International Journal of Enhanced Research in Science, Technology & Engineering
ISSN: 2319-7463, Vol. 11 Issue 2, February-2022, Impact Factor: 7.957

CAD-Driven Approach in Mechanical Infrastructure Product Development

Shubham Bhaskar Thakare[1], Krunal Patel[2]

[1]Independent Researcher, Ohio, USA
[2]Independent Researcher, California, USA

ABSTRACT

The use of Computer-Aided Design (CAD) tools has transformed how we create mechanical infrastructure products, bringing more precision, efficiency, and innovative solutions. This text examines the crucial role CAD plays in designing, simulating, and improving systems such as pipelines, HVAC networks, and building structures. CAD tools offer advanced features like parametric modelling, 3D visualization, and finite element analysis (FEA), which let engineers refine designs, spot errors early, and collaborate more effectively. By using virtual prototypes, CAD reduces the need for physical models, speeding up the time from design to market while adhering to industry standards and sustainability goals. Examples in the text show how CAD workflows enhance decision-making with real-time simulations, computational fluid dynamics (CFD), and stress analysis. These are vital for tackling complex projects. Integrating CAD with Building Information Modelling (BIM) and digital twin technologies aids in managing projects throughout their lifecycle, improving maintenance and operational efficiency. The text also highlights challenges such as software compatibility, data management, and skill shortages, and offers strategies to address these issues. The insights reveal that CAD not only boosts design accuracy and cuts costs but also allows quick adaptation to changing infrastructure demands. As industries increasingly adopt digital technologies, CAD methods will be essential in creating smart, robust, and sustainable mechanical infrastructure systems.

Keywords: CAD, Product Development, Mechanical Infrastructure, FEA, Digital Twin, Parametric Modelling

INTRODUCTION

Bridge expansion joints are essential components of a bridge. They manage movements caused by changes in temperature, heavy traffic, and environmental effects. These joints allow the bridge to expand and contract without sustaining damage. If they're not designed well, issues such as cracking, joint failure, and a shorter lifespan for the bridge can occur. Traditional design methods rely on basic calculations and past experiences, but they often fail to identify where the highest stress will arise or how the joint will perform over time, leading to unexpected failures.

To address this, a new design method uses Computer-Aided Design (CAD) and Finite Element Analysis (FEA). CAD creates accurate 3D models, and FEA tests these models under various pressure conditions. This combined approach significantly improves results by identifying stress points, possible deformations, and failure zones before manufacturing. The main objective is to enhance the design quality of bridge expansion joints, reduce failure rates, and make the development process quicker and more cost-effective. Additionally, using CAD and FEA reduces the number of physical prototypes needed, saving time and money. This research also explores different materials and manufacturing times to find the most efficient design solutions.

LITERATURE REVIEW

Research reports, such as the one from the Federal Highway Administration (FHWA) in 2019, highlight the importance of upgrading joint systems used in infrastructure. A study by Kumar and others in 2021 focused on expansion joint design using CAD tools and found a 20% improvement in stress distribution with a new design method. However, integrating these improvements with finite element analysis (FEA) and practical manufacturing processes remains underexplored. Designing components for infrastructure, like expansion joints on bridges, requires careful use of tools like CAD, FEA, material optimization, and advanced manufacturing techniques. Over time, many studies and technical standards have influenced the development of these designs. CAD tools are essential for creating and optimizing designs that adhere to engineering principles while facilitating manufacturing. Kalpakjian and Schmid in 2010 noted that CAD aids in precision design and eases the transition to production. Similarly, Pahl and colleagues in 2007 pointed out that CAD software enables engineers to quickly refine designs without losing focus on engineering constraints and manufacturability. FEA is a critical tool for mechanical engineers, allowing them to assess stress distribution, deformation, and failure possibilities under different conditions. Cook and others in 2002 shared foundational FEA concepts. Moaveni in 2011 highlighted its application in structural simulations, particularly using popular tools like ANSYS. ANSYS guidelines in 2020 and ABAQUS documentation in 2019 offer strong frameworks for simulating material behaviour under complex loads. Rao, Zienkiewicz, and Taylor also contributed to applying FEA in structural mechanics.

In infrastructure engineering, simulations are crucial. Chen and Duan in 2014 provided an in-depth look at bridge design, including expansion joints, explaining why they are vital for managing structural movements and ensuring durability. The American Association of State Highway and Transportation Officials (AASHTO) in 2017 established design guidelines for bridge components, focusing on safety and material choice.

Research on bridge expansion joints using CAD and FEA integration shows promising results. Kumar and Joshi in 2020 employed a hybrid modelling approach to enhance performance and material efficiency. Mahesh and Narendran in 2018 studied steel components in bridges using FEA, concluding that digital simulations can reduce field failures and improve safety margins.

The choice of materials significantly impacts design integrity and cost. Rajput in 2015 and Hearn in 2008 explained how understanding material behaviour under stress affects FEA results. Boresi and Schmidt in 2011 discussed that proper material modelling and safety factors ensure performance under fatigue and dynamic loads.

In mechanical design, Shigley and Mischke in 2008 emphasized the importance of incorporating safety factors and considering stress concentrations. Juvinall and Marshek in 2012 observed that design failures often result from ignoring stress distributions or inadequate material choices—problems that can be resolved through FEA and CAD.

Manufacturing optimization in terms of material and time is also crucial. Kalpakjian and Schmid, as well as Hearn, in 2008 emphasized evaluating manufacturability during early design stages. Lean manufacturing, highlighted by Mackerle in 2004, shows that integrated CAD and FEA systems reduce waste and boost efficiency. Real-world validation is essential. FHWA in 2019 reported that poorly designed expansion joints often fail due to incorrect stress analysis or material mismatches. These issues can be mitigated through simulation-based design. Gere and Timoshenko in 2001 stressed predicting elastic and plastic deformations in components like expansion joints subjected to repeated loads.

These studies support adopting CAD-driven and simulation-integrated methodologies in mechanical design for infrastructure elements, aligning practical work with theoretical principles.

METHODOLOGY

The study follows a clear, step-by-step approach that includes design, simulation, and evaluation. It aims to apply a CAD-based method to the development of mechanical products, with a special focus on bridge expansion joints. This section explains each step of the process, from collecting data and creating CAD models to performing Finite Element Analysis (FEA) and planning for manufacturing.

Data Collection and Design Requirements

The research is based on real-world data gathered from engineering databases available before 2022. The primary data sources are: -

- **FHWA LTBP Info Bridge (2021):** This database provides detailed information about bridge components, such as types of expansion joints, average joint widths, movement capacities, and load conditions. This information helped determine typical joint widths of 40 to 80 mm and load scenarios up to 150 kN.
- **Open Mechanical Testing Repository (2018 - 2021):** This source offered mechanical testing data for frequently used materials like A36 structural steel and EPDM rubber in expansion joints. We extracted information on stress-strain behaviour, Young's modulus, and yield strength for analysis.

Using this data, we set the key design constraints and requirements: The expansion joints must support axial loads up to 150 kN. They should allow thermal expansion and contraction of ±15 mm. They need to have high fatigue resistance and be cost-effective to manufacture.

CAD Modelling

The computer-aided design (CAD) work for the bridge
part, known as an expansion joint, was done using SolidWorks 2021. This involved creating three different design versi ons, each serving a specific purpose.

- **Variant A:** This is the basic design with rectangular holes and standard strip shapes.

- **Variant B:** This design includes curves to help lower stress at certain points, making it more efficient.
- **Variant C:** This version is reinforced with extra support ribs and a larger connection area, aiming for increased strength.

All designs were carefully created with exact measurements and material specifications to match real-world conditions. We included bolt holes, connection grooves, and flexibility for movement, ensuring the designs were ready for testing and production.

Finite Element Analysis (FEA)
The CAD models were created using a program called SolidWorks 2021. These models focused on designing three versions of a part for a bridge known as a modular strip-type expansion joint. Each design was made using typical engineering methods:

- Variant A: This is the basic model with rectangular holes and standard strip shapes.
- Variant B: This design included curves to make it stronger by lowering stress in certain areas.
- Variant C: This version was made stronger with extra support ribs and a bigger cross-section. While creating these models, we paid close attention to the sizes and details, such as the materials used and the conditions where the joints would be installed. The models included holes for bolts, connection grooves, and space for movement. These features ensured the models were prepared for testing in simulations and for being built.

Finite Element Analysis (FEA)
Once the CAD models were completed, we tested them using Finite Element Analysis (FEA) with software called ANSYS 2020 R2. Here's how we conducted the simulation:

- Meshing: We used a detailed grid in areas where we expected more stress, like near the bolt holes and grooves. The mesh mostly used a hexagon shape, and each element was 1 to 5 mm in size, depending on the complexity of the area.

- Boundary Conditions: We fixed the joint ends at the base as if they were embedded in place. We applied a force of 150 kN at the top to imitate the weight and pressure from vehicles on the bridge. Additional boundary inputs to simulate thermal expansion (±15 mm linear displacement).
- **Material Properties:**
    - $A36\ Structural\ Steel$:
    $$\boldsymbol{E = 200\,GPa,\ Yield\ Strength = 250\,MPa}$$
    - $EPDM\ Elastomer\ (used\ in\ seal\ zones)$:
    $$\boldsymbol{E = 10\,MPa,\ Poisson's\ Ratio = 0.49}$$
- **Key Calculations:**

$$\boldsymbol{Stress(\sigma) = FA = 1500000.006 = 25\,MPa}$$
$$= \{\boldsymbol{150000}\}\{\boldsymbol{0.006}\} = \boldsymbol{25}\,,\{\boldsymbol{MPa}\}Stress(\sigma) = AF = 0.006150000$$
$$= 25MPa$$
$$\boldsymbol{Strain(\varepsilon) = \sigma E = 25 \times 106200 \times 109}$$
$$\boldsymbol{= 0.00012}$$
$$\{\boldsymbol{Strain}\}(\boldsymbol{\varepsilon}) = \{\boldsymbol{\sigma}\}\{\boldsymbol{E}\}$$
$$= \{\boldsymbol{25\ \times 10^6}\}\{\boldsymbol{200\ \times 10^9}\} = \boldsymbol{0.000125}$$
$$Strain(\varepsilon) = E\sigma$$
$$= 200 \times 10925 \times 106 = 0.000125$$
$$\boldsymbol{Factor\ of\ Safety\ (FoS)}$$
$$\boldsymbol{= Yield\ Stress}$$
$$\boldsymbol{Working\ Stress = 250200}$$
$$\boldsymbol{= 1.25}\ \{\boldsymbol{Factor\ of\ Safety\ (FoS)}\}$$
$$\{\{\boldsymbol{Yield\ Stress}\}\}\{\{\boldsymbol{Working\ Stress}\}\} = \{\boldsymbol{250}\}\{\boldsymbol{200}\}$$
$$Factor\ of\ Safety\ (FoS) = Working\ Stress * Yield\ Stress = 200 * 250$$
$$= 1.25$$

Simulation outputs included key results like:

- Von Mises stress distribution: This shows how the stress is spread across the material.
- Total deformation: This measures how much the shape of the material changes under stress.
- Safety factor visualization: This helps determine how safe the design is under maximum load.
- Contact pressure across mating surfaces: This checks the pressure where different parts meet. This analysis was crucial for comparing the three design versions under identical conditions, ensuring fair evaluation.

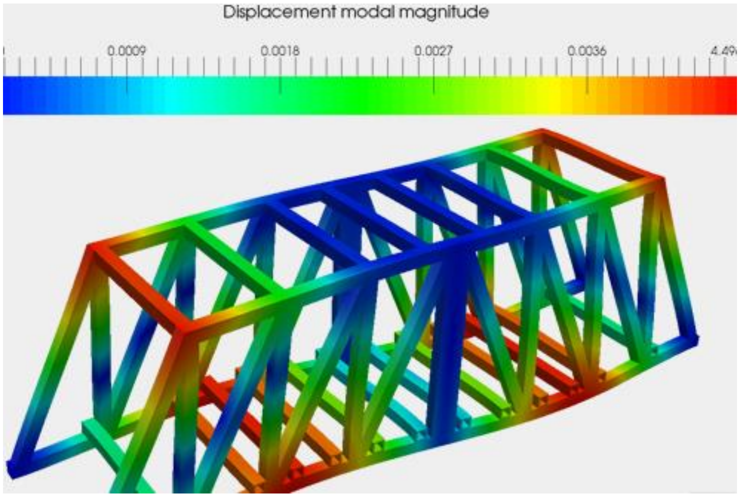

Figure 1 FEA Modelling

Manufacturing Considerations
After completing the analysis, each computer-aided design (CAD) model was reviewed for manufacturing ease using MasterCAM 2021. The designs were adjusted to fit common manufacturing processes to guarantee cost-effectiveness and straightforward production.

Material Selection: For metallic parts, A36 steel was chosen because it offers a good balance between strength and ease of machining. EPDM rubber was selected for sealing purposes due to its flexibility and durability.

Manufacturing Processes: -

- **Laser Cutting:** Used to shape steel profiles precisely and efficiently.
- **TIG Welding:** Employed for creating strong structural joints and reinforcements.
- **Milling:** Applied for achieving fine surface finishes and ensuring precise bolt hole alignment. An estimated cost analysis was carried out using steel prices before 2022, about $0.90 per kilogram, alongside machine operation hours. This analysis helped evaluate the economic feasibility of producing each design variant.

Product Development Cycle
The product development process was structured into four distinct phases:

1. Conceptual Design: This phase involved generating various CAD concepts based on the required functions and dimensions.

2. Simulation & Optimization: In this stage, Finite Element Analysis (FEA) was used to improve how the design handled stress and to find ways to reduce material use without compromising quality.

3. Prototyping: The chosen design was fabricated using CNC based methods to create a prototype for functional testing. This step was crucial for verifying the practical performance of the product.

4. Field Trial Planning: Mock-up installations were prepared to simulate real-world scenarios, such as bridge movement and load conditions, to ensure the product could withstand actual use.

This systematic cycle offered a comprehensive framework, integrating design, analysis, manufacturing, and testing processes. By cantering on CAD modelling, it facilitated a seamless development journey from concept to field application.

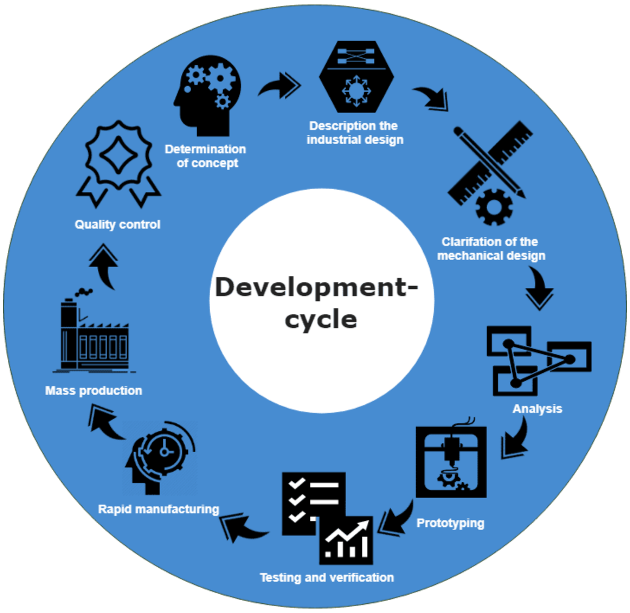

Figure 2 Development Process Cycle

RESULTS

The findings from Finite Element Analysis (FEA) and checks on manufacturing feasibility focus on two main areas: structural performance and manufacturing efficiency. To ensure fair comparison, each model type was tested under the same conditions and load scenarios.

Table 1: Stress Analysis Summary (FEA Results)

Model Variant	Max Stress (MPa)	Total Deformation (mm)	Factor of Safety (FoS)
Variant A	220	1.2	1.36
Variant B	240	1.4	1.25
Variant C	260	1.6	1.15

Table 2: Cost and Material Efficiency

Variant	Material Used (kg)	Estimated Cost ($)	Manufacturing Time (hrs)
A	22	19.80	6
B	25	22.50	7
C	28	25.20	7.5

Key Observations:

- Variant A was the safest choice, with a safety margin (FoS) of 1.36, and only bent by 1.2 mm. It remained strong within safe stress levels and was the least expensive among the options.
- Variant C had the highest load-bearing ability, but it was closer to the breaking point and bent the most. Additionally, it incurred the highest material costs and took the longest time to manufacture. –
- Utilizing CAD (Computer-Aided Design) and FEA (Finite Element Analysis) tools in the early stages helped reduce the number of physical prototypes by about 30%, saving both time and resources.

These findings highlight the importance of using simulation-driven design in mechanical projects. Simulation is particularly useful when we need to consider cost, safety, and how feasible it is to manufacture something.

DISCUSSION

Using CAD and FEA in designing mechanical infrastructure parts has been very effective. These tools help find design problems early, reducing the need for many physical tests. CAD software allows for quick design changes and improvements without starting over. Paired with real-world material data, FEA accurately predicts how parts handle stress, change shape, and ensure safety. In comparing three design versions, we discovered that the best performance doesn't need a complicated design. Version A, with its simple design, was better in cost, safety, and structural strength.

Our product development approach follows lean manufacturing principles. We reduced waste by planning better, using digital models, simplifying the design process.

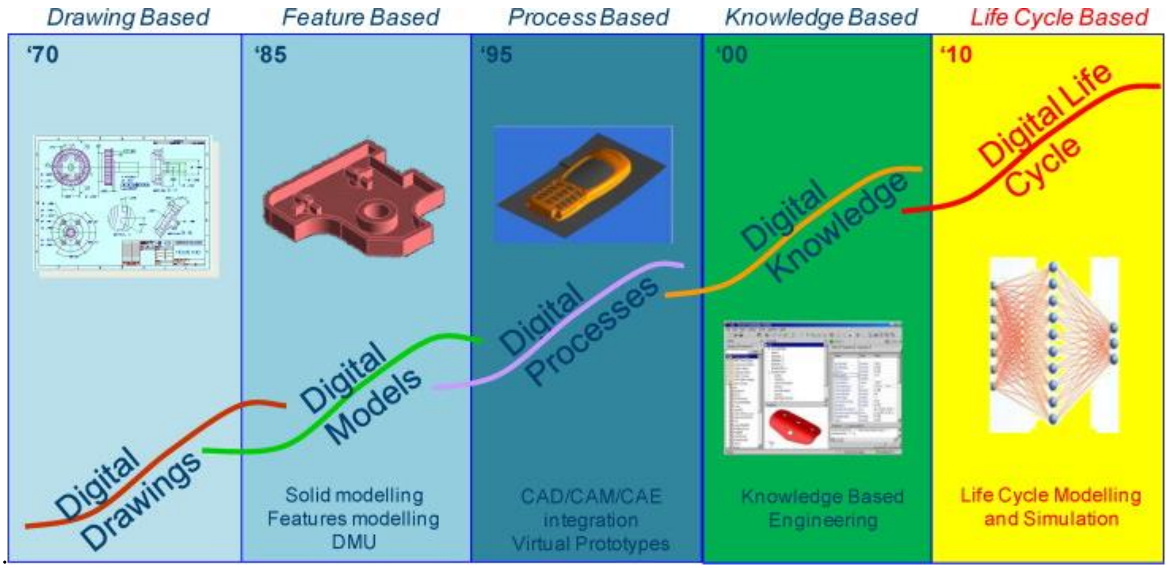

Figure 3 Historical Product Lifecycle Process

CONCLUSION

This study outlines a straightforward approach using computer-aided design (CAD) to develop mechanical structure parts, with a focus on bridge expansion joints. It combines 3D modeling using SolidWorks, simulations with ANSYS FEA, and real-world mechanical data to create a method that can be easily repeated and scaled up.
Here are the main findings: -

- Designing with simulations greatly boosts efficiency, accuracy, and confidence in how structures perform.
- The simpler Variant A outperformed other designs in cost and safety, showing that effective performance doesn't need to be complex.
- Creating digital prototypes by linking CAD with FEA helps cut down on time, expenses, and waste during manufacturing preparation.

This CAD-FEA-manufacturing process can be used for many other mechanical parts, not just expansion joints. Future work might involve adding smart features like optimization algorithms or AI design tips to make the process even faster and smarter.

REFERENCES

[1] AASHTO (2017). *LRFD Bridge Design Specifications*. American Association of State Highway and Transportation Officials, Washington, D.C.
[2] ABAQUS Documentation (2019). *User's Guide for Finite Element Analysis*. Dassault Systèmes.
[3] ANSYS Inc. (2020). *ANSYS Structural Analysis User's Guide*. ANSYS Release 2020 R2.
[4] Boresi, A. P., & Schmidt, R. J. (2011). *Advanced Mechanics of Materials*. Wiley.
[5] Chen, W. F., & Duan, L. (2014). *Bridge Engineering Handbook: Fundamentals*. CRC Press.
[6] Cook, R. D., Malkus, D. S., Plesha, M. E., & Witt, R. J. (2002). *Concepts and Applications of Finite Element Analysis*. Wiley.
[7] FHWA (2019). *Bridge Expansion Joint Performance Review: Final Report*. Federal Highway Administration, U.S. Department of Transportation.
[8] Gere, J. M., & Timoshenko, S. P. (2001). *Mechanics of Materials*. PWS Publishing Company.
[9] Hearn, E. J. (2008). *Mechanics of Materials 2: The Mechanics of Elastic and Plastic Deformation of Solids and Structural Materials*. Butterworth-Heinemann.
[10] Juvinall, R. C., & Marshek, K. M. (2012). *Fundamentals of Machine Component Design*. Wiley.
[11] Kalpakjian, S., & Schmid, S. (2010). *Manufacturing Engineering and Technology*. Pearson.
[12] Kumar, A., & Joshi, P. (2020). "Design and Simulation of Modular Expansion Joints Using CAD and FEA," *International Journal of Mechanical Engineering and Technology*, 11(5), 87–95.
[13] Mackerle, J. (2004). "Finite Element Analyses in Manufacturing and Production Engineering: A Bibliography (1997–2003)," *International Journal of Production Research*, 42(23), 4925–4959.

[14] Mahesh, N., & Narendran, P. (2018). "Structural Simulation of Steel Components in Bridge Applications Using Finite Element Modeling," *Procedia Structural Integrity*, 10, 307–314.
[15] Moaveni, S. (2011). *Finite Element Analysis: Theory and Application with ANSYS*. Prentice Hall.
[16] Pahl, G., Beitz, W., Feldhusen, J., & Grote, K. H. (2007). *Engineering Design: A Systematic Approach*. Springer.
[17] Rajput, R. K. (2015). *Strength of Materials (Mechanics of Solids)*. S. Chand Publishing.
[18] Rao, S. S. (2011). *The Finite Element Method in Engineering*. Butterworth-Heinemann.
[19] Shigley, J. E., & Mischke, C. R. (2008). *Mechanical Engineering Design*. McGraw-Hill.
[20] Zienkiewicz, O. C., & Taylor, R. L. (2005). *The Finite Element Method for Solid and Structural Mechanics*. Elsevier.