Comparison of ATENA and Other FE Software for Concrete Structures

Ahmad Delpasand, Rostislav Chudoba

April 16, 2025

1 Introduction and Overview

When analyzing reinforced concrete structures, the choice of finite element (FE) software can greatly influence the ease and accuracy of capturing complex nonlinear behaviors. ATENA is a specialized FE package dedicated to concrete, whereas Abaqus, ANSYS, LIMFES, and InfoGraph are more general-purpose or design-oriented tools. This comparison highlights why ATENA's concrete-focused approach makes it a superior choice for graduate students and researchers dealing with cracking, crushing, and reinforcement interaction in concrete structures. We contrast general features and nonlinear capabilities of the previously mentioned packages, emphasizing how ATENA's specialization in concrete leads to advantages in modeling realistic concrete behavior.

2 Specialization and Focus

ATENA (developed by Cervenka Consulting) is explicitly designed for nonlinear analysis of concrete and reinforced concrete structures. Unlike Abaqus and ANSYS, which are multi-purpose FE platforms for a wide range of materials and physics, ATENA focuses on quasi-brittle materials (concrete, masonry, rock, etc.), providing an environment tailored to civil engineers. According to its developers, ATENA "belongs to the category of advanced analysis software such as Abaqus or ANSYS, but it focuses on reinforced concrete structures and brittle materials" [1]. This specialization means that many default settings, material models, and solution methods in ATENA are optimized for concrete behavior. In fact, ATENA can automatically derives concrete material properties from a given concrete strength class using *fib* Model Code 2010 formulae, making setup more straightforward [1]. As one researcher states, the program is "specially designed for concrete, making it easier for users since well-set default values are initially given, and even with severe cracking, it showed consistent convergence" [1].

By contrast, Abaqus (Dassault Systèmes) and ANSYS are general FE suites used mainly across aerospace, mechanical, and civil engineering disciplines. They certainly can model concrete, but they are not exclusively devoted to it – users must manually calibrate many concrete material parameters (e.g. tension stiffening, fracture energy) for realistic results.

InfoGraph (InfoGraph GmbH) falls into another group of integrated structural analysis software primarily aimed at practicing engineers for design verification. It supports concrete design to various codes (e.g., Eurocode 2, national annexes) and includes nonlinear analysis features as an extension of its design focus [2]. Meanwhile, LIMFES (a finite element code developed by N. Kerkeni at RWTH Aachen/H+P Ingenieure) is a specialized tool like ATENA, used for detailed nonlinear analysis of concrete bridges and structures [3]. However, LIMFES is more of an in-house or research code, whereas ATENA is a widely distributed commercial software with over 2,500 installations worldwide [1].

In summary, ATENA's singular focus on concrete gives it an edge in concrete structures modelling, while Abaqus/ANSYS provide broader applicability (however require more effort and expertise for concrete), InfoGraph caters to design-code-oriented analysis, and LIMFES serves expert-level use.

3 Nonlinear Concrete Behavior: Cracking, Crushing, and Reinforcement

Capturing concrete's nonlinear behavior—cracking in tension, crushing in compression, and interaction with reinforcement—is essential for realistic analysis. Here, ATENA excels with built-in constitutive models and solution algorithms specifically tuned for these phenomena. ATENA can directly simulate concrete cracking and crushing progressions, and reinforcement yielding under load [1]. It employs advanced material models that allow concrete to crack, with strain-softening, aggregate interlock, post-cracking tensile resistance and proper energy dissipation to avoid mesh sensitivity. In ATENA, cracks are not just an abstract reduction in stiffness; the software provides discrete crack visualization during and after analysis, showing crack paths, widths, and propagation stages [1]. This allows researchers and students to visually inspect crack development, aiding in understanding and validating numerical results against physical ones. Reinforcement in ATENA can be modeled as discrete rebars with bond-slip behavior or as smeared reinforcement of a composite material, with rigorous tracking of the interaction between steel and concrete [1].

General FE codes like Abaqus and ANSYS also have capabilities for modelling concrete cracking and crushing, but they use more generic frameworks. Abaqus offers the Concrete Damaged Plasticity (CDP) model which mimic concrete cracking and crushing in an averaged sense by degrading stiffness once a failure criterion is reached. However, it does not explicitly form individual cracks, instead reducing material stiffness via damage variables. Users must input tension stiffening data to approximate crack effects and rebar tension tie-action. Although Abaqus can predict overall load drops and inelastic deformations due to cracking, visualization of crack paths is not done straightforward rather indicated by tensile damage or plastic strain contours [4].

ANSYS historically provided a dedicated concrete element, SOLID65, with a smeared crack model capable of cracking in multiple directions and crushing in compression. This model enables crack initiation and propagation modeling in a smeared manner. Reinforcement can also be modeled as either smeared or discrete elements. Nevertheless, achieving convergence for highly nonlinear crack growth can be challenging, and careful mesh and parameter choices are necessary for accurate results [5].

InfoGraph, oriented toward structural engineering, incorporates nonlinear FE analysis to support structural design checks. It includes damage models by De Vree, Mazars, and a combined plasticity-damage model from Lubliner, Lee, and Fenves (analogous to Abaqus CDP model). This allows InfoGraph to perform 3D nonlinear analyses of concrete members, predicting crack zones and stiffness degradation effectively. However, detailed crack path visualization or the variety of concrete material models are more limited compared to ATENA [2].

LIMFES is developed explicitly for nonlinear fracture analysis of concrete structures, particularly bridges. It incorporates sophisticated concrete models such as Bažant's old Microplane Model M4, aiming to simulate brittle failure and crack development with high fidelity. LIMFES has been utilized extensively to validate shear failure predictions and crack patterns both in research and practical bridge assessments [3].

In summary, ATENA and LIMFES provide specialized handling tools for concrete inelasticity, Abaqus and ANSYS offer general but powerful nonlinear modeling (needing customization) for concrete, and InfoGraph delivers intermediate-level nonlinear analysis tailored for practical design tasks.

4 Depth of Material Modeling

A clear advantage of ATENA is the breadth of concrete material models available and ready to use. ATENA comes with multiple constitutive models for concrete: a crack band model based on fracture energy, a fracture-plastic model combining plasticity and smeared cracking, and a more recent variant of Microplane Model M7 [6]. These advanced models allow researchers to simulate salient aspects of concrete behavior, such as tension softening, compression yielding, and confinement effects, with confidence in their validation for concrete. By inputting concrete compressive strength, ATENA internally generates default parameters for these models using established code formulae, making it highly convenient, especially for students [1].

In contrast, Abaqus provides the Concrete Damaged Plasticity (CDP) model, a continuum damage-plasticity formulation, requiring users to specify numerous parameters, including dilation angle, eccentricity, biaxial strength ratio, and fracture energy. Accurate parameter setting typically involves extensive calibration studies or literature revision [4]. Abaqus allows custom material definitions through user subroutines (UMAT/VUMAT), but this requires advanced expertise.

ANSYS offers built-in material models, such as the linear-exponential crack model in SOLID65 and a microplane-based model alike M7. These models, while powerful, require scripting in APDL and precise definition of material tables, making them less accessible for casual or student users [5].

InfoGraph simplifies material modeling with predefined nonlinear material behaviors aligning with code models. It includes standard nonlinear stress-strain curves and damage models like Mazars and Lubliner-Fenves, facilitating quick setup for structural engineers, albeit with fewer advanced model options compared to ATENA [2].

LIMFES employs research-grade material modeling, notably the Microplane M4 model, allowing detailed representation of concrete behavior, yet its application demands significant expertise and is mainly utilized in specialized research settings [3].

5 Ease of Use and Learning Curve

For students and newcomers, the learning curve of an FE package is a crucial point. ATENA is often praised for its user-friendly graphical interface tailored to structural concrete modeling [1]. Its GUI and workflow align with how one designs a concrete experiment or structure. Users can easily define beams, columns, shells, or 3D solids, assign concrete properties simply by specifying compressive strength (with the software auto-calculating detailed properties), and add reinforcement either as individual bars (from a library of rebar sizes) or as smeared layers [1]. Unique built-in features of the software includes crack monitoring and runtime visualization of the cracking process. [1]. According to Richard Malm, "a novice user can rather easily create advanced models in ATENA" [1], underscoring its approachability for students. Additionally, the documentation and examples provided are all concrete-specific, which significantly shortens the learning curve for users within this domain.

By contrast, general-purpose FE packages like Abaqus and ANSYS have notably steeper learning curves for detailed nonlinear analysis. Their broad applicability means users must familiarize themselves with numerous options and complex preprocessing environments. In Abaqus/CAE or ANSYS Workbench, setting up a reinforced concrete model involves several detailed steps: defining concrete material parameters (like dilation angles or shear transfer coefficients), specifying rebar as embedded elements or layers, and carefully selecting analysis parameters for convergence (e.g., viscosity parameters in Abaqus Concrete Damaged Plasticity (CDP) model) [4, 7]. Although manageable for experienced analysts, this complexity may challenge students without substantial guidance. Moreover, troubleshooting convergence or unrealistic crack patterns can require extensive adjustments to meshes or material properties, a frustration ATENA mitigates by employing robust default algorithms optimized specifically for concrete cracking problems [1].

InfoGraph has an advantage in ease-of-use specifically for structural engineers, providing an intuitive, Windows-based interface with CAD-like modeling capabilities and integrated design-analysis workflows. Students with an engineering background can quickly perform linear analyses and easily activate nonlinear analysis options without scripting. While user-friendly, InfoGraph is somewhat limited in modeling flexibility compared to ATENA or general FE codes [2].

LIMFES, being an in-house research tool without a commercial-grade graphical interface, is the most challenging for newcomers to learn unless guided by experienced users.

In summary, ATENA provides an optimal balance of specialization and user-friendliness, enabling students to quickly learn nonlinear concrete FE analysis essentials, allowing more focus on interpreting results rather than overcoming software complexities.

6 Research vs. Industry Use

Another point of comparison is how well each software serves academic research needs versus practical industry applications. ATENA has strong adoption in academia for concrete research. Its ability to simulate detailed concrete failure processes (and visualize them) makes it a favorite for thesis projects on topics like shear failure, anchorage behavior, fiber-reinforced concrete, etc. Numerous journal papers and dissertations have used ATENA to validate new theories or investigate complex behaviors, with well-documented accuracy in modeling brittle fracture in concrete mechanics literature [8]. At the same time, ATENA is used in industry by specialized consulting firms for advanced analysis—for instance, assessing structures that lie outside code provisions or optimizing reinforcement using nonlinear redistribution [1]. However, it is not commonly found in everyday design offices, as standard design software or simpler methods typically suffice for routine projects. It truly shines in advanced applications where understanding the "real" behavior is crucial; ATENA's motto emphasizes helping engineers understand "real structural behavior or true load carrying capacity" beyond code formulas [1].

Abaqus and ANSYS are extensively utilized in both academia and industry, though typically not in routine concrete design. In academic research, these tools are popular for studying concrete in general simulations or when coupling with other physics, such as soil interaction or dynamic events (blast, impact). For industry, civil engineering firms generally do not use Abaqus/ANSYS for standard building designs as they are inefficient for code compliance checks. Instead, these tools are preferred in specialized analyses like concrete dams under seismic events or nuclear containment structures, often within specialized consulting or when conventional code-based tools are inadequate.

InfoGraph, conversely, is oriented towards industry practitioners for routine, code-compliant design. It is widely used, especially in Europe, for designing concrete and composite structures according to Eurocode and national standards [9]. Its strength is the integrated workflow: modeling, analysis (linear or nonlinear), and direct reinforcement requirement checks per code. For research, InfoGraph is less common unless the focus is on evaluating code methods or assessing structural performance according to guidelines.

LIMFES occupies a unique niche, having originated academically but applied in industry for advanced bridge recalculations [10]. Its use in industry is limited, primarily to the developers' organization and collaborators. For a PhD student associated directly with such groups, LIMFES might be beneficial for research specifically involving bridge shear failure.

In summary, ATENA is highly suitable for academic concrete research and has a foothold in specialized industry problems. Abaqus and ANSYS are essential for multipurpose simulations, InfoGraph is optimized for code-driven industry design workflows, and LIMFES is a specialized tool confined to expert research-industry crossover use.

7 Design Code Support

Support for design codes and standards varies significantly among these programs. ATENA is not a design-code checking tool per se – it does not automatically tell you if a beam meets ACI or Eurocode requirements. Instead, it allows you to numerically test a design. In practice, ATENA complements code-based design: engineers can design a structure with conventional methods and then use ATENA to verify critical parts or investigate failure modes that codes only approximate [1]. For example, ATENA can reveal how much load beyond the code-predicted capacity a structure can carry by redistributing forces after cracking, potentially leading to more economical reinforcement designs or justifying the safety of existing structures with smaller safety factors [1]. It even automatically captures internal force redistribution due to cracking, resulting in reinforcement savings in design [1]. However, formal code checks (such as calculating required rebar area or checking serviceability crack width limits) must be performed manually by interpreting ATENA's results.

InfoGraph, conversely, has extensive built-in code support. It continuously updates its software to comply with the latest codes, as evidenced by updates adding checks for EN 1992-2 and various national annexes [2]. With InfoGraph, detailed code check reports can be obtained: after a nonlinear analysis, it reports remaining safety factors or needed reinforcement according to the code formulas, taking cracked state stiffness into account. InfoGraph can perform nonlinear serviceability analyses (accounting for crack formation) and check crack widths against code limits. Additionally, it supports designs under fire conditions, seismic pushover analyses, and other conditions prescribed by codes, all within a single platform [2]. This comprehensive functionality makes InfoGraph particularly attractive for industry practitioners who must ensure compliance with building regulations.

ANSYS and Abaqus do not include any built-in code-specific design checks, as they are purely analysis tools. Any verification against design codes, such as comparing bending capacity (ϕM_n) to demand, must be performed manually or via custom scripts. In research contexts, this is typically not problematic since researchers often explore behavior beyond code requirements. However, for projects focused explicitly on design, additional effort would be needed to link Abaqus or ANSYS results to code criteria through external calculations or scripts.

Similarly, LIMFES is an analysis tool without built-in code checks. Its use in bridge assessments implies that engineers interpret its results to satisfy code-based assessment procedures. German bridge recalculation guidelines, for instance, allow nonlinear analysis as a final recourse, and LIMFES fits precisely that role [3].

In summary, if direct design code support is essential, InfoGraph clearly stands out. ATENA, by contrast, is oriented toward deeper insight beyond codes and thus pairs indirectly with code-based design. Abaqus and ANSYS remain neutral concerning code compliance, whereas LIMFES is employed within defined assessment procedures but does not independently provide code-checking features.

8 Multiphysics and Extended Capabilities

Many advanced studies of concrete structures involve multiphysics scenarios—such as thermal effects (fire heating), hydro-thermal phenomena (concrete drying shrinkage or moisture transport), or dynamic events. Here, the general finite element (FE) packages have a clear advantage in breadth. ANSYS and Abaqus are full multiphysics platforms: Abaqus can couple stress analysis with heat transfer (for fire simulation or hydration heat in mass concrete), pore pressure (for soil or seepage in concrete dams), acoustics, and even run computational fluid dynamics (CFD) or electromagnetic simulations through related solvers [4]. ANSYS, as a suite, offers structural, thermal, fluid, and electromagnetic solvers that can be coupled; for instance, one could simulate a concrete dam with fluid pressure loading and transient thermal gradients, all within ANSYS [7].

If a student's project involves scenarios such as concrete subjected to fire or concrete interacting with fluid (e.g., wave impact on a sea wall), these general tools provide a flexible framework. ATENA, being specialized, still supports some multiphysics but within the realm of structural concrete. Notably, ATENA has modules for high-temperature analysis of concrete (to simulate fire damage in concrete structures) and durability aspects like reinforcement corrosion and concrete moisture transport [1]. These are specific to civil engineering needs—for instance, one can apply fire loading in ATENA to observe how a concrete column spalls and loses capacity [1]. ATENA might not handle fluid flow or advanced CFD, but it is certainly capable of coupled thermal-stress analysis for fire and can incorporate creep and shrinkage of concrete (per codes or user input) to study long-term behavior. It also has support for dynamic analysis (transient loads, eigenfrequencies) although it's not typically used for high-speed impacts or explosions (where Abaqus Explicit or LS-DYNA in ANSYS are more prevalent).

InfoGraph includes a dedicated module for structural fire analysis, which applies the Eurocode parametric fire curve and performs nonlinear heat transfer combined with structural analysis for steel, concrete, or timber structures [11]. This is a targeted form of multiphysics (thermal-mechanical for fire scenarios). InfoGraph can also handle prestress loss, creep, and shrinkage within the context of design. More complex interactions, such as fluid dynamics, are outside InfoGraph's scope. LIMFES is primarily a structural FE solver; any multiphysics (such as thermal gradients in bridge analysis) would likely be handled by applying temperature loads rather than through fully coupled field equations [12].

In conclusion, Abaqus and ANSYS are the preferred solutions if true multiphysics or unusual physics couplings are required, offering unmatched versatility beyond structural mechanics. However, if the interest is strictly concrete structural behavior under environmental conditions such as loading and temperature, ATENA, and to a lesser extent InfoGraph, provide built-in specialized capabilities to adequately address these scenarios.

9 Comparison Table of Key Features

Aspect	ATENA	Abaqus	LIMFES	InfoGraph	ANSYS
Concrete Specialization	Yes, concrete-focused defaults [1]	No, general-purpose [4]	Yes, concrete bridges [12]	Partial, RC focus [13]	No, general-purpose [7]
Nonlinear Analysis	Robust, explicit cracks [1]	Advanced, cracks smeared [4]	High fidelity, cracks explicit [14]	Intermediate, cracks smeared [13]	Advanced, cracks smeared [7]
Material Modeling Depth	High, multiple built-in models [15, 1]	Moderate, one built-in model [16]	High, research-grade [17]	Moderate, simplified [13]	Moderate, user-calibrated [7]
Ease of Use/Learning	Easy, concrete-centric GUI [1]	Steep, general GUI [4]	Difficult, expert-only	Easy, user-friendly GUI [13]	Moderate, mixed GUI/script [7]
Research vs Industry	Excellent research, specialized industry [1]	Excellent research, niche industry [4]	Niche research, specialized industry [12]	Limited research, strong industry [13]	Excellent research, niche industry [7]
Design Code Support	Minimal, indirect [1]	None	None, assessment only [12]	Extensive, direct code checks [13]	None
Multiphysics Support	Concrete-specific multiphysics [1]	Comprehensive multiphysics [4]	Limited	Structural fire analysis [13]	Comprehensive multiphysics [7]
Community & Support	Dedicated, concrete-specific [1]	Huge, general FE [4]	Small, internal use	Moderate, European [13]	Huge, general FE [7]

Table 1: Comparison of key features for concrete FE analysis software.

10 Summary and Recommendation

In conclusion, ATENA distinguishes itself as the premier choice for nonlinear analysis of concrete structures due to its specialized focus, rich concrete material modeling, and user-friendly design for structural engineers. For a Master's or PhD student whose work revolves around concrete behavior (cracking, crushing, reinforcement interactions), ATENA provides an environment where one can set up sophisticated models relatively quickly and obtain realistic, insightful results. The ability to visualize crack development and rely on proven concrete models (without painstaking calibration) means more time can be spent on interpreting structural behavior and less on debugging the FE model. ATENA's niche focus does not significantly limit its scope for concrete structures—it has been applied successfully from lab-scale specimens up to large bridge assessments—and it continues to be improved with state-of-the-art concrete research (e.g., new fracture models and durability simulations).

On the other hand, Abaqus and ANSYS remain excellent tools for cases where general FE capabilities are needed in addition to concrete modeling (for example, if your project involves thermal or dynamic effects beyond ATENA's range, or if you need to simulate something like a composite steel-concrete system with many material types). They are powerful but come with a higher overhead for the user and do not inherently provide the concrete-specific conveniences that ATENA does. InfoGraph is a strong option when the goal is to adhere strictly to design codes and produce an immediately actionable design—it will let you know if your concrete section passes or fails according to the code, and it can account for nonlinear behavior in that process. However, it won't give as much detailed insight into the failure mechanics as ATENA, nor is it intended for the exploratory "what-if" studies that researchers often conduct. LIMFES, while very capable in the concrete niche, is not widely accessible; if one happens to collaborate with the developers or focus on bridge shear failure research, it could be of interest, but for most students ATENA would be the practical choice among specialized tools.

Recommendation: For graduate students and researchers focusing on concrete structures, ATENA is the top recommendation due to its combination of advanced concrete modeling and relative ease of use. It enables deep investigation into concrete behavior (crack patterns, failure modes, reinforcement performance) with confidence and support from a community that speaks the language of concrete mechanics. Students can trust ATENA's results as it has been validated extensively in both academic and real-world scenarios [18, 3]. After mastering ATENA, one can always expand to Abaqus or ANSYS for broader applications, but ATENA will likely cover 90% of what is needed for concrete-centric research. In an industrial context, we suggest using ATENA to supplement traditional design workflows: design with code-based tools (or InfoGraph for an integrated approach) and then use ATENA for advanced verification of critical parts. This way, one gets the best of both worlds—code compliance and a true understanding of the structure's behavior. Overall, ATENA's professor-approved specialization in concrete makes it an invaluable tool for those aiming to push the frontiers of concrete structural analysis in both academia and industry [1].

References

[1] Cervenka Consulting. *ATENA*. 2025. URL: https://www.cervenka.cz/products/atena/.

[2] InfoGraph GmbH. *Developments — InfoGraph Software für die Tragwerksplanung*. 2025. URL: https://www.infograph.eu.

[3] H+P Ingenieure GmbH. *Nachrechnung von Bestandsbrücken*. 2025. URL: https://www.huping.de.

[4] Dassault Systèmes. *Abaqus Documentation*. 2025.

[5] ANSYS Inc. *Concrete Modeling with SOLID65 Element*. 2025.

[6] ResearchGate. *Which software is the best to model a nonlinear concrete (ABAQUS, ANSYS, ADINA, ATENA)?* URL: https://www.researchgate.net/post/Which_software_is_the_best_to_model_a_non-linear_concrete_ABAQUS_ANSYS_ADINA_ATENA_etc.

[7] Ansys Inc. *ANSYS Documentation*. User's Manuals.

[8] FEA - JKP Static. *Documentation on accuracy in brittle fracture modeling*.

[9] InfoGraph GmbH. *Structural design and code compliance documentation*.

[10] H+P Ingenieure GmbH. *Bridge recalculation documentation*.

[11] InfoGraph GmbH. *Structural Analysis for Fire Scenarios Manual*. 2024.

[12] RWTH Aachen and H+P Ingenieure. *LIMFES Documentation*. 2024.

[13] InfoGraph GmbH. *Structural Analysis Software*. https://www.infograph.eu.

[14] Zdeněk P. Bažant. *Microplane Model M4*. IABSE Workshop, Helsinki, 2015.

[15] *ResearchGate Discussions on Concrete Models in FE*. https://www.researchgate.net.

[16] Dassault Systèmes. *Calibration of Concrete Damaged Plasticity Parameters*. Abaqus Documentation.

[17] Zdeněk P. Bažant. *Microplane Models for Concrete*. IABSE Workshop, Helsinki, 2015.

[18] FEA - JKP Static. *Documentation and Validation studies*. 2023.