CivilFEM: Extending ANSYS Capabilities in Civil Engineering

The state-of-the-art technology for the civil engineering world

Dr. Dimitrios Sofialidis
Technical Manager
SimTec
ANSYS+CivilFEM is the most advanced, comprehensive and reputable finite element analysis and design software package available for the structural engineering projects.

It combines the state-of-the-art general purpose structural analysis features of ANSYS (ISO-9001) with high-end civil engineering-specific structural analysis capabilities of CivilFEM, making it a unique and powerful tool for a wide range of civil engineering projects.

As both programs are completely integrated, CivilFEM supports all types of advanced analysis supported by ANSYS running as a unique software and executable. ANSYS is the finite element software used by 90 of the TOP100 engineering firms in the world with 30 years of constant growing (NASDAQ ANSS).
CivilFEM INTRO

**CivilFEM** using **ANSYS** pre, post and solving capabilities, adds more than **400 new features** and specific utilities for the Civil engineering field.

**CivilFEM** can be purchased as "add-on" to any **ANSYS** product.

<table>
<thead>
<tr>
<th>ANSYS PREPROCESSOR</th>
<th>PREPROCESSOR</th>
<th>ADITIONAL OR COMPLEMENTARY ANALYSIS</th>
<th>CivilFEM INTRO + Specific Modules</th>
</tr>
</thead>
<tbody>
<tr>
<td>SOLUTION</td>
<td>POSTPROCESSOR</td>
<td><strong>CivilFEM</strong> (Features added to Ansys by CivilFEM)</td>
<td></td>
</tr>
</tbody>
</table>
Positioning and Advantages

- Ansys+CivilFEM goal is to be a technological leader and different product with unique capabilities that is able to change the construction/civil engineering design and technological process.
- It means a new culture in this industry.
- Product according new/emerging advanced high end customer needs.
- Only one product for most of customer needs and departments.
CAE for Civil Engineering

Only one software to cover all the industry needs and requirements.
ANSYS+CivilFEM Q&A

• **Q.A. (Quality Assurance):**
  CivilFEM provides a similar Q.A. than ANSYS Inc.

• **Q & A includes:**
  a) Error reporting.
  b) Service Packs: bugs are fixed apart of including some more new features.

In each new release, we must check the verification example set supplied by ANSYS Inc. (400) plus the ANSYS/CivilFEM ones (800).

Total around 5,000 verification tests including the original ANSYS verifications in Pittsburgh.
Typical Projects for ANSYS+CivilFEM

- Industrial **buildings**, high rise buildings and sport stadiums.
- **Seismic** calculations.
- Nuclear, wind and thermal **power plants**.
- Off–shore and **marine** structures.
- **Bridges** (concrete, steel, cable, etc).
- Prestressed and non linear concrete structures.
- **Tunnels**.
- **Foundations** (slabs, piles, walls, etc).
- **Geological** and **Soil** mechanics problems.
- **Dams** (concrete, earth, etc).
- Cable structures, special buildings, etc.
- **Quality control, forensic structural analysis, project modifications.**
H18: Pretensado de 1m en cables de retenida
ANSYS+CivilFEM Projects
CivilFEM INTRO Module

www.civilfem.com/intro
CivilFEM INTRO main features

- 32 and/or 64 bit and parallel processor
- Material library & Section library
- Easy Beam & Shell Postprocessor
- Smart Load Combination tool and algebraic combinations
- Code Checking & Design
- Shell/Solid Reinforcement
- Seismic tools
- Composite sections

- Export–Import of solid & beam sections
- Connection with Robot, SAP2000, DXF and Lidar
- HTML and Excel connection
- Integration with FLAC\textsuperscript{3D} (advanced soil mechanics)
CivilFEM Setup Options

Most international codes implemented

Intuitive definition of any units system
Steel Codes
Reinforced Concrete Codes
Prestressed Concrete Codes
Seismic Codes
CivilFEM Material Library

ANSYS & CivilFEM material editing from one window

Automatic updating of time dependent material properties

Generation of user-defined material library
CivilFEM Types of Materials

• Structural steel
• Concrete
• Reinforcing Steel
• Prestressing Steel
• Soil
• Rock
• User
CivilFEM Material Library

A complete library of materials:
- CEB-FIP Model code
- Eurocode No.2
- Eurocode No.3
- ACI 318 - 05
- AISC
- British Standard
- UNE
- ASTM
- EHE
- EA
- Chinese code GB-50010
- Chinese code GB-50017
- Brazilian Code NBR6118
- Indian Standard 456
- Russian SP 52/1001/2004
User Material Library

The material properties are stored in a file named by the user.

Possibility of adding user databases in CivilFEM’s material library.
CivilFEM Material Library

User defined material name

Birth time of materials

Death time of materials
CivilFEM Material Library

<table>
<thead>
<tr>
<th>Name</th>
<th>Reference</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fe 360</td>
<td>1</td>
<td>Structural steel</td>
</tr>
<tr>
<td>AC30/37</td>
<td>2</td>
<td>Concrete</td>
</tr>
<tr>
<td>S400</td>
<td>3</td>
<td>Reinforcing steel</td>
</tr>
<tr>
<td>Y1570C</td>
<td>4</td>
<td>Prestressing steel</td>
</tr>
<tr>
<td>GGC</td>
<td>5</td>
<td>Soil</td>
</tr>
<tr>
<td>Molina</td>
<td>6</td>
<td>Rock</td>
</tr>
</tbody>
</table>

Non-Linear Materials

With time-dependent material properties
Simulation of Construction Process

New features for the study of **evolutive construction process analysis** (material birth and death)

- Distinguishing characteristic of CivilFEM: Time of birth.
- Construction Process through material definition (activation and deactivation time).
CivilFEM Material Library

Visualization of time-dependent material properties
CivilFEM Material Library
Advanced Concrete Modeling

- Interface for SOLID65 element (non-linear reinforced concrete element) properties definition
CivilFEM Material Library

Advanced Concrete Modeling

Simplified and Accurate Creep Modeling

a) Step by step method
b) Effective modulus method

- American Code ACI 209R-92
- EHE
- CEB -90 CODE
- EUROCODE 2
CivilFEM Material Library

Stress-strain curve for structural behavior

User modification and graphical control of stress-strain relationships
CivilFEM Material Library

Structural Steel

Thickness dependent diagram and properties
CivilFEM Material Library

Soils and rocks library

- Library with mechanical, elastic and plastic properties of characteristics soils and rocks (around 100).
- Definition, editing and modification of properties not contemplated by ANSYS (Atterberg limits, Hoek & Brown coefficients, etc).
- Correlations among geotechnical parameters from tests results (elasticity module versus SPT, etc)
CivilFEM Sections Explorer

Definition of any type of cross sections using a single window

Compose cross sections by merging
Hot Rolled Shapes Library

“Library of more than 4,000 shapes”

Quick search of adequate shape

Addition of user defined hot rolled shapes

Properties:

- $A = 7.80E-03 \text{ m}^2$
- $I_{xx} = 591.800E-09 \text{ m}^4$
- $I_{yy} = 20.030E-06 \text{ m}^4$
- $I_{zz} = 56.960E-06 \text{ m}^4$
- $W_{yy} = 236.300E-06 \text{ m}^3$
- $W_{zz} = 559.600E-06 \text{ m}^3$
- $W_{xy} = 395.800E-06 \text{ m}^3$
- $W_{yz} = 642.500E-06 \text{ m}^3$

Dimensions:

- $h_1 = 170.000E-03 \text{ m}$
- $r_1 = 18.000E-03 \text{ m}$
- $r_2 = 0.000E-03 \text{ m}$
- $d = 134.000E-03 \text{ m}$
Hot Rolled Shapes Library

Up to 3 properties with maximum and minimum values can be specified in the search

Automatic search of appropriate shape
Introduction of user’s own hot rolled shapes library into CivilFEM library by reading a file or by menu

Just define the shape and dimensions of the section
User Hot Rolled Shapes Library

Mechanical properties are automatically calculated
Steel Sections by Plates

Any generic cross section shape can be defined

User friendly definition of plate properties that forms the section
Steel Section Edition

• You may define a section reading its properties from library
• Then you can modify its dimensions redefining this section as "section by dimensions or by plates".
• And you can even edit its plates to transform the section from an existing one (changing dimensions).
Sections by dimensions

Automatic definition of section shape

Automatic definition of preliminary reinforcement

Helpful information about the section geometry
Capturing 2D Sections

An easy way to define any generic composite beam section from a 2D ANSYS meshed drawing - each element corresponds to a tessella.

Each tessella adopts initially the material assigned to its corresponding element.

You can change the material of any tessella.
User Database of Cross Sections

The cross section properties are stored in a file named by the user.

Properties of the cross section are read and listed in the active code and unit system.
Section Merging

• It’s possible to create a section as a composition of two existing sections.
• The merged section will take into account all the properties of the two initial sections.
Variable Cross Sections

Cross section number

Tapered beams

Real constant list
Import Geometry (I)

Improving the capabilities to import external geometry, includes *translators*, which permits the direct generation of ANSYS/CivilFEM models from:

- DXF files exported by the majority of CAD softwares used by the Civil Engineering.
- Terrain surfaces in LiDAR ASCII_GRID (SIG).
Patterns and Ensembles

- The new Pattern concept introduced in CivilFEM incorporates an own algebra of structures design.

- By means of the definition of Patterns or cell-structures, it is possible to make complex beam models.

- The Patterns algebra allows the handling of more complex structures (Ensembles) that can in turn be further combined using the same logic or algebra.
Standardized Structures (I)
Standardized Structures (I)
Standardized Structures (I)
Standardized Structures

Truss Catalog
Standardized Structures (II)

**Patterns standardized**

Base: any regular polygon

Predesign of lateral pattern faces

The *Pattern* can be totally personalized
Standardized Structures (III)

Generation of open Ensembles

Pattern
Standardized Structures (IV)

Generation of closed Ensembles
Standardized Structures (V)

Ensembles operations

- Ensemble to Ensemble copying, moving, rotating and scaling
- Ensemble to Solid linking

180°
Concrete Shell Reinforcement

- Variable depth
- Variable reinforcement amount in X & Y directions and in Top & Bottom faces for each vertex
- Non orthogonal reinforcements and variable orientation
- $T_X, T_Y, T_{XY}, M_X, M_Y, M_{XY}, N_X, N_Y$ Reinforcement
Other features

- Surface generation using variable thickness shell
- Methods: Wood Armer, CEB, Virtual frames (Axial + Bending Method), Cappra*
Shell Bending Design

An icon shows the reinforcement location
CivilFEM Load Combinations

- Problem example 1: Loads in building
  - What is the maximum moment in section A-A?
  - Where should the variable load be located?
  - Should the wind blow from right to left or from left to right?
  - Is the dead load favorable or unfavorable?
CivilFEM Load Combinations

- Problem example 2: Mobile loads
  - Where must be located the two engines to obtain the maximum stresses at point P?
  - What is the maximum bending moment at section A-A?
  - What are the concomitant values?
Load Combination Explorer

• Any load combination scenario can be quickly and easily simulated by using this window.

You can be sure of covering all the possibilities
CivilFEM Load Combinations

(Example of the smart combination algorithm application)

For any load distribution over the structure

To obtain the envelop of maximum vertical displacements at all nodes
Beam & Shell Utilities

- Automatic output of forces and moments
- Direct plot and list of results
  - Forces
  - Moments
  - Stresses
  - Strains
- HTML and Excel format for listing
- Stress and strain distribution inside any ANSYS beam cross section
Capturing Solid Sections

Automatic generation of cross section

It allows checking & design of beam cross sections captured from a 3D solid model

ANSYS 3D model
Forces and Moments

- Beam Utilities
  - Graph Results
    - Forces & Moments
    - Stress & Strain
    - Section Results
  - List Results
    - Forces & Moments
    - Stress & Strain
- Results title automatically displayed
- An icon showing sign criterion and results description is always displayed

ANSYS 10.0A1
NOV 14 2006
13:42:04
LINE STRESS
STEP=3
SUB =2
TIME=22
CFETAB_ICFETAB_J
MIN =-69977
ELEM=963
MAX =70290
ELEM=983
-69977
-54392
-38807
-23222
-7636
7949
23534
39119
54705
70290

B215 EVAP D

INTEGIBER, S.A.

CivilFEM®
Stresses and Strains

The icon shows the actual section and the stress point location.
Connection with MS-EXCEL

- Any array parameter defined in ANSYS/CivilFEM.
- CivilFEM utility for exporting families of arrays.
- Automatic graphic generation in Excel.
- Configuration options of Excel graphics from CivilFEM windows.
Export Results in HTML

HTML export utility

• It’s possible to easily create a table in HTML from any Ansys or CivilFEM result
Export Results in TXT

** ANSYS **
Release 10.0
** civiFEM **
Wednesday 18/11, January 25, 2006

FORCES AND MOMENTS LIST
Load Step 1, Substep 1, Alternative 3

BEAM ELEMENTS FORCES AND MOMENTS

<table>
<thead>
<tr>
<th>Element</th>
<th>End</th>
<th>AXIAL FORCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>I</td>
<td>-102.714E+03</td>
</tr>
<tr>
<td>2</td>
<td>I</td>
<td>102.714E+03</td>
</tr>
<tr>
<td>3</td>
<td>I</td>
<td>-488.144E+03</td>
</tr>
<tr>
<td>4</td>
<td>I</td>
<td>-104.167E+03</td>
</tr>
<tr>
<td>5</td>
<td>I</td>
<td>104.167E+03</td>
</tr>
<tr>
<td>6</td>
<td>I</td>
<td>-208.333E+03</td>
</tr>
<tr>
<td>7</td>
<td>I</td>
<td></td>
</tr>
</tbody>
</table>
Beam Sections Results

Stress and strains inside beam cross sections of any beam created in CivilFEM, thanks to the discretization of the cross sections into tessellas.
Concrete Checking and Design

• **Codes**
  - EUROCODE 2
  - ACI 318
  - EHE
  - CEB-FIP (Model Code)
  - British Standard 8110
  - Australian code (AS 3600)
  - Chinese code (GB-50010/ GB-50017)
  - Brazilian Code NBR6118
  - Indian Standard 456
  - Russian SP 52/1001/2004

• **Capabilities**
  - Axial+Biaxial Bending Interaction Diagram (3D)
  - Axial+Biaxial Bending Checking
  - Axial+Biaxial Bending Design
  - Shear & Torsion Checking
  - Shear & Torsion Design
  - Shell Reinforcement
CivilFEM checks the structure with the initial reinforcement amount and shows the following results for each element:
- Elements that are **OK** and **NO OK** according to code specifications
- Safety factors
Axial Biaxial Bending Design (3D)

- In the design process, all the scalable reinforcements (defined by the user) are multiplied by an optimization factor $\omega$ that makes the safety factor/ratio of the section as close as possible to 1.00

- The $\omega$ factor is searched in a range of values specified by the user

  \[ \omega_{\text{min}} < \omega < \omega_{\text{max}} \]

- The results of a reinforcement design with CivilFEM are the obtained reinforcement amount and the $\omega$ factor for each element end
Cracking Checking

- CivilFEM automatically checks the structure against cracking according to codes.

- Code Checking
  - ACI
  - Check by code
    - Beams & Solid
      - 2D Axial + Bend
      - 3D Axial + Bend
      - Shear & Torsion
    - Cracking
    - Decompression
  - Shells
  - Design by code
  - Utilities
  - Beam Results
Steel Checking

- Codes
  - EUROCODE 3 (European)
  - EA (Spanish)
  - AISC-LRFD (American)
  - British Standard 5950:1985
  - British Standard 5950:2000
  - Chinese Code GB50017-2003
  - Others
Eurocode 3 Checking

• **Types of checking**
  - Tension (1D)
  - Compression (1D)
  - Bending (2D)
  - Shear (2D)
  - Bending+Shear (2D)
  - Bending+Axial (3D)
  - Bending+Axial+Shear (3D)
  - Compression Buckling (1D)
  - Lateral Buckling (2D)
  - Lateral Buckling in Bending+Tension (3D)
  - Buckling in Bending+Compression (3D)
Seismic Analysis

- Utilities to make easier the seismic spectral analysis:
  - Spectrum definition according to codes (Spanish, Eurocode 8, Italian, Chinese GB50011 and others under request)
  - Automatic mode combinations according to code specifications (SRSS, CQC)
  - CFRAPPN seismic sign utility

Simply define the spectrum and select the number of modes and the seismic analysis will be extracted.
Seismic Analysis

The treatment of response spectrum has been rewritten and six new codes have been incorporated:

- AASHTO
- CALTRANS Seismic Design Criteria
- French code PS 92
- Greek code EAK 2000
- Indian code IS 1856
- Uniform Building Code 1997
Seismic Analysis: Ground acceleration

EAK 2000
Greek Code for Seismic Resistant Structures

Seismic Risk Zones
- Zone I
- Zone II
- Zone III
- Zone IV

EAK 2000 Greek Code for Seismic Resistant Structures
Risk Zones: [ ]
Seismic Analysis

Push-Over and Retrofit Analysis

From the random spectrum definition, it is possible to perform a *Push-Over* calculation which allows:

- Analyze the evolutive behavior of the structure
- Determine its weak points.
- Analyze the *Retrofit* effect.
Seismic Analysis

SOIL STRUCTURE INTERACTION
For more information, you may contact:

Dr. Dimitris Sofialidis
SimTec Ltd.
Technical Manager
dimitris@simtec.gr

www.simtec.gr