Volume 12, No.11, November 2025

Journal of Global Research in Mathematical Archives

UGC Approved Journal

ISSN 2320 – 5822

RESEARCH PAPER

Available online at http://www.jgrma.info

COMPARATIVE ASSESSMENT OF PIPING STRESS ANALYSIS SOFTWARE: CAESAR-II, AUTOPIPE, AND ANSYS APPROACHES

Dr. Chintal Kumar Patel[1]

[1] Associate Professor, CSE, Geetanjali Institute of Technical Studies

chintal.patel@gits.ac.in

Abstract: Piping stress analysis software is a type of assessment needed to provide the structural reliability and safety of complex piping system under high pressure, temperature, and dynamic loading conditions. The given paper presents an extensive review of comparative analysis of three most popular tools CAESAR-II, AutoPIPE and ANSYS which are commonly used in the field of industrial design and analysis. Both software use different computational techniques and modeling tools to estimate stress, deformation and failure propensity in piping systems. The CAESAR-II and AutoPIPE mainly adhere to code-based analysis techniques of flexibility with more lean, global-level stress analysis in concurrence with international design standards like ASME B31.1 and B31.3. ANSYS on the other hand employs Finite Element Method (FEM) to determine the stress distribution detail and nonlinear response analysis in three-dimensional analysis and offers enhanced accuracy in localized analysis. This paper discuss their advantages in analysis, modeling, weaknesses, and their use in industries. In addition, the review incorporates new findings in the fields of fluid-structure interaction, thermal analysis, and fatigue analysis of piping systems. The purpose of this comparative analysis is to examine how computational predictions and real-world outcomes differ and how to improve them. It is suggested that artificial intelligence, hybrid FEM code coupling and auto simulation workflow integration is introduced in the future to advance efficiency, accuracy and predictive maintenance in the design of modern piping systems.

Keywords: Piping Stress Analysis, CAESAR-II, AutoPIPE, ANSYS, Finite Element Method (FEM), Flexibility Analysis, Structural Integrity, Code Compliance.

1 INTRODUCTION

The amount of stress a system of pipes is subjected to due to forces and moments is determined through engineering research known as pipe stress analysis. Determining the type of loading, the pipe material used, and the internal and external elements that can affect the planned and anticipated system are part of the study. Once the analysis confirms that the pipes can withstand the loads, it may identify potential problems or weak points in the system. Pipelines are an essential component of infrastructure for transferring fluids such as oil, gas, and water. With an impeccable safety record, they are valued for their dependability, efficiency, and affordability in delivering oil and gas globally [1]. Coal, oil, and gas are among the fossil fuels that today supply more than 80% of the world's energy. When corrosion occurs on the pipeline walls, it compromises the pipeline's integrity. Pipelines are vulnerable to a wide range of internal and external factors that can accelerate corrosion and cause cracks and other faults.

The fundamental stress intensity limits for the aforementioned stress categories are established by applying limit design theory in conjunction with appropriate safety factors. Pretend that the pipes are elastic and completely plastic, meaning they won't harden under stress [2]. The pipe burst at the strain imposed by an applied force that makes the primary membrane stress equal to the material's yield stress, Sy. This yield stress must be present over the whole cross-section for piping to fail when bent. This won't happen until the load exceeds the pipe's yield moment plus a factor called the cross-section's form factor.

The CAESAR II module, which replicates municipal-scale mass retrofitting efforts that incorporate various seismic and energy improvement techniques, is informed by vulnerability assessments of the current building stock. The core of the CAESAR II tool consists of "Seismic Impact scenarios" that end users request depending on hazard intensities and significant risk elements to be addressed in the simulation object [3][4]. Predicted building and population damage thresholds are part of the model's output. Customization of impact scenarios is possible based on the desired level of territorial information, data availability, and individual requirements.

Although ANSYS Multiphysics can display stress distribution in three dimensions, AutoPIPE, one of the industry's favorite finite element programs for calculating stress magnitudes on riser-modeled nodes, has this disadvantage [5]. Engineering understanding of the impact of external loadings acting along the riser can be greatly enhanced by a three-dimensional perspective of the riser's stress distribution. Developing a finite element stress analysis to inform the various decisions that go into the system can improve the overall design accuracy of a riser system. ANSYS [6] is a finite element analysis program that is part of an all-purpose FEA software suite. Finite Element Analysis (FEA) is a numerical technique that divides a complex system into small, user-defined bits.

The software generates a thorough description of the system's behavior by solving the equations that regulate the elements' behavior. A user-friendly, front-end finite element analysis tool, the ANSYS Workbench environment is compatible with Design Model and CAD systems [7].

1.1 Structure of the paper

The paper is structured as follows: Section 2 covers fundamentals of piping stress analysis; Section 3 reviews major analysis software computational approaches of CAESAR-II, AutoPIPE, and ANSYS, Section 4 summarizes relevant literature; and Section 5 concludes with key findings and future directions.

2 FUNDAMENTALS OF PIPING STRESS ANALYSIS

The term "piping stress analysis" (PSA) refers to calculations that account for static and dynamic loads from gravity, temperature fluctuations, internal and external pressure, variations in flow velocity, and ground shocks. The minimum requirements for stress analysis are established by the standards and recommendations [8]. Stress analysis indicates whether the pipe system will break during engineering design.

2.1 Overview of Piping Systems and Stress Mechanisms

Piping systems transport fluids under varying pressure and temperature, which subject them to complex mechanical stresses. These stresses arise from internal pressure, thermal expansion, weight, and external forces, requiring careful analysis to ensure system safety and code compliance.

2.2 Design Codes and Standards (e.g., ASME B31.1, B31.3)

Design codes and standards provide the fundamental rules and guidelines that ensure the safety, reliability, and uniformity of piping systems[9]. They define accepted engineering practices that designers must follow to meet regulatory and industry requirements.

- **Codes:** A "code" is a set of regulations that the government has decided upon and put into effect [10]. Ensuring public and industrial safety during a particular activity or with a specific piece of equipment is the goal of every code. The same groups that work on standards also tend to work on codes.
- **Standards:** System, component, and method variances can be costly, inconvenient, and confusing; standardization can and does alleviate these problems [11]. It is the user's responsibility to ensure that documents prepared by a competent organization adhere to sound engineering practice.

2.2.1 Importance of Codes and Standards, for Instance

- Allowable bending at the ends of pipe and joints, and Allowable Rotations[12].
- These allowable limits are calculated allowances. Compliance ensures that these stress levels are attained under stress concentration factors: calculated stresses Sci or %) and stress limits.
- Ensure that these stress levels are attained under tolerable stress conditions at individual structural joint elements under different operating conditions to mitigate failure risk.

2.2.2 ASME B31

According to ASME B31, pipelines are categorized based on the level of danger associated with the fluid they carry. In category "D," the criteria for design, examination, and testing are less stringent since the risk is lower. On the other hand, in category "M," the risk is greater, and the standards are more stringent [13]. The design conditions, including potential pressures, temperatures, and loads the pipelines may encounter, are accounted for throughout the calculations.

2.3 Key Parameters in Stress Analysis (Pressure, Temperature, Loads)

The key parameters to be considered while receiving the bending moment and axial forces include internal pressure, temperature difference, and external mechanical loads[14]. The stress analysis must therefore consider coupled effects of pressure and temperature, and the interaction between axial and bending stresses to ensure safe operation of the bonded piping system (figure 1).

Figure 1: Key parameters in stress analysis

- **Pressure:** The internal pressure on the pipe bends increases as the fluid flows through it. The mismatch in surface area between the pipe bend's intrados and extrados produces uneven thrust forces [15]. Large stress concentrations are produced on the pipe bend wall as a result of these forces distorting the pipe bend's cross-section.
- **Temperature:** A ship's plumbing system is influenced by a number of factors, including the ship's weight, pressure, temperature, the hull's bending, the forced displacement caused by the hull's thermal deformation, and the inertia forces caused by wave-induced ship motion [16][17]. The longitudinal and hoop stresses of the cylindrical pressure vessel are used to assess the stress caused by the pipe pressure. Internal pressure is the only load that cannot be expressed by the longitudinal axial force and equivalent bending moment.
- **Loads:** Sustained loads include dead weight, internal pressure, and other applied axial loads (i.e., those unrelated to temperature, accelerations, etc.). Factors that contribute to environmental loads include earthquakes, waves, winds, snow, and ice buildup due to precipitation or sea spray [18]. The stress limitations for sustained-or occasional stresses should be satisfied by environmental loads, which are classified as either continuous or intermittent in nature.

2.4 Challenges in Modern Piping Design

Semi-structured interviews and case studies were used to compile the challenges associated with piping prefabrication, including design standardization, economies of scale, the use of BIM software in prefabrication, the absence of fabrication facilities, the availability and quality of fittings and valves, procurement authority over specific equipment [19], workforce capacity, design and construction specifications, and relevant policies.

Green construction and prefabrication have received greater attention from policymakers in recent years. The main focus of prefabrication programs, however, was on structural components such as slabs and walls rather than the entire building. Green building policies, on the other hand, focus on reducing energy use throughout the facility's operations and maintenance. Few regulations exist to encourage the prefabrication of pipes. Reduced tolerances are required during pipe prefabrication and installation due to the high expense of rework. Hence, highly trained workers are required. It is challenging to get economies of scale due to the intense rivalry among MEP contractors. One obstacle to pipe prefabrication is the availability of suitable fittings and valves, as different fittings and valves use connectors of varying sizes. The availability of the fabrication facility, support for the relevant specifications, and the BIM software for piping prefabrication pose further problems. The outcome shown that these obstacles are manageable.

3 PIPING STRESS ANALYSIS SOFTWARE TOOLS

The performance, safety, and dependability of pipe systems under different operating conditions can be assessed through piping stress analysis, which uses sophisticated tools and procedures. These instruments are useful for managing loads, identifying high-stress areas, and verifying compliance with regulations. Here is a rundown of some of the most popular methods and tools: [20].

3.1 Evolution of Piping Analysis Tools

Different software tools adopt varied approaches to piping stress analysis. CAESAR-II and AutoPIPE use code-based flexibility methods for rapid system evaluation, while ANSYS applies finite element modelling for detailed local stress assessment.

3.2 CAESAR-II

Effective and precise evaluation of piping systems under weight, pressure, heat, seismic activity, and other static and dynamic loads is made possible with a comprehensive pipe stress analysis program called CAESAR II. Systems with pipes of any size or complexity can be analyzed using it. No other program comes close to CAESAR II in terms of the unique calculating methods and analytical tools it incorporates [21]. The results produced by CAESAR II, whether creating a new system or troubleshooting an existing one, provide a comprehensive description of the system's behavior, based on rules and design restrictions derived from widely recognized industry standards. Figure 2 illustrates a typical piping system, which was simulated in CAESAR-II, with colored stress distributions on the different parts of the pipe. The three-dimensional arrangement is complete with the supports, restraints, and node points, and on the right side, there is a stress scale that shows the ranges of stress levels in the system. Below that, the displacement and restraint summary tables are shown, which were produced after the analysis was performed.

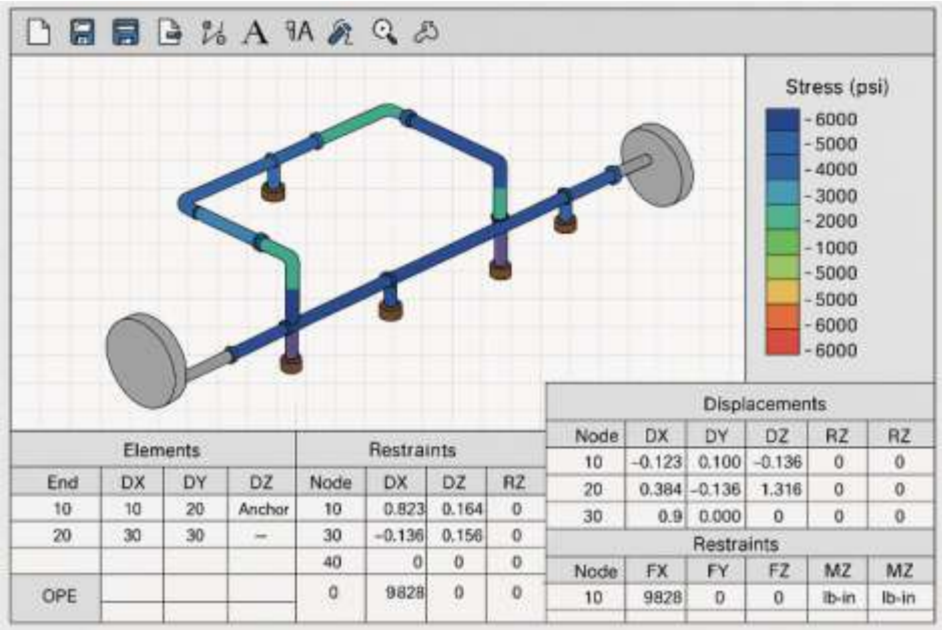

Figure 2: CAESAR-II Piping Stress Analysis Model

3.2.1 Computational Methodology and Interface

The first thing to do is gauge the pipe:

Q = A x v
Q – Flow rate
A – Area in meter
V – Velocity in m/s

The outcomes these methods produce are considered correct, but this isn't confirmed by comparison. Linear stress analysis can be used to ensure that the geometry remains in the linear elastic range—that is, the part returns to its original shape when the load is removed—as long as rotations and displacements are negligible relative to the geometry. For this kind of analysis, one typical design goal is the FOS. It is possible to figure out the pipe's thickness using equation (1) and the information below:

$$T = \frac{WP * D_o}{(2 * f * E) + WP} + C \tag{1}$$

CAESAR II can be used to analyze stress in the auxiliary steam piping since the auxiliary steam piping, air heater, and carbon blowing systems comply with piping rules, ensuring a safe assessment of the stresses in the piping system [22]. Its results show the standard methods that expand plant life and improve quality control and can find the pressure drop by using the method in equation (2).

$$\Delta P = \frac{f * V^2 * L}{2 * g * d} \tag{2}$$

The forces in the pipe are first studied with CAESAR-II, and then the results are checked for FEM in Solidworks.

3.3 AutoPIPE

Auto-Pipe is a planning tool that allows to do this automatically. The work mainly focuses on pipeline designs, but more general topologies can also be employed. The figure 3 illustrates the AutoPIPE interface displaying a modeled piping section with defined supports, bends, and node points. The right-side dialog shows the settings used to assign properties such as flexibility, stiffness, and loading conditions. AutoPIPE provides an intuitive environment for engineers to build, modify, and analyze piping layouts with accurate stress and displacement calculations.

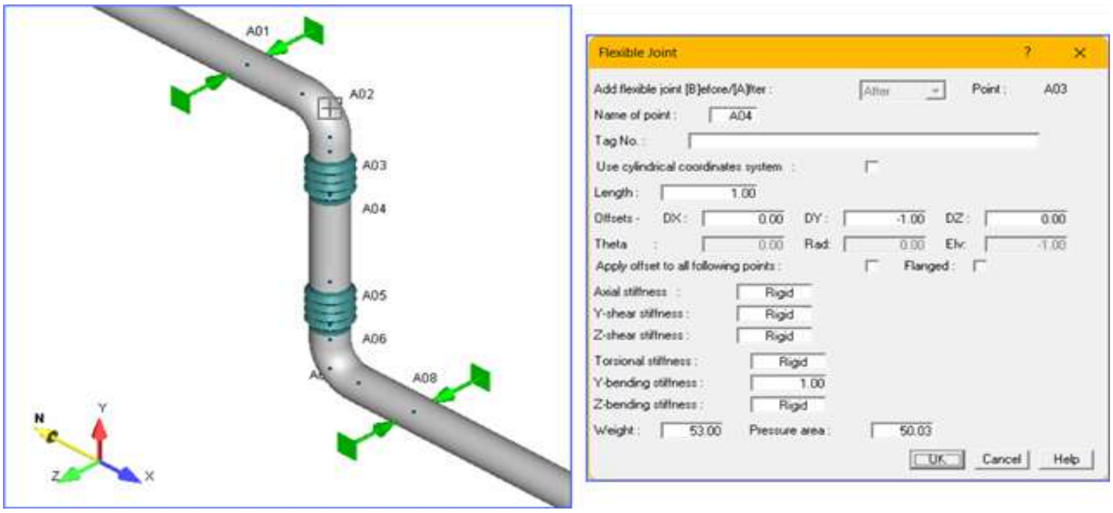

Figure 3: AutoPIPE Piping Stress Analysis Interface

The following parts make up Auto-Pipe:

- A high-level programming language called X describes a collection of very generic tasks that should be translated onto devices (i.e., computing nodes) that are connected in a general way.
- A collection of devices that tasks run on, including both generic and specific devices (like Xilinx Virtex II) [23]. Devices might use a mix of single-core CPUs, chip multiprocessors, and field-programmable gate arrays (FPGAs).
- The process of assigning jobs to either general or specific instruments.
- A platform for modeling and study of system performance with different job and device mappings.
- An assembly of loadable modules that includes interface modules, FPGA bitmaps, and built computational workloads. It allow the implementation of an application when coupled with the right hardware.

3.3.1 Modeling Environment

Auto-Pipe relies on a programming interface comparable to LabVIEW and other graphical programming languages. The development of streaming apps can also be made easier by leveraging the familiar syntax of conventionally sequential programming languages through a variety of projects. The majority of them use a set of features available in languages like C, C++, or Java. Both of these initiatives aim to make it easier to create streaming algorithms for use in hardware and software [24].

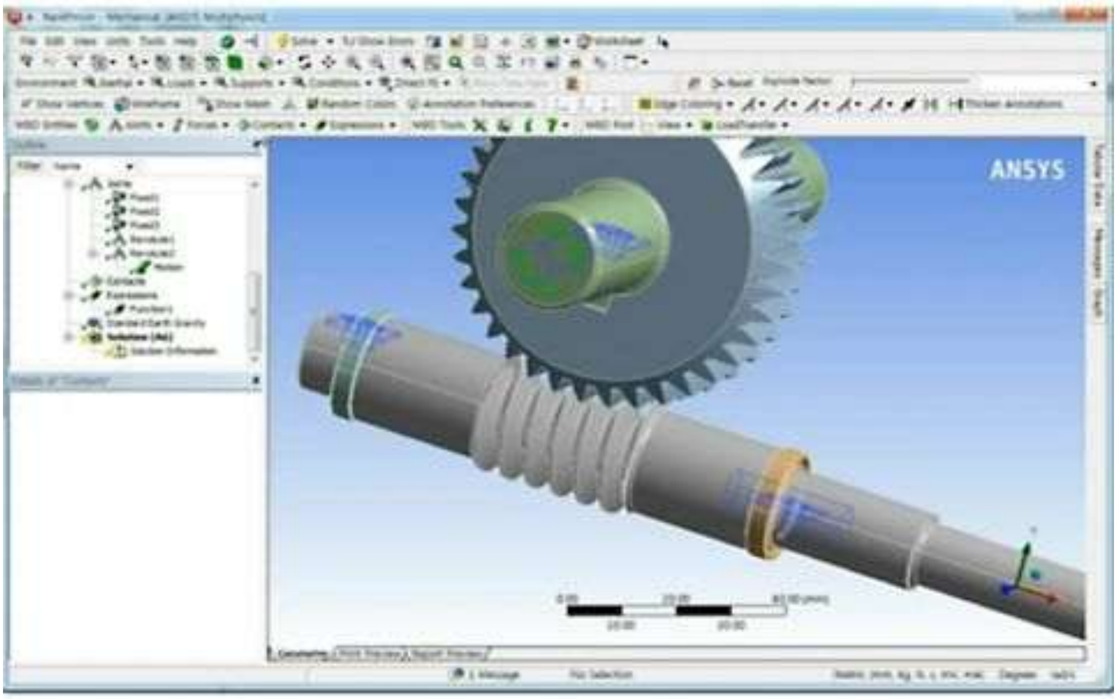

Figure 4: ANSYS Mechanical Interface

3.4 ANSYS

The new owners of SASI rebranded the company's flagship software as ANSYS since they saw it as the core offering. ANSYS is a general-purpose finite element modeling program that can handle a wide range of mechanical problems numerically [25]. This figure 4 shows the ANSYS Mechanical workspace displaying a detailed 3D model of a worm gear assembly. The interface includes geometry trees, toolbars, and analysis settings, allowing engineers to visualize, refine, and simulate mechanical components. The model is shown with annotations and measurement tools that help perform stress, deformation, and contact analyses in ANSYS.

Heat transfer, fluid dynamics, electromagnetic and acoustic problems, linear and non-linear static and dynamic structure analysis, and more mechanical problems.

3.4.1 Finite Element Method (FEM) Approach

Finite element methods (FEM) and other numerical approaches are necessary to resolve this deformation [26]. One numerical model that has applications in many different academic disciplines is the "finite element measurement" method. The finite element approach is commonly used to handle problems involving grid deformation, temperature change, phase flow, and magnetic current power, between many others. How to use the finite element approach successfully: A problem's solution according to the finite element method's detailed steps [27]:

- **Step 1:** Determine the initial conditions, limit values, and governing equations
- **Step 2:** Domain discretization
- **Step 3:** The element equations must be determined.
- **Step 4:** Compile systemic equations
- **Step 5:** Problem boundary conditions are imposed.:
- **Step 6:** Resolving universal equations
- **Step 7:** Results presentation

3.5 Comparative Evaluation of piping stress analysis

Piping stress analysis is still a crucial part of engineering design, guaranteeing the efficiency, dependability, and safety of piping systems in a variety of applications and industries [28]. Table 1 summarises the strengths and shortcomings of the currently available best practices, tools, and methodologies based on the comparative analysis throughout this evaluation. The numerous features and adaptability of CAESAR II and AutoPIPE make them highly sought-after platforms for engineering. The challenges of geometry and dynamic loads, however, are formidable.

Table 1: Comparative Evaluation of different Softwares for Piping Stress Analysis

Parameter	**CAESAR-II**	**AutoPIPE**	**ANSYS**
Analysis Approach	Code-based beam element analysis	Beam element with CAD integration	Finite Element Method (FEM)
Strengths	Fast system-level evaluation; wide code compliance; easy reporting	Strong CAD connectivity; efficient for complex piping layouts; broad code coverage	Highly accurate; detailed 3D and nonlinear stress evaluation
Limitations	Limited for local 3D or nonlinear effects	Less suitable for detailed local stress analysis	Requires expert modelling; time-consuming

3.6 Best Practices of Comparative Piping Stress Analysis

Effective piping stress analysis requires a structured approach that ensures accuracy, safety, and consistency across different software tools. Following best practices helps engineers achieve reliable results while minimizing errors and improving workflow efficiency.

- **Amassing Sufficient Information:** This technique lowers the possibility of simulation errors since accurate evaluations take into account information about the size of the pipes, connectors, material, work conditions, and environment.
- **Observing Standards and Codes:** The relevant standards of practice, which may be ASME B31, ISO 14692 or the API codes, must never be ignored in terms of safety and regulation. The engineers are aware of these codes and this aids them in making good design decisions.
- **Reasonable Representation of Loads:** Accurate identification and recording are required for thermal, pressure, weight, wind, seismic, and transient loads. Failure to identify key load requirements may result in a false signal and may even break down the system.

- **Application of Simple Models Where Necessary:** A detailed model allows high accuracy, but can be excessive in terms of computer time, and may be excessive in terms of errors. It is an efficient way of maintaining completeness of the task through simplification of models with the critical accuracy requirements.
- **Validation of Input Data and Results:** Every item of input data must be compared to the real world, manual calculations, and other software tools. The same is true for the results. This can be helpful in the identification of mistakes and ensuring correct output.
- **Integration with Design Tools:** Select simple-to-integrate applications, such as CAESAR II or AutoPIPE, so that the design and analysis team can operate without difficulties. Bi-directional data interchange decreases the possibility of data mismatch and also requires less time.
- **Dynamic Analysis for Complex Systems:** Dynamic analysis should be used to systems that might be exposed to changing physical conditions, such as earthquakes or dynamic pressure caused by sudden changes in flow [29].
- **Optimization of Support Systems:** Install supports in the beam efficiently to reduce stresses and displacements by using modeling techniques. When supports are positioned correctly, the system may settle over time and avoid incurring unnecessary loads.
- **Regular Training and Skill Development:** Provide funding for engineers' training so they may stay up to date on the newest tools, technology, and standards. The software, ANSYS, Auto PIPE, and CAESAR II all include options that can increase analysis quality and speed.

4 LITERATURE REVIEW

The literature review is based on the recent study of the computational methods and numerical models, which are applicable to the stress analysis of piping through the CAESAR-II, AutoPIPE, and ANSYS tools.

Yin *et al.* (2025) models the high-temperature saturated steam flow in an adiabatic pipe with three 90-degree right-angle bends using a one-way Fluid-Structure Interaction (FSI) technique, which increases local energy dissipation. Supersonic flow (Ma = 1.77) is produced at the outer elbow as the flow transitions through the compressibility of the steam at low pressure (0.8 MPa). The mechanical reactions are highly influenced by boundary conditions. For example, when the intake pressure is high, the fluid's kinetic energy surges suddenly, increasing the elbow impact stress to 430.39 MPa, exceeding the material's yield point. With a maximum strain of 12.09 mm, the second elbow becomes a structural weakness as a result of the cumulative upstream vortices [30].

Kemmler *et al*. (2025) provides a highly detailed fluid-coupled micromechanical technique that uses the discrete element and lattice Boltzmann methods to simulate the installation of a suction bucket in three dimensions. Quantitative and qualitative concordance with experimental data provide credence to the suggested methodology and demonstrate the physical validity of the outcomes. In this way, the paper demonstrates that significant local scenario evaluations in three dimensions on a large scale can be carried out with minimal assumptions made by macromechanical models [31].

Xia *et al.* (2025) The fine particle migration of pipe, the particle loss process, and the corresponding variation in permeability coefficient were effectively simulated using the discrete element program MatDEM and the proposed technique. To ascertain the overall fluid pressure inside each pore and the fluid flow through the pore mouths, a pore density flow approach was first put forward. Pore-jamming is a phenomena that happens simultaneously with small particle movement. The model provides a useful method for statistically analyzing the seepage process in pipes and exploring its mechanisms at the pore scale [32].

Yaseen *et al.* (2024) proposed two fin section types—circular and conical—and investigated two fin configuration types—inline (IA) and staggered (SA)—with 36 fins in each case. All of the fins are made of aluminum and have predetermined measurements. With the same amount of power and mass flow rate, circular pin fins have a larger pressure drop and a maximum temperature that is 0.46 degrees lower than cone fins, particularly in staggered configurations. In addition, the maximum temperature for staggered layouts is up to 1.17% lower for circular fins and 2.035% lower for cone pin fins compared to inline formations [33].

Urcelay *et al.* (2024) The development of delamination in fatigue-loaded composite wind turbine blades requires up-to-date information. A state-of-the-art fatigue propagation model is used to construct a novel cohesive model for delamination fatigue initiation. This model calculates the number of cycles until delaminations are introduced using data from initiation S-N curves. A new ANSYS Mechanical APDL user-defined cohesive element is created by combining the two models. The evolution of the crack growth rate is accurately anticipated in test conditions where fatigue propagation predominates, such as those with repeated delaminations. It is shown that fatigue propagation after initiation can be modeled, and in situations where fatigue initiation is the main factor, an adequate prediction is also achieved [34].

Kubiak and Fotovat (2023). Harmonic loads acting in the plane of the plate are applied to the plates that have boundary conditions that are simply supported. The equation of motion is derived from the static equilibrium path using a novel SIM that incorporates LS fitting. Solving these equations allows one to determine the dynamic buckling loads using the Budiansky-Hutchinson and Volmir criteria, as well as the dynamic reactions of GFRP laminated plates with varying layer configurations. Utilizing the commercial FEM, specifically ANSYS software, static equilibrium routes are derived. Once again, using ANSYS software, compare SIM results with FEM results, and verify that, when considering all sorts of material couplings, dynamic responses are valid [35].

Table 2 This table presents a consolidated overview of recent studies, highlighting research focus, simulation approach, key outcomes, limitations, and suggested advancements in piping stress modeling and analysis.

Table 2: Review of Recent Studies Related to Piping Stress and Simulation Techniques

Reference	Study On	Approach	Key Findings	Challenges / Limitations	Future Directions
Yin et al., (2025)	Mechanical reaction and steam flow characteristics in the discharge pipes of nuclear safety valves	One-way FSI simulation using adiabatic pipe model with 90° bends	Supersonic flow (Ma = 1.77) happens at the outer elbow due to centrifugal forces and Dean vortices, while the second elbow is a structural weak spot.	High local stress (430.39 MPa) exceeds material yield; limited to one-way FSI (no feedback from structure to fluid)	Extend to two-way FSI models; experimental validation; material optimization for high-stress regions
Kemmler et al., (2025)	Suction bucket installation in marine foundations	3D fluid-coupled micromechanical model using LBM + DEM	Model validated with experiments; successfully captures local fluidization and suction-driven installation phenomena	Computationally expensive for large-scale scenarios	Apply to real-scale offshore foundations; integrate multi-physics coupling for sediment-fluid interactions
Xia et al., (2025)	Fine particle migration and piping seepage process	Pore density flow method implemented in MatDEM software	Simulated fine particle migration, permeability change, and pore-jamming; effective pore-scale modeling	Focused mainly on microscale phenomena; lacks coupling with macroscopic soil behavior	Extend to multi-scale simulations; couple with continuum models for practical engineering applications
Yaseen et al., (2024)	Thermal-fluid performance of microelectronic cooling fins	Comparative analysis of circular and conical fin geometries in inline and staggered arrangements	Circular fins yield 0.46% lower maximum temperature; staggered layouts enhance cooling but increase pressure drop	Trade-off between cooling efficiency and pressure loss; limited experimental verification	Optimize fin spacing and geometry; use advanced materials or nanofluids for enhanced heat dissipation
Urcelay et al., (2024)	Fatigue delamination in composite wind turbine blades	An ANSYS Mechanical APDL-based cohesive model of fatigue start and propagation	Accurately predicts delamination growth and crack evolution under fatigue loading	Complex calibration needed; computationally intensive for large structures	Apply model to large-scale blade simulations; improve computational efficiency and fatigue prediction accuracy
Kubiak & Fotovat, (2023)	Laminated composite plate dynamic buckling	Validation with ANSYS FEM of equation derivation using SIM and LS fitting	Simplified method accurately predicts dynamic responses; reduced computation time	Limited to simply supported boundary conditions; specific to harmonic loading	Extend method to other boundary/loading types; explore nonlinear and transient effects

5 CONCLUSION AND FUTURE WORK

This review reveals that CAESAR-II, AutoPIPE and ANSYS offer useful functions to analyze piping-stress analysis but their usefulness is determined by the complexity of design and analysis needs. The CAESAR-II is still very efficient in system level-stress testing with full-fleet ASME code compliance with timely and dependable results to validate the design. AutoPIPE has an easy-to-use interface and advanced CAD integration, which is appropriate in the case of complex pipeline networks and fast iteration. ANSYS, which is founded on Finite Element Method, offers accurate and localized stress and deformation data, especially on non linear or transient scenarios, although it demands more expertise and computing capabilities. On the whole, CAESAR-II and AutoPIPE would be the most suitable solutions in large-scale industrial systems, when compliance and speed is a priority, whereas ANSYS proves to be the best choice in terms of detailed mechanical analysis and high-level design verification. Nevertheless, the existing tools are struggling to model coupled thermal, dynamic and fluid structure effects in a high computing capacity.

Future efforts in this area should be focused on the incorporation of hybrid computational models that integrate code based design checks with more detailed finite element analysis to achieve greater reliability. The use of AI and machine learning algorithms can make the model calibration process, load prediction, and fault detection real-time automation. Standardization of data exchange formats and API integration between CAESAR-II, AutoPIPE and ANSYS should also be prioritized in order to enhance interoperability. All in all, the future of the piping stress analysis sphere is to develop adaptability, intelligent and interoperable tools that can model multi-physics complex scenarios with high efficiency to promote more affordable and safer pipeline engineering.

REFERENCES

[1] U. Sarwar *et al.*, "Mechanics Enhancing pipeline integrity : a comprehensive review of deep learning-enabled finite element analysis for stress corrosion cracking prediction ABSTRACT," vol. 2060, 2024, doi: 10.1080/19942060.2024.2302906.

[2] W. S. Sun, "CHAPTER B4 STRESS ANALYSIS OF PIPING SYSTEMS," pp. 107–214, 2004.

[3] G. Zuccaro, D. De Gregorio, M. F. Leone, S. Sessa, S. Nardone, and F. L. Perelli, "CAESAR II Tool : Complementary Analyses for Emergency Planning Based on Seismic Risks Impact Evaluations," 2021.

[4] P. B. Patel, "Thermal Efficiency and Design Considerations in Liquid Cooling Systems," *Int. J. Eng. Sci. Math.*, vol. 10, no. 3, pp. 118–125, 2021.

[5] L. K. Loon, "Universiti Teknologi PETRONAS Bandar Sri Iskandar 31750 Tronoh Perak Darul Ridzuan," no. January, 2009.

[6] Vaidehi Shah, "Next-Gen Emergency Communication Using Low-Power Wide-Area and Software-Defined WANS," *Int. J. Adv. Res. Sci. Commun. Technol.*, pp. 600–609, Sep. 2022, doi: 10.48175/IJARSCT-8349M.

[7] K. T. B. Jayachandraiah, "Modeling and Analysis of the Crankshaft Using Ansys Software," pp. 84–89, 2013.

[8] V. Thakaran, "A Comparative Study of Piping Stress Analysis Methods with Different Tools , Techniques , and Best Practices," pp. 675–684, 2022, doi: 10.48175/IJARSCT-7868D.

[9] S. K. Chintagunta, "AI in Code , Testing , and Deployment : A Survey on Productivity Enhancement in Modern Software Engineering," *Int. J. Res. Anal. Rev.*, vol. 10, no. 4, pp. 747–752, 2023.

[10] S. P. Kalava, "Enhancing Software Development with AI-Driven Code Reviews," *North Am. J. Eng. Res.*, vol. 5, no. 2, pp. 1–7, 2024.

[11] S. Koorse, M. Roy, M. Janardhana, and S. Seetharamu, "AN OVERVIEW OF STRESS ANALYSIS OF HIGH-ENERGY PIPELINE SYSTEMS USED IN THERMAL POWER PLANTS," pp. 538–542, 2014.

[12] V. Thakran, "A Review of Flexibility Analysis in Piping Systems : Insights from CAESAR-II Applications," *TIJER - Int. Res. J.*, vol. 10, no. 10, pp. 68–73, 2023.

[13] D. Coverdale *et al.*, "Industrial Piping System : Design and Corrosion Protection," pp. 1–28, 2025.

[14] Z. Lin and S. Li, "Heliyon Study on interfacial mechanical properties of bonded pipe joint under dynamic load," *Heliyon*, vol. 10, no. 16, p. e36369, 2024, doi: 10.1016/j.heliyon.2024.e36369.

[15] T. Arumugam, S. Devi, V. Kumar, S. Karuppanan, and M. Ovinis, "applied sciences The Influence of Axial Compressive Stress and Internal Pressure on a Pipeline Network : A Review," 2023.

[16] S. Hwang, M. Kim, and J. Lee, "Thermal Stress Analysis of Process Piping System Installed on LNG Vessel Subject to Hull Design Loads," 2020.

[17] V. Panchal, "Thermal and Power Management Challenges in High-Performance Mobile Processors," *Int. J. Innov. Res. Sci. Eng. Technol.*, vol. 13, no. 11, Nov. 2024, doi: 10.15680/IJIRSET.2024.1311014.

[18] D. N. Veritas, "STRUCTURAL ANALYSIS OF PIPING SYSTEMS OCTOBER 2008," no. October, 2008.

[19] X. Li, Z. Li, and G. Wu, "Modular and Offsite Construction of Piping: Current Barriers and Route," *Appl. Sci.*, vol. 7, no. 6, p. 547, May 2017, doi: 10.3390/app7060547.

[20] K. Huang, J. Wu, M. Hu, H. Xiang, and Z. Zhang, "Optimization and Stress Analysis of Underground Oil Pipelines in High and Steep Slope Areas," pp. 477–483, 2015.

[21] R. Baskaran, "PRODUCT ANALYSIS AND STRESS ANALYSIS OF PROCESS PIPING ROUTING," no. September, 2019.

[22] R. Gengadevi, "Validation of Piping Stresses With Caesar Ii and Fem and Comparision of Results," vol. 1, no. 1, pp. 16–20, 2020.

[23] M. A. Franklin, E. J. Tyson, J. Buckley, P. Crowley, J. Maschmeyer, and S. Louis, "Auto-Pipe and the X Language : A Pipeline Design Tool and Description Language," 2006.

[24] A. R. Duggasani, "The Integration of Java-Based Applications in Software Development and AI: A Conceptual Review," *Int. J. Adv. Res. Sci. Commun. Technol.*, vol. 5, no. 1, p. 11, 2025.

[25] P. Nagra, "FEM analysis and optimization of crankshaft using ANSYS — A review," vol. 6, no. 6, pp. 378–383, 2019.

[26] A. Khaleel, H. Abdul, and H. Abdul, "Application of Finite Element Technique : A Review Study Application of Finite Element Technique : A Review Study," no. April, 2023, doi: 10.31699/IJCPE.2023.1.13.

[27] I. Erhunmwun and U. B. Ikponmwosa, "Review on finite element method," no. November 2017, 2018, doi: 10.4314/jasem.v21i5.30.

[28] R. Patel and P. B. Patel, "The Role of Simulation & Engineering Software in Optimizing Mechanical System Performance," *TIJER – Int. Res. J.*, vol. 11, no. 6, pp. 1–6, 2024.

[29] N. Jaćimović, "Analysis of piping stress intensification factors based on numerical models," *Int. J. Press. Vessel. Pip.*, vol. 163, pp. 8–14, Jun. 2018, doi: 10.1016/j.ijpvp.2018.03.009.

[30] S. Yin, X. Ji, X. Lu, and Q. Zhang, "International Journal of Advanced Nuclear Reactor Design and Technology Simulation study of steam flow behavior and piping mechanical response analysis in safety valve discharge pipes with fluid-structure interaction," *Int. J. Adv. Nucl. React. Des. Technol.*, vol. 7, no. 3, pp. 308–318, 2025, doi: 10.1016/j.jandt.2025.07.006.

[31] S. Kemmler *et al.*, "Computers and Geotechnics A fully-resolved micromechanical simulation of piping erosion during a suction bucket installation," *Comput. Geotech.*, vol. 186, no. December 2024, p. 107375, 2025, doi:

10.1016/j.compgeo.2025.107375.

[32] W. Xia, C. Liu, H. Liu, T. Zhao, and Y. Zhu, "Rock Mechanics Bulletin Modeling of particle migration in piping based on an improved discrete element method," *Rock Mech. Bull.*, vol. 4, no. 1, p. 100151, 2025, doi: 10.1016/j.rockmb.2024.100151.

[33] S. J. Yaseen, Z. K. Radhi, R. L. Natoosh, R. Al-sabur, R. Z. Homod, and H. I. Mohammed, "A computational search for the optimal microelectronic heat sink using ANSYS Icepak," *Int. J. Thermofluids*, vol. 23, no. July, p. 100759, 2024, doi: 10.1016/j.ijft.2024.100759.

[34] I. Urcelay, B. Lau, V. Bak, A. Turon, and E. Lindgaard, "An ANSYS user cohesive element for the modelling of fatigue initiation and propagation of delaminations in composite structures," *Eng. Fract. Mech.*, vol. 308, no. July, p. 110337, 2024, doi: 10.1016/j.engfracmech.2024.110337.

[35] T. Kubiak and M. B. Fotovat, "Dynamic response and dynamic buckling of general laminated plates : A semi-inverse method," *Compos. Struct.*, vol. 324, no. May, p. 117548, 2023, doi: 10.1016/j.compstruct.2023.117548.