plate (Support Case 1)	1.22	1.22	1.05
Xk (cm^2/m) at center of plate (Support Case 1)	6.31	6.08	6.52
Zk (cm^2/m) at center of plate (Support Case 2)	1.70	1.71	1.66
Xk (cm^2/m) at center of plate (Support Case 2)	5.17	4.85	5.24

Computer Files

The project files for this example are "B1_FS" & "B2_SS". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Sofistik and Cedrus-5.

B2.2. Skew Plate under Transverse Loading

Problem Description

A two dimensional skew plate, subjected to vertical loading is examined (Figure B2.2.1).

Geometry, Properties and Loads

The plate of dimensions 6.0m x 8.48[⌞6.0]m has 30cm thickness and 45° skew angle. Material is reinforced concrete of modulus of elasticity 29GPa and Poisson ratio v=0.20. The concrete strength class is C20/25 while the class of reinforcing steel is B500C. The centre distance of the reinforcement from the plate edge is 20mm. The short sides of the plate are fixed while the long sides are free. The plate is subjected to its self-weight (ε_c=25.0kN/m^3), to a uniformly distributed permanent load 2kN/m^2 and to a uniformly distributed variable load 5kN/m^2. Two cases of loading combinations are examined. In the first one (Comb1) the value of the participation factor of all loads is equal to 1.0. In the second case (Comb2) the participation factor of permanent loads is equal to 1.35 while the corresponding one of the variable load is equal to 1.50.
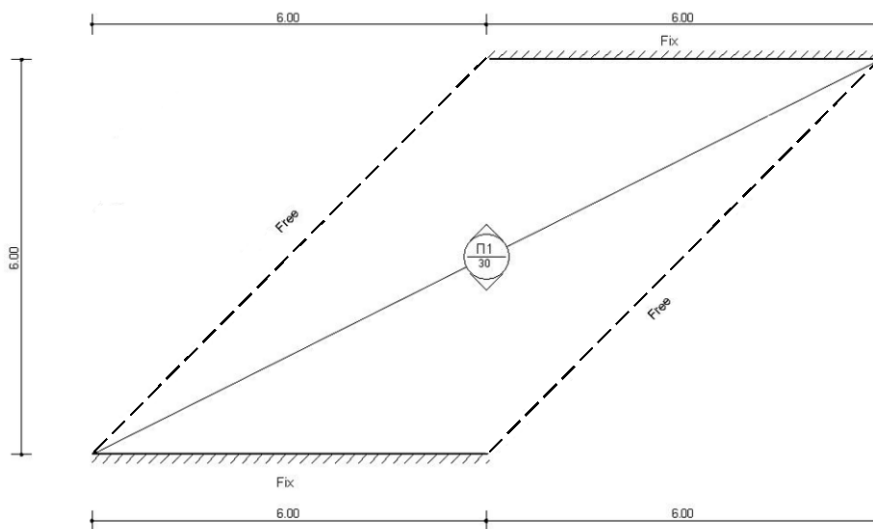

Figure B2.2.1. Skew Plate

Assumptions

During analysis, Kirchhoff theory of plates has been adopted.

Technical Features in ScadaPro Tested

➢ Analysis of plate elements under transverse loading

➢ Fixed boundary conditions

➢ Plate elements design

45

Results Comparison

Vertical displacements due to combination Comb1 and bending moments, reinforcement due to combination Comb2 obtained from ScadaPro analysis are presented in Tables B2.2.1 ÷ B2.2.3 respectively, and compared with those obtained from Sofistik and Cedrus-5 software. Moreover, in Figures B2.2.2 ÷ B2.2.3 contour lines of top and bottom transverse reinforcement respectively, are presented for all of the aforementioned software.
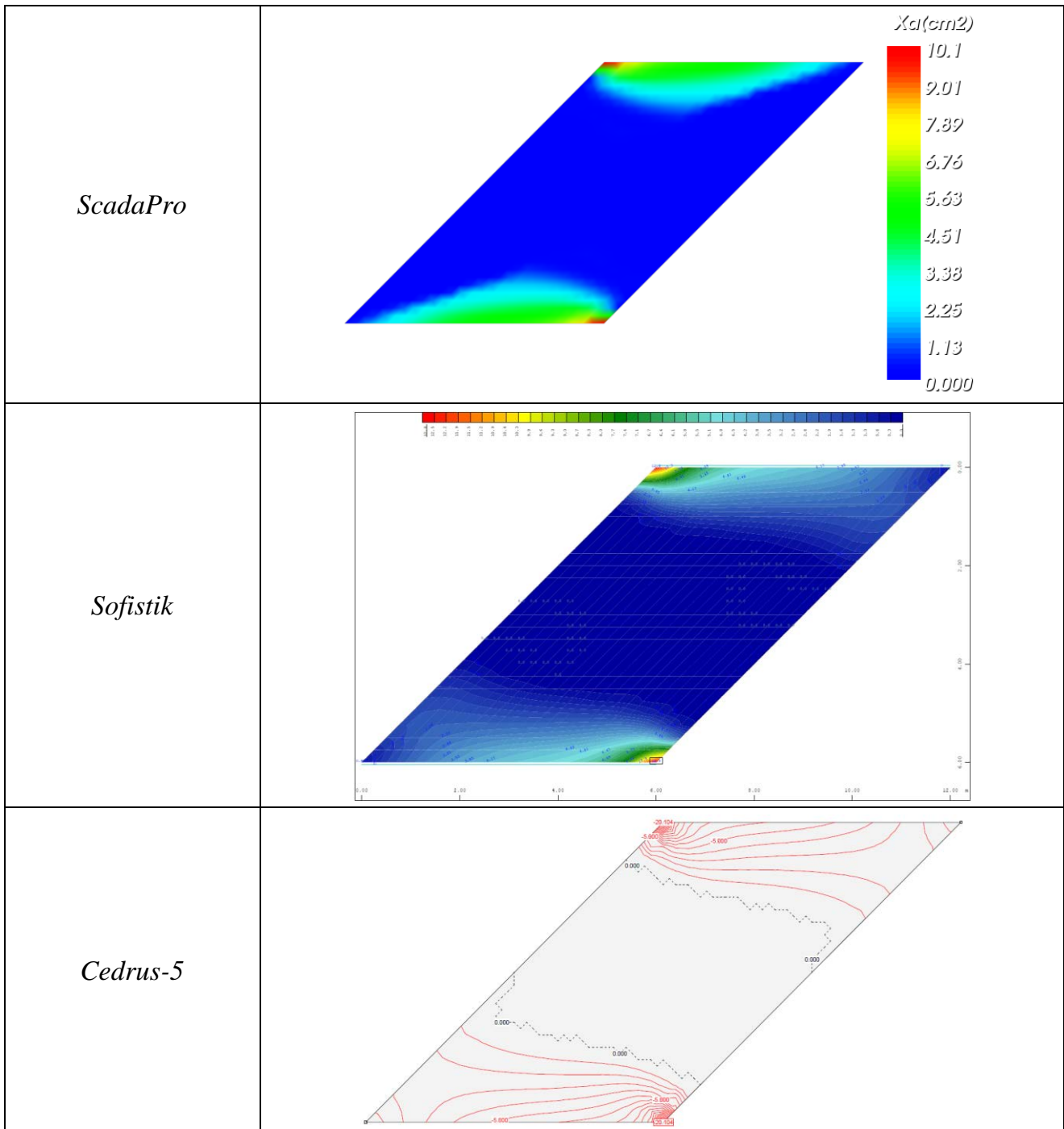
ScadaPro	
Sofistik	
Cedrus-5	

Figure B2.2.2. Top transverse reinforcement Xα (cm²/m) according to ScadaPro, Sofistik and Cedrus-5 software (Comb2).
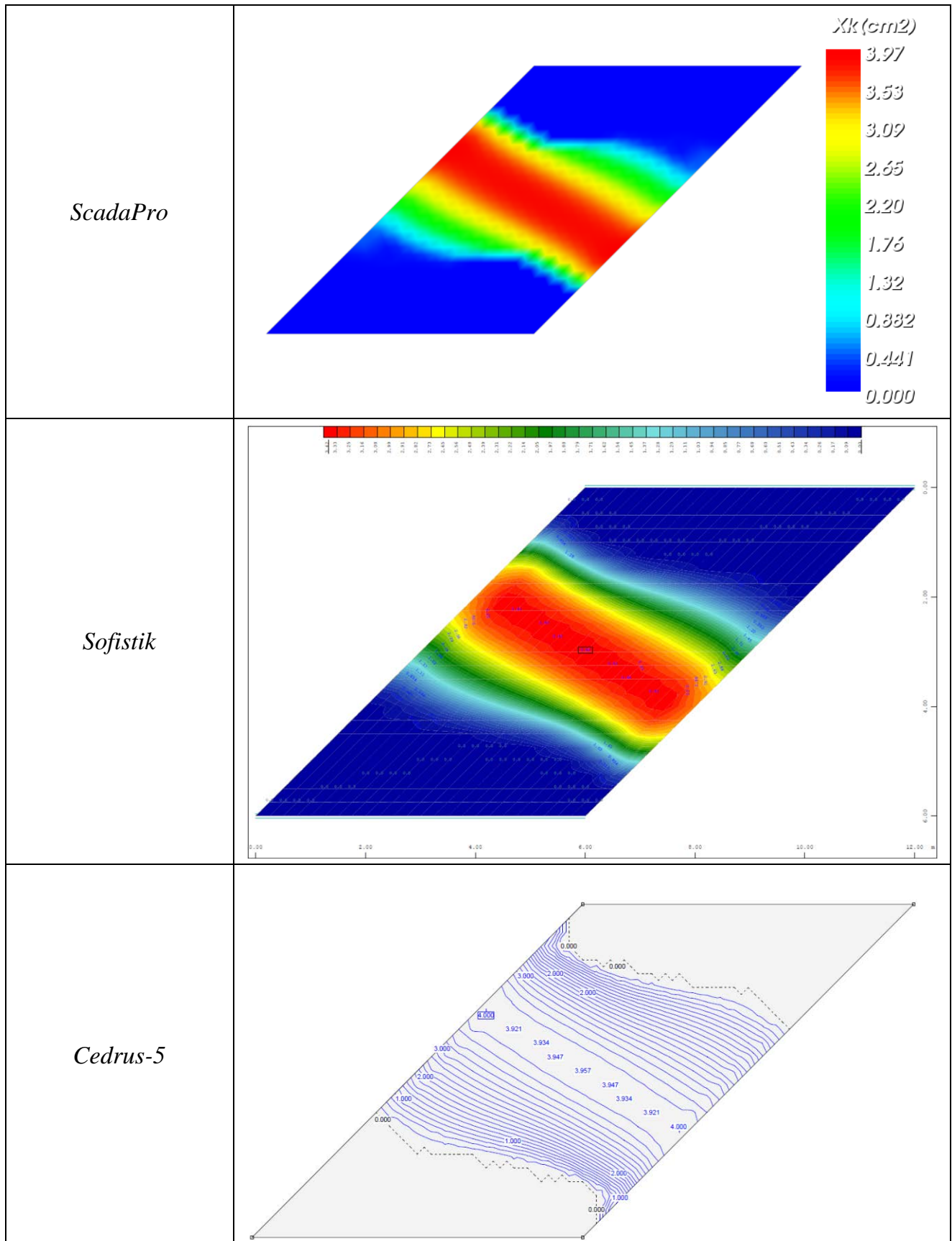
ScadaPro	
Sofistik	
Cedrus-5	

Figure B2.2.3. Bottom transverse reinforcement Xk (cm^2/m) according to ScadaPro, Sofistik and Cedrus-5 software (Comb2).

Table B2.2.1. Comparison of Results for Vertical Displacements (Comb1).

Vertical Displacement (mm) at	ScadaPro	Sofistik	Cedrus-5
center point of plate	1.04	1.08	1.05
middle point of free side	1.33	1.19	1.14

Table B2.2.2. Comparison of Results for Bending Moments (Comb2).

Bending Moment	ScadaPro	Sofistik	Cedrus-5
Myy (kNm/m) at center of plate	35.00	34.94	35.03
Mxx (kNm/m) at middle point of free side	8.44	9.60	8.23

Table B2.2.3 Comparison of Results for Reinforcement (Comb2).

Reinforcement	ScadaPro	Sofistik	Cedrus-5
Xk (cm^2/m) at center of plate	3.97	3.44	3.91
Zk (cm^2/m) at middle point of free side	1.95	1.56	1.64

Computer Files

The project file for this example is "B2_FIX". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Sofistik and Cedrus-5.

B2.3. Square Plate with Internal Supports

Problem Description

A two dimensional square plate with internal supports, subjected to vertical loading is examined (Figure B2.3.1).

Geometry, Properties and Loads

The plate of overall dimensions 10.00m x 10.00m has 20cm thickness with four internal supports. All boundary of the plate (externally and internally) are simply supported. Material is reinforced concrete of modulus of elasticity 29GPa and Poisson ratio $v=0.20$. The concrete strength class is C20/25 while the class of reinforcing steel is B500C. The centre distance of the reinforcement from the plate edge is 20mm.

The plate is subjected to its self-weight ($\varepsilon_c=25.0$kN/m^3), to a uniformly distributed permanent load 2kN/m^2 and to a uniformly distributed variable load 5kN/m^2.

Two One case of loading combinations are examined. In the first one (Comb1) the value of the participation factor of all loads is equal to 1.0. In the second case (Comb2) the participation factor of permanent loads is equal to 1.35 while the corresponding one of the variable load is equal to 1.50.

For the analysis of the plates two cases are studied. According to the first one (Case1) the system is analyzed using "Markus Tables" [9], which are usually applied in engineering practice. In the second case (Case2), the system is analyzed using plate finite elements.
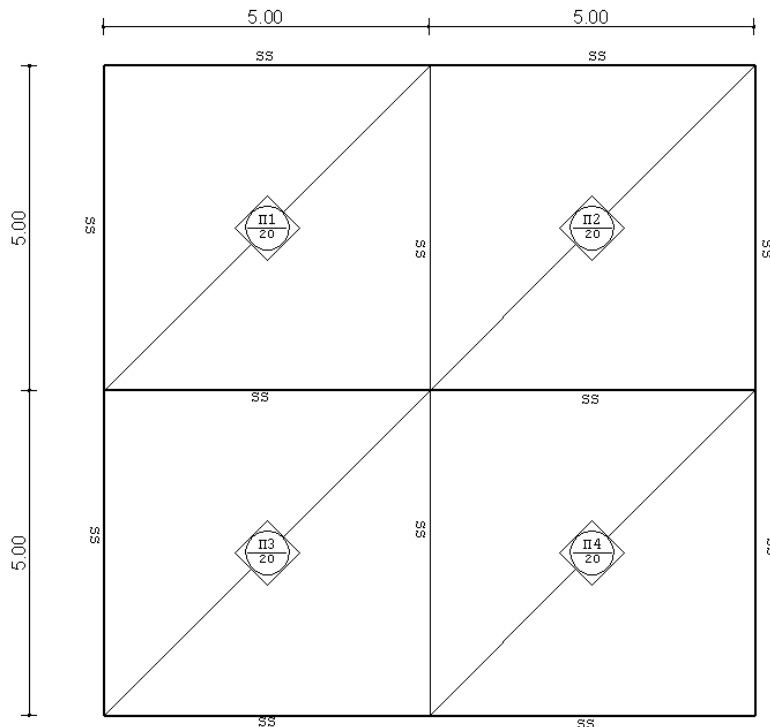

Figure B2.3.1. Square Plate with Internal Supports.

Assumptions

During analysis, Kirchhoff theory of plates has been adopted.

Technical Features in ScadaPro Tested

➢ Analysis of plate elements under transverse loading

➢ Simply supported boundary conditions

➢ Plate elements design

Results Comparison

Vertical displacement due to combination Comb1 obtained from ScadaPro analysis is presented in Figures B2.3.2 for Case 2, where the max values are compared with those obtained from Sofistik software.
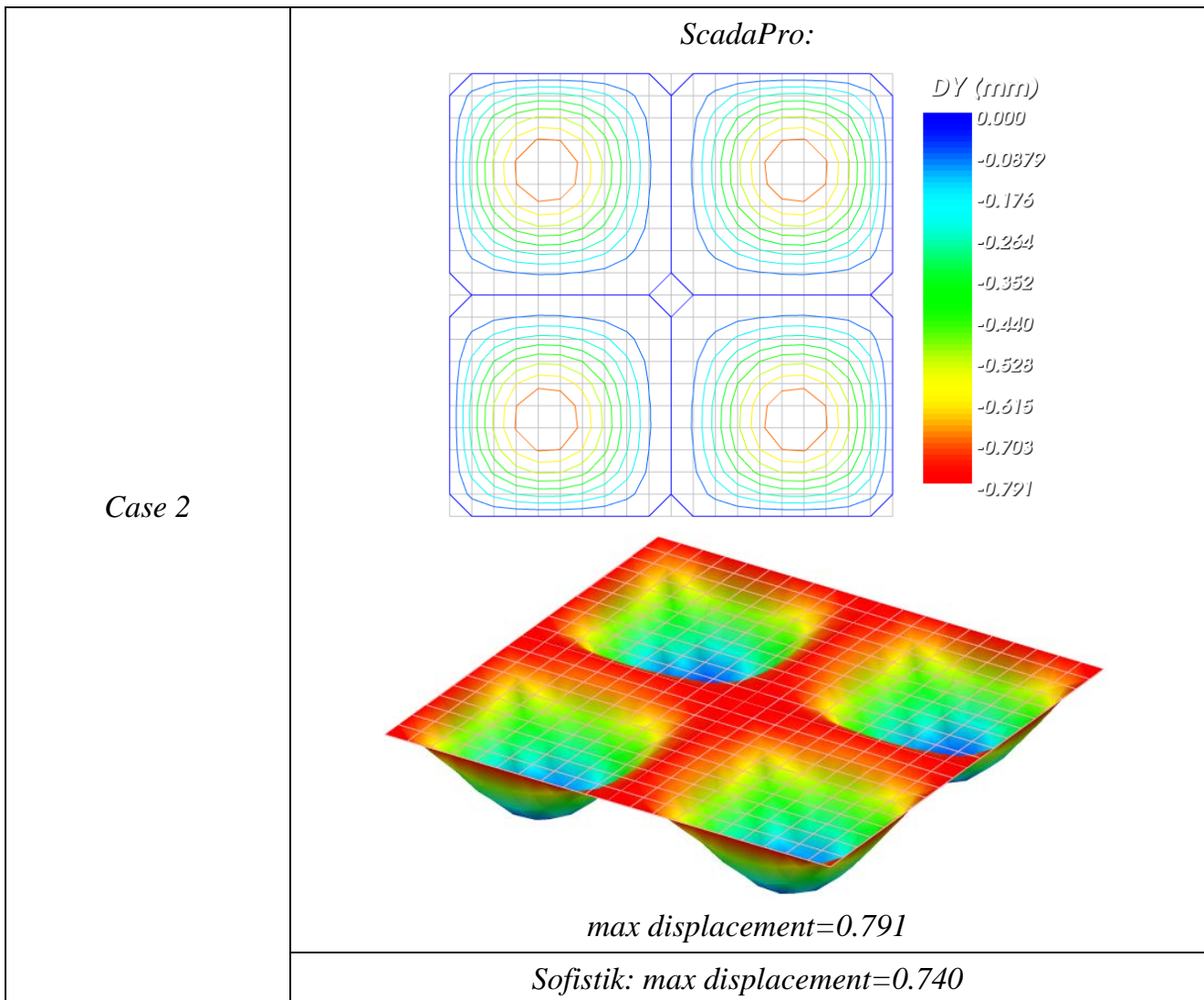
	ScadaPro:
Case 2	*max displacement=0.791*
	Sofistik: max displacement=0.740

Figure B2.3.2. Vertical displacements (mm) for Case 2 (Comb1).

Moreover, in Figures B2.3.3 contour lines of bottom transverse reinforcement due to combination Comb2 and for Case 2 are presented obtained from ScadaPro analysis and compared with those obtained from Sofistik software.
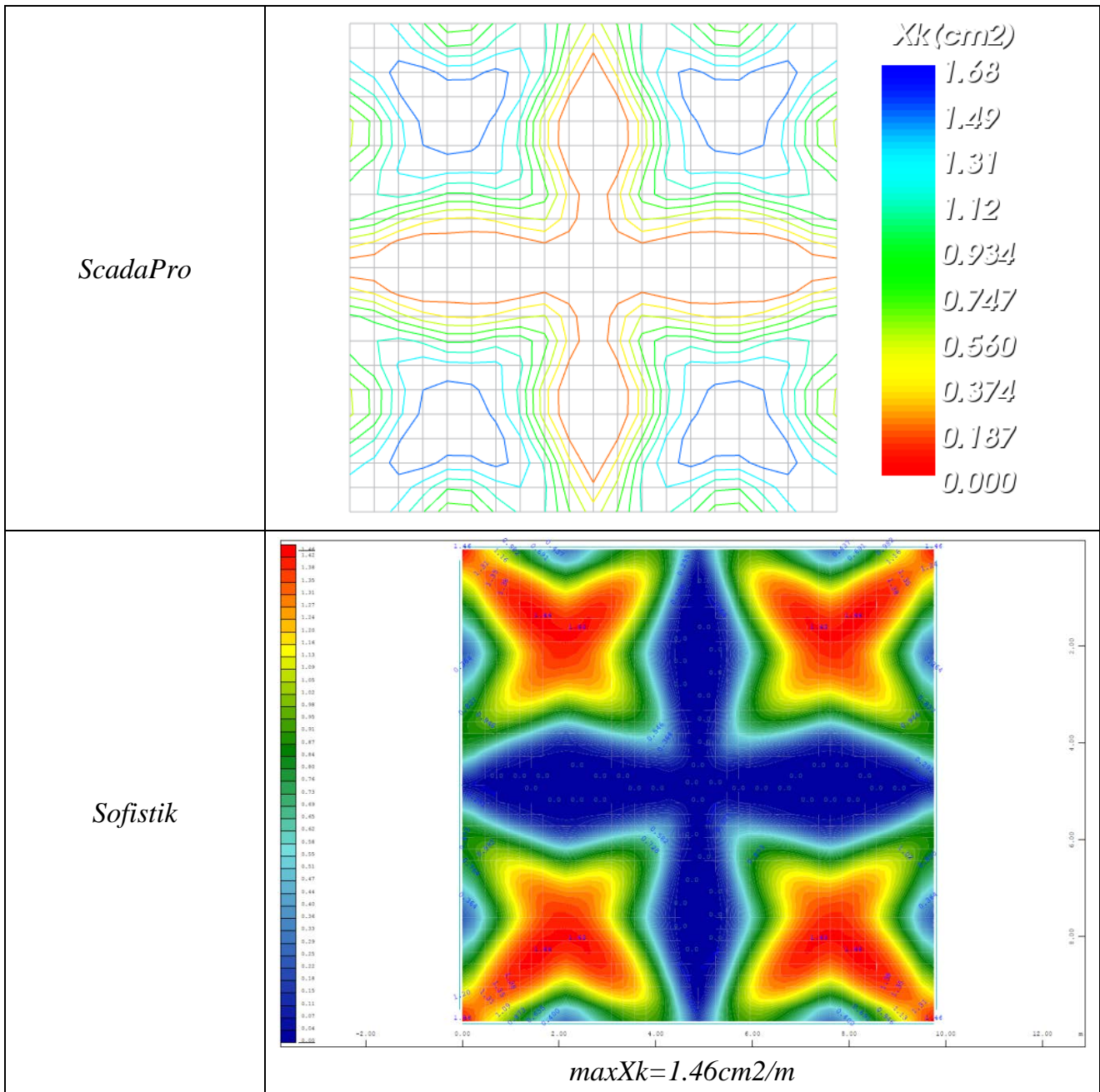
ScadaPro	
Sofistik	 *maxXk=1.46cm2/m*

Figure B2.3.3. Bottom transverse reinforcement Xk (cm^2/m) for Case 2 according to ScadaPro and Sofistik Analysis.

Finally, in Table B2.3.1 the max reinforcement (cm^2/m) due to combination Comb2 of both, at the span and at the support of individual plate obtained from ScadaPro analysis for Cases 1 and 2, are presented.

Table B2.3.1. Comparison of Results for Case 1 (Comb2).

max Reinforcement (cm^2/m) - ScadaPro		
Case 1 "Markus Tables"	at the span of individual plate	1.88
	at the support of individual plate	3.39
Case 2 "FEM"	at the span of individual plate	1.68
	at the support of individual plate	2.55

Computer Files

The project files for this example are "B3_C1", "B3_C2. These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Sofistik.

B2.4. Stiffened Plate

Problem Description

A two dimensional rectangular plate stiffened by a rectangular beam, subjected to vertical loading is examined (Figure B2.3.1).

Geometry, Properties and Loads

A rectangular plate with dimensions $l_{px} \times l_{py} = 18.0 \times 9.0 \; m$ subjected to a uniform load $g = 10kN / m^2$ and stiffened by a rectangular beam of *1.0 m* width eccentrically placed with respect to the centerline of the plate (Fig.B2.4.1) has been studied. The plate is clamped along its small edges, while the other two edges are free.

Four cases are studied. According to the first one (Case1), a finite element solution is employed using beam elements for the beam taking into account vertical eccentricity and plate finite elements for the plate (results are obtained from ScadaPro and SAP2000 software). In the second case (Case2), a finite element solution is employed using plate elements for the plate and shell elements for the beam (results are obtained from Nastran software). In the third case (Case3), a finite element solution is employed using solid elements for both the beam and the plate (results are obtained from Nastran software). Finally, according to the fourth case (Case4), a boundary element solution (BEM) is employed using boundary elements for both, the beam and the plate, and using the structural model presented by Sapountzakis and Mokos in [18]. In Fig.B2.4.2 the structural models for all four cases are presented.
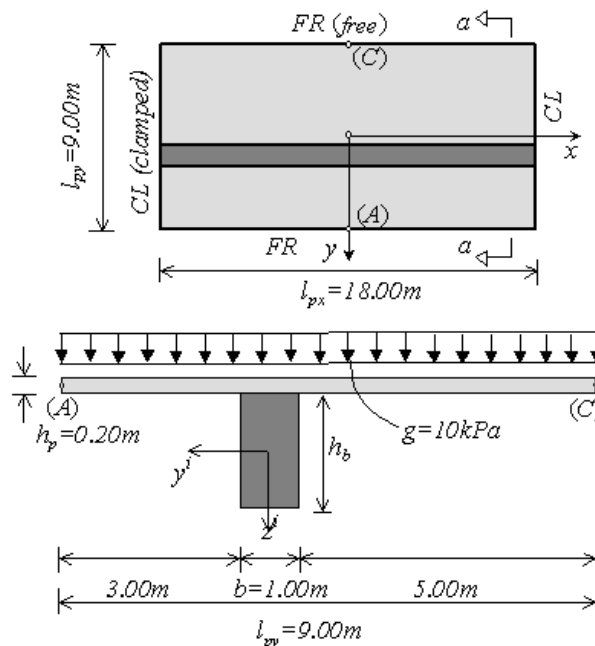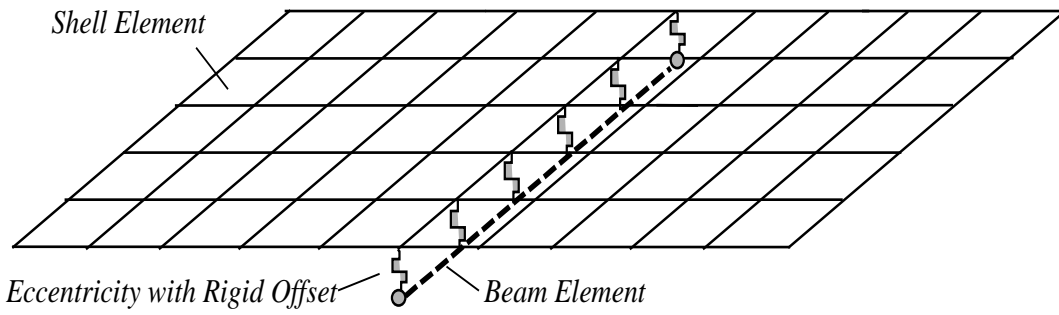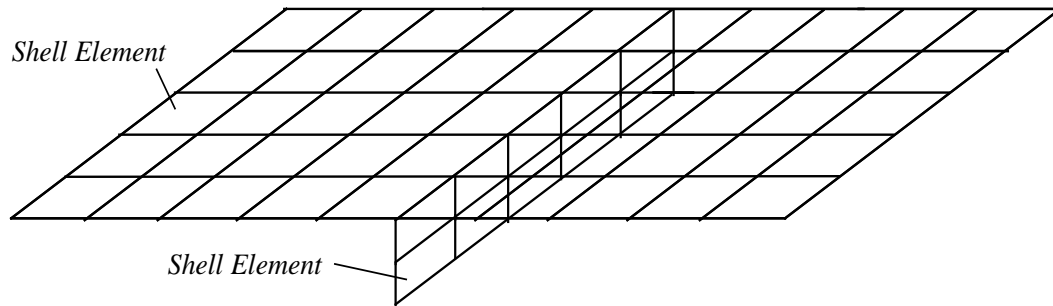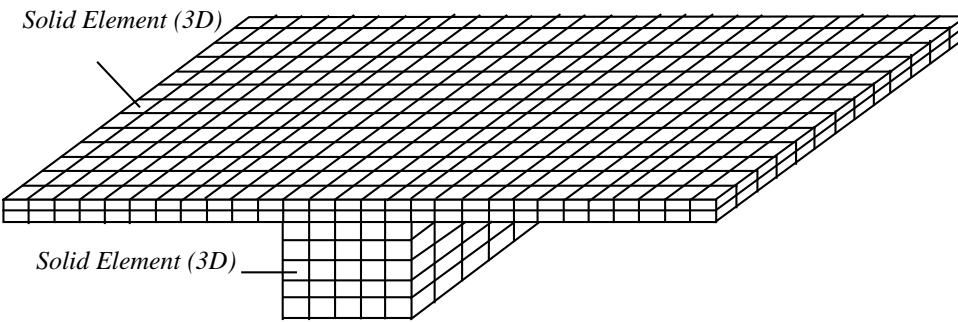

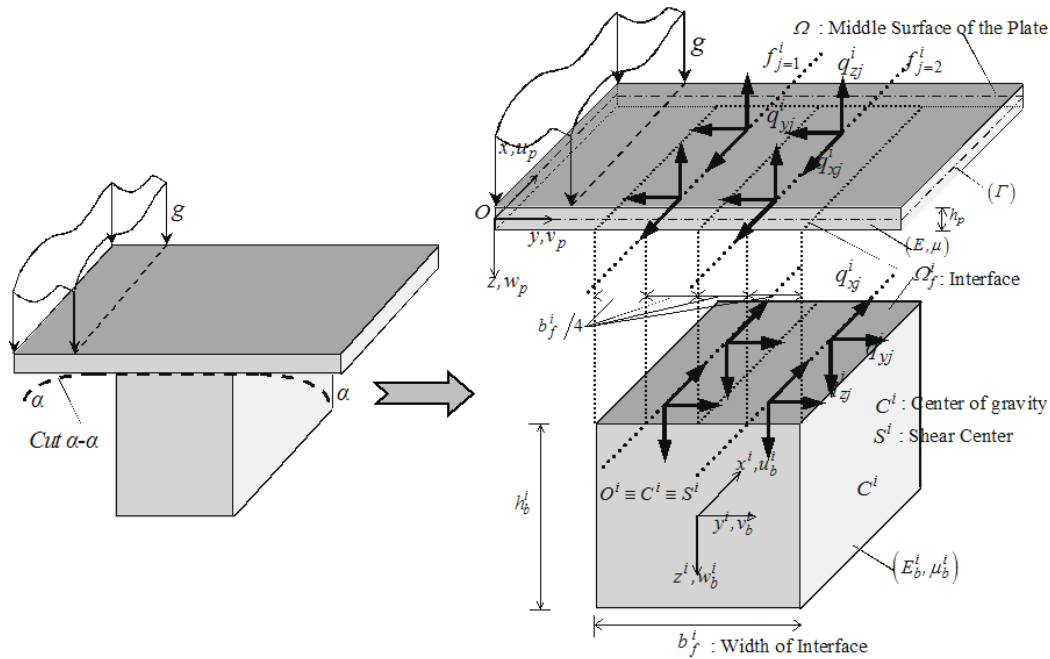*Figure B2.4.1. Plan view (a) and section a-a (b) of the stiffened plate.*

Case 1

Case 2

Case 3

Case 4

Figure B2.4.2. Structural models for all four cases of the stiffened plate.

Assumptions

During analysis, Kirchhoff theory of plate and Bernoulli theory of beam has been adopted.

Technical Features in ScadaPro Tested

➢ Analysis of stiffened plate under transverse loading

➢ Three – dimensional analysis of shell and beam elements (simultaneously)

Results Comparison

Vertical displacements for all four cases are presented in Table B2.4.1. Moreover, for $h_b = 1.0m$, in Fig.B2.4.3 the deflection surfaces of the stiffened plate and in Fig.B2.4.4 the bending moments (Case1) of the beam are also presented.

Table B2.4.1. Deflections (cm) of the stiffened plate.

h_b (m)	FEM				BEM
	Case1		Case2	Case3	Case4
	ScadaPro	SAP2000	Nastran	Nastran	Sapountzakis & Mokos [18]
Center of the Plate					
0.5	1.83	1.87	1.83	1.71	2.56
1.0	0.58	0.61	0.62	0.51	0.77
1.5	0.35	0.38	0.39	0.28	0.43
2.0	0.28	0.31	0.32	0.19	0.31
Middle of the free edge A of the plate					
0.5	1.77	1.74	1.85	1.68	2.30
1.0	0.97	0.94	0.97	0.70	0.96
1.5	0.86	0.84	0.85	0.59	0.82
2.0	0.83	0.81	0.82	0.57	0.80
Middle of the free edge C of the plate					
0.5	5.24	5.21	5.10	4.61	5.84
1.0	3.95	3.90	3.91	3.44	4.07
1.5	3.69	3.64	3.67	3.13	3.66
2.0	3.60	3.55	3.58	2.99	3.51

max $w_p = 3.95$cm , Case1 (ScadaPro)

max $w_p = 3.91$cm , Case1 (SAP2000)

max $w_p = 3.92$cm , Case2 (Nastran)

max $w_p = 3.44$cm , Case3 (Nastran)

max $w_p = 4.07$cm , Case4 ([18])

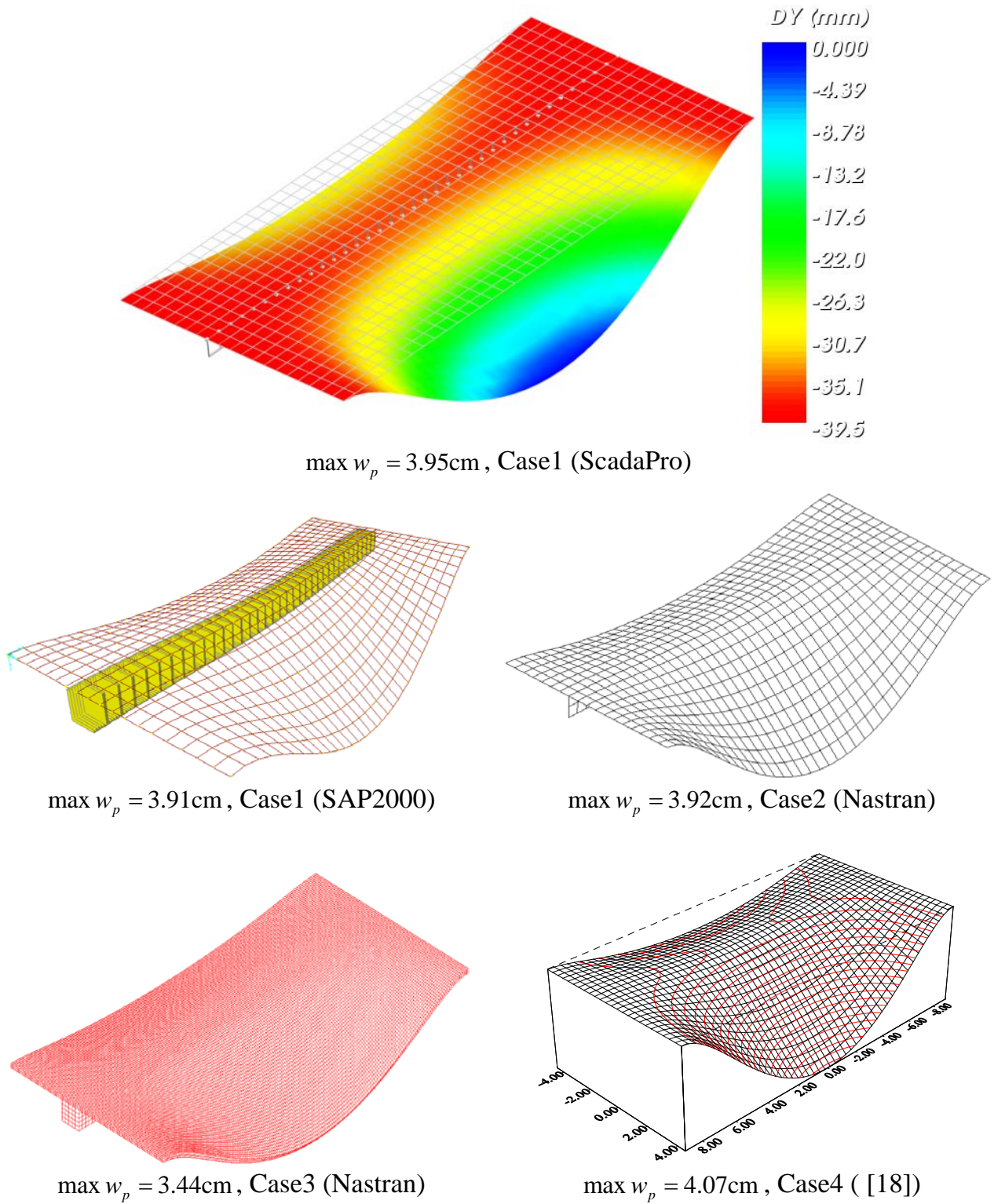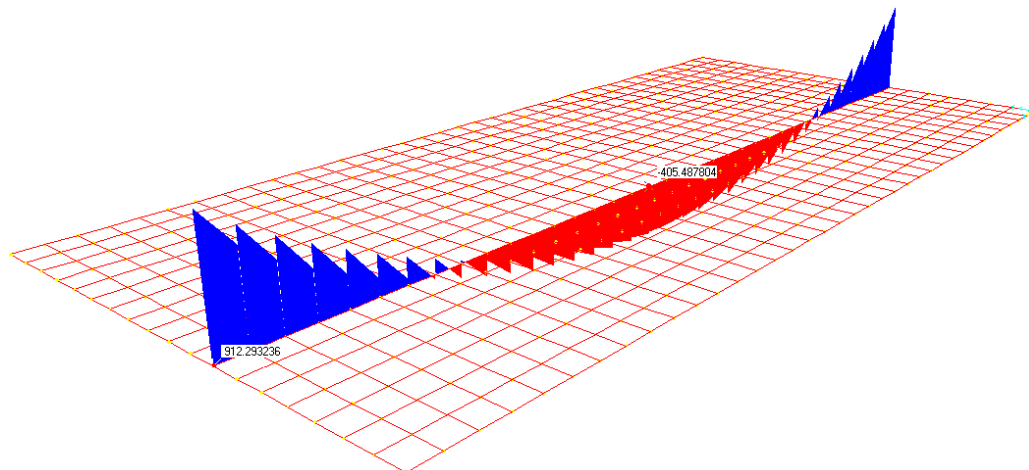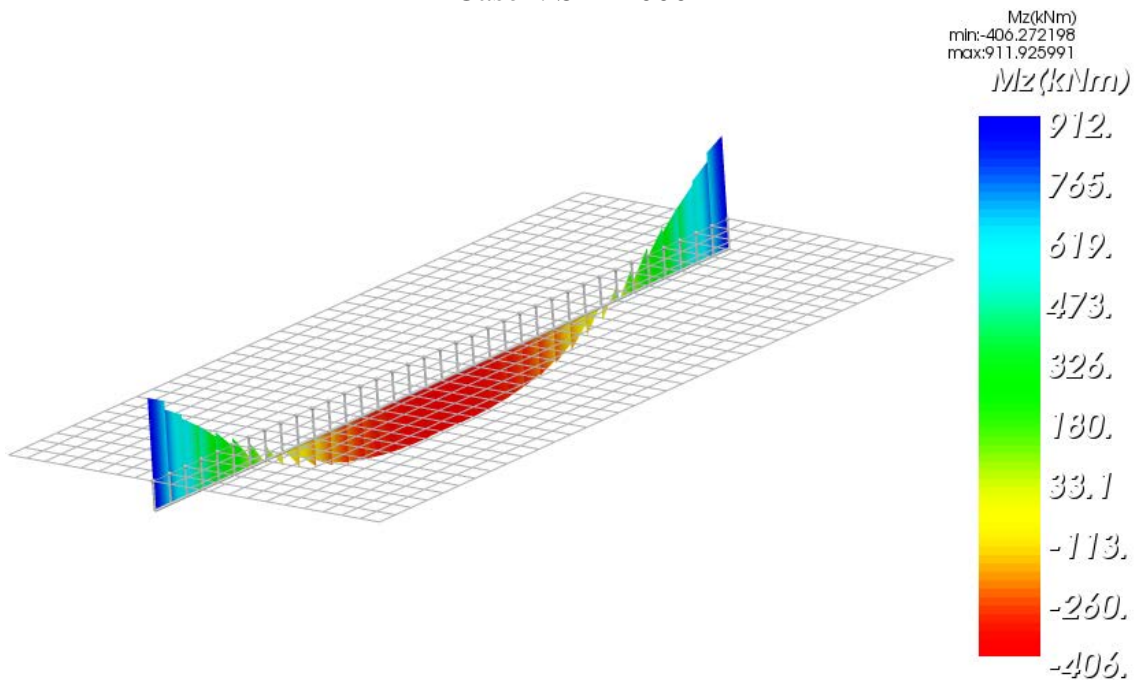*Figure B2.4.3. Deflection surfaces and corresponding maximum values of the stiffened plate for $h_b = 1.0m$.*

(maxMz: 912.3kNm/-405.5kNm)
Case1: SAP2000

(maxMz: 911.9kNm/-406.3kNm)
Case1: ScadaPro

Figure B2.4.4. Bending moments of the beam for $h_b = 1.0m$ (Case1).

Computer Files

The project files for this example are "B4_H50", "B4_H100", "B4_H150", "B4_H200". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from SAP2000, Nastran and the structural model presented by Sapountzakis and Mokos in [18].

B3. Analysis & Design of Steel Structures

B3.1. Three Dimensional Steel Truss Roof

Problem Description

A three dimensional steel truss roof subjected to vertical loading is examined (Figure B3.1.1).

Geometry, Properties and Loads

The cross section of top and bottom flange of main typical trusses is SHS150x10, of diagonals is SHS90x5. The cross section of top and bottom flange of secondary trusses is CHS60.3x4, of diagonals is CHS48.3x4.

The cross section of rafters is IPE-160 and of stiff joints is LEQ100x10. Modulus of elasticity of the material is 210GPa and Poisson ratio is 0.30. The class of structural steel is S235.

The main trusses are simply supported. The vertical movement of the edge of the rafters is forbidden.

Self-weight is taken into account (ε_s=78.5kN/m^3). Rafters are subjected to permanent distributed vertical load of 1.430kN/m and 0.715kN/m for middle and outside rafters, respectively (Figure B3.1.2.a). Rafters are also subjected to variable distributed vertical load of 1.773kN/m and 0.887kN/m for middle and outside rafters, respectively (Figure B3.1.2.b). Two cases of loading combinations are examined. In the first one (Comb1) the value of the participation factor of all loads is equal to 1.0. In the second case (Comb2) the participation factor of permanent loads is equal to 1.35 while the corresponding one of the variable load is equal to 1.50.
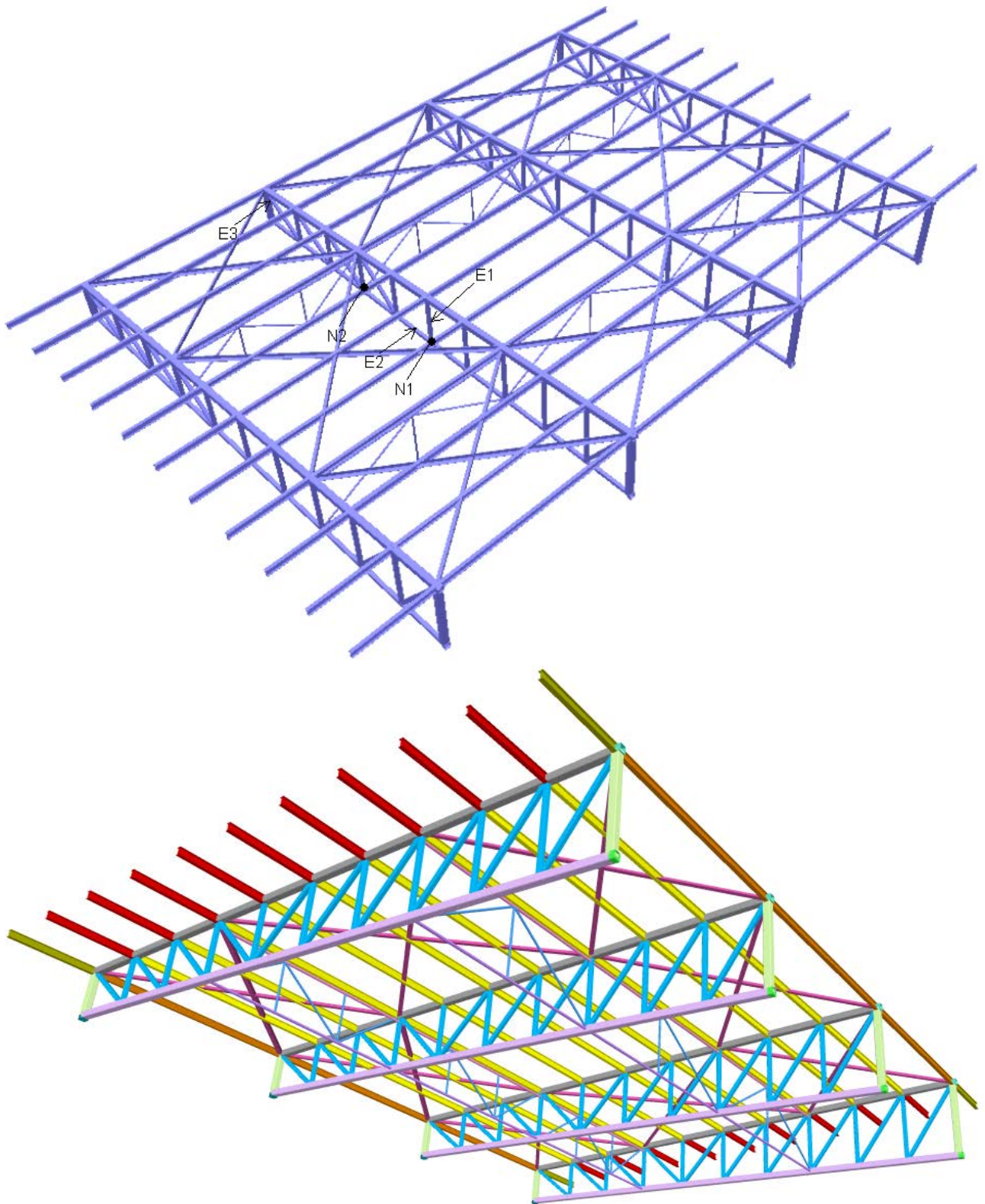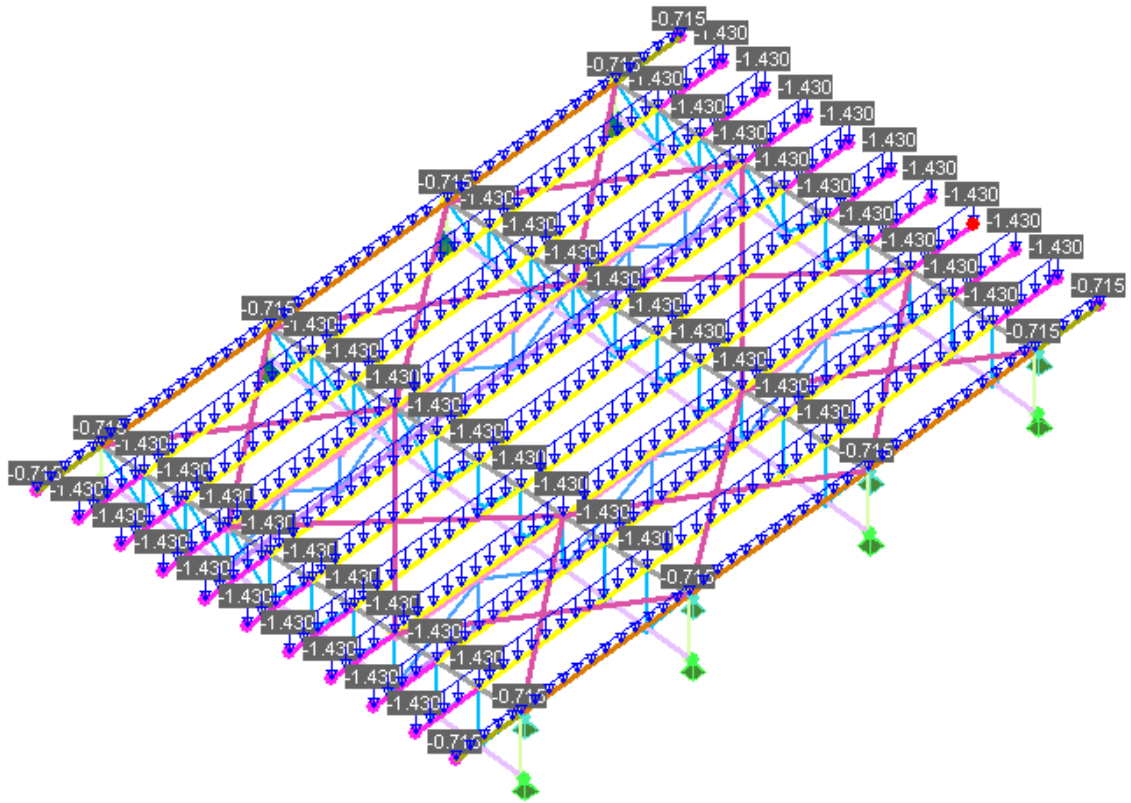
Figure B3.1.1. Three Dimensional Steel Truss Roof.

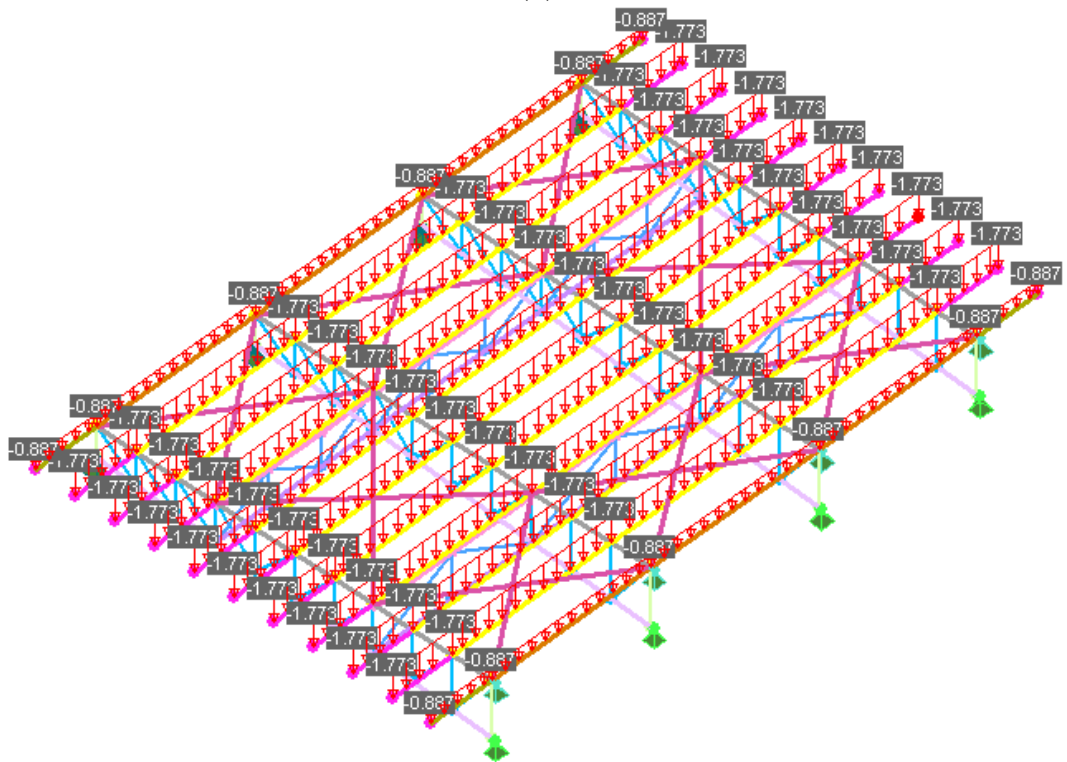(including self-weight)

(*a*)

(*b*)

Figure B3.1.2. Permanent (a) and variable (b) distributed vertical load of the truss roof.

Assumptions

During analysis, shear deformations are neglected. Section properties are automatically recovered from the cross section library of ScadaPro.

Technical Features in ScadaPro Tested

> ➢ Three – dimensional analysis of beam elements

Results Comparison

Maximum values of vertical displacements (Comb1) and axial forces (Comb2) along principal axis of cross section obtained from ScadaPro analysis, are presented in Tables B3.1.1 & B3.1.2 respectively and compared with those obtained from Statik-5 software.

Table B3.1.1. Comparison of Results for vertical Displacements (Comb1)

Vertical Displacements DY(mm) at	ScadaPro	Statik-5
Node 'N1'	5.49	5.61
Node 'N2'	4.80	4.92

Table B3.1.2. Comparison of Results for Axial forces (Comb2)

Axial Forces N(kN) at	ScadaPro	Statik-5
Element 'E1'	50.48	52.07
Element 'E2'	118.84	120.84
Element 'E3'	106.56	109.00

Computer Files

The project file for this example is "B5". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Statik-5.

B3.2. Three Dimensional Frame Structure

Problem Description

A three dimensional steel frame subjected to vertical and horizontal loading is examined (Figure B3.2.1).
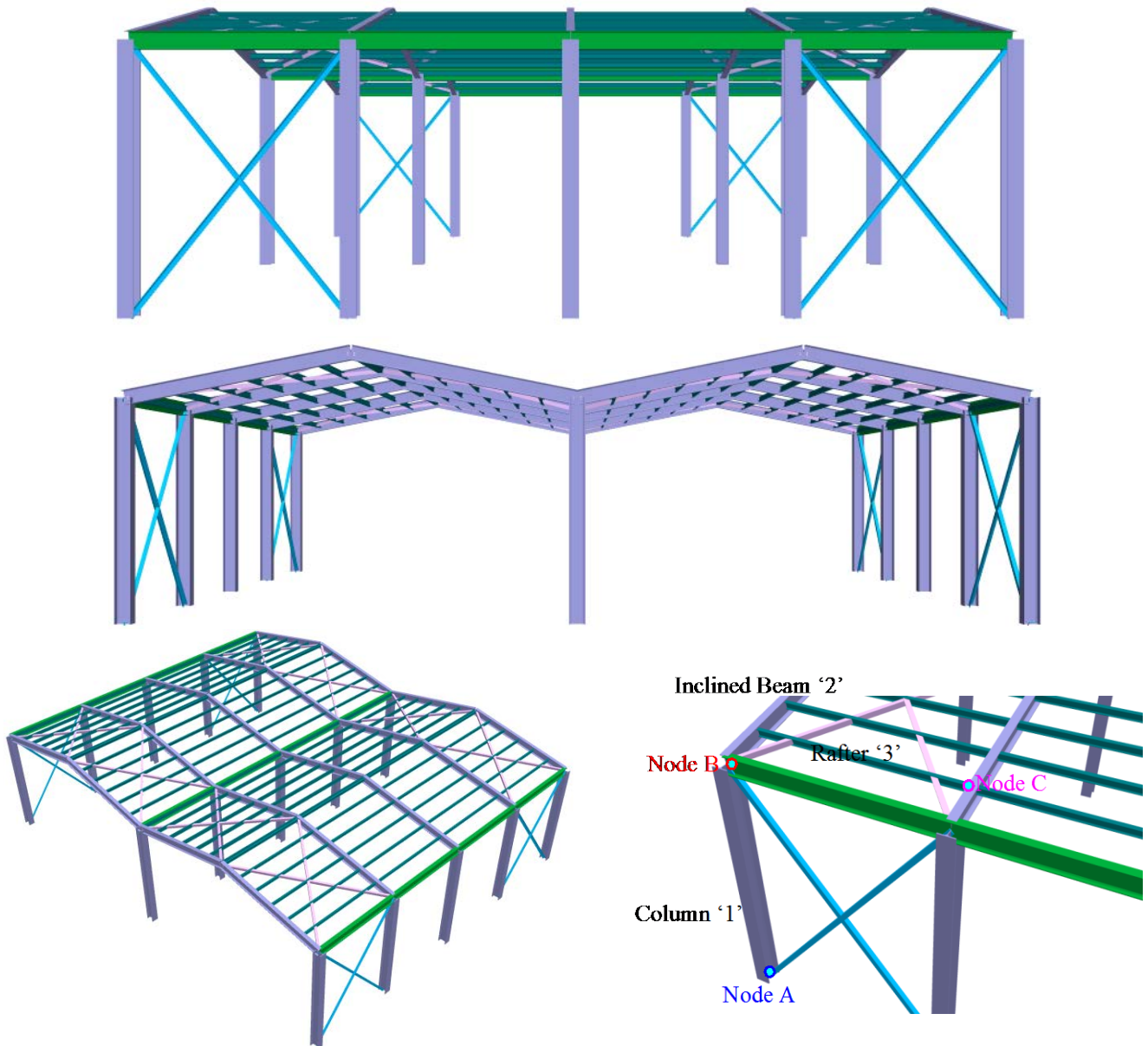

Figure B3.2.1. Three Dimensional Steel Frame Structure.

Geometry, Properties and Loads

The cross section of columns is HEB-360, of inclined and horizontal beams is IPE-330, of rafters is IPE-140 and of stiff joints is LEQ100x10. Modulus of elasticity of the material is 210GPa and Poisson ratio is 0.30. The class of structural steel is S235.

The base of the columns is fixed.

Self-weight is taken into account (ε_s=78.5kN/m^3), rafters are subjected to permanent uniformly distributed vertical load of 2.3kN/m (Figure B3.2.2.a) and columns of one side are subjected to variable uniformly distributed horizontal loads at X-axis of 8kN/m and 4kN/m for middle and outside columns (Figure B3.2.2.b), respectively.

The participation factor of permanent loads is equal to 1.35 while the corresponding one of the variable load is equal to 1.50 (CombA).

Assumptions

During analysis, shear deformations are neglected. Section properties are automatically recovered from the cross section library of ScadaPro.

Technical Features in ScadaPro Tested

> ➢ Three – dimensional analysis of beam elements

> ➢ Verification of steel cross section

Results Comparison

Axial forces and Bending Moments along principal axis of cross section and the respectively maximum degree of utilization for cross-section design obtained from ScadaPro analysis, are presented in Tables B3.2.1 & B3.2.3.

Table B3.2.1. Comparison of Results (CombA)

	ScadaPro	Sofistik
Axial forces N (kN) at Column '1' (node A)	62.94	65.50
Bending Moment My (kNm) at Inclined beam '2' (node B)	69.50	79.37
Bending Moment My (kNm) at Rafter '3' (node C)	5.42	5.14

Table B3.2.3. Results for Maximum Degree of Utilization for Cross-Section Design (CombA)

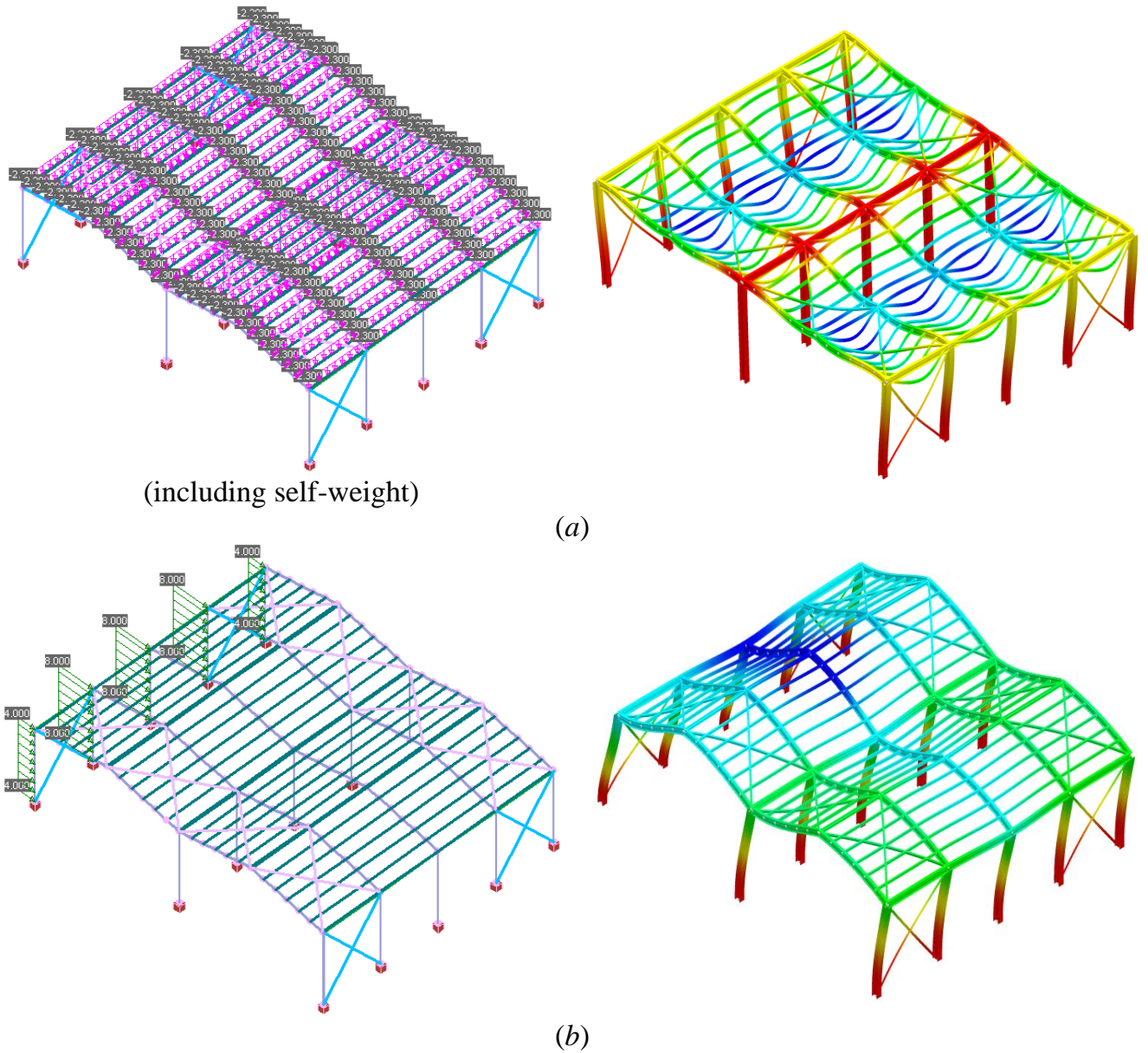| Maximum degree of utilization *n* at	ScadaPro
Column '1'	0.11
Inclined beam '2'	0.36
Rafter '3'	0.44

(including self-weight)

(*a*)

(*b*)

Figure B3.2.2. Permanent (a) and variable (b) distributed load and corresponding deflection lines of the frame structure.

Computer Files

The project file for this example is "B6". These files are provided as part of the ScadaPro installation.

Conclusion

The result comparison reflects acceptable agreement between ScadaPro software and results obtained from Sofistik.

B4. References

[1]. *ABAQUS/CAE 6.11: User's Manual, Online Documentation Help*, Dassault Systèmes USA (2011).

[2]. Argyris, J. and Mlejnek, H.P., *Die Methode der Finiten Elemente*, Vol. I, II, III, Vieweg,Braunschweig (1986, 1987, 1988).

[3]. Bathe K.J., *Numerical Methods in Finite Element Analysis*, Prentice-Hall (1976).

[4]. *CSI Analysis Reference Manual for SAP2000, ETABS, SAFE and CSiBridge*, Computers and Structures, Inc., Berkeley, USA (1995).

[5]. *ETABS: Integrated Building Design Software, User Interface Reference Manual (version 8)*, Computers and Structures, Inc, (2002).

[6]. *ETABS: Version 8: Integrated building design software, Structural and earthquake engineering software*, Computers and Structures, Inc, (2003).

[7]. Gautam Puri, *Python Scripts for Abaqus (Learn by Example)*, Gautam Puri, USA (2011).

[8]. Hibbitt, Karlsson and Sorensen, Inc. *ABAQUS Analysis user's manual (v 6.7)*, Pawtucket, Rhode Island (2007).

[9]. Markus B, *Theorie und Berechung rotationssymetriesche Bauwerke*, Wernre Verlag, Düsseldorf (1967).

[10]. MSC/NASTRAN, *Finite element modeling and postprocessing system*, USA.

[11]. Pilkey W.D. *Analysis and Design of Elastic Beams -Computational Methods*, Wiley, New York (2002).

[12]. *SAP2000: Integrated finite element analysis and design of structures: analysis reference*, Computers and Structures, Inc., Berkeley, (1998).

[13]. *SAP2000: Integrated Finite Element Analysis and Design of Structures: Concrete Design Manual; Steel Design Manual*, Computers and Structures, Inc., (1998).

[14]. *SAP2000: Software Verification Manual*, Computers and Structures, Inc., (2006).

[15]. *SAP2000: Three Dimensional Static and Dynamic Finite Element Analysis and Design of Structures*, Computers and Structures, Inc, (1998).

[16]. Sapountzakis E.J. and Dourakopoulos J.A. Shear Deformation Effect in Flexural-Torsional Vibrations of Composite Beams by BEM, *Journal of Vibration and Control*, 16(12), 1763-1789 (2010).

[17]. Sapountzakis E.J. and Mokos V.G. A BEM Solution to Transverse Shear Loading of Beams, *Computational Mechanics,* 36, 384-397 (2005).

[18]. Sapountzakis E.J. and Mokos V.G. An Improved Model for the Analysis of Plates Stiffened by Parallel Beams with Deformable Connection, *Computers and Structures*, Vol.86, 2166-2181 (2008).

[19]. Sapountzakis, E.J. and Mokos, V.G., Nonuniform torsion of composite bars by boundary element method, *Journal of Engineering Mechanics* (ASCE), 127(9), 945-953 (2001).

[20]. Schueller, W., *Building Support Structures: Analysis and Design Using SAP2000 Software*, Computer and Structures, Inc., (2008).

[21]. Timoshenko, S. and Woinowsky-Krieger, S., *Theory of Plates and Shells,* 2nd ed., McGraw-Hill Book Company, New York (1959).

[22]. Timoshenko, S. and Young, D. H., *Vibration Problems in Engineering*, Van Nostrand, New Jersey (1955).

[23]. Timoshenko, S. and Young, D.H., *Vibration Problems in Engineering*, New Jersey, Van Nostrand (1955).

[24]. Timoshenko, S.P. and Goodier, J.N., *Theory of Elasticity*, 3rd edn, McGraw-Hill, New York (1984).

[25]. Vlasov, V.Z.: *Thin-walled elastic beams*, Israel Program for Scientific Translations, Jerusalem (1961).

[26]. Wilson E.L., Hollings J. P. and Dovey H. H., *ETABS: Three Dimensional Analysis of Building Systems*, National Information Service for Earthquake Engineering /Computer Applications, Earthquake Engineering Research Center, University of California (1979).

[27]. Wilson E.L., *SAP: A General Structural Analysis Program*, Report No. UC SESM 70- 20, Structural Engineering Laboratory, University of California, Berkeley (1970).

[28]. Wilson E.L., *SOLID SAP: A Static Analysis Program for Three Dimensional Solid Structures*, Report No. UC SESM 71-19, Structural Engineering Laboratory, University of California, Berkeley (1972).

[29]. *SOFiSTiK: AQUA - Materials and Cross Sections (v.16.12),* SOFiSTiK AG, Oberschleissheim, (2012).

[30]. *SOFiSTiK: SOFiMSHA - Import and Export of Finite Elements and Beam Structures (v.16.06),* SOFiSTiK AG, Oberschleissheim, (2012).

[31]. *SOFiSTiK: SOFiLOAD – Loadgenerator for Finite Elements and Frameworks (v.15.26),* SOFiSTiK AG, Oberschleissheim, (2012).

[32]. *SOFiSTiK: ASE - General Static Analysis of Finite Element Structures (v.27.07),* SOFiSTiK AG, Oberschleissheim, (2012).

[33]. *SOFiSTiK: MAXIMA - Superposition (v.16.07),* SOFiSTiK AG, Oberschleissheim, (2012).

[34]. *SOFiSTiK: BEMESS - Design of Plates and Shells (v. 27.07),* SOFiSTiK AG, Oberschleissheim, (2012).

[35]. *SOFiSTiK: AQB - Design of Cross Sections and of Prestressed Concrete and Composite Cross Sections (v.13.82),* SOFiSTiK AG, Oberschleissheim, (2012).

[36]. *CEDRUS-5: Manual,* Cubus Hellas Ltd (2004)

[37]. *STATIK-5: Manual,* Cubus Hellas Ltd (2004)