ANSYS+CivilFEM: High-End Solution For Advanced Civil Engineering Projects

Miguel Ángel Moreno
Ingeciber S.A.
Enrique Monteagudo
Ingeciber S.A.
Isabella Maia
Ingeciber S.A.
Madrid, Spain

Abstract:
CivilFEM performs the best customization of the well-known Finite Element Program ANSYS. The combination of both programs, totally integrated, provides the Construction and Civil Engineering fields with the possibility of applying high-end technology to a wide range of projects. Using the same windows graphic user interface and sharing input data and results, makes it very easy for the user to apply them for solving difficult Civil Engineering problems. The ability to generate finite element models of any complex three-dimensional civil structure with non-linear behavior and construction process simulation, means a new and efficient approach to run advanced analysis on PC’s.

Introduction
CivilFEM is at the present time one of the most advanced tools that engineers can embrace, a project that is committed with a time and with a permanent vocation of investigation and development.

CivilFEM capabilities include a unique and extensive materials and sections library for concrete and steel structures. In addition, the user may introduce any shape or material into the corresponding CivilFEM libraries. A user-friendly beam and shell postprocessor includes listing and plotting section geometry, reinforcements, beam results or stresses and strains inside the cross-section.

The skilled combination module, selects loads and coefficients for logic code combinations. Results embrace concomitance at element and global level as well as worst load arrangements in beam, shell and solid elements.

In the field of Seismic Analysis, the designer is provided with an automatic seismic design module according to Eurocode No.8, UBC, NCSE-94 (Spanish Code) and others to make the seismic spectrum analysis faster and easier.

CivilFEM also performs concrete and steel code checking and design according to Eurocodes (EC2 and EC3), American (ACI318 and AISC-LRFD), British (BS8110 and BS5950-1995 and 2001), Spanish codes (EHE and EA), Model Code (CEB-FIP) and other main codes. With regards to serviceability, CivilFEM conducts for the aforementioned codes cracking analysis.

CivilFEM specialized add-on modules include non-linear concrete, soil mechanics, prestressed concrete, bridges and dams features. The non-linear concrete module supports large deflection buckling, non linear deformations, moment-curvature diagrams, cracking and transient effects. For soil mechanics, slope stability analysis, libraries of soils and rocks properties, screens analysis and design, nonlinear behavior analysis and ballast module computation, will allow the user to carry out advanced geotechnical problems.

The features and prestige of ANSYS in the finite element world, combined with CivilFEM's capabilities and specialized modules, translates into a fully integrated state of the art Civil Engineering working tool.
This advantageous and unique bundle product represents the possibility of applying the analysis with finite elements to a wide range of problems according to the hardships and requirements of this field.

**CivilFEM Integration with ANSYS**

CivilFEM works inside Ansys program. That is, all Ansys tools may be used with CivilFEM (APDL, UIDL, optimization, graphical output, ...), the CivilFEM menus are integrated inside the Ansys Main Menu, CivilFEM Help is managed as Ansys Help System, CivilFEM commands are generated by CivilFEM menus and written to the Ansys log file, and so on. Ansys results are stored in CivilFEM results file in addition to data related to the cross section. You may switch between Ansys and CivilFEM processors at any time, and mix commands from both to generate, solve and postprocess your model easily.

CivilFEM implementation is based in the use of dynamic link libraries (DLLs). These libraries contain the CivilFEM code and are incorporated into Ansys executable file in execution time, working as if they were part of Ansys. This structure allows to connect with the DLLs by means of ANSYS external commands. These commands are declared in file ans_ext.tbl CivilFEM menus are elaborated using Ansys UIDL features. All CivilFEM windows are written in Visual C++ and Dll’s in Fortran 95.

Like Ansys, CivilFEM may work in any system of units. However, the user must specify the units system before any CivilFEM calculation is done, since specific code formulations and CivilFEM tools depend on units. A library with the most usual (American and European) units in Civil Engineering is available. Furthermore you may also define any other unit system (not available in the library) by selecting the "User Defined Units" (See Figure 1)

*Figure 1 - CivilFEM units library*
Civil Material Library

CivilFEM defines the mechanical properties required by Ansys and the specific properties needed for code checking of usual civil materials, in accordance with code specifications. This library includes materials from most usual standards. Nevertheless, the user is provided with the necessary tools to add materials into this library. Using this tool, modified materials or new materials from any code may be included. CivilFEM material's library has five classes of properties for all CivilFEM materials:

- General properties: Common to all the materials.
- Structural analysis diagram: stress-strain diagram used when solving the model.
- Design diagram: stress-strain diagram used when checking or designing a cross section.
- Specific properties depending on the material: steel, concrete, soils, rocks,…
- Specific properties depending on codes: EC2, EC3, ACI, BS8110, BS5950-2001, LRFD,…

Moreover, user materials may be saved in CivilFEM’s material library (See Figure 2). Material properties in CivilFEM are time dependent (See Figure 3), which means that the user can solved the same model at different ages. For each one of these scenarios, materials used in the model will be active or not, depending upon the age of the material. The program will calculate the material age by subtracting from the age solicited the birth time of the material. With this capability, the user can simulate construction processes, not only at element level but inside the cross section as well.
Figure 3 - CivilFEM time dependent material properties

For the non linear structural behavior, a graphical control of stress-strain curves enables the user to select the graph that best represents the different types of loading of the structural analysis. The user may also add or delete points, or simply create his/her own curve (See Figure 4). Different relationships between stresses and strains are also provided for sections design, with the same possibilities as for the structural stress-strain curve.
Section Library

All cross sections in CivilFEM are divided automatically into points and tessellas. A tesella is a subdivision of the cross section and each tesella’s vertex is a point. This particular characteristic of CivilFEM’s cross sections, allows the program to analyze and provide results inside the cross section. Cross sections in CivilFEM can be defined using eight different procedures:

1) The library of hot rolled shapes:
   This library contains the most international sections libraries, such as hot rolled shapes distributed in Europe by ARBED, as well as all the hot rolled shapes included in the AISC LRFD (over 4000 hot rolled steel shapes are included). These last two series include all shapes groups in metric and U.S. units.

2) User library of hot rolled shapes:
   CivilFEM enables the user to add more hot rolled shapes into the existing library. This capability is two folded. The user may either read mechanical or geometric properties from a file or introduce the geometry corresponding to the selected hot rolled section using a specific CivilFEM window. In the latter case, CivilFEM will automatically calculate the mechanical properties for the section entered by the user (See Figure 5). Using this tool, shapes from any country may be added.
3) Definition of sections by dimensions:

The most usual welded steel sections as well as the most usual concrete sections may be defined by dimensions. In the case of reinforced concrete, a preliminary reinforcement may be defined using the predefined options offered. Nevertheless, user reinforcement layout for bending, shear or torsion is also available in CivilFEM.

4) Definition of sections by plates:

Each plate adopts the active material. This feature allows defining a generic steel section built up with 2 to 100 plates with the possibility of having different materials. For each plate, the ends coordinates, thickness and restraint conditions need to be specified.

5) Definition of sections starting from a 2D mesh of virtual elements MESH200:

This feature allows the definition of sections with any shape and even with mixed materials (different concrete strengths or concrete with structural steel) using Ansys elements MESH200. Each teselum (subdivision of the cross section) will adopt the material assigned to the corresponding element once the cross section is captured by CivilFEM. Export of CivilFEM sections into Ansys are also possible.

6) Definition of beam cross section inside a 3D solid model of finite elements:

Each teselum adopts the material from the finite element to which is associated. The SOLID SECTION (cross section coming from a 3D ANSYS model) and the CROSS SECTION (transverse section as commonly understood in engineering) are defined simultaneously and their numbers coincide. This capability allows to build and solve a complete 3D solid model of the structure, and then select sections inside this model to be processed by CivilFEM. This feature allows code checking and design of all user-defined sections of 3D beam, shell or solid finite element models.
7) Composition of sections:

Any section defined by any of the above means can be combined with any other concrete or steel section and form a new section (See Figure 6).

![Figure 6 - CivilFEM section composition](image)

8) User data base of sections:

Any section defined by any of the above means can be stored in a user data base for later use.

CivilFEM uses its Beam & Shell property window to easily select the section for element ends i and j. The corresponding element type must also be entered in this window, and the real constants will automatically be calculated by CivilFEM and sent to ANSYS (See Figure 7).
Load Combination

The smart combination module of CivilFEM is able to select which loads and coefficients must be combined to reach the worst condition at any node or element of the structure. All possible load combinations and coefficients are treated independently for every node and element of the model. That is to say, for different nodes or elements the worst load combination and coefficient selected by CivilFEM may differ. Only the targets (any ANSYS or CivilFEM result to maximize or minimize) and the combination rules must be specified. It has been developed specifically for solving complex code combinations for civil engineering needs. Furthermore, CivilFEM also provides the user with all the concomitant values of the chosen target. The hexagon (Shell63) shown in the figure, (See Figure 8) is the solution given by CivilFEM when asked for the pressure load to maximized negative displacement in a particular corner. We have selected the CivilFEM option Compatible (which means that the load can be place in all, any or none of the model elements) and the displacement at the chosen node as the result to be maximize by CivilFEM (target). The worst load configuration possible to achieve the target, the contribution of each loaded element to the total displacement as well as all the displacement concomitant values are provided by CivilFEM.
**Figure 8 - Displacements on shell for load configuration producing Uzmax at corner**

**Beam & Shell Utilities**

This capability includes tools to postprocess beam and shell elements. CivilFEM provides the user with automatic lists and plots of section’s geometry, specific code properties, code checking results or strength properties through Ansys graphics window or using specific CivilFEM windows.

Moreover, CivilFEM plots automatically forces and moments graphics for beam and shell elements in addition to stresses and strains, always including a legend and an icon with information about the direction and sign convention of the results shown. For stress plots, the user may specify among the many points defined inside CivilFEM cross sections the one for which the stress must be displayed.

For Ansys beams, pipes or spar elements the user may plot normal stresses (from axial and bending) inside the beam cross sections (See Figure 9).
Figure 9 - Stresses inside the beam cross section

Figure 10 - Steel structure code checking
Code checking

Code checking in CivilFEM is carried out through its postprocessor. All the checking capabilities specified in each of the codes shown in the introductory paragraph are available in CivilFEM. The user can postprocess these results easily, by plotting elements that pass or do not pass the solicited check or by plotting the checking criterion. Moreover, in the case of Eurocode No.3, the effect of local buckling on slender sections (class 4 sections) is also analyzed by CivilFEM (See Figure 11). Most of the intermediate values necessary to carry out the requested check are also available. For concrete, along with other checks, the program is also able to check and design beam elements with generic reinforced concrete cross section under axial load plus biaxial bending.

![Figure 11 - Eurocode No.3 class 4 section](image)

Shell Reinforcement

Shell Reinforcement capability works with Ansys shell43, shell63 and shell93 elements. Two methods are available for checking and design shell models:

- Wood Armer MX, MY, MXY Reinforcement: Reinforcement design is made using the Wood-Armer method for MX, MY, and MXY moments at each node.
- Full, TX, TY, TXY, MX, MY, MXY, NX, NY Reinforcement: Reinforcement design is made by using the method proposed in the CEB-FIP Model Code (See Figure 12). An example of this reinforcement design is the AENA tunnel at Madrid Barajas airport (See Figure 13).

Non orthogonal reinforcement and variable mechanical covers as well as different thickness for all shell vertices is available.
Figure 12 - Reinforcement design in shell elements using CEB-FIP method

Figure 13 - AENA reinforced concrete tunnel calculated with ANSYS+CivilFEM
**Seismic Analysis**

This set of CivilFEM tools are designed to make faster and easier the seismic structural analysis by means of windows. Only three easy steps are necessary when carrying out a modal spectral analysis in CivilFEM; automatic definition of seismic spectrum, mode extraction and mode combination according to specifications. CivilFEM includes two seismic codes; Eurocode No.8 and Spanish code (NCSE-94). In the future other local and/or countries seismic codes will be introduced.

For each one of the three directions (longitudinal, transverse and vertical), the combination of the results obtained for the different modes is carried out through the square root of the sum of the squares of the variable considered.

Once combined the significant modes in each one of the three directions (longitudinal, transverse and vertical), the result obtained for these directions is combined again automatically by square root of the sum of squares, obtaining a unique solution.

![Figure 14 - Automatic spectrum definition for seismic analysis](image)

**Non Linear Concrete Module**

This CivilFEM specialized module permits the user to account for geometrical changes due to construction processes, changes in the material with respect to time as well as cracking and yielding phenomena and carry out complex Civil Engineering analysis with the click of a button. The non linear analysis is carried out in a transparent way to the user. This module considers the true nonlinear stress-strain curve of each of the materials forming the section as well as their variation with time. Each internal point is treated independently, allowing a detailed analysis of the cross section behavior and the consideration of a section with its real shape and not as a simple parameter set.
With this specialized module it will be possible for example to analyze large deflection buckling of concrete beam elements. This feature allows to reach the static equilibrium over deformed geometry with beam elements, taking into account the non linear behavior and possible cracking of concrete sections.

Moreover, the non linear analysis in CivilFEM accounts for non linear redistribution, finding the modified laws of bending moments in a beam element model, and considering plastic redistribution.

With the non linear deformations capability, it is possible to obtain the deformed geometry in beam element models accounting for all nonlinear concrete behavior.

CivilFEM also includes a nonlinear moment-curvature diagram with nonlinear moment redistribution analysis of beam structures (See Figure 15). Nevertheless, CivilFEM can obtain moment-curvature laws for reinforced concrete and generic mixed materials sections analyzed with this module.

Analysis of cracking and yielding phenomena is also possible through this specialized module. This feature is design to check the amount of yielding and cracking of the cross section.

![Figure 15 - Non linear analysis, cracked concrete section](image)

**Soil Mechanics Module**

This CivilFEM specialized module will allow the user to conduct soil mechanic related analysis in a user friendly way. A list of the different analysis the user is able to perform using CivilFEM soil mechanics module are described hereafter:

- **Earth Pressure**: This feature allows to define dry or flooded soil properties, and introduce the pressures that these soils induce on the structures as nodal forces and pressures at element faces for beam, shell and 2D/3D solid elements. Loads from active earth pressure and soil weight may be generated. The earth active pressure is calculated by using the Rankine theory. The soil weight is computed taking into account the depth and density of the soil over each point of the structure. The water table may be located at any level (See Figure 16).
Figure 16 - Footing design with soil mechanics module

- Slope Stability Analysis: This feature allows calculating the safety on deep sliding phenomena defined over an Ansys model of finite elements. Using a set of commands or windows, it is possible to define mechanical soil properties. Furthermore, the user is able to calculate safety coefficients and define sliding surfaces attached to them.

- Variation of the Elasticity Modulus with the Stress Level: This utility allows to introduce two non-linearities in the material simultaneously: plastic behavior according to Drucker-Prager or according to Hoek and Brown and variation of the elasticity modulus with the stress level.

- Ballast Modulus Calculation: CivilFEM will calculate for the user a theoretical value of the ballast module, by taking into account the geometry of the foundation and the properties of the soil laying underneath. With this capability, the user will be able to introduce automatically the ballast module given by the program and will avoid the hazard of having to model the soil underneath the structure. The estimation of this ballast coefficient, that will allow to approximate the elastic behavior of the soil (defined by its parameters $E$ and $\nu$), will be carried out using the Winkler model.

- Screens Analysis and Design: With this utility the user will be able to easily calculate screens of unit width, using precise models of beams (for the screen) and springs (for the soil).

- Seepage Analysis: CivilFEM, with the help of the ANSYS thermal capabilities and using an original optimization procedure, allows to obtain the saturation line (2D problem) or the saturation surface (3D problem). Also, it allows to solve the inverse problem (obtaining the permeability coefficients from the data measured “in situ”).

In addition to the soil mechanics analysis aforementioned, the user is also provided with helpful libraries of elastic and plastic soils properties. These libraries allow to automatically define Ansys elastic and plastic mechanical properties of most usual soil types as well as the definition and handling of geotechnical material properties (soils and rocks) which are not contemplated by ANSYS (Atterberg limit, Hoek and Brown coefficient, etc). It also allows the definition of correlations among properties (e.g. elasticity modulus as a function of SPT) and the possibility of incorporating predefined material libraries.

Other advance geotechnical calculations and models are under development and will be released soon.
Future Modules

Ingeciber is currently working in the development of three new specialized modules that will be available shortly. These three modules are described hereafter and will include the following capabilities:

1) Prestressed module:

   With this module prestressed concrete structures will be easy to model, analyze and postprocess using CivilFEM tools. The most important prestressed related capabilities will be available in this specialized module and are described below:

   - 3D Tendon Geometry Editor: The Tendon Geometry Editor allows the definition and edition of geometric and strength properties of all tendons of a structure. This geometry may be shown and edited either graphically or by coordinates. It’s possible to reference the tendon geometry to existing elements or to local coordinate systems of the model. The tendon editor works with the following patterns: straight, parabolic and defined by 2D/3D Bézier curves. Moreover, it is possible to perform automatically tangential adjustments among different types of curves.

   - Loss of Prestress: The loss of prestress calculation allows to obtain instant and delayed prestress losses for the tendons defined with the editor.

   - 2D Deviation Forces on 2D/3D Beam Elements: Once the loss of prestress has been calculated, it is possible to obtain the deviation forces produced by the prestress tendons on any 2D or 3D beam structure defined in Ansys. The deviation forces are defined as a specific load set located on nodes and/or distributed on beams. This is done with a single command. 2D deviation forces are calculated with the assumption that the tendons are in the same plane in which deviation forces are contained.

   - 3D Deviation Forces on Beams, Solids and Shells: Provided the user has calculated the loss of prestress, it is possible to reach, with a single command, the deviation forces produced by the tendons on any beam, shell or solid structure defined in Ansys. The deviation forces will be calculated as a set of loads located at nodes and/or distributed on elements of 3D models. These forces will be calculated in 3D using the actual tendon geometry.

   - Prestressed Non-linear Cable Analysis: This feature makes easy the generation of cable finite elements inside three-dimensional structures in order to represent the prestress in non-linear analysis.

   - Prestressed Sections Code Checking: This feature allows to check the section safety against bi-axial bending + axial forces over 2D or 3D beam elements and 3D model sections (See Figure 18).

![Figure 17 - Slope stability analysis](image-url)
2) Bridge module:

In this module CivilFEM will include very useful but indispensable capabilities for bridge designers. In this field of Civil Engineering, it is particularly important to account for linear as well as superficial mobile loads. For this reason, CivilFEM will generate these loads automatically for the user. On the one hand, the linear and superficial mobile load generator will automatically obtain the loads corresponding to the various load hypothesis that represent a load model over 2D or 3D beam structures. This calculation is performed from the definition of the loads that form the load model and their possible path on the structure in the case of linear mobile loads or the surfaces on which they move in the case of surface mobile loads.

To facilitate the model generation, the user will be provided with libraries of bridge sections by dimensions and common bridge components. The first library, will contain many parametrically defined common sections of steel and reinforced/prestressed concrete bridges. The second library, will allow to automatically generate and/or calculate structures that are also considered when calculating bridges: footings, piles, abutments, piers, bearing systems, frames and continuous beam models (See Figure 19).
3) Dams module:

This advance module will allow the user to easily conduct typical analysis for dams. The most important dams related capabilities will be available in this specialized module and are described hereafter:

- Construction Thermal Analysis: These set of tools make easy the thermal-structural analysis of concrete dams construction process.
- Automatic Load Generator: Automatic load generation and hypothesis combination.
- Coupling of Consolidation Effects with Stress Analysis: Coupling of seepage phenomena in porous soils with strains and stresses.
- Dynamic Loads from Fluid-Structure Interaction: Dynamic load generation on structures due to the fluid-structure interaction (See Figure 20).

![Figure 20 - Dams analysis model](Image)

**Conclusion**

The bundle product ANSYS+CivilFEM brings the new high technology features of the computer aided engineering world widely used in other technological industries into the Civil Engineering field. This features facilitate to perform easily, advanced finite element analysis through its unique capabilities. CivilFEM’s powerful tools allow the users to tackle new high-end engineering problems and overcome them easily.