Providing Integrated Structural and Civil Engineering Application using ANSYS Workbench SDK

ESOP System

Pawel Piechnik
Project Manager
Robobat
Overview of the Presentation

• Goal
  - to show simplicity of integrating ROBOBAT civil engineering software with ANSYS Workbench using SDK

• Contents
  - ROBOBAT and its products
  - ESOP system – integrated environment
Robobat

- A software author in the Civil & Structural Engineering market since 1988
  - Sold in excess of 14000 licenses worldwide
  - Employ in excess of 150 staff,
  - Turnover $13M in 2005
  - Interoperable solutions for analysis, code design, modeling and drafting of civil engineering structures
  - Robobat is also an authorized ANSYS reseller
Today we show integration of our software ESOP with ANSYS WB. This solution can also integrate with our other applications:

- **Robot Millennium**
  structural engineering analysis and code design

- **RCAD Steel**
  drafting tool for steel based on Autocad

- **RCAD Concrete**
  drafting tool for reinforced concrete based on Autocad

The integration of ESOP with ANSYS Workbench opens up the complete civil engineering process to the user.
• MS Excel based environment dedicated to creation of advanced engineering calculations

• Environment for Rapid Application Development

• Library of engineering applications
ESOP – Environment for User’s engineering calculations

Editorial tools:
- variables registration and recognition
- units processing
- dynamic tables
- templates
- preferences & personalization
- dedicated controls (hyperlinks, combo boxes, etc …)

Data access:
- „transparent” access to databases (via user controls and formulas)
- Provided databases: profiles, materials, bolts, loads ....
- adding of user databases

Inherited Excel functionality
- Formulas
- Objects
- VBA debugger
- Equations …
### PROTECTED STEEL ELEMENT IN FIRE

Calculation of the evolution of steel temperature and thermal properties at elevated temperature

**Profile:** UB 305 x 102 x 25

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface exposed to fire</td>
<td>0.991</td>
<td>m²</td>
</tr>
<tr>
<td>Cross-sectional area</td>
<td>0.00316</td>
<td>m²</td>
</tr>
</tbody>
</table>

**Thermal properties of steel**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unit mass of steel</td>
<td>7950</td>
<td>kg/m³</td>
</tr>
</tbody>
</table>

**Thermal properties of fire protection material**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thickness of protection</td>
<td>0.016</td>
<td>m</td>
</tr>
<tr>
<td>Unit mass of protection</td>
<td>800</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Specific heat</td>
<td>1700</td>
<td>J/kgK</td>
</tr>
<tr>
<td>Thermal conductivity</td>
<td>0.2</td>
<td>W/m²K</td>
</tr>
<tr>
<td>Section factor for protected steel member</td>
<td>225.0633</td>
<td>m⁻¹</td>
</tr>
</tbody>
</table>

**Nominal steel temperature in time**

- **Exposure time (t):** 30 min
- **Nominal temperature time after time (t):** 374.74

**Graph:** Temperature increase in protected steel element

- **Temperature (°C):** 0 to 1000
- **Time (min):** 0 to 60
ESOP – RCAD environment

- Interface design based on Excel UI, + editorial tools

- Engineering tools
  - FEM grid generator
  - FEM solver
  - HTML note generator
  - 3D viewer

- Databases
  - profiles, materials, bolts, loads ....

- Developer tools
  - project templates (C++, VB)
  - COM communication templates
  - hardware & software protection

- Localization
  - national versions
  - localized databases
  - national codes verification
ESOP – Library of Applications

• More than 400 commercial modules

• Current domains of application:
  - static analysis
  - dynamic analysis
  - steel design
  - RC design
  - timber design
  - masonry design
  - strength of materials
  - physics
  - geotechnics

• Localization:
  - 5 national versions
  - localized databases
  - national codes verification
ESOP – FEM solver – Slab analysis

- advanced calculations
- simplicity of use
ESOP – Static analysis

Bar structures

Shells, Plates

Other

Sections
ESOP – Steel

Connections

Verifications
ESOP – Reinforced Concrete

Stairs

Cantilevers, punching

Floors
ESOP – Timber

- parameterization
- static analysis
- code verification
ESOP – Masonry

Walls
ESOP – Vertical solutions

Wave Loader for Robot Millennium
- load generation
- parameterization and visualization
ESOP in ANSYS Workbench

- ESOP as a part of ANSYS WB Projects
  - ESOP ActiveX inside ANSYS WB
  - integration by ANSYS SDK
  - automated installation

- Communication with ANSYS
  - based on COM interfaces
  - models generation
  - data and results analysis
  - automation for ANSYS
  - additional user calculations
ESOP in ANSYS Workbench
ESOP in ANSYS Workbench

- Multi-span beam - static calculations

Beam:
section: W 24x76
material: STEEL

- Moment of inertia with respect to y-axis
- Young modulus

<table>
<thead>
<tr>
<th>Supports positions</th>
<th>[m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$x_1$</td>
<td>4.0</td>
</tr>
<tr>
<td>$x_2$</td>
<td>8.1</td>
</tr>
<tr>
<td>$x_3$</td>
<td>16.2</td>
</tr>
<tr>
<td>$x_4$</td>
<td>30.0</td>
</tr>
</tbody>
</table>

Load:
Concentrated forces:
- No. 1: 244.7 [kN] at [m] 16.3

Forces - uniform load:
- No. 1: 71.0 [kN/m] at [m] 16.3
Welded plate – simulation and code verification
**Verification**

**Bearing on the concrete:**

\[ F_p = 14.0 \text{ [MPa]} \]

Concrete allowable bearing strength

\[ f_y \leq 0.19 \times f_p \]

\[ A = 0.47 \text{ [mm]} \]

Concrete bearing distance

\[ A = 1.5 \left( 1 - \frac{1}{1 - 2.66 \frac{P \times A + M^2}{f_p \times b^2 \times (N^2)_0^2}} \right) \]

**Bending of the base plate by concrete bearing**

\[ F_{bh} = 198.0 \text{ [kN]} \]

Allowable bending stresses

\[ M_{pl} = 14579.3 \text{ [N.m/m]} \]

Base plate bending moment

\[ t_{p,req} = 20.86 \text{ [mm]} \]

Required base plate thickness

\[ t_{p,req}^2 \leq t_p = 50.80 \text{ verified - ratio = 0.41} \]

**Bending of the base plate by anchor bolts**

\[ T = 179.9 \text{ [N]} \]

Total tension force in anchor bolts

\[ T = \frac{F_p \times A \times B}{2} - P \]

\[ T_s = 179.9 \text{ [N]} \]

Tension force in single anchor bolts

\[ t_{p,req} = 1.86 \text{ [mm]} \]

Required base plate thickness

\[ t_{p,req}^2 \leq t_p = 50.80 \text{ verified - ratio = 0.03} \]
Bolted tube joint – simulation and code verification

**Tubes connection**

<table>
<thead>
<tr>
<th>Code</th>
<th>Proprietary© 2006 ANSYS, Inc.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status</td>
<td>not verified</td>
</tr>
<tr>
<td>Ratio</td>
<td>0.62</td>
</tr>
</tbody>
</table>

**Forces**

- $N = 200,000$ [kN] Axial force

**End plate**

- $h = 220$ [mm] Height
- $t = 20$ [mm] Thickness of the plate

**Section**

- $R = 20$ mm Thickness

**Materials**

- STAL 18G2 385
- $f_y = 305,0$ [MPa] Design resistance

**Geometrical verification**

- $2t = 6a$ Total plate thickness
- $4h > 6a$ not verified
- $7,500 < 55,000 < 120,000$ verified
- $58,000 < 56,000 < 36,000$ verified

**Bolts’ resistance**

- $S_w = 3.6$ [MN] Design resistance of bolts
- $S_{cy} = 0.95 S_w$ (Table 17)

**End plate thickness**

- $c = 58$ [mm] Distance between the hole and the weld edge
- $d = 126$ [mm] Width of the end plate connection
- $t_{min} = 4$ [mm] Minimal end-plate thickness
- $20,000 > 3,900$ verified

**Tension capacity**

- $\beta = 1.0$ Leverage effect coefficient
- $\beta = 2.67 - \frac{1}{\psi}$ (84)
Model generation in ANSYS WB

Simulation

Static analysis
Nearest Future:  
ESOP for ANSYS Workbench

- Creation of complementary user calculations and modules cooperating with ANSYS WB, in MS Excel:
  - Getting information from ANSYS WB with use of MS Excel formulas
  - Library of VB procedures for ANSYS WB management
Nearest Future: ESOP for ANSYS Workbench

• Easy getting information from ANSYS WB:
  - Set of wizards, enabling:
    - easy question definition
    - generation of Excel formulas, getting data from ANSYS
  - Reach library of wizards/formulas

=AWB_GET_STRESS(""; "MAX", "VAL", "MISES", "Temperature")
Welds verification

\[
\begin{align*}
\text{Data:} & \\
\theta &= 6.0 \text{ [mm]} \quad \text{Weld's thickness} \\
L &= 100.0 \text{ [mm]} \quad \text{Weld's length} \\
\alpha &= 2
\end{align*}
\]

Coefficient:

\[
\alpha = 0.93 \quad \text{Reduction coefficient}
\]

Material: S275

\[
f_r = 275.0 \text{ [MPa]} \quad \text{Resistance}
\]

Efforts:

\[
\begin{align*}
N &= 1000.0 \text{ [N]} \quad \text{Axial force} \\
T &= 1000.0 \text{ [N]} \quad \text{Shearing force}
\end{align*}
\]

Angle:

\[
\phi = 45.0 \text{ [Deg]} \quad \text{Angle between the axes}
\]

Verifications:

\[
\begin{align*}
F &= 707.1 \text{ [N]} \quad \text{Effort taken by the formulas}
\end{align*}
\]

\[
\begin{align*}
\sigma &= 1.5 \quad f_r = 275.0 \text{ [MPa]} \\
\sigma &= 1.6 \quad f_r = 275.0 \text{ [MPa]}
\end{align*}
\]

\[
\begin{align*}
\sigma &= \frac{F}{A} \\
\sigma &= \frac{0.75 + 0.1 \cos \theta}{2} \quad \text{Valid} \\
\sigma &= \frac{F}{2 \sin \phi} \quad \text{Valid}
\end{align*}
\]

Selection from database:

=AWB_Get_Parameter(„THICK“)

Values from databases:

=AWB_Get_Parameter(„FX“)

Units processing (automatic)
Nearest Future: ESOP for ANSYS Workbench

- Easy setting information in ANSYS WB:
  - Set of wizards, enabling:
    - easy question definition
    - generation of VB procedures, setting information in ANSYS
  - Reach library of wizards/procedures

```vba
Public Sub SetForces(Fx As Double, Fy As Double, Fz As Double)
  Set EsopOS = CreateObject("EsopOS.Application")
  Set WB = EsopOS.AnysysWB
  Set DS = WB.AppletList.Applet("USApplet").App

  Dim E As DSEnvironmentAuto
  Set E = DS.Tres.Projects(1).Models(1).Environments(1)

  Dim S As DSSelectionLib.IWESelection
  Set S = DS.SelectionManager

  Dim L As DSLoadAuto
  Set L = E.AddLoad(S, _id_VertexForce)
  L.DefineBy = Components
  L.ComponentX = Fx
  L.ComponentY = Fy
  L.ComponentZ = Fz
End Sub
```
Nearest Future: ESOP Expert SDK (also in Ansys WB)

- C# templates in .Net environment for quick development of advanced engineering applications

- Providing ready-made functional objects and templates like:
  - Frame container (windows, toolbars, ...)
  - 2D / 3D Graphical Viewer
  - HTML note generator
  - HTML note viewer and composer
  - Dedicated editorial controls ...
  - Serializing and saving mechanism (XML format)
  - Databases access
ESOP System is the environment allowing for easy creation of complementary functionalities and calculations for ANSYS products, by end-users and 3rd party companies.