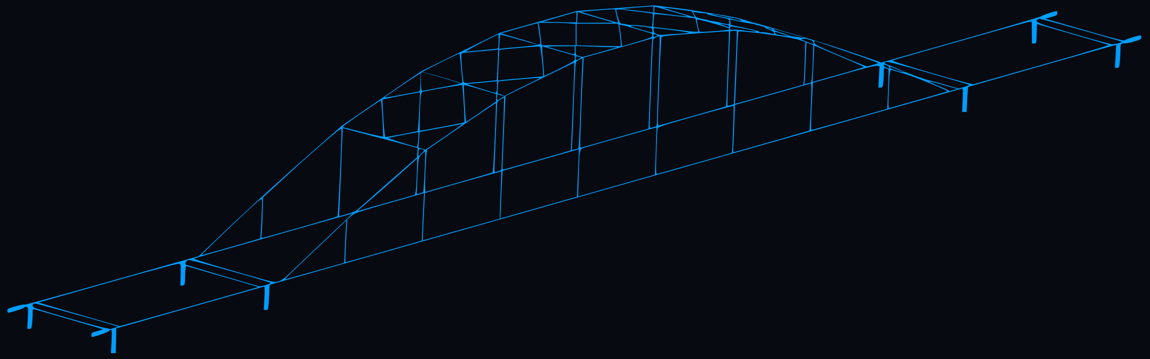
DESIGN AND DEVELOPMENT OF A SCENARIO ANALYSIS TOOL FOR A BRIDGE USING A PHYSICS-BASED DIGITAL TWIN



HEMANAND KALYANASUNDARAM

DESIGN AND DEVELOPMENT OF A SCENARIO ANALYSIS TOOL FOR A BRIDGE USING A PHYSICS-BASED DIGITAL TWIN | HEMANAND KALYANASUNDARAM

UNIVERSITY OF TWENTE

Design and development of a scenario analysis tool for a bridge using a physics-based digital twin

Hemanand Kalyanasundaram

Graduation committee members:

Chairman and Director of PDEng. Program	
Prof.dr.ir. D.J. Schipper	University of Twente
Supervisors	
Prof.dr.ir. T. Tinga	University of Twente
dr.ir. R. Loendersloot	University of Twente
Members	
dr.ir. A. Hartmann	University of Twente
ing. M. Bosveld	RWS Oost Nederland

This work was performed at the Dynamics Based Maintenance (DBM) group, Faculty of Engineering Technology, University of Twente, Enschede, the Netherlands as a part of the Kunstwerken in Control (KiC) project.

Hemanand Kalyanasundaram
Design and development of a scenario analysis tool for a bridge using a physics-based digital twin.
Cover design: Sathya Prabha Suresh Kumar
Printed by Gildeprint, Enschede, The Netherlands

DESIGN AND DEVELOPMENT OF A SCENARIO ANALYSIS TOOL FOR A BRIDGE USING A PHYSICS-BASED DIGITAL TWIN

PDEng Thesis

to obtain the degree of
Professional Doctorate in Engineering (PDEng) at the University of Twente,
on the authority of the rector magnificus,
prof. dr. T.T.M. Palstra,
on account of the decision of the graduation committee,
to be defended
on Wednesday the 9 of December 2020 at 13.00 hours

by

Hemanand Kalyanasundaram
born on the 19 August 1990
in Aruppukottai, India

This PDEng Thesis has been approved by:

Thesis Supervisor: Prof.dr.ir. T. Tinga
Co-supervisor(s): Dr.ir. R. Loendersloot

Summary

Bridges are a vital part of a country's infrastructure. In the Netherlands, there are approximately 3200 bridges and they support the traffic flow by providing passage over the highways, canals, rivers etc. Rijkswaterstaat (RWS), the Dutch infrastructure and water management board owns and maintains many bridges throughout the country. Many of these bridges are more than half a century old, and there is a mismatch between current loads and their designed capacity due to increasing traffic and heavier vehicles. This mismatch often leads to structural damage and failures before achieving the designed life and the cost of maintenance increases as the bridge ages. The bridge maintenance obstructs the traffic and leads to economic losses. It is essential to maintain the structure in time, else the damages worsen and eventually lead to structural failure, economic losses and loss of life.

RWS follows a risk-based maintenance strategy. The inspectors visually assess the bridge's structural elements to get inputs for the risk assessment. The condition-based assessment allocates a damage number from zero to six based on expert opinion, where zero is good condition and six being very bad condition. The assessment is combined with the risk matrix provides the quality to support maintenance decisions. This subjective assessment affects the risk estimation negatively. It does not provide insights on the bridge's structural performance change, due to traffic load changes, and damage progress over time. These insights could help the asset managers to improve maintenance plan, and optimize resource allocation. The assets managers are interested in knowing the consequence on the bridge's structural performance because of a maintenance action or damage. They are also interested in gaining insights on damage progress and the loads on structural performance over time.

The ‘Kunstwerken in Control’ (KiC) project consortium is established and funded to develop methods to monitor and assess the bridges to assist in maintenance planning. The Hengelo branch of the RWS is the main stakeholder in KiC project and the user. The goal of this PDEng project is to design and develop a tool to assess a bridge's structural performance under different loading and damage scenarios. The tool is developed and validated for a case study bridge called Tankinkbrug. A measurement campaign was done on bridge to collect readings for validation.

A scenario analysis tool is designed to meet the requirements. The components of the tool are identified and explained in chapter 4.2. In this prototype development, the assessment of the bridge deck and superstructure are focused. The knowledge question “*how to assess the structural performance of a bridge?*” is answered using the deflection influence lines (DIL). A damaged state DIL is compared with the reference state DIL to calculate the change in the bridge’s structural performance.

A physics-based digital twin model is developed using the finite element method to replicate the DIL. It is validated using the measurement campaign readings and incorporated in the scenario analysis tool. The tool is designed to consider different damage and loading scenarios and predict the DILs. Percentage difference between the reference and damaged state DIL is used as the key performance indicator (KPI). It is proved in chapter 6.4 that the KPI shows the damage in the structure and locates it. KPI’s sensitivity to damage severity quantifies the structural performance change. KPI can serve as an insight, and thresholds can be set by the user to support their maintenance decision. Provision to consider the degradation models is included in the tool to study the effects on structural performance over time. A graphical user interface is designed to take the inputs from the user’s and display the results (See chapter 6.1). The tool is developed using open-source resources for economic viability. Recommendations for further tool development are listed in chapter 7.2.

Table of contents

List of figures

List of tables

List of abbreviations

KiC	Kunstwerken in Control
UT	University of Twente
RWS	Rijkswaterstaat
IoT	Internet of Things
DBM	Dynamics Based maintenance
SHM	Structural Health Monitoring
DIL	Displacement Influence Line
FEA	Finite Element Analysis
CAD	Computer Aided Design
IL	Influence Line
KPI	Key Performance Indicator
LVDT	Linear Variable Differential Transformer
VDS	Virtual Displacement Sensor

1 Introduction

1.1 Background and motivation

In the Netherlands, many bridges are more than half a century old. There is a mismatch between current loads and its designed capacity due to increasing traffic and heavier vehicles. This mismatch often leads to structural damage and failures before achieving the design life. Not fixing the damages and the potential damage situations through maintenance in time lead to structural failure and consequently loss of life, like the mishap of the Genoa bridge collapse in Italy. Ideally, maintenance needs to be performed just in time, since doing very early maintenance does not allow the user to exploit the structure to its fullest capacity. On the other hand, delaying maintenance increases the risk of failure and increases the cost of maintenance. The maintenance cost also increases as the bridge ages [1]. The bridge inspection and maintenance require full or partial closure of the bridge to the traffic resulting in economic losses. This creates economic interest among the asset owners.

The ‘Kunstwerken in Control’ (KiC) project is established and funded to develop methods to monitor and assess bridges and to assist in maintenance planning. The project members are the University of Twente (UT), Rijkswaterstaat (RWS), Province Overijssel, Strukton, Saxion, Centric, Twente 47, and Antea Group. The KiC focuses on the Internet of Things (IoT) and a digital twin development. As a part of KiC project, a collaboration between the Hengelo branch of RWS, Dynamic based maintenance group of UT is made and this PDEng. project is created to identify a value adding method using digital twin to the RWS Oost Nederland’s (Hengelo) maintenance practices.

RWS, the Dutch infrastructure and water management board owns and maintains many bridges throughout the Netherlands. RWS follows a risk-based maintenance strategy where periodic visual inspection of a bridge is carried out to assess the bridge condition. The assessment of the bridge structure is based on expert opinion and is therefore subjective. The asset managers need to make maintenance decisions considering the available budget over an asset lifecycle based on this subjective assessment. The bridge inspection, maintenance planning, maintenance activities, and all other related activities must be carried out within the available resources. For a damage scenario, it is difficult to assess whether the structure can still be safely used within the designed loading capacity without assessing the structural performance. Knowledge of the structural performance can be used to support maintenance decisions and make the best use out of the available funds. This creates an interest in asset managers of RWS Hengelo to assess the change in structural performance due to damage. Identifying the change in structural performance over time due to damages provides insights to plan the maintenance activity considering the resources' availability. Based on the subjective assessment, asset managers cannot predict the structural performance change over time as damage progresses. They need insights on bridge performance considering multiple damages and loading scenarios to adapt their maintenance strategy. It is neither practical nor advisable to damage the bridge to study its behavior in real life, especially when it is still in use. These challenges can be addressed by a digital twin model that replicates the bridge in a virtual environment. Different scenarios can be modelled and analyzed in the virtual model to gain insights hence risk can be assessed more objectively rather than based on subjective assessments.

In this project, a tool to assess the structural performance of a bridge under different loading and damage scenarios is designed and developed using a digital twin model. The performance change, in other words, the consequence on structural performance is quantified. The quantified performance change can provide relevant insights to the asset managers.

1.2 Design objectives and scope

Through meetings with the stakeholders of the KiC project, and RWS Hengelo asset managers the following needs and problem statement are identified:

- RWS asset managers are interested in gaining insights on the consequence of their possible maintenance actions on a bridge structure to support their maintenance decisions.
- KiC envisions to use digital tools to create value for the asset owners.

This design project therefore focuses on developing a physics-based digital model of the bridge deck and the superstructure using sensor data from a case study bridge to achieve the following objective:
“Design and develop a tool using a physics-based digital model of a bridge, to assess the consequence of the bridge structural performance due to damage or maintenance actions on the bridge structure.”

1.3 Approach

A literature study and university course selection are done to gather the knowledge related to project requirements and answer the knowledge question *“how to assess the structural performance of a bridge?”*.
The concept and the tool prototype will be developed for a case study bridge. To design the tool, an iterative approach is considered. The tool will be divided into subsystems and components. The subsystems and components are developed and

improved iteratively, then connected to meet the requirements. Finally, the tool will be validated using the case study bridge measurements.

1.4 Thesis outline

This report provides details of the tool development has the following outline. Chapter 2 is used to discuss the literature review on bridges, structural health monitoring of bridges, digital twins, and current practice at RWS. Information required to answer the knowledge question is gathered. In chapter 3 the stakeholders' needs and requirements are discussed. The tool has to be designed to fulfill the requirements. Chapter 4 explains the concept and basic designs of the tool that fulfill these requirements. The components required to develop the tool are created and discussed in Chapter 5. The assembly and the validation of the tool is described in Chapter 6. The requirements set in Chapter 3 are discussed again to verify their compliance. In Chapter 7 conclusion and recommendations are provided.

2 Literature review

A literature study was done to gather the required knowledge on bridge structures, digital twin, Structural health monitoring, and RWS current practices. This helps to answers the knowledge question, and, define the requirements.

2.1 Bridges

A bridge is a structure built to span physical obstacles without closing the way underneath such as a body of water, valley, or road to provide passage over the obstacle. The bridges can be classified under different categories, considering parameters such as material, length, construction, etc. Based on the structural arrangement basic types are identified as girder, cable-stayed, suspension and arch bridges [2]. Though bridges can be classified, every bridge is a unique prototype with unique structural shapes and arrangements, a combination of materials, and dimensions that are highly influenced by traffic, geographical, and fiscal parameters. A bridge has to withstand multiple loads acting on it during its design life; self-weight, traffic, and environmental loads such as thermal, wind, chemical, etc. Eurocodes [3] and American codes [4] provide the design standards and guidelines for the bridge design. The bridge has to fulfill its function. If the bridge system is no longer capable of fulfilling its function, it is a failure.

The difference between damage, defect, and failure is presented below [5]: Damage is when the structure is no longer operating in its ideal condition, but it can still function satisfactorily, but in a suboptimal manner. The damages on the structure may grow at an accelerated pace due to multiple parameters over time and lead to failure. A defect is inherent in the material, and statistically all materials will contain a known amount of defects. This means that the structure will operate at its optimum if the constituent materials include defects. Failure is the incapability of the system to fulfill its function.

To avoid bridge damage leading to its failure, inspection and maintenance are done. The consequences of failure can often be seen as a good indicator of the importance of a bridge structure, given its form, function, and location within a transport network. They can range from casualties and injuries to structural damage, reduction in network functionality and may also extend into environmental as well as societal impact [6]. To avoid these consequences, the maintenance of a bridge is vital. Identifying potential failure in early stages and doing maintenance just in time improves a bridge's life and possibly extends its lifetime beyond the designed period. This reduces the maintenance cost by fully utilizing the bridge structure.

The national road network in the Netherlands consists of around 3200 kilometers of road, of which 2200 kilometers are highways. There are approximately 3200 bridges within this network, where the exact construction year is unknown for around 100 bridges. Almost all bridges and viaducts are primarily concrete structures. About one hundred are mainly steel structures, aqueducts, or moveable bridges [7]. The maintenance cost increases due to a mismatch in the designed capacity and increased traffic loads. As the bridges' age the maintenance cost increases and most Dutch bridges are more than 30 years old [8].

Rijkswaterstaat (RWS), the Dutch infrastructure and water management board, follows a risk-based maintenance strategy. The risk level is determined by the probability of failure occurrence and its consequence. The size of the risk is scaled qualitatively, scale ranges from 1 (negligible) to 5 (unacceptable) as shown in Table 2.1. This scale guidelines used in the object risk analysis and condition assessment of structural elements to assess the risk. Object risk analysis (ORA) is done on the bridge structural element, it has six steps and are explained in [9].

Table 2.1 Risk matrix.

Chance	Consequence			
	Neglect	Serious	Very serious	Catastrophic
Chance of falling is unacceptable (calamity)	3. Increased	4. High	5. Unacceptable	5. Unacceptable
Chance of failing is very high	3. Increased	3. Increased	4. High	5. Unacceptable
Chance of failing is high	2. Limited	3. Increased	3. Increased	4. High
Higher than immediately after delivery the accepted probability of failure is approached	1. Neglect	2. Limited	3. Increased	3. Increased
Higher than immediately after delivery but within the acceptable probability of failure	1. Neglect	1. Neglect	2. Limited	2. Limited
Not higher than immediately after delivery	1. Neglect	1. Neglect	1. Neglect	1. Neglect

Periodic inspection of the bridge structure is carried out to collect the necessary information. Three levels of inspection are done; daily inspection, condition inspection every two years, and maintenance inspection every six years is done to assess the bridge [10]. During the inspection the condition of each structural element is assesed visually and the status is indicated from 0 (good) to 6 (poor condition) (see Table 2.2 condition level column). Also the inputs required for the ORA is collected. The individual elements are assessed based on the expert opinion and the reference documents available with RWS for the assessed structural element.

The quality status of the structure is assessed by combining the condition assesment and the risk matix scale. The quality represents the extent to which the structural condition meets the performance requirements (risk level). The asset manager supports the maintenance plan and decisions based on the risk assesment obtained from the ORA and quality status indicator of the structure [7-10].

Table 2.2 Quality status indicator (Condition vs Risk)

Condition of the structural element	**Risk Level**				
	1	2	3	4	5
0. In very good condition	0	0	0	0	0
1. In good condition	1	1	1	1	1
2. In good order	2	2	2	2	2
3. In fair condition. Risk as in reference documents	3	3	3	3	3
4. In poor condition. Does not meet reference documents	3	3	4	4	4
5. In poor condition. Does not meet the minimum acceptable level	3	3	5	5	5
6. In very poor condition. Extreme risk; do not meet any requirements.	3	3	6	6	6

In condition assessment based on visual inspection, difficult to assess the effects of load and damage progress over time on structural performance. This affects the maintenance planning negatively.

A structural performance assessment method that can consider the effect of different loadings and structural degradation over time is required to fulfill these gaps. This can provide more insights to assess the risk. Further it can be used to quantify the consequence of a maintenance action. Therefore, a physics-based method has be used to assess the bridge performance as it considers different failure mechanisms of the bridge.

2.2 Digital twin

A digital twin has different definitions and classifications based on the industry, see Table 2.3.

Table 2.3 Digital twin definitions and interpretations.

Digital twin	**Reference**
"The digital twin is a set of virtual information constructs that fully describes a potential or actual physical manufactured product from the micro atomic level to the macro geometrical level. At its optimum, any information that could be obtained from inspecting a physically manufactured product can be obtained from its digital twin."	[11]
"A digital replica of a product or system maintained as a virtual equivalent throughout the lifespan of the physical product. A dynamic software model that uses sensors and other data to analyze its state, respond to changes, and improve operations."	[12]
"A digital twin is an integrated multiphysics, multiscale, probabilistic simulation of an as-built vehicle or system that uses the best available physical models, sensor updates, fleet history, etc., to mirror the life of its corresponding flying twin"	[13]
"Coupled model of the real machine that operates in the cloud platform and simulates the health condition with an integrated knowledge from both data-driven analytical algorithms as well as other available physical knowledge"	[14]
"Digital twin is a real mapping of all components in the product life cycle using physical data, virtual data and interaction data between them"	[15]
"A dynamic virtual representation of a physical object or system across its lifecycle, using real-time data to enable understanding, learning, and reasoning"	[16]
"Using a digital copy of the physical system to perform real-time optimization"	[17]
"A digital twin is a digital replica of a living or non-living physical entity. By bridging the physical and the virtual world, data is transmitted seamlessly allowing the virtual entity to exist simultaneously with the physical entity."	[18]

Among these, a general definition mostly recognized and being used was given by Glaessegen and Stargel [13]. The digital twin consists of three parts: a physical product, a virtual product, and connected data that tie the physical and virtual products [19].

In summary, a digital twin model has to replicate at least one feature of the physical product. The digital twin shall accommodate the changes in the physical system and external parameters affecting the system; it can predict the changes in the feature due to changes in parameters. In this project, the feature shall be the physics-based damage sensitive property of a bridge system.

2.3 Structural health monitoring

Structural health monitoring (SHM) techniques monitor a system and detect any damages on the system. SHM techniques provide various methods to assess the structural condition nondestructively by diagnosing the structure's response due to the loads acting on it. SHM axioms are important to understand since they guide the design and implementation of the SHM system. SHM axioms will provide guidance to the virtual monitoring system development in the digital model. A number of axioms (7) are formulated by Worden in [20], from that a few most relevant to the digital model development are presented below:

Axiom 1: “All materials have flaws and defects.” Metals are never perfect single crystal with a perfect lattice structure. The manufacturing process affects the materials’ quality at the micro structural level. In engineering applications, the effects of these defects are subsumed into the average material properties such as yield stress or fatigue limit.

Axiom 2: “The assessment of a damage requires a comparison between two systems.” The assessment is done by comparing the structural condition with the baseline or reference of the structure. The baseline can be the pristine structure without any damage or design limits or an instance in the structure’s lifecycle. In a digital model, a structural condition can be modeled and compared with the reference state to assess performance.

Axiom 3: “Sensors cannot measure damage.” Feature extraction through signal processing and statistical classification are necessary to convert sensor data into damage information. The sensors measure the response of the system to its operational and environmental input. In the digital model development, the model's response has to be matched with the bridge response.

SHM is classified into four levels, based on the damage identification, as presented below [21]:

Level 1 – Detection: Detection of damage presence in the structure.

Level 2 – Localization: Localize and locate the detected damage.

Level 3 – Assessment: Quantify and assess the located damage.

Level 4 - Prediction : Estimation of remaining service life.

In order to provide insights on a bridge structural performance to the users, at least level 3 damage identification has to be met. Level 3 SHM enables quantification of the structural performance change due to a structural change at a specific location on the bridge. A suitable SHM level 3 damage sensitive feature must be selected as a bridge’s performance measuring parameter. This feature shall be replicated in the digital model. The bridge’s static and/or dynamic responses are monitored using sensors. The response signal is used to extract damage sensitive parameters. A few damage sensitive features are explored further.

2.3.1 Natural frequency

The natural frequency is the frequency at which the system will when oscillate unaffectedly by external forces. It depends on the mass and stiffness of the structure. Structural changes and damages are detected by monitoring changes in the natural frequencies. Using accelerometer readings, the structure's excitation response is monitored, and the natural frequency is extracted from it. The response is recorded either by applying a known excitation force or by operational modal analysis. Elimination of the environmental effects on the natural frequency is a challenge. It is possible to achieve level 1 SHM by using the natural frequency as a parameter. [22]

2.3.2 Modal damping

Structural damping is defined as a measure of energy dissipation in a vibrating structure and its ability to bring the structural system to its inert state gradually. Modal damping as a feature is investigated since the damages such as cracks affects the damping ratio. Damping is difficult to estimate and damping levels are nonlinearly influenced by vibration amplitude, operational and environmental effect making it more complicated [23].

2.3.3 Modal shapes

Modal shapes are the deformation shape of the structure when it is vibrating at a natural frequency. A mode shape contains the spatial information and using it as a damage sensitive feature level 2 SHM is achievable [22]. It is less affected by the environmental effects compared to natural frequencies. Unlike the natural frequency, multiple sensors are required on bridge to monitor the modal shape, making it difficult for direct monitoring [23].

2.3.4 Modal curvatures

Modal curvature (the 2nd derivative of the velocity, ν'') utilizes the relation between the bending moment (*M*) and flexural rigidity (*EI*) [24].

$$\nu'' = \frac{M}{EI} \tag{Eq.2.1}$$

The modal curvature change is used to identify and locate the damage. Modal curvature methods requires many sensors to define higher modes and the performance depends on the number of modes considered for evaluation. Using modal curvature alone is not recommended for damage identification, it has to be combined with other methods [25].

2.3.5 Influence lines

The influence lines (IL) represent the response of the structure at a fixed point as a function of the location of the load. A structural responses such as deflection, stress, shear force, bending moment, strain at a specific point of the structure are extracted or derived as a load (force, moment) moves over the structure. Influence lines are a static property and have extensive applications starting from the design of the bridges, existing structure performance assessment, estimating the ultimate capacity of the bridge, damage detection and localization [26]. Stress and deflection influence line based damage detection are discussed in [26-29] and a few points are presented below:

1. Influence lines are a static global property of the bridges, and it is not needed to consider the effects of structural mass.
2. Challenges in the number of sensor required to increase the damage detection accuracy can be overcome using influence line since in theory, only one sensor is required to get the complete IL.
3. Level 3 SHM is achievable using displacement influence lines. Multiple damage or structural modification in the beam-like structure can be quantified. It is not needed to combine it with other features to improve its level.

These characteristics of the influence lines makes it more suitable to replicate as a feature in the model development than the dynamic properties. The deflection influence line is obtained using sensors such as linear variable displacement transducers (LVDT) and the stress influence line can be obtained using sensors such as strain gauges.

2.4 Summary

In this chapter, the general aspects of bridges and the RWS maintenance decision making method have been studied. The need for a performance assessment method was identified. The digital twin models and SHM techniques have been explored to discover a method to assess the bridge structural performance.

3 Stakeholder analysis and project requirements

A stakeholder is defined as an individual, group of people, organization or other entity with a direct or indirect interest in a system. The members of the 'Kunstwerken in Control' (KiC) project are thus the stakeholders. The Dynamics Based Maintenance (DBM) group of the University of Twente (UT), the KiC project manager and the Rijkswaterstaat (RWS) branch located at Hengelo are directly involved in this project. Other members such as the Pervasive Systems group of the University of Twente, Strukton, Saxion, Centric and Antea Group are also part of the project. The project focuses on multiple methods to promote IoT in the maintenance of bridges and viaducts. Meetings with the stakeholders helped to identify the needs listed in Table 3.1.

Table 3.1 Needs of the Stakeholders.

Stakeholder	**Needs**
RWS	Method to assess the structure to understand the consequence of maintenance actions. Note: A maintenance action is considered a structural modification on the bridge and a plan to fix/not fix the bridge's damage.
KiC, DBM, UT, RWS.	Report on digital twin development and a prototype or proof of concept.

The stakeholders' needs can be translated into requirements by exploring the user needs and the steps to be followed to achieve them. This is explored following the process flow as in Figure 3.1. Analyzing stakeholders' needs and converting it into requirements for the system is essential in the design process. This explains the tool's capabilities based on the user requirements and the user's actions performed on the tool. A functional analysis is required to understand and achieve the requirements further. The tool functions will be defined using the layout of Figure 3.1.

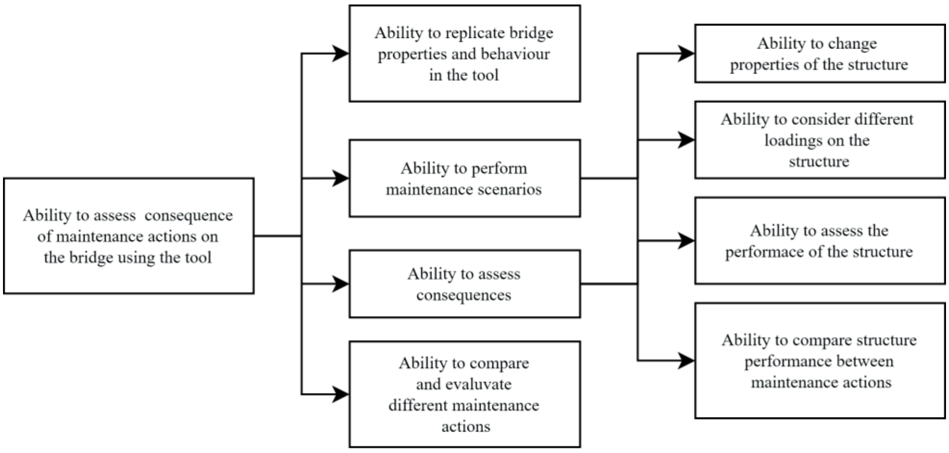


Figure 3.1 Translating RWS needs into design requirements.

Comparing maintenance scenarios on the real bridge is not viable due to the bridge's cost and traffic obstruction. To study the consequences of maintenance actions, it is necessary to have a model with provisions to modify the bridge structural elements and carry out a structural performance assessment. A physics-based digital model is a very convenient and economical solution to achieve this. It is identified as a requirement for RWS to assess the maintenance action consequences.

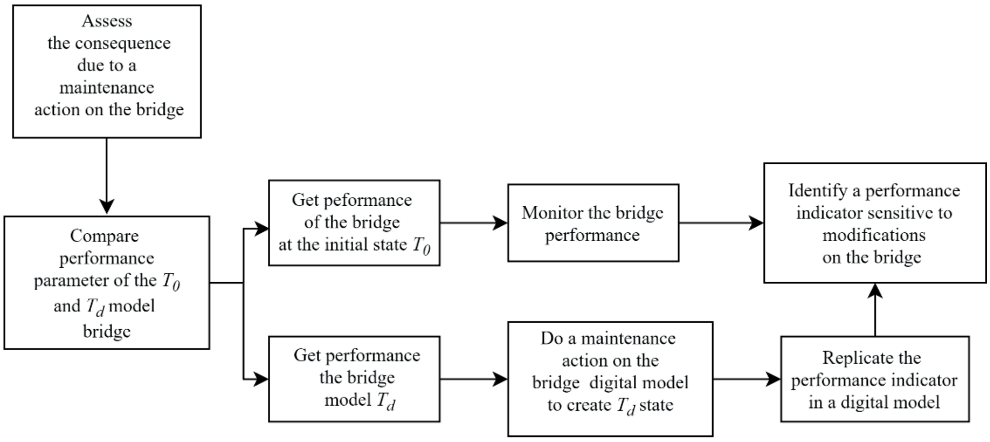


Figure 3.2 Functional analysis of the tool.

The functional analysis of the tool to be developed is shown in Figure 3.2. The functional analysis provides the layout for the actions to be performed to get the final output. Every action has to be designed and developed as a function performed by the tool's subsystems. The first step is to identify the performance indicator, use it to monitor the bridge, and replicate it in a digital model. The requirements for the tool are set based on literature study and meetings with the stakeholders.

Translating needs and functional analysis provided insights to define the requirements. Table 3.2 shows the stakeholder requirements (ShR) and system requirements (SyR). Stakeholder requirements structures the user expectations on the output of the project by defining deliverables from the user’s perspective. Stakeholder requirements set the directions to define the system requirements. SyRs define the technical parameters and measures from the developer perspective to meet the stakeholder requirements.

Table 3.2 List of requirements.

No	Type	Requirements
ShR	1	A prototype tool shall be designed to study the consequence of the maintenance actions on bridges.
	2	The tool shall be used without depending on the expensive licensed software to make it affordable for RWS.
	3	The tool output shall be integrable with the current structural assessment process followed by RWS Hengelo.
	4	The tool shall be accessible through a graphical user interface.
	5	The tool user interface shall be designed for the user knowledge level of 'higher professional education' (in dutch: Hoger beroepsonderwijs - HBO).
	6	The tool shall be developed within the duration of PDEng project (1 year).
	7	The tool shall be submitted as a package to install and operate.
SyR	1	A method shall be formed for the bridge structural performance assessment.
	2	A physics-based model shall be developed to replicate the case study bridge behaviour under different vehicle loads.
	3	A physics-based parameter shall be identified to assess the performance of the structure.
	4	The performance parameter shall be sensitive to structural changes and loads acting on the structure.
	5	The performance parameter shall be measurable and monitorable on the bridge.
	6	The tool and the model shall be developed for a case study bridge.

	7	The tool shall be validated using measurement campaign readings of the case study bridge and replicate the physics-based parameter as accurately as possible.
	8	The model shall have the provisions to alter the physical properties to replicate structural changes due to maintenance action.
	9	The model shall have provisions to include degradation models to study changes over time in the structure.
	10	The model shall have provisions to consider load variations on the bridge to evaluate its effects on the physics-based parameter.
	11	The model shall contain a Key Performance Indicator (KPI) to compare the performance parameter of the bridge at different conditions and to assess and quantify the consequence.
	12	The output of the tool shall be readable/visualizable to by the user.
	13	The tool shall be scalable to add bridge elements and concept to other bridges.

To meet the requirements, a design methodology is followed and for the application process is described in chapter 4. Design choices made during the design process are often checked and verified to meet the requirements. If a design choice does not meet the requirements, possible alternatives are identified to fix it. In the next chapters, the requirements are linked with tool development and design choices.

A few requirements are difficult to fulfil in some situations due to a lack of resources, information, time, money, etc. Based on the stakeholder, it is identified that the requirement ShR 2 is given high importance. To meet ShR2, either the tool has to be developed from scratch or suitable open-source alternatives have to be identified and utilized for the tool development. ShR 2 requires a significant amount of time available to develop the tool, affecting other requirements. In section 4.3.2 challenges in using opensource tools are discussed.

4 System design

In this chapter, the tool is designed and the design decisions are explained. Firstly, the scenario analysis concept is presented for the bridge system. Then the basic design is done based on the requirements. Tools and components to build the system are explored and selected using design methodologies and decision-making frameworks.

4.1 Scenario analysis

Scenario analysis is a process of analyzing future events by considering possible alternative outcomes. A virtual/digital bridge system replicating the bridge's properties and predicting the system's response for a change in the system parameters will enable scenario analysis; the predicted virtual system response should match the bridge response. The inputs of the scenario analysis tool can be the capacity and load parameters. The output is the predicted response. The concept of the scenario analysis tool is shown schematically in Figure 4.1. Varying the load or capacity parameters of the bridge system will change the response of the system. The predicted bridge response can be a useful insight for the stakeholder to decide on the maintenance actions applied to bridge system.

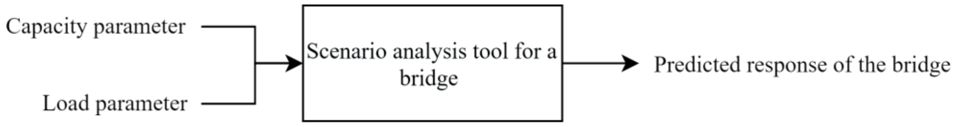


Figure 4.1 Concept of the scenario analysis tool

The scenario analysis tool will fulfil ShR 1. The user can model the maintenance actions using the input parameters to change the capacity of the bridge's structural members and apply a specific load on the structure to predict the response.

The bridge's structural strength properties depend on the structural members' dimensions, boundary conditions, and material properties. A maintenance action or damage can be modeled in the scenario analysis tool by changing the stiffness of the structural members. A structural maintenance action could be reinforcing the bridge's structural members or fixing damage in it thereby increasing the structural stiffness; Damage is modelled by reducing the stiffness of the structural member [30]. The structure can be restored by changing the stiffness to the initial value or strengthened by increasing the stiffness further. The structure's bending stiffness depends on the modulus of elasticity E (a material property) and the second moment of inertia I (a geometrical property). Degradation mechanisms can be linked to material property changes and material loss in the structure. Degradation mechanisms model the damage progress over time and the scenario analysis tool can predict the resulting structural response change over time.

Also the load parameters also can be varied to create a scenario. By applying a specific vehicle load on the bridge, the response of the structure can be predicted. Different types of loads can be combined. Both capacity and load parameter can be varied together to predict the response. The predicted response can be compared with the designed response limits or a different scenario's performance to identify the variation. The variation in performance is the consequence of the parameter change. This quantified consequence can serve as an insight to the user.

4.2 Basic design

The base of the scenario analysis tool is the digital bridge system. A digital system replicating the physical system's properties, which allows simulations to predict the real system's behavior due to a parameter change, is the system's twin model. The digital twin of the bridge will enable scenario analyses where the user can change the load and capacity parameters and predict the behavior through simulations. The digital twin for the bridge is divided into subsystems and unit/components. The subsystem can operate independently and multiple subsystems form the system. A set of components makes a subsystem. Based on the requirements and functional analysis, four subsystems and one component are identified in the digital twin system as shown in Figure 4.2. The GUI, physics-based model, KPI calculation, and the bridge response prediction module are the subsystems. A physics-based parameter (damage sensitive feature) sensitive to structural modifications or damage is considered a component. The key performance indicator (KPI) calculation is a subsystem that quantifies the variations of the performance parameter by comparing two scenarios' performance parameters. The KPI provides insight to the user. The physics-based model subsystem considers the capacity and loading inputs from the user and predicts the performance parameter of a scenario. The graphical user interface (GUI) subsystem provides the user access to operate the scenario analysis tool. The user can provide inputs to create scenarios and read the outputs in the GUI.

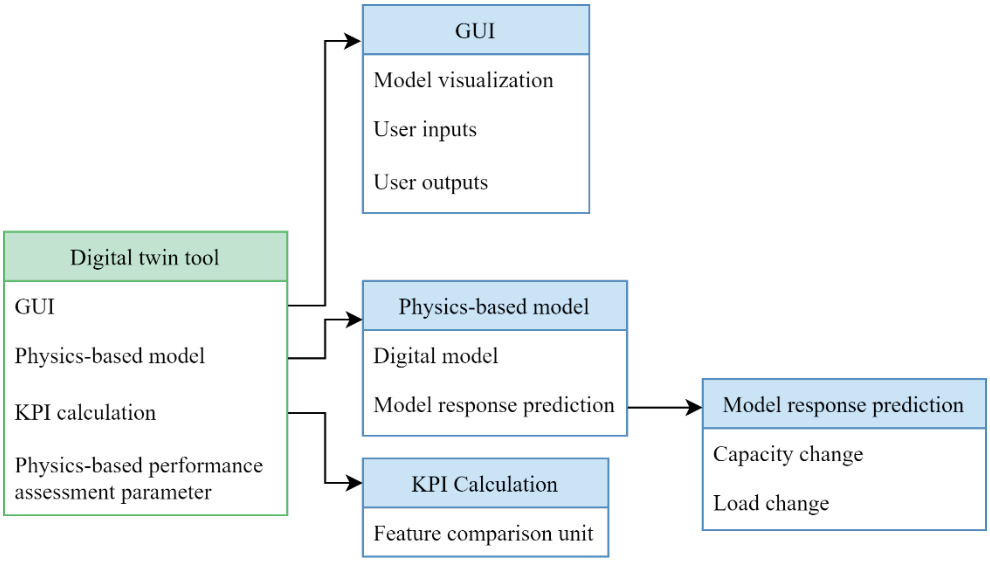


Figure 4.2 Subsystems and components.

Once the subsystems and component are defined, the solution space to develop them are explored. A solution space contains the possible options that would fit the subsystem and component functions. The solution space for the subsystems and components of the tool is shown in Figure 4.3. To develop the tool the underlined components are selected in their category. The reasons for the selection of these subsystems and components are presented in the next sections.

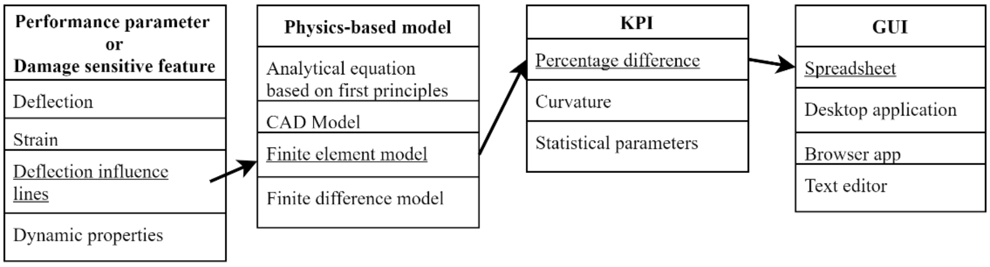


Figure 4.3 Solution space for tool development.

4.2.1 Performance parameter selection

The performance parameters or damage sensitive features are discussed in chapter 2.3. They are either static or dynamic properties of the structure. Among them, the static property deflection influence line (DIL) is selected as the performance parameter. It is the deflection at a point in the structure as a function of load position on the structure. As discussed in chapter 2.3, based on the literature [26-29, 31], key points that make (DIL) suitable over other features for the system design are:

1. The deflection influence lines are more sensitive than the dynamic properties such as modal frequency change for stiffness changes in beam-like structures.
2. Deflection influence lines are a promising feature in detecting, locating and quantifying the damage achieving level 3 of structural health monitoring.

3. Deflection influence lines are based on the stiffness and boundary conditions of the bridge structure and It is not needed to consider the mass of the structure.
4. Considering a capacity scenario, the deflection influence lines of that scenario can be extended to different loading scenarios using the principle of superposition, provided that the structure is within elastic limits.
5. A static method ensures better measurements using sensors in terms of accuracy and direct measurement.
6. Displacement sensors such as linear variable differential transducers (LVDTs) monitors the DILs at the connected location on the bridge.

These properties make the deflection influence line a suitable feature to assess consequence due to structural modifications. It matches the physics-based performance parameter requirement (SyR 3), which is sensitive to the structural changes (SyR 4). It is monitorable (SyR 5). It is also will be utilized to fulfill tool development and validation (SyR 6 and 7).

4.2.2 Physics-based model selection

A physics-based model shall be capable of replicating the properties of the bridge digitally. The model shall consider the user's inputs and predict the bridge's response parameter, in this case, the DIL. The model shall be scalable, allowing it to be modified and extended when there is a change in the bridge structure (SyR 13). Considering these criteria, an option is selected from Figure 4.3 to build the subsystems. The selections are discussed below:

An analytical equation based on first principles is not a viable solution for a physics-based digital twin model development. Though equations can replicate the parameters and be faster in operation, it is difficult and time-consuming to develop one for complicated structures. Scalability, considering different loading scenarios and accounting for structural modifications is difficult to achieve using analytical equations.

A CAD model can represent the bridge by replicating dimensions, material details and it can accommodate new data. However, it does not fulfill the requirements to predict the structure's response under different load and capacity scenarios.

A numerical model based on first principles fits the purpose to achieve a solution with reasonable accuracy. Handling complex geometry is a bottleneck in the finite difference method [32]. On the other hand, the Finite Element (FE) method is a proven technique to create a physics-based simulation of structures. It is often considered that numerical models such as the finite element model are time-consuming to solve. But the availability of computational power and fast solvers makes the FE method a viable solution to create a physics-based digital model. After considering all these options, it is decided to select the finite element method to develop the model. An FE model of a case study bridge will be developed and it will be validated using measurement campaign readings of the bridge. The FE model of the case study bridge fits the system requirements such as SyR 2, 4, 8, 9, 10, and 12 to act as digital twin model, to replicate the performance parameter, and accommodate the maintenance scenario analysis setup. FE solver selection and developing a model to solve using the FE solver are vital steps in the tool design. FE model development is a time-consuming task; the solver and the type of analysis will affect the working hours required to create a FE model. Although time required for model preparation can be reduced by automation, it is not a task for the concept and prototype development stage project. The ShR 2 requirement affects the FE solver selection, as it mentions the time available to develop the tool. Considering the requirements, the FE solver selection is discussed in detail in section 4.3.2.

4.2.3 Key Performance Indicator (KPI) selection

A KPI is used to quantify the consequence of the maintenance action or damage on the bridge structure. The selected performance parameter DIL is used in the calculation of the KPI. The DIL after the maintenance action or damage is calculated using the FE model. The calculated DILs are compared with the reference state DIL and the percentage difference between them is calculated. For the first design iteration, The percentage difference between the two DILs is set as the KPI (SyR 11).

$$KPI = \left(1 - \frac{Y_{DIL\_damaged}}{Y_{DIL\_T_0}}\right) \times 100\% \qquad \text{(Eq. 4.1)}$$

The amplitude (*Y*) of the DIL for the load position (*X*) is obtained. Two DILs amplitudes $Y_{DIL\_damaged}$ and $Y_{DIL\_T_0}$ for a load position (*X*) is compared to calculate the KPI. The KPI calculated plotted against the load position (*X*) to obtain the KPI curve.

The user can set the reference DIL based on the design calculations, rules guidelines or the actual DIL of the bridge using a measurement campaign. In this prototype, the reference state DILs are set using the measurement campaign readings; The measurement campaign therefore named the T_0 state of the bridge.

Other options mentioned in the solution space (see Figure 4.3) shall be explored in the next design iterations to identify the most suitable.

4.2.4 GUI selection

The graphical User Interface (GUI) provides the user with a window to access the scenario analysis tool components. The user provides inputs required to create a scenario. The inputs are the scenario name, working directory and load and capacity parameters. The user initiates the calculation through the GUI. The outputs, calculate performance parameter DIL and the KPI are also accessed through the GUI.

The user knowledge level specified in ShR 5 is considered in the design of the GUI. The bridge structure will be present in the GUI with scaled markings to identify a structural member location and make it easy to modify its structural parameter. Creating a desktop application and browser-based application is attractive and they provide better visualization than a spreadsheet application. However, a desktop or browser-based application development will be time-consuming. Thus, the spreadsheet is selected as GUI for the prototype. It is a suitable choice considering the user's familiarity in using a spreadsheet and the time available for tool development.

4.2.5 Basic design summary

The FE model that matches the DIL of the real bridge for the applied proof-loading is the validated physics-based digital twin model. The digital twin model is a subsystem of the scenario analysis tool. Capacity and load scenarios can be modelled in the digital twin to predict the DILs. The KPI is quantifying the consequence of a maintenance action or damage relative to the reference state. The representation of the full concept is shown in Figure 4.4.

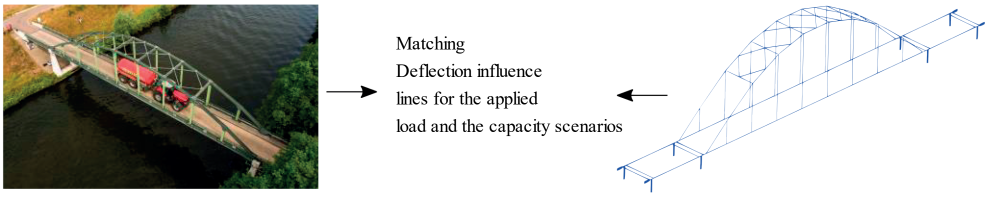


Figure 4.4 Concept of Physics-based digital twin model.

4.3 Components selection

Multiple components are put together to make the subsystem. The components have to be selected based on the requirements. If a component is available and meets the requirements, it is selected off-the-shelf (e.g., FE solver). The components that are not available are built from scratch (e.g., KPI calculator). The components programming language, FE solver and the case study selection are discussed in this section.

4.3.1 Programing language selection

A software tool is developed using programming languages. The set of instructions are written in a programming language to execute the functions of the components and subsystems. It also the programming language that connects the components and subsystems making the tool complete. If a new component has to be created from scratch, it will be created using the selected programming language. The ShR 2 emphasizes reducing the license cost of the software. This is considered in the programming language selection.

Based on the literature [33][34], it can be concluded that the programing language affects the development time, cost and computing performance of the tool. The literatures [33][34] compared the programming languages C, C++, Python, and MATLAB on multiple criteria such as industrial acceptance, academic acceptance, the purpose of language, ease of use, ability of language etc. As a result, Python is selected as the preferred language to develop a software prototype. Python is an open-source language and there are many libraries available in Python to implement them as off-the-shelf components. This speeds up the prototype development. Python is used as an application programming interface (API) in several FE solvers and CAD modelling tools. Considering the above points and the requirements, it is decided to use Python to develop the tool. Python fulfils the requirements of not depending on licensed programs to reduce cost (ShR 2) and scalability (SyR 13).

4.3.2 Solver selection

The FE solver is essential to develop a physics-based digital twin. The FE solver component has to meet multiple criteria to fulfil the stakeholder and system requirements. The criteria are solver capability, license cost, ease of development, pre and post-processing, support and scripting language. To select the solver a multi-criteria analysis (MCA) is done. That is a method that structures a decision problem in terms of several possible alternatives and assesses each of them under various criteria simultaneously. Several MCA methods to rank, compare and select options based on the chosen criteria [34] exist. From those, the ratio weighting method is used to do the analysis for solver selection. The weights for the criteria are assigned considering the stakeholder and system requirements as given in Table 4.1. The criteria are allocated with weights based on their importance. The scores vary from 1 to 10, 1 being poor and 10 being good in the category. Solvers are scored based on collected literature, community forum discussions and experienced user suggestions. The result is computed based on the summation of the product of the weight and score. The highest scoring solver is selected. The criteria that are considered are explained below.

Capabilities: Finite element solvers capability to consider, linear static, natural frequency, thermal, model composites, solid, shell, beam, and bar elements are taken into account for scoring. Though few solvers have much more capabilities such as nonlinearities, contact definition, etc. these capabilities are not accounted for scoring since they are not used in this project.

License Cost: The lower the price, the higher the score is given. Open source solvers always scores the highest.

Ease of development: User-friendliness of the solver, ease of installation, ease of problem setup, solver command language, options to extract required results, ease of integration with other programming modules are accounted to score the FE solver. Choosing a solver such as code_aster or Elmer will cost a lot of time due to challenges in performing the required actions such as modelling, result extraction, etc. Solvers such as Abaqus, Ansys are easier to learn and the development time is less comparatively.

Table 4.1 Solver selection MCA.

Criteria/FEA Software	**Weight Factor**	**Abaqus**	**ANSYS APDL**	**Salome-Meca/ code_aster**	**Elmer**	**Openseas**	**Solidworks Simulation**
Capabilities	0.2	10	10	10	10	10	10
License cost	0.3	3	3	10	10	7	5
Ease of use	0.1	7	8	4	4	4	8
Pre and post-processing	0.1	9	9	7	4	4	7
Support	0.1	10	10	8	6	6	10
Scripting language	0.2	8	5	9	7	5	5
Result	1	7.1	6.6	8.7	7.8	6.5	7

Pre and post-processing: Availability of pre and post-processing modules along with the solver as a package and graphical user interface are accounted. A solver that has readily available options to prepare models and process results scores higher.

Support: Based on the user community and developer support offered score is given. Better support is given a higher score. Commercial software has an advantage due to exclusive customer support.

Scripting language: Python is selected to develop the tool. A solver with a Python application programming interface (API) is given the highest score. The code_aster solver score is 9 (instead of 10) due to the difficulties in learning the French keywords. Abaqus scored 9 (instead of 10) since the API is still not updated from the Python 2 to Python 3 version. Support for the Python 2 version ends in 2020.

Other solvers have Python APIs, however, knowledge of the native language of the solver is essential to write subroutines.

The multicriteria analysis in Table 4.1 proves that Salome_Meca/code_aster suits best to develop the tool. Factors such as open-source code, integrated pre-postprocessing, a standalone solver for customization and Python API are strong points for Salome_Meca/code_aster. More details about Salome_Meca/code_aster are provided in Appendix D.

4.4 Case study

The concept will be validated using a case study. The digital twin model will be developed for the case study bridge which is called the Tankinkbrug. It was built in the year 1952, over the Twente canal to connect the villages Goor and Delden in the Overijssel province of the Netherlands. The Tankinkbrug shown in Figure 4.5, is a continuous tied-arch bridge with three spans, two sets of pillars support the riding deck at a distance of 8.9 m from both abutments. One set of the pillar has roller support while the other provides simple support. The bridge length is 58.96 m and the total width is 4.37 m with a 3 m riding deck. The riding deck is of concrete and has two steel beams as a superstructure. It has a steel bow arch connected to the beams through 8 steel columns on each side with wind bracings connecting the arch bows.

Figure 4.5 The Tankinkbrug.

The Tankinkbrug is selected as a case study since it is not used extensively at present. So, it is easy to conduct measurement campaigns with controlled loadings on the bridge. Multiple similar bridges are in use to cross the Twente canal [35]; the experience gained from the Tankinkbrug case study might provide insights and knowledge to work with the other bridges.

4.5 Summary

In this chapter, the components of the digital twin system were identified. The solution space was explored to choose the components, a concept design was presented and tools to realize the concept have been selected through literature study and MCA. To validate the concept, the selected tools are used to develop the case study bridge's digital twin. The requirements ShR 2, 4, and 7 have been met. Furthermore, the component selection and subsystem design comply with SyR 1, 2, 3, 4, 5, 6, 8, 9, 10, 11 and 12. The requirements will be validated by running sample scenario cases. The FE model development and other component development will be discussed in further chapters.

5 FE model development

In this chapter, the development of the digital model using the finite element method is explained. The steps in the model development such as model simplification, selection of elements, mesh convergence, boundary conditions, model calibration and validation are discussed. This model represents the bridge's physical properties and replicates its behavior under loads. The boundary conditions are calibrated using the measurement campaign readings, to make the model replicate the deflection influence line (DIL) feature as accurately as possible. The validated model is then physics-based digital twin of the bridge. The digital twin will be used to model the capacity and the loading scenarios and predict the DILs.

Before preparing the FE model, it is necessary to know the expected outcome of the model. Decisions on model preparation are made based on the output requirements and time constraints. In this case, deflection influence lines are the output from the model and design decisions are made towards achieving that result as accurately as possible within a reasonable computational time.

5.1 Model simplification and element selection

Preparing the model with extensive details is time-consuming and more importantly computationally expensive. It is essential to identify the level of detail required to get the output. Previous studies on tied-arch bridges [36][37][38], the load distribution on the structural members of a tied-arch bridge, and the required output are considered to simplify the FE model and select suitable FE elements. Details like the brackets, gusset plates, rivet joints and connections are not considered in the modelling. The geometrical properties of the concrete sections are simplified and replaced with an equivalent section assuming isotropic material properties and uniform thickness in the cross-section. The isotropic material model properties considered in the FE model are listed in Table 5.1.

Table 5.1 Material properties considered in the FE model.

	Steel	Concrete
Grade	S235	C25/30
E [N/m^2]	2.1×10^{11}	3.15×10^{10}
Poisson ratio	0.3	0.2

The matching equivalent section is calculated by matching the flexural rigidity (*EI*) of the equivalent section with the bridge deck. First, the material properties are simplified, from a steel-concrete section to the full-concrete section. The transformed area is calculated using the procedure followed in [39]. In this method, the assumptions are perfect bonding between the steel reinforcements and concrete, and the materials are below the elastic limit. The steel reinforcements are transformed to the equivalent concrete area using the modular ratio (R_m).

$$R_m = \frac{E_{Steel}}{E_{concrete}} = 6.67 \qquad \text{(Eq. 5.1)}$$

Where properties as given in Table 5.1 are assigned,

E_{steel} = Elastic modulus of the steel

$E_{concrete}$ = Elastic modulus of the concrete

The transformed area of the steel is assumed to be concentrated at the same point in the cross-section as the total steel area.

The equivalent area is calculated as follows:

$$A_{eqc} = (R_m - 1)A_{steel} \qquad \text{(Eq. 5.2)}$$

A_{eqc} = Equivalent area made of concrete material

A_{steel} = Total cross-section area of the steel reinforcements

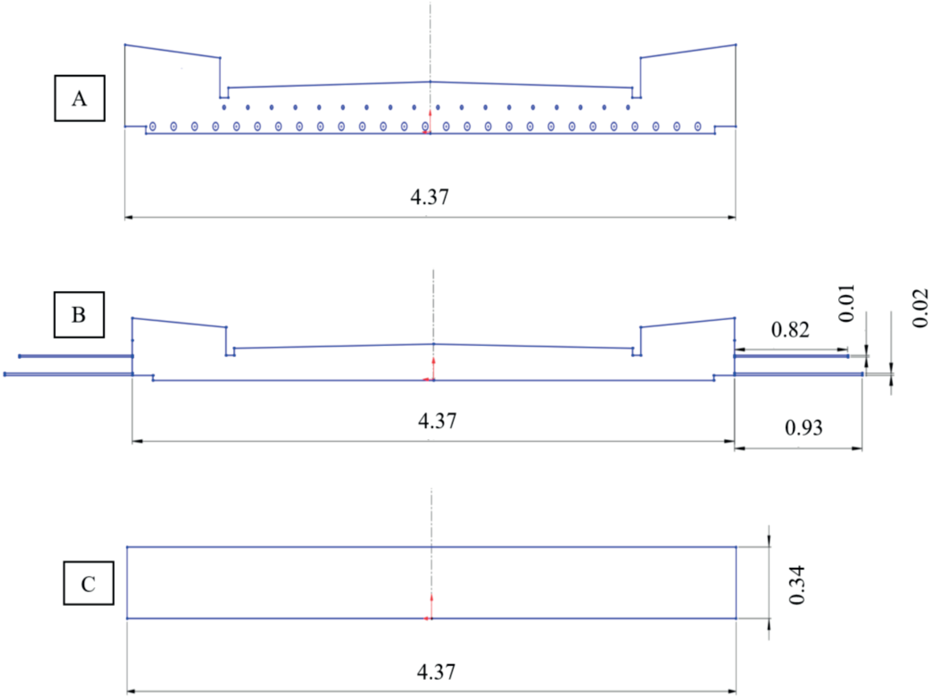


Figure 5.1 Reinforced concrete model simplification.

Figure 5.1 shows the simplification steps:- A: shows the deck with steel reinforcements, B: the transformed material equivalent section, C: the geometrical equivalent section. The simplified section is calculated by fixing the width as 4.37 m (same as the deck width) and considering the elastic modulus of the concrete material to match the flexural rigidity (*EI*) of the section shown in Figure 5.1- B. Detailed calculations of the model simplification can be found in Appendix B. The calculated thickness value is considered in the Kirchhoff plate element. The concrete deck is modelled using Kirchhoff plate elements since the deck's thickness is less than 10% of the width and length of the bridge deck [40][41]. The deck girder, wind bracings, and arch are of steel material and modeled with Euler-Bernoulli beam elements. The steel hangers are modelled as bar elements assuming they only transfer axial loads [42].

5.2 Mesh convergence

A mesh convergence study is done to select the optimal mesh size. Simulations were done by varying the mesh size and keeping other parameters at a fixed value to find the optimum mesh size to get deflection results. The converged mesh makes deflection results independent of mesh size, but a very fine mesh needs more computational power to solve. The maximum deflection value in the vertical direction is recorded at every mesh density. The deflection values against the mesh size are shown in Figure 5.2.

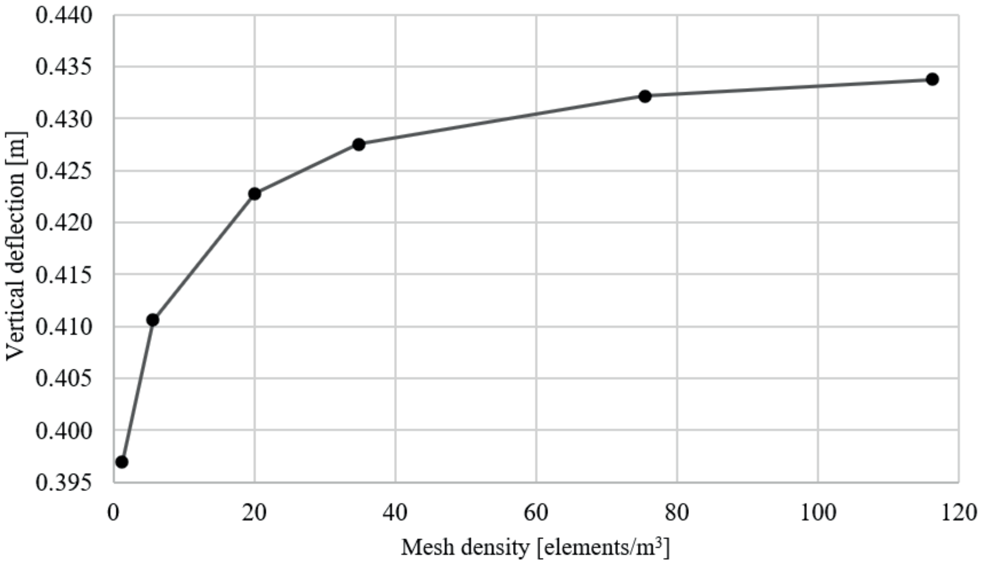


Figure 5.2 Mesh convergence.

The percentage difference is calculated by comparing the deflection values obtained using the current mesh density and the previous mesh density and listed in The selected mesh deflection has a mesh density of 35 elements/m^3 and deviates less than 2%. This mesh is used for the prototype to accelerate the development process. A converged higher density mesh shall be selected for the next design iteration.

Table 5.2.

The zero percentage difference in deflection indicates that the mesh has converged and the result is independent from mesh density:

$$\epsilon_{i+1} = \frac{D_{i+1} - D_i}{D_{i+1}} \times 100\ \% \qquad \text{(Eq. 5.3)}$$

ϵ = percentage [%]

D = Deflection [m]

i = mesh iteration number

The selected mesh deflection has a mesh density of 35 elements/m^3 and deviates less than 2%. This mesh is used for the prototype to accelerate the development process. A converged higher density mesh shall be selected for the next design iteration.

Table 5.2 Mesh density and deflection value convergence

Mesh iteration number (i)	Mesh density [elements/m^3]	Deflection [m]	Percentage Difference [%]
1	1	-0.396916	-
2	6	-0.410617	3.34
3	20	-0.422808	2.88
4	35	-0.427506	1.10
5	75	-0.432156	1.07
6	116	-0.433746	0.37

5.3 Boundary conditions

The model is prepared based on the structural arrangement of the bridge and its kinematics. It is assumed that the main girders are attached firmly to the side of the decks with no relative movement between them. The girders pass through the deck as depicted in Figure 5.3. Nodes in that section are grouped to translate in unison in the longitudinal (*X*) and transversal (*Z*) direction as they would behave in real life as is shown with a tied nodes representation in the zoomed section of Figure 5.4. In total, eight groups are identified and modelled; the regions of grouped nodes are highlighted in red in Figure 5.4.

The deck is supported by the bearings. The bearing supports and abutments act as the boundaries of the superstructure. Fixing the nodes at the bearing support does not represent the true behavior since the bearings allow rotation of the beam girder about their pivotal point. The bearings and abutment boundaries are represented using spring elements as is shown in Figure 5.4. The discrete spring element has six degrees of freedom at each node. Stiffness values and boundary conditions can be assigned at each node. One end is fixed and the other end is connected to the beam girder where stiffness values are assigned. The spring element stiffness can be adjusted to match the rotation of the bearings [43, 44] .

The deflection is based on the bearing support boundary condition and the structural stiffness contribution from the deck, main girder, columns, arches, brackets, rivets, wind bracings, fasteners and steel reinforcements. The model is simplified to reduce model preparation time and computational effort, thus a few structural elements contributing to stiffness are not modelled. The model does not include brackets, rivets, wind bracings, fasteners and transversal reinforcements that are on the real bridge. The spring boundaries stiffness can be calibrated to compensate for the missing elements [43, 45]. Furthermore, the spring boundary conditions are assumed to be symmetrical about the longitudinal axis and the same stiffness values are used for north and south bearings.

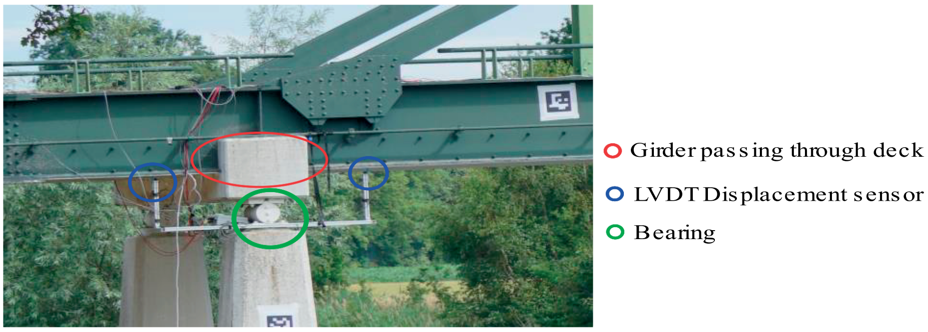


Figure 5.3 Bearing and girder arrangement.

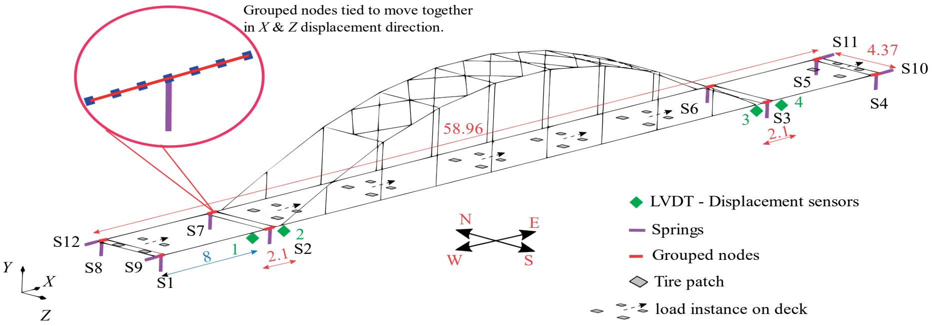


Figure 5.4 Boundary conditions, sensors and loads.

5.4 Model updating and model validation

In this section, the FE model is updated to match a proof load bridge response and then the updated model will be validated by applying a different proof load. If the model output matches the measured bridge response for the applied proof load during the validation step the model is calibrated.

The model spring stiffnesses are adjusted iteratively in a trial and error manner. Only the spring elements representing the boundaries are used to calibrate the model. The case study bridge is in use since 1952, so the bridge material properties, dimensions and boundary conditions most likely have changed over the years due to maintenance actions, damages, material degradation, environmental conditions etc. Thus, the exact material properties, the stiffness values, and dimensions of the bridge at the measurement campaign do not match the design documents, or in other words the model does not match the real bridge. Identifying the exact details and modelling the bridge will consume a lot of resources. Considering the above challenges, it is decided to model the bridge using the designed properties. Only the spring boundaries are calibrated to compensate for the discrepancies due to on-site conditions and missing structural elements.

The deflection values of the model have to match the bridge readings for the applied quasi-static loadings. The deflection influence lines at LVDT location 1,2,3 and 4 are compared for calibration. Considering the measurement campaign readings the following condition is assumed, the vehicle is moving slowly on the bridge with negligible dynamic effects. This is a reasonable assumption since the vehicle moves approximately 5 km/h, and crosses the bridge without sudden changes in velocity. Considering the bridge deck width and vehicle width, there is not enough room for the vehicle to deviate transversely, so it is reasonable to assume that it is moving longitudinally in a straight line. The 10-ton proof loading vehicle axle loads are shown in Figure 5.5. The information from the vehicle manufacturer’s data book is used to calculate the axle loads. The vehicle information and axle load calculations are explained in Appendix A.2.

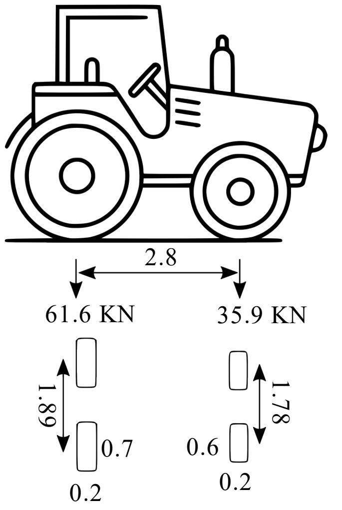


Figure 5.5 10 ton proof loading vehicle (dimensions are in meters).

The springs stiffness mentioned in Table 5.3 are assigned to the FE model and the10-ton proof loading vehicle is then moved from the west to the east side of the bridge. The load is applied to the model and moved step by step quasi-statically. The vehicle load is placed at different locations at multiple instances on the bridge deck representing the vehicle movement. A few load instances are shown in Figure 5.4. The patch loads shown in Figure 5.5 are applied on the bridge at every instance. The nodal displacement at the point of interest for every instance is tracked to extract the influence line. Figure 5.6 shows the LVDT deflection readings from the measurement campaign plotted against the position of the load on the bridge. The FE model's nodal displacements are recorded in the model by (virtually) moving the proof loads as used in the experiments. The model provides virtual LVDTs (VLVDT) readings. The FE model readings are compared with the measurement campaign readings.

Table 5.3 Spring boundary stiffness for the calibrated FE model.

Springs	X [Nm]	Y [Nm]	Z [Nm]	RX [Nm/rad]	RY [Nm/rad]	RZ [Nm/rad]
S1	Free	Fixed	Fixed	Free	Free	3.0×10^{11}
S2	Free	Fixed	Fixed	Free	Free	6.0×10^{10}
S3	Free	Fixed	Fixed	Free	Free	8.0×10^{11}
S4	Free	Fixed	Fixed	Free	Free	3.0×10^{10}
S5	Free	Fixed	Fixed	Free	Free	3.0×10^{10}
S6	Free	Fixed	Fixed	Free	Free	8.0×10^{11}
S7	Free	Fixed	Fixed	Free	Free	6.0×10^{10}
S8	Free	Fixed	Fixed	Free	Free	3.0×10^{11}
S9	1.0×10^{5}	Free	Free	Free	Free	Free
S10	1.0×10^{5}	Free	Free	Free	Free	Free
S11	1.0×10^{5}	Free	Free	Free	Free	Free
S12	1.0×10^{5}	Free	Free	Free	Free	Free

The updated FE model deflection in the vertical direction (*Y*) matches closely with the measurement campaign readings for the 10-ton proof loading vehicle. It still has to be validated. The updated (calibrated) model is loaded with a different proof-loading scenario; the measurements and model response have to match to validate the model. If the deflection response matches the bridge readings the model is considered as a validated model. It will then be used as the physics-based digital twin replicating the case study bridge's deflection influence lines when the bridge vertical deflections are under the elastic limits.

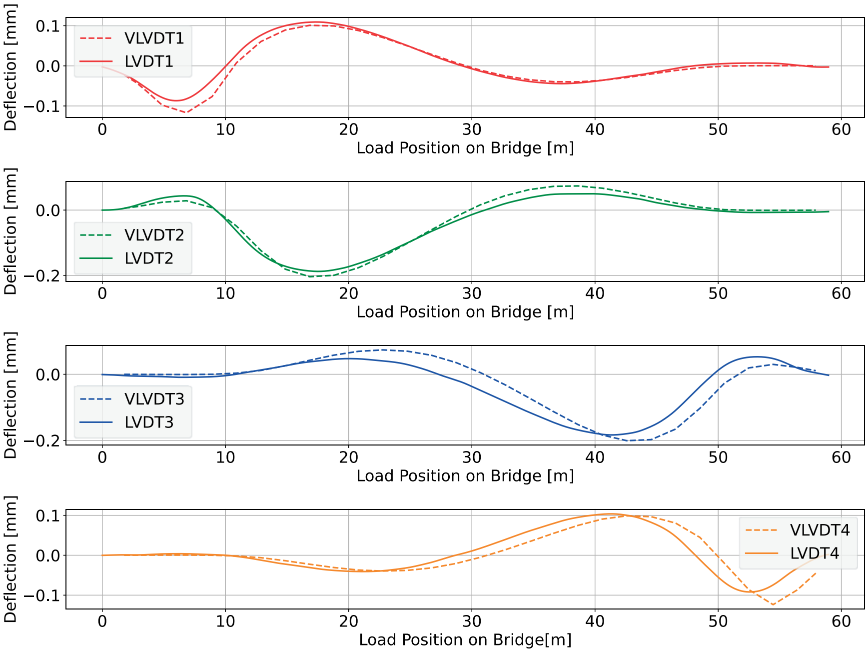


Figure 5.6 VLVDT and LVDT readings comparison for the 10-ton load.

A 37-ton proof-loading vehicle shown in Figure 5.7 is moved quasi-statically from the bridge's west to east. The reading from the measurement campaign is truncated at both ends to the axle distance of the proof-loading vehicle. Only the parts when the vehicle is completely on the bridge are considered. The deflection influence lines from the bridge's LVDT and model's VLVDT are compared in Figure 5.8 for the 37-ton proof-loading vehicle. The deflection influence lines of the model and the case study match closely. The model accuracy can be improved further by calibrating the spring stiffness values. For the prototype development and concept demonstration, the current model is considered as the validated FE model. The physics-based digital twin reflects the deflection influence lines the same as the case study bridge for the applied load.

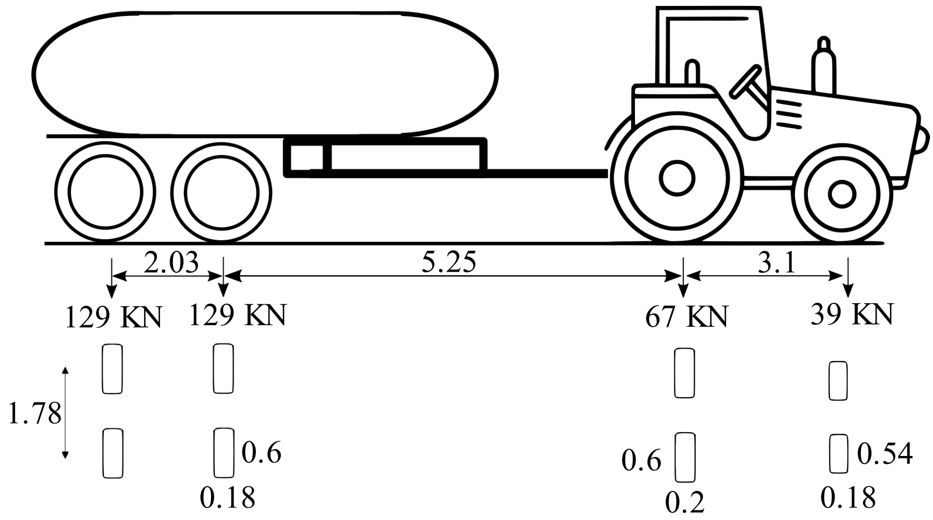


Figure 5.7 37-Ton Proof loading vehicles.

5.5 Discussion of FE results

The simulation results follow the profile of the measured results from the bridge, but they do not match accurately throughout every load positions, as can be seen in Figure 5.6 and Figure 5.8. The readings of VLVDT1 and VLVDT2 match better with the experimental readings compared to those of VLVDT3 and VLVDT4. The FE model deflection values are higher than the measured results, so it is conservative. Therefore the calibrated spring boundary values are set as the validated model. This is considered a reasonable accuracy for the prototype and concept validation. The springs stiffness need to be calibrated to improve the model further, if necessary. The FE model uses a mesh of less than 2% variations. Hence, the results could further be improved by increasing the mesh density. Whether improving the FE model further is necessary depends on the sensitivity of the DIL to the damage severity. At this stage of design, this information is not available.

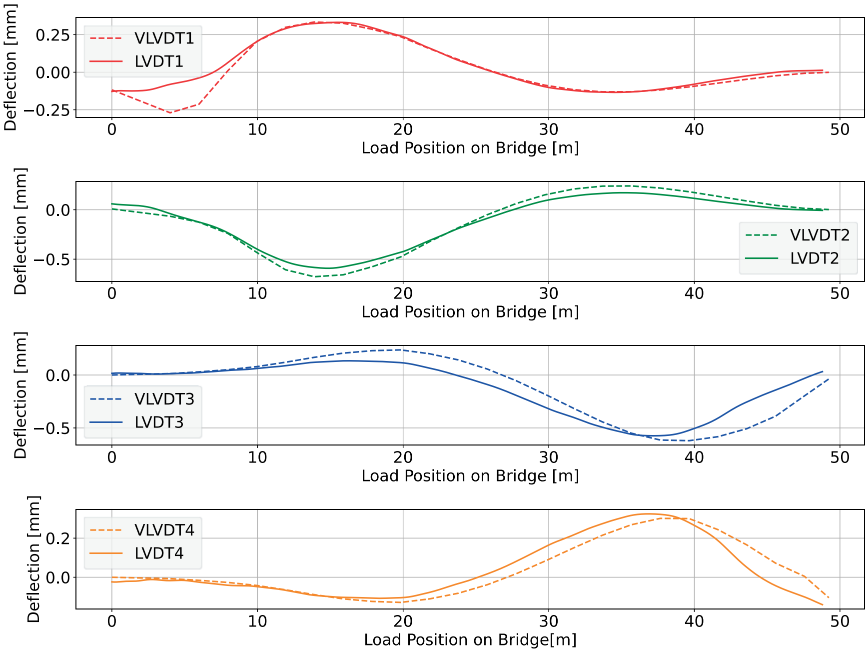


Figure 5.8 VLVDT and LVDT readings comparison for the 37-ton load.

5.6 Summary

In this chapter, the bridge's simplified FE model was developed and calibrated using the measurement campaign readings from the 10 ton proof-loading vehicle. The calibrated model has been validated using the 37-ton proof-loading vehicle. This fulfills the requirements for the physics-based model development (SyR 6), and validation of the model using measurement campaign reading of the case study bridge (SyR 7). The validated digital twin model will be used to create the scenario analysis modules in the tool and the tool design is explained in the next chapter.

6 Scenario analysis tool

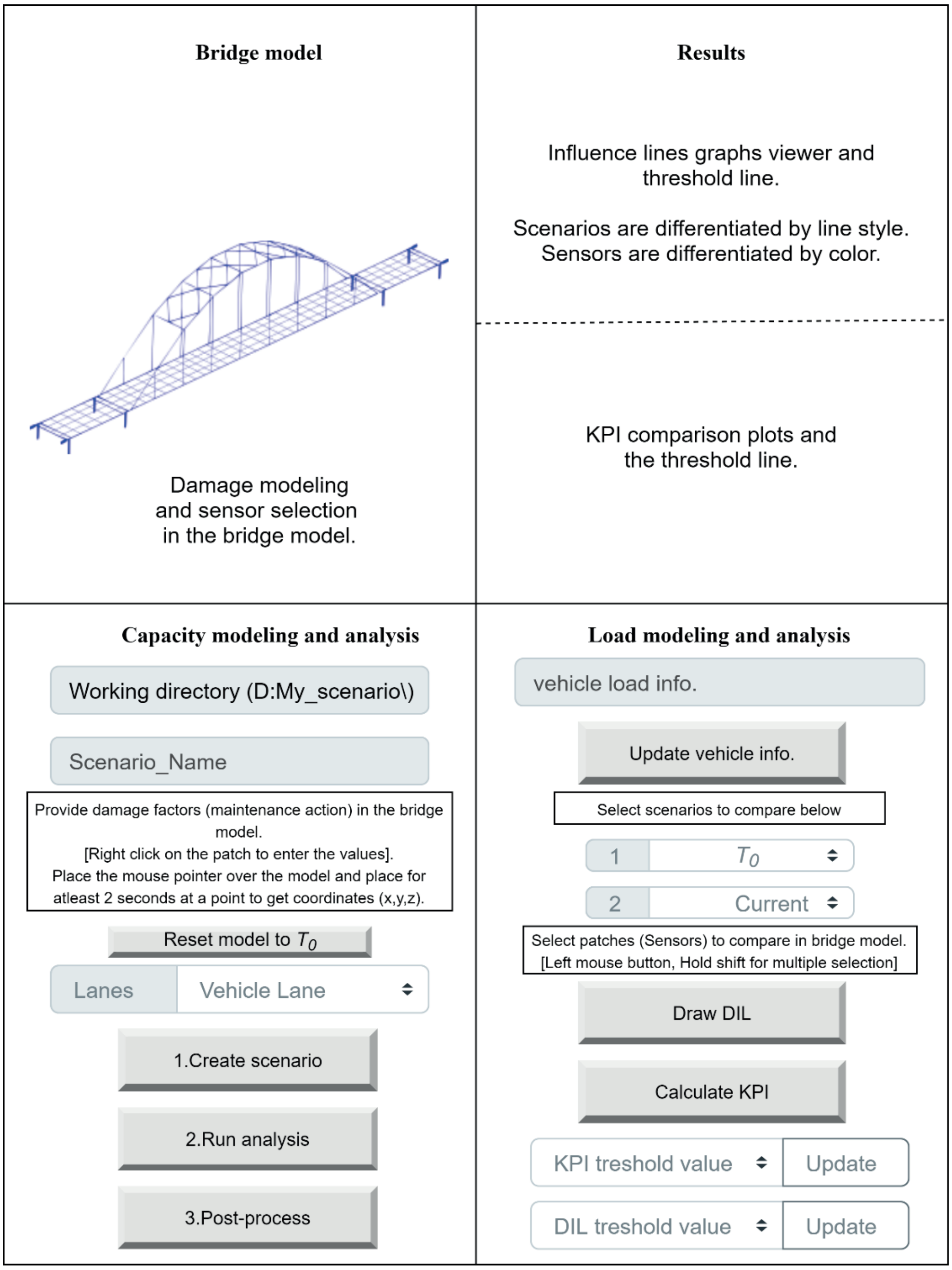
In this chapter, the final product of this project, i.e. the scenario analysis tool, its subsystems and the GUI are discussed. This chapter provides the necessary details on the components to understand their functions. The concept and the tool are validated by modelling a few damage scenarios and quantifying the consequences. The tool's detailed schematic and the connection between components that are required to apply the tool are presented in a user manual in Appendix C.

6.1 Tool Graphical User Interface (GUI)

The tool has been developed considering the user knowledge set in ShR 5. The user does not need to have extensive knowledge of physics-based failure mechanism and tool backend programming to operate the tool. The scenario analysis tool GUI design is show in Figure 6.1. It shows the essential options that are required for the user to interact with the tool. The main window contains four sections: bridge model, capacity modeling and analysis, load modelling and analysis, and results.

The scenario modeling and use of the GUI are explained below. In the section 'Capacity modeling & analysis' the following steps should be taken:

1. Set working directory where all the files created will be stored.
2. Define the scenario name to label the scenario files and store them in the working directory.
3. Model the damage or the maintenance action in the bridge model (capacity scenarios). The FE model is validated to match the T_0 condition and has been incorporated in the tool. The flexural rigidity (EI) factor is one (1) when the model is 100% matching the T_0 condition. Damage can be modelled by reducing the factor to less than one and a maintenance action to strengthen



Bridge model

Damage modeling and sensor selection in the bridge model.

Results

Influence lines graphs viewer and threshold line.

Scenarios are differentiated by line style. Sensors are differentiated by color.

KPI comparison plots and the threshold line.

Capacity modeling and analysis

Working directory (D:My_scenario\)

Scenario_Name

Provide damage factors (maintenance action) in the bridge model.
[Right click on the patch to enter the values].
Place the mouse pointer over the model and place for atleast 2 seconds at a point to get coordinates (x,y,z).

Load modeling and analysis

vehicle load info.

Select scenarios to compare below

Select patches (Sensors) to compare in bridge model.
[Left mouse button, Hold shift for multiple selection]

Figure 6.1 Scenario analysis tool GUI.

1. the structure can be modelled by increasing the factor greater than one. For example, to reduce strength by 20% from T_0, an input factor 0.8 has to be entered at the respective patch in the model. Right click on the patch to enter the factor values on the model.
2. Selection of load moving lane, either mid lane or vehicle lane. If mid lane is selected, a 1 kN (1000 N) load is moved at the center line of the bridge. For vehicle lane selection the information on vehicle axial load and distance has to be provided as per the format [(Axel No., Load in N, Distance from 1st axel in m),….].

After providing the inputs, the user can create the scenario, run the analysis and post process the results. After post-processing, the influence lines are ready to be extrapolated to different loading scenarios and the KPI calculation can be performed. The 'load modelling and analysis' window section has options to provide the inputs for loading scenarios and KPI calculation. If there are no changes in the capacity inputs, different loadings can be analyzed by changing the vehicle load information and updating it. The KPI is calculated by comparing the current modelled scenario with the T_0 scenario. The sensor on the patches can be selected using the left-mouse button. The user can click the button 'Draw DIL' to visualize the influence lines in the 'Results' window. Threshold value inputs can be given by the user to draw straight line markers on the plot. The user can calculate the KPI of the selected sensors by clicking the 'Calculate KPI' button and get the KPI plots in the result window.

6.2 Physics-based digital twin subsystem

The validated digital twin model obtained in section 5.4 is used to develop the scenario analysis tool. To create the scenario analysis tool the model needs to be equipped with element groups to take inputs from the user. The elements are grouped and a virtual displacement sensor (VDS) or (VLVDT) node is assigned to each group. The displacement influence line (DIL) is extracted at the VDS. To consider the capacity input, the element group stiffness (*EI*) is assigned with a factor value that can be changed by the user. For the reference state $T_{0,}$ factors are set to one in every group, the factors can be increased for a structural reinforcing maintenance action or decreased to represent damage by the user. To consider loading inputs from the user, nodes inline are grouped to create a line. Two lines of nodes are created; the lines are two meters apart on the bridge deck and form a lane for a vehicle axle to travel. A load of 1 kN (1000 N) is applied to the axle in the digital twin model for all loading scenarios. Using the principle of superposition, the DILs are extrapolated from the 1 kN DILs to the user defined loads. The load is moved across the bridge quasi-statically and the DILs at all the virtual displacement sensor nodes are extracted as output. Post-processing of the FE results is done to extract the DILs. The post-processing and superposition extrapolation components are created using Python and placed in this subsystem. The outputs of this subsystem are sent to the KPI subsystem. The model replicates the physical properties (SyR 8), offers provision to include a degradation model (SyR 9) and considers the different loads on the structure (SyR 10) , which means that these three requirements are fulfilled by this subsystem.

6.3 KPI calculator subsystem

The predicted DILs for a scenario and the reference state DILs are used in the KPI calculation. The reference state DILs are extrapolated using the principle of superposition to the user load inputs. The predicted DILs and extrapolated reference DILs are compared to calculate the KPI curve. The percentage difference between the two curves (Eq. 4.1) is plotted to quantify the consequence due to a scenario.

6.4 Design validation

In this section, three damage scenarios are modelled and KPIs are calculated to validate the scenario analysis concept. Three different damage scenarios are considered by reducing the stiffness in an element group. The damaged element group is in the south beam girder, the group has a meter length, and the group's midpoint is at 8 m from the west end of the bridge. The 10-ton proof loading vehicle is moved from the west end to the east end of the bridge in all three scenarios to keep the loading parameters the same. The location of the damage and location of virtual displacement sensors (VLVDT) used to extract the feature are shown in Figure 6.2. The stiffness (*EI*) of the element group is reduced by 10%, 20%, and 25% from the T_0 state, respectively, by specifying 0.9, 0.8, and 0.75 as factors at that element group to create three different example scenarios; The DILs from the sensor placed on that element group and at three other places away from the element group (VLVDT 2, 3 and 4) will be extracted for KPI calculation. The three scenario analyses are run and the results are compared to the T_0 reference state DILs.

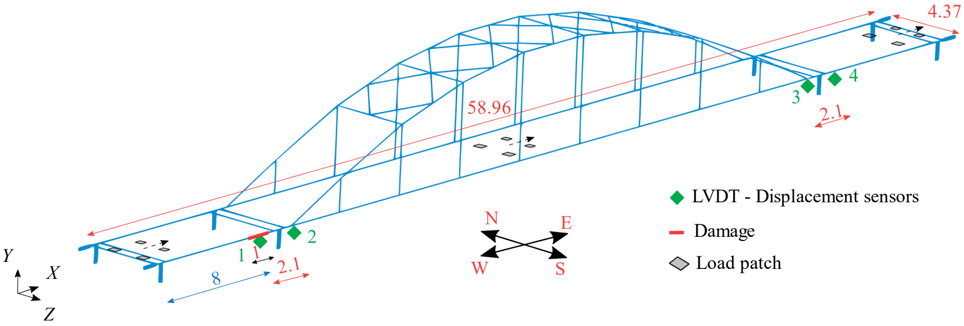


Figure 6.2 VLVDT and the damage modelled on the digital twin.

The 10% damage scenario influence lines of VLVDT 1,2,3, and 4 are compared with corresponding T_0 state readings. VLVDT 1 is placed on the damaged section, and the rest of the sensors are away from the damage, as depicted in Figure 6.2. The DILs are plotted in Figure 6.3 and the percentage difference KPI is plotted in Figure 6.4.

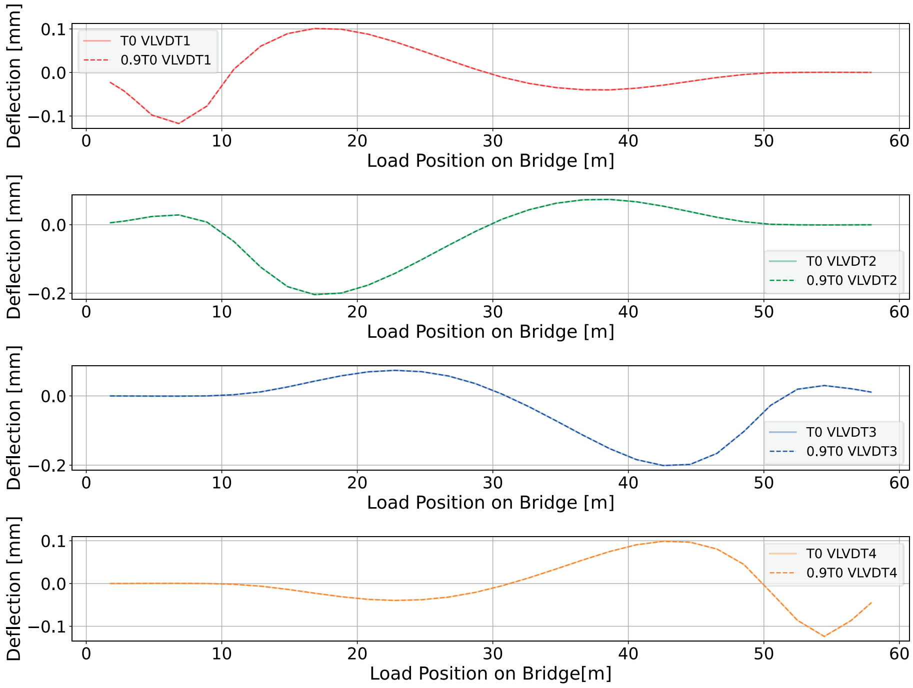


Figure 6.3 Comparing T_0 and 10% damage element group (0.9 T_0) VLVDTs.

The influence lines look identical for the naked eye, but the KPI percentages shows a peak around the damage location, see Figure 6.4. This proves that the influence lines are sensitive to damage. The percentage difference KPI shows a peak near the damage even when the sensor is not at the damage location. This proves the DIL are not only sensitive to damage (SHM Level 1) in a robust way, but can also be used to estimate the damage location (SHM Level 2). To fulfill SHM level 3, the damage severity has to be quantified. The KPI of VLVDT1 is more sensitive than the other VLVDTs since it is on the damaged patch. In the scenario analysis, the VLVDT on the damaged patch is compared to T_0 to calculate the KPI and quantify the damage severity.

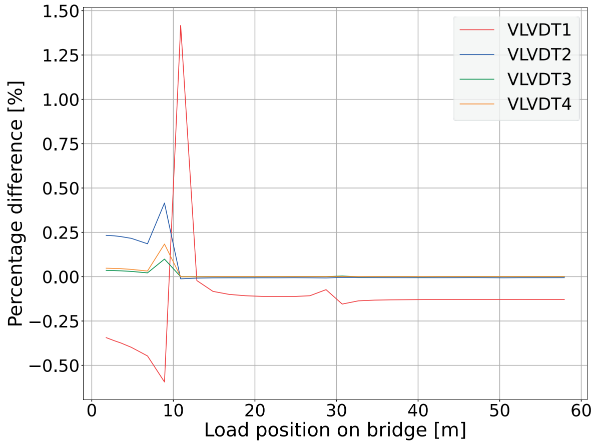


Figure 6.4 T_0 state and 10% Damage comparison of KPI for all LVDTs.

Figure 6.5 shows the VLVDT1 influence lines for the different damage severities. The influence lines look identical in other locations except near the damage. Near the damage (at 8 m) a slight variation can be seen in Figure 6.5. The KPI needs to be calculated to assess the performance and quantify the consequence.

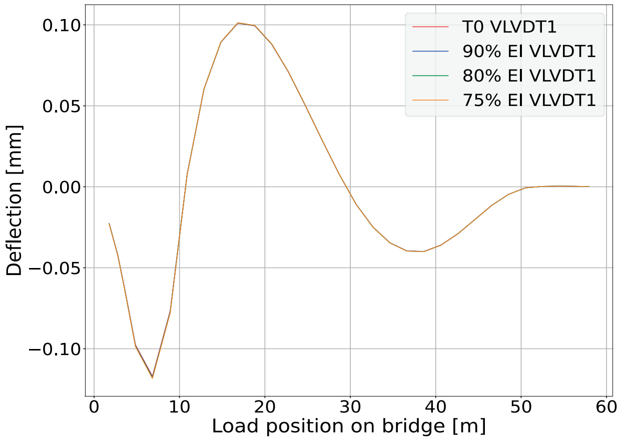


Figure 6.5 DILs of VLVDT1 for varying damage severity.

The scenarios' KPIs are calculated by comparing the damaged state DILs with the T_0 reference state DIL as plotted in Figure 6.6. This reveals that the more severe the damage, the higher the peak is in the KPI curve at the damage location. This proves that the proposed KPI is sensitive to damage severity, thus achieving SHM Level 3. This percentage difference in DILs is the quantified consequence due to a maintenance action or damage.

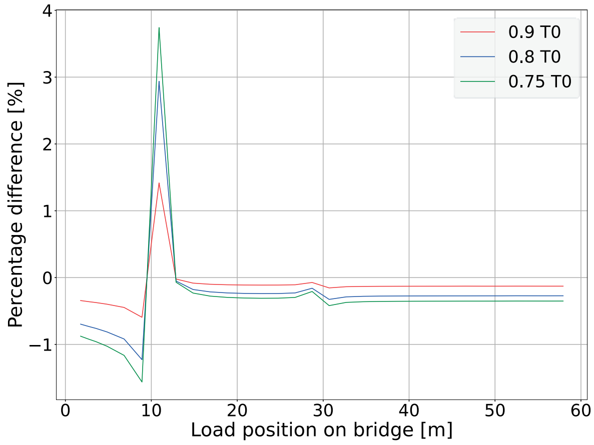


Figure 6.6 KPI comparison of different damage severity.

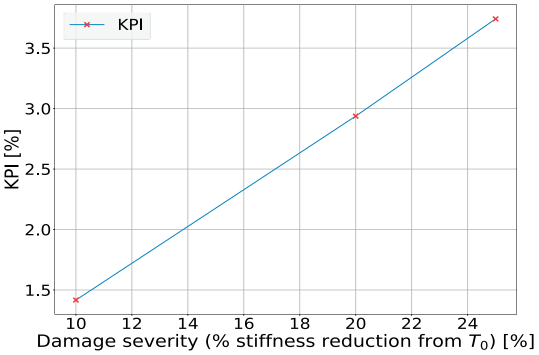


Figure 6.7 Damage severity vs KPI near the damage location.

Finally, Figure 6.7 shows the relation between damage severity and the KPI at the location of the damage. The KPI percentage increases proportional to the damage percentage increase, compared to the reference state T_0. This relation can be used to quantify the damage consequence and thresholds on the KPI can be set by the user to support maintenance decisions.

In the GUI of the scenario analysis tool as discussed in section 6.1, based on the user selected sensors, the KPI of the scenarios will be calculated similarly. In Figure 6.1, the calculated result will be displayed in the Results window for the user.

6.5 Summary

The damage scenario analysis results prove that the displacement influence line feature is sensitive to damage. Thus, the DILs can be used for performance assessment of the bridge structure. The KPI can be used to quantify the consequence due to a maintenance action or damage. This validates the scenario analysis concept and tool design; it fulfils and validates the requirements ShR 1 and 2, and all the SyRs. The validated tool will be submitted as a package to install and operate (ShR 7).

6.6 Requirements checklist

Table 6.1 shows the checklist of the requirements. The compliance of the requirements is stated in the corresponding columns and reference is made to the related chapters that explain the tool design. The partially fulfilled and unfulfilled requirements are explained with the reason for not fulfilling.

Table 6.1 Requirements verification.

Type	**No.**	**Requirements**	**Compliance**
ShR	1	A prototype tool shall be designed to study the consequence of maintenance actions on bridges.	Satisfied. The tool has been designed and validated. Refer to sections from 6.1to 6.4
	2	The tool shall be used without depending on expensive licensed software to make it affordable for RWS.	Satisfied. Open-source components are used to develop the tool. See section 4.3
	3	The tool output shall be integrable with the current structural assessment process followed by RWS Hengelo.	The current RWS practice uses expert opinion based scores to assess structures. The outputs of a physics-based KPI percentage number to makes it easier for the user to implement it as scoring. Implementation not done and verified.
	4	The tool shall be accessible through a graphical user interface.	Satisfied. Refer to section 6.1.
	5	The tool shall be usable by the user with at least 'higher professional education' (in dutch: Hoger beroepsonderwijs - HBO) level knowledge.	Satisfied in design. See section 4.5 and 6.1. The tool has a simple GUI hiding the backend wok from the users. It has to be verified.
	6	The tool shall be developed within the duration of the PDEng. program with at least 60 credit hours.	The concept is developed and validated.
	7	The tool should be submitted as a package to install and operate.	Partially satisfied. The concept, tool design and validation makes it possible that the tool can be packaged to install and operate. The development of the post-process modules and GUI are not fulfilled due to time constraints.

SyR	1	A method shall be formed for the bridge structural performance assessment.	Satisfied. See sections 4.5, 6.3 and 6.4
	2	A physics-based model shall be developed to replicate the case study bridge behavior under different vehicle loads.	Satisfied. The FE model replicates the deflection behavior under vehicle loads. See section 5.4
	3	A physics-based parameter shall be identified to assess the performance of the structure.	Satisfied. Deflection influence line (DIL) is used as the parameter. See sections 4.5, 6.3 and 6.4
	4	The performance parameter shall be sensitive to structural changes and loads acting on the structure.	Satisfied. Proof for DIL as a sensitive parameter for load and structural changes is validated. See section 6.4.
	5	The performance parameter shall be monitorable.	Satisfied. See sections 5.3. The DIL is monitorable using displacement sensors such as LVDT.
	6	The tool and the model shall be developed for a case study bridge.	Satisfied. The case study bridge is in section 4.4, and the model is validated in section 5.4.
	7	The tool shall be validated using measurement campaign readings of case study bridge and replicate the physics-based parameter as accurate as possible.	Satisfied. See section 5.4.
	8	The model shall have provisions to alter the physical properties to replicate structural changes due to maintenance action.	Satisfied. The provisions are given in section 6.1 and it is validated by changing properties in section 6.4

	9	The model shall have provisions to include degradation models to study changes over time in the structure.	Satisfied. See sections 6.1 & 6.4. The maintenance action or damage is modeled as stiffness reduction. It can be linked to degradation models.
	10	The model shall have provisions to consider load variations on the bridge to consider its effects on the physics-based parameter.	Satisfied. See section 6.1and 6.2, where a provision to consider different loads are given. Section 0 where different proof-loads are applied to validate the model.
	11	The tool shall contain a performance indicator (KPI) to compare the performance parameter of the bridge at different conditions to assess and quantify the consequence.	Satisfied. See sections 6.3 and 6.4.
	12	The output of the tool shall be readable/visualizable to by the user.	Partially satisfied. See sections 6.1. and 6.4. The concept GUI is designed to display the KPI plots. This is not implemented in the tool yet.
	13	The tool shall be scalable to add bridge elements and the concept to other bridges.	The FE model is scalable for a bridge. The concept is scalable following the same design cycle, but this has not been validated yet.

7 Conclusions and recommendations

The digital twin model of the case study bridge has been developed. The conclusions and recommendation for future development are given in this chapter.

7.1 Conclusions

The project's aim was to design a tool to assess the consequence of a maintenance action on a bridge. The first step is to identify a physics-based method to measure the bridge's performance to assess the consequence. The knowledge question "*how to assess the structural performance of a bridge?*" is answered using the displacement influence lines. The displacement influence line (DIL) is identified as the performance measurement feature.

A scenario analysis tool to predict the performance feature of the case study bridge was designed and the concept has been validated using the digital twin model of the bridge. The tool predicts the performance feature for the created scenario using the digital twin model. The model can predict the feature when the bridge elements are within linear elastic limits. It is validated using the measurement campaign readings of the case study bridge. It can consider capacity parameters, maintenance actions and damage, load parameters (vehicle loads) to create scenarios. The tool predicts the performance feature for the created scenario using the digital twin model.

A percentage difference KPI is identified to assess and quantify the consequence. The KPI is calculated by comparing a scenario DIL with the reference state DIL. The quantified KPI provides insights on the structural performance to the user. The user can set thresholds on the KPI. The user can use these insights to support the maintenance decision.

Economic viability is achieved using open-source tools. The tool is modular and separated into several subsystems. New subsystems can be developed and linked to the tool. This enables the scalability of the concept. The concept can be applied to other bridges following the same design cycle. The GUI is designed for the users to access the tool, to create scenarios and to read the results. The requirements checklist is presented in Table 6.1, showing that the digital twin concept and the scenario analysis concept meet most of the requirements. The requirements such as scalability to the other bridges and implementation with RWS current maintenance practices are yet to be verified.

7.2 Recommendation

The prototype can be further be developed by iterating over the design of subsystems. The following recommendations provide directions to develop the tool and concept further:

- More design iterations on the subsystem and component level can be performed. The FE model can be made more accurate by choosing the converged mesh, and a KPI's other than the percentage difference can be explored to quantify the damages. Model updating algorithms shall be developed to replace the trial and error calibration method.
- In this prototype, the KPI is calculated by comparing the damaged bridge to the reference state to assess the consequence of the structural changes. The reference state is set as the measurement campaign instance, instead of the newly-built situation. Therefore, an assessment of the reference state bridge has to be done to study if that reference condition is still acceptable

- Effects due to environmental loadings such as temperature are not considered in the prototype. More research is required to understand the environmental effects.
- Suitable degradation models can be developed or selected and then used as the capacity input in the tool. The predicted performance can be used to assess the consequences by comparing it them with the reference state performance. Using a degradation model enables a structural performance assessment over time.
- The concept of influence lines is scalable to other bridges. Similar to the displacement influence line feature, the stress influence line can be used to assess the structure’s performance.

References

[1] H. Klatter and J. Van Noortwijk, “Life-cycle cost approach to bridge management in the Netherlands,” *IBMC*, vol. 9, pp. 179–188, 2003.

[2] W. F. Chen and L. Duan, *Bridge engineering Handbook Fundamentals*, 2nd ed. Taylor & Francis, 2014.

[3] “Euro Codes.” [Online]. Available: https://eurocodes.jrc.ec.europa.eu/showpage.php?id=14.

[4] American Association of State Highway and Transportation Officials, *AASHTO LRFD Bridge design specifications*. 2012.

[5] K. Worden and J. M. Dulieu-Barton, “An Overview of Intelligent Fault Detection in Systems and Structures,” *Struct. Heal. Monit.*, vol. 3, no. 1, pp. 85–98, 2004.

[6] B. M. Imam and M. K. Chryssanthopoulos, “Causes and consequences of metallic bridge failures,” *Struct. Eng. Int. J. Int. Assoc. Bridg. Struct. Eng.*, vol. 22, no. 1, pp. 93–98, 2012.

[7] M.-J. Kallen, “Markov processes for maintenance optimization of civil infrastructure in the Netherlands,” Delft University of Technology, 2007.

[8] H. Klatter, A. Vrouwenvelder, and J. van Noortwijk, “Societal aspects of bridge management and safety in the Netherlands,” in *Klatter2006SocietalAO*, 2006, pp. 397–398.

[9] Rijkswaterstaat, “Guidelines on Performance-based Risk Analyses (PRA),” *ProBo Support Desk*, 2018.

[10] T. C. Viana da Rocha Feiri, “Risk Assessment Model based on RAMS Criteria,” University of Twente, 2015.

[11] M. Grieves, “Origins of the Digital Twin Concept,” *Rev. Obstet. y Ginecol. Venez.*, vol. 23, no. August, pp. 889–896, 2016.

[12] D. Kadleček, “Digital Twins.” [Online]. Available:

http://www.uaetu.com/tools/download.ashx?site=main&id=137. [Accessed: 03-Nov-2020].

[13] E. H. Glaessgen and D. S. Stargel, "The digital twin paradigm for future NASA and U.S. Air force vehicles," *Collect. Tech. Pap. - AIAA/ASME/ASCE/AHS/ASC Struct. Struct. Dyn. Mater. Conf.*, pp. 1–14, 2012.

[14] J. Lee, E. Lapira, B. Bagheri, and H. an Kao, "Recent advances and trends in predictive manufacturing systems in big data environment," *Manuf. Lett.*, vol. 1, no. 1, pp. 38–41, 2013.

[15] S. C.-Y. Lu *et al.*, "Digital twin-driven product design framework," *Int. J. Prod. Res.*, no. February, pp. 1–19, 2018.

[16] B. Ruth *et al.*, "Customer experience challenges: bringing together digital, physical and social realms," *J. Serv. Manag.*, vol. 29, no. 5, pp. 776–808, Jan. 2018.

[17] B. Schleich, K. Wärmefjord, R. Söderberg, and S. Wartzack, "Geometrical Variations Management 4.0: Towards next generation geometry assurance," *Procedia CIRP*, vol. 75, pp. 3–10, 2018.

[18] A. El Saddik, "The Convergence of Multimedia Technologies," *IEEE Multimed.*, vol. 25, no. June, pp. 87–92, 2018.

[19] F. Tao, J. Cheng, Q. Qi, M. Zhang, H. Zhang, and F. Sui, "Digital twin-driven product design, manufacturing and service with big data," *Int. J. Adv. Manuf. Technol.*, vol. 94, no. 9–12, pp. 3563–3576, 2018.

[20] K. Worden, R. C. Farrar, G. Manson, and G. Park, "The fundamental axioms of structural health monitoring," *Proc. R. Soc. A.*, vol. 463, no. 2082, pp. 1639–1664, 2007.

[21] H. Wenzel, *Health Monitoring of Bridges*. John Wiley & Sons, Ltd, 2009.

[22] A. Malekjafarian, P. J. McGetrick, and E. J. OBrien, "A Review of Indirect Bridge Monitoring Using Passing Vehicles," *Shock Vib.*, vol. 2015, no. 1, pp. 1–16, 2015.

[23] J. J. Moughty and J. R. Casas, “A State of the Art Review of Modal-Based Damage Detection in Bridges: Development, Challenges, and Solutions,” *Appl. Sci.*, vol. 7, no. 5, p. 510, 2017.

[24] M. M. Abdel Wahab and G. De Roeck, “Damage detection in bridges using modal curvatures: Application to a real damage scenario,” *Journal of Sound and Vibration*, vol. 226, no. 2. Journal of Sound and vibration, pp. 217–235, 1999.

[25] P. C. Chang, A. Flatau, and S. C. Liu, “Review paper: Health monitoring of civil infrastructure,” *Struct. Heal. Monit.*, vol. 2, no. 3, pp. 257–267, 2003.

[26] Z. W. Chen, Q. L. Cai, and S. Zhu, “Damage quantification of beam structures using deflection influence lines,” *Struct. Control Heal. Monit.*, vol. 25, no. 11, pp. 1–17, 2018.

[27] Z. Chen *et al.*, “Damage Detection in Long Suspension Bridges Using Stress Influence Lines,” pp. 1–11, 2015.

[28] I. Štimac, A. Mihanović, and I. Kožar, “Damage detection from analysis of displacement influence lines,” in *SICON - International conference on bridges*, 2006, pp. 1001–1008.

[29] S. Zhang, “Damage Detection in Beam Bridges Using Quasi-static Displacement Influence Lines,” *Appl. Sci.*, vol. 9, no. 9, 2019.

[30] A. Teughels and G. De Roeck, “Structural damage identification of the highway bridge Z24 by FE model updating,” *J. Sound Vib.*, vol. 278, no. 3, pp. 589–610, 2004.

[31] A. Strauss, R. Wendner, K. Bergmeister, and D. M. Frangopol, “Monitoring and influence lines based performance indicators,” *Appl. Stat. Probab. Civ. Eng. -Proceedings 11th Int. Conf. Appl. Stat. Probab. Civ. Eng.*, no. February 2018, pp. 1059–1068, 2011.

[32] J. Peiró and S. Sherwin, “Finite Difference, Finite Element and Finite Volume Methods for Partial Differential Equations,” *Handb. Mater. Model.*, vol. M, pp. 2415–2446, 2005.

[33] H. Fangohr, “A comparison of C, MATLAB, and Python as teaching languages in engineering,” *Lect. Notes Comput. Sci. (including Subser. Lect. Notes Artif. Intell. Lect. Notes Bioinformatics)*, vol. 3039, pp. 1210–1217, 2004.

[34] B. D. Rouyendegh and A. Yıldızbaşı, “Multi-criteria decision making approach for evaluation of the performance of computer programming languages in higher education,” no. March, pp. 1–10, 2018.

[35] “List of bridges over the Twente canals - Wikipedia.” [Online]. Available: https://nl.wikipedia.org/wiki/Lijst_van_bruggen_over_de_Twentekanalen. [Accessed: 27-Aug-2020].

[36] X. Jiang, L. Wang, and H. Zhang, “Static and Dynamic Analysis of A through Tied Arch Bridge,” *IOP Conf. Ser. Earth Environ. Sci.*, vol. 252, no. 5, 2019.

[37] T. Zordan, B. Briseghella, and T. Liu, “Finite element model updating of a tied-arch bridge using Douglas-Reid method and Rosenbrock optimization algorithm,” *J. Traffic Transp. Eng. (English Ed.*, vol. 1, no. 4, pp. 280–292, 2014.

[38] B. Briseghella, N. Gallino, C. Gentile, and T. Zordan, “Finite element modelling of a tied arch bridge from operational modal analysis,” in *ARCH’07*, 2007, no. September.

[39] J. G MacGregor and J. K Wight, *Reinforced concrete Mechanics and Design*, 6th ed. 2012.

[40] SCIA Engineering, “Advanced concept training FEA,” 2015. [Online]. Available: https://downloads.scia.net/support/sciaengineer/manuals/15/fem/[eng]advanced concept training - fem 2011.pdf.

[41] E. Oñate, *Structural Analysis with the Finite Element Method Linear Statics Volume 2. Beams, Plates and Shells*, vol. First Edit. 2013.

[42] J. E. Finke, “Static and dynamic characterization of tied arch bridges,” Missouri university of science and technology, 2016.

[43] H. Tran-Ngoc, S. Khatir, G. De Roeck, T. Bui-Tien, L. Nguyen-Ngoc, and M. Abdel Wahab, "Model updating for nam O bridge using particle swarm optimization algorithm and genetic algorithm," *Sensors (Switzerland)*, vol. 18, no. 12, 2018.

[44] F. Magalhães, Á. Cunha, and E. Caetano, "Dynamic monitoring of a long span arch bridge," *Eng. Struct.*, vol. 30, no. 11, pp. 3034–3044, 2008.

[45] C. Crosti and D. Duthinh, "A nonlinear model for gusset plate connections," *Eng. Struct.*, vol. 62–63, no. February, pp. 135–147, 2014.

[46] John Deere, "10 Ton proof loading_1." [Online]. Available: http://manuals.deere.com/omview/OMETN78900_19/?tM=%0Ahttp://manuals.deere.com/omview/OMETN78900_19/OULXBER,0001A41_19_20120306.html%0A. [Accessed: 05-Nov-2020].

[47] John Deere, "10 Ton Proof Loading_2." [Online]. Available: http://manuals.deere.com/omview/OMETN78900_19/OULXBER,0001909_19_20120426.html.

[48] Massey Ferguson, "37-Ton Proof Loading." [Online]. Available: https://maplelanefarmservice.ca/resources/specsheets/massey ferguson/8700/Massey_Ferguson_8700_Series_Specsheet.pdf.

[49] F. F. Udoeyo, *Structural Analysis*. 2020.

A Bridge and proof load details

A.1 Case study – Tankinkbrug

The Tankinkbrug, which is owned by Rijkwaterstaat, is situated over the Twente canal between the villages Delden and Goor. It is a continuous tied-arch bridge with two sets of pillars supports at a distance of 8.9 m from both ends. One set of pillars has roller support while the other provides simple support. The bridge’s total length is of 58.96 m and the total width is 4.37 m with 3 m riding deck. The riding deck is of concrete and has two steel beams as a superstructure. It has a steel bow arch which is connected to the beams through steel columns. This type of structure is called Langer Tied arch bridge, and it is designed in such a way that the arch columns only carry the axial loads. The Tankinkbrug with proof loading is shown in Figure A.1.

Figure A.1 Tankinkbrug with proof loading vehicle.

In July 2019, a measurement campaign was carried out considering different loading scenarios on the bridge. The structure was monitored for deflection and dynamic response when a proof-loading truck moves over the bridge. Linear variable differential transformers (LVDTs) and accelerometers were used to monitor the bridge response. The sensor layout on the bridge is shown in Figure A.2. The LVDT and the accelerometer fixed on the bridge are shown in Figure A.3 and Figure A.4.

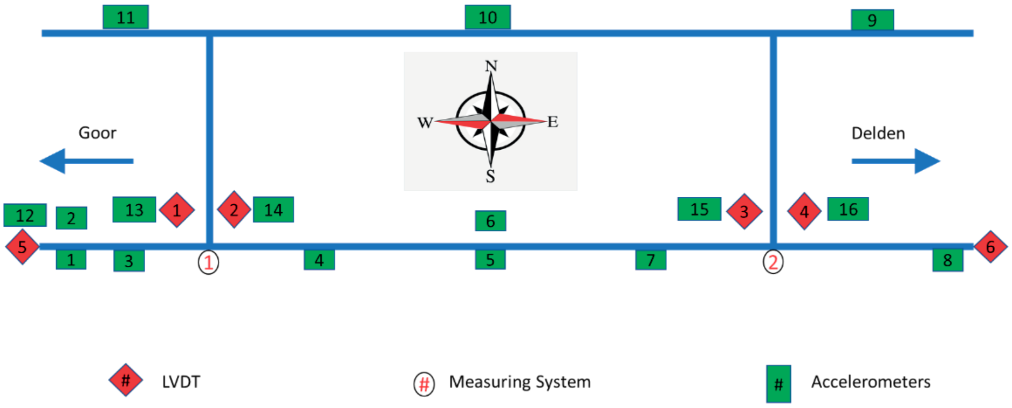


Figure A.2 Sensor Layout.

Figure A.3 LVDTs 1&2 arrangement on the bridge.

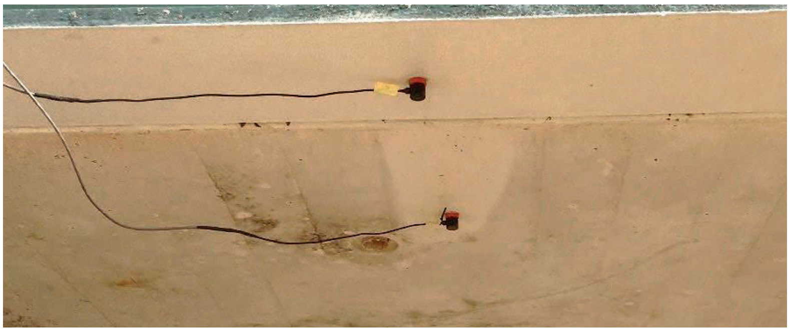


Figure A.4 Accelerometers attached to the bridge.

Multiple trials were done to record the response of the bridge. Three proof loading vehicles were used to exert loading on the riding deck. Four loading combinations were used during the measurement campaign:10 Ton (green tractor), 20 Ton (blue tractor), 30 Ton (red tractor), and 37 Ton (red tractor). The 20 Ton (blue tractor) load is used in combination with two 900 kg static loads to create multiple scenarios. The static loads are placed at ¼, ½, and ¾ distance of the bridge, and the 20 Ton tractor is moved to record the response. The vehicles are moved at a low speed of 5 km/h and a high speed of 12 km/h. Vehicle load, vehicle speed, static loads, and static loads position are the parameters varied to create scenarios. Twenty different scenarios are executed, and a few are repeated three times, leading to 52 readings in total. In all the scenarios, the proof loading vehicle is moved from the west end to the east end of the bridge to record the response.

Among the three proof loading vehicles, only the 10 Ton (green tractor) and the 37 Ton (red tractor) poof loading vehicle are used in the digital twin model development. The total load of the proof loading vehicle and tractor model is available from the measurement campaign data. To model the loadings in the FE model, the axle loads and contact patch details must be calculated from the available information. Unavailable information has to be calculated based on the manufacturer information or has to be estimated with reasonable limits.

A.2 Proof loading vehicle details

Only the total weight of the proof loading vehicle is available. However, the axial loads and tire patch information are necessary to apply the FE model's load and calibrate it. The required information is calculated using manufacturer information or estimated. Below provides the details of the calculation.

10-ton vehicle (Green) calculation

10-ton proof loading vehicle (Green) model is John Deere 6190 R. The exact tare weight is 9940 kg weight [43, 44]. This vehicle is used to in the FE model updating. Using the tare weight and dimensions the horizontal center of gravity(HCG) is calculated. The vehicle is in equilibrium. So based on the manufacturer info,

- Tare weight of the tractor = 7360 kg
- Front axle load = 2710 kg
- Rear axle load = 4650 kg
- Wheelbase = 2.8 m.
- HCG from the front wheel = 2.8× (4650/7360) = 1.77 m

Using the HCG assuming it does not change when the tractor weighs 9940 kg the axle loads are calculated:

- Front axle load = Total weight × (Wheel base-HCG)/Wheelbase = 3660 kg
- Rear axle load = Total weight – Front axle load = 6280 kg

It is assumed that the axle load is equally distributed to the wheels attached. The load distributed on the wheel is converted to a pressure distribution based on the contact patch of the tire.

Based on the tire model, the following information is gathered from the datasheet:

- Front tire width = 0.6 m
- Rear tire width = 0.7 m

It is also assumed the contact patch is a rectangle and the pressure distribution is uniform over the contact area. The patch width is estimated based on the available pictures of the proof loading tractor. The contact patch length is assumed as 200 mm which is approximately one-third of the patch width. The calculated load and vehicle information are presented in Figure A.5.

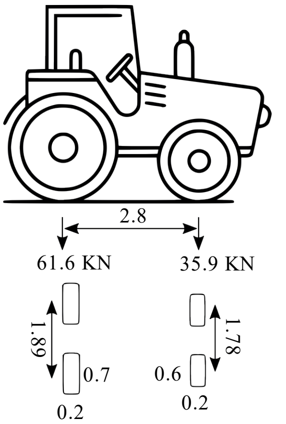


Figure A.5 10-ton proof loading vehicle dimensions [in meters].

37-ton vehicle (Red) calculation

The tractor model is a Massey Ferguson 8727 [48]. The tractor with trailer and ballast together weighs 37.2 tons. The weight of the empty tractor with the trailer is 26.7 tons as gathered from the measurement. The trailer blast load is 10.5 tons. The weight of the tractor is 10.8 tons identified from the manufacturer's data. The tare weight axle loads could not be found in the official documents. To approximate, the percentage of axle weight distribution similar to the 10-ton tractor is assumed. The total load of 10.8 tons distributed 36.8% to the front axle and 63.2% to the rear axle. It is assumed that the trailer load of 10.5 tons is distributed equally among the four tires. The axle distances are collected from the vehicle owner. The vehicle is shown in Figure A.6.

- Tractor and trailer weight from measurements = 26700 kg
- Tractor weight from manufacturer = 10800 kg
- Tractor front axle load = $0.368 \times 10800 = 3976$ kg
- Tractor rear axle load = $0.632 \times 10800 = 6823$ kg
- Trailer weight = 26700 -10800 = 15900 kg
- Trailer ballast load = 10500 kg

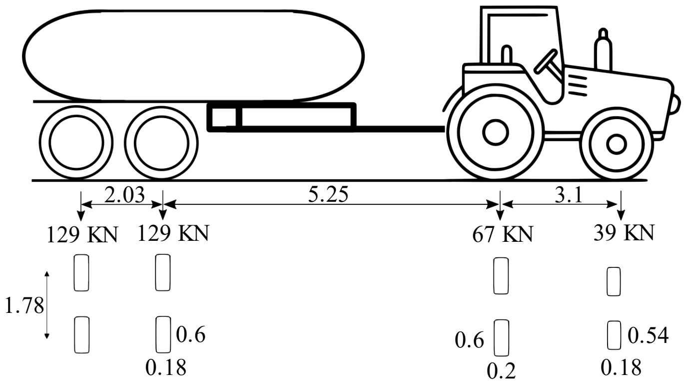


Figure A.6 37-ton proof loading vehicle.

B FE model simplification

The finite element model of the bridge is simplified to reduce the computational and modelling effort required. The deflection is based on the flexural rigidity (*EI)* based on the elastic modulus (*E*) and the second moment of inertia (*I*). The simplified section has to match the flexural rigidity of the deck section. First, the equivalent deck section is calculated by simplifying the material properties. Then the material equivalent deck is simplified to a rectangular deck, and the thickness is calculated to match the required *EI* by fixing the width to 4.37m.

The concrete deck is reinforced with steel bars. To calculate the flexural rigidity *EI,* the composite section has to be transformed into a homogeneous section with one material. The steel section is transformed to an equivalent concrete section. The steel's transformed is assumed to be concentrated at the same point in the cross-section as the real steel area [39].

Figure B.1 shows the deck reinforcement arrangement. Longitudinal reinforcement contributing to the bending stiffness is considered for calculation.

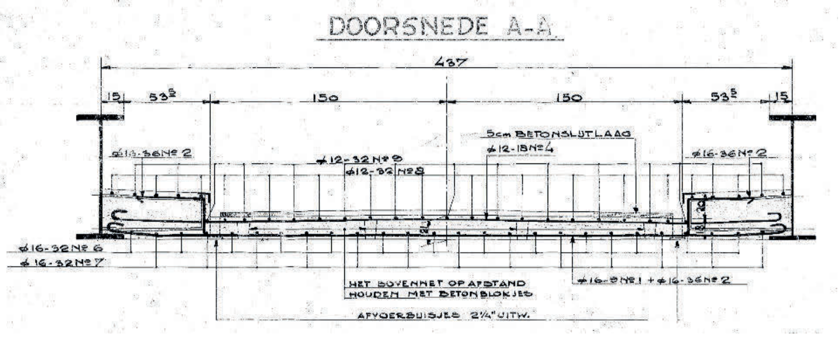


Figure B.1 Deck reinforcement arrangement

The material properties are listed in Table B.1 considered in the simplification.

Table B.1Material properties

	Steel	Concrete
Material Grade	S235	C25/30
Youngs modulus E [N/m^2]	2.1×10^{11}	3.15×10^{10}
Poisson ratio	0.3	0.2
Density [kg/m^3]	7850	2500

Steel reinforcements cross-section area calculation presented in Table B.2.

Table B.2 Steel reinforcement

Steel reinforcements	Diameter [m]	Numbers	Area [m2]
Type 1	0.012	31	0.00350424
Type 2	0.016	26	0.00522496
Total steel reinforcement cross-section area			0.0087292

Concrete cross section area = Total deck area – steel reinforcement area

= 1.28 – 0.00873

= 1.27127 m^2

Modular ratio calculation

$$R_m = \frac{E_{Steel}}{E_{concrete}} = \frac{2.1 \times 10^{11}}{3.15 \times 10^{10}} = 6.67$$

Equivalent concrete area top = $(R_m - 1)A_{stl\_Top} = 0.01988$ m^2

Equivalent concrete area bottom =$(R_m - 1)A_{stl\_Bottom} = 0.02963$ m^2

E_{steel} – Elastic modulus of the steel [N/m^2]

$E_{concrete}$ – Elastic modulus of the concrete [N/m^2]

A_{stl} – Cross-section area of the steel reinforcements [m^2]

R_m– modular ratio

The transformed section and the moment of inertia calculated using Solid Works option are shown in Figure B.2 and Figure B.3.

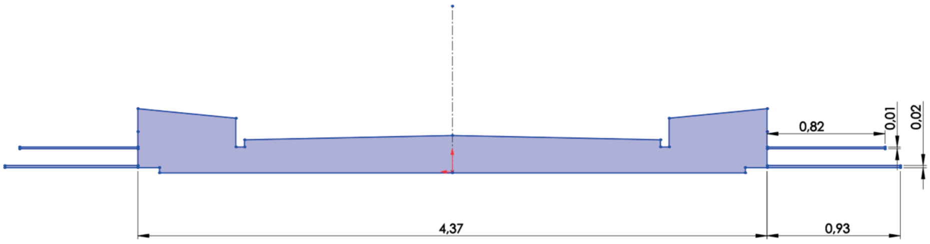
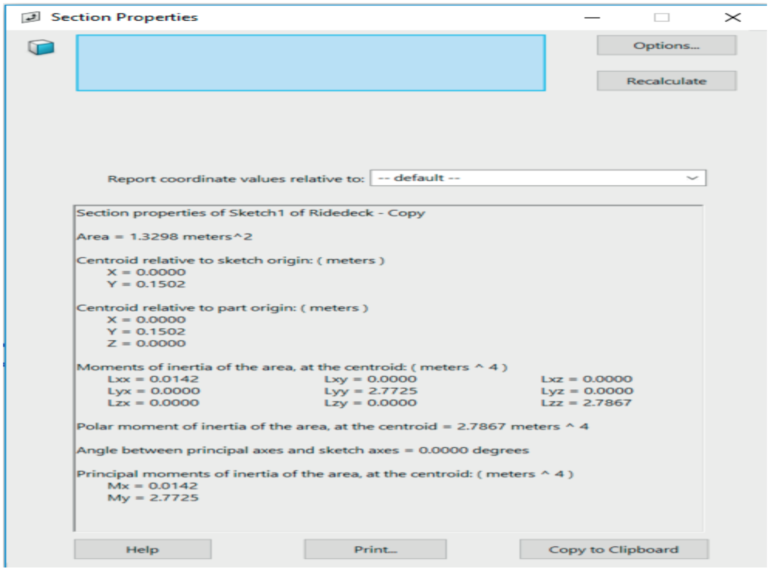


Figure B.2 Transformed deck section.



Report coordinate values relative to: -- default --

```
Section properties of Sketch1 of Ridedeck - Copy

Area = 1.3298 meters^2

Centroid relative to sketch origin: ( meters )
      X = 0.0000
      Y = 0.1502

Centroid relative to part origin: ( meters )
      X = 0.0000
      Y = 0.1502
      Z = 0.0000

Moments of inertia of the area, at the centroid: ( meters ^ 4 )
      Lxx = 0.0142                 Lxy = 0.0000                 Lxz = 0.0000
      Lyx = 0.0000                 Lyy = 2.7725                 Lyz = 0.0000
      Lzx = 0.0000                 Lzy = 0.0000                 Lzz = 2.7867

Polar moment of inertia of the area, at the centroid = 2.7867 meters ^ 4

Angle between principal axes and sketch axes = 0.0000 degrees

Principal moments of inertia of the area, at the centroid: ( meters ^ 4 )
      Mx = 0.0142
      My = 2.7725
```

Figure B.3 Moment of inertia of transformed section.

Moment of inertia $I_{xx} = 0.0142$ m^4

The equivalent section has to match the moment of inertia.

The bridge deck width B= 4.37 m

The section’s thickness calculated for a rectangular shape with 4.37m width and 0.0142 m^4 moments of inertia is as follows:

$$Deck\ thickness\ = = \left(\frac{I_{xx} \times 12}{B}\right)^{\frac{1}{3}} = 0.34\ \mathrm{m}$$

Similarly, the transversal reinforced concrete beam girder is also simplified.

The transverse beam is shown in Figure B.4. The main girder reinforcement passes through the deck and reinforces the beam. The equivalent concrete area is calculated to transform the steel reinforcement. The moment of inertia of the transformed section is used to calculate flexural rigidity by considering the concrete's elastic modulus.

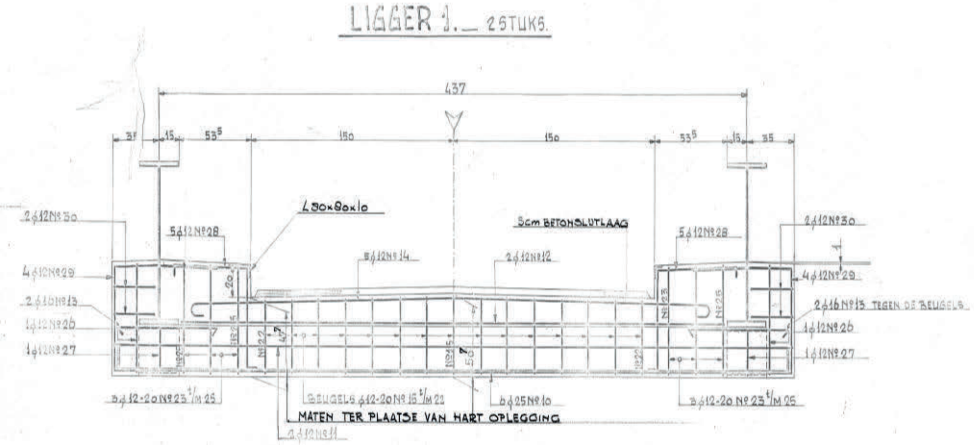


Figure B.4 Transverse deck beam cross-section view.

The simplified section width is fixed as 4.37m and the thickness is identified as 0.67. The deck and transverse beam are simplified in geometry and material properties and equivalent section are identified. The simplified thickness values are assigned to the FE model bridge deck.

C Tool detail design

In this appendix, the scenario analysis software tool's detail design is presented. The tool's subsystems, components and their connections, input and output parameters of the subsystems are discussed. Scenario analysis helps to evaluate the bridge's structural performance due to damage and/or load acting on it. Evaluating structural performance at different states and comparing it with a reference state provides insights to support the maintenance decision. The bridge's digital twin at the measurement campaign state (T_0) is set as the reference state in the scenario analysis module.

A scenario is created by changing the element stiffness in the FE model and applying a load on it to observe the structural response. The DILs sensitivity to damage is utilized to quantify the change between the T_0 state and a new scenario state T_d. For the same load moving on the bridge, the percentage difference between T_0 and T_d state DILs is the performance change in the bridge structure. The load applied can be varied to study the structure's response and create loading scenarios. In the scenario analysis module, the user can alter the capacity and load parameters to study the changes in DILs. The detailed design of the tool is explained in the next sections.

C.1 Tool design layout

The tool has GUI subsystem, Physics-Based model subsystem and KPI calculation subsystem. Each subsystem has multiple components to enable its functions.

The GUI subsystem components are user input and output, Physics-based model subsystem components include the reference state digital twin, scenario analysis template, FE solver, and post-processing. KPI calculation subsystem contains the analytical components predicting the loading scenarios and comparison component. These components and the subsystems are interlinked to create the output based on the user input. The connections are made based on the process outline presented below:

Step 1: Take user input to model damage scenarios in the FE model using GUI.

Step 2: Use scenario templates and the user input to create the scenario setup files.

Step 3: Send the scenario files to the FE solver, run the solver to solve and get the FE results.

Step 4: Extract displacement influence lines (DILs) from the FE result.

Step 5: Based on the user load input using the principle of superposition extrapolate the DILs to the loading scenarios.

Step 6: Use the reference state DILs stored to extrapolate for the user input.

Step 7: Compare the extrapolated reference state DIL and scenario DILs to calculate percentage KPI.

Step 8: Display the results. Reference state DILs, scenario DILs and the calculated KPI are presented to the user in the GUI output window.

These steps are repeated based on the user input; if there is no change in capacity parameters the process starts from step 5 and continue till the end to create the output. This reduces the computational effort required to study a scenario. In a new scenario, where the capacity parameter changes, all the steps will be repeated to predict the response.

C.2 Installation

One time user input, installation directory path is given as input. The installation directory path must not contain space (ASCII 32) character for the tool to work. The templates, code_aster solver, analytical solver and the T_0 calibrated model are stored in the installation directory. Mesh groups are added to the calibrated model to enable the scenario analysis template. This template is used to create a scenario analysis setup. The template only takes damage scenarios and unit point load or unit axle load based on user input. The FE model component to create the scenario analysis has two parts, the mesh and the comm template. The mesh is the skeleton representing the shape and structural members of the bridge. The mesh elements are grouped to assign material properties, sectional properties, exert force and boundary conditions on it. In the mesh template, the deck and beam girders are grouped and a virtual displacement sensor (VDS) is fixed at the mid of every group to extract DILs. The elements are grouped in mesh to form 1m^2 size patches on the deck and 1 m length in the beam girders in this prototype. The groups' size can be reduced as small as the element size of the mesh to model damages. Considering the time limit of this project, it is decided to model 1m^2 patches. In the comm template, the damage is modelled with stiffness reduction in material properties. The factors given by the user in the GUI bridge model are multiplied with the modulus of elasticity to reduce flexural rigidity (*EI*).

Using the mesh group names the corresponding stiffness for that groups can be assigned. User inputs are linked to the group names and corresponding stiffness reduction factors are multiplied with the T_0 state elastic modulus. To apply loads, the vehicle lanes and the mid lane are defined as groups. The mid lane is designated to move a point load over it. The vehicle lanes are designated to move the axle loads over them. Mid lane line runs at the center deck's center, and the vehicle lane consists of two lines that are 2 meters apart on the deck. The vehicle lane distance is decided based on the width of the deck. The lanes can be defined along any longitudinal node line, but only two lane options are given for the prototype.. The DILs are the lines from VDS by moving the load on mid lane. The field DILs are the lines from VDS by moving the axle load on the vehicle lane. T_0 reference state DILs and field DILs extracted for all VDS and placed as databases in the installation directory. These databases of DILs will be used to compare it with damaged scenario influence lines and evaluate structural performance.

The analytical solver extrapolates the results based on the user input loading parameters. The analytical solver also contains the part that calculates the KPI. The installation directory stores the templates, the FE solver component, analytical solver, reference state DILs as shown in Figure C.1 and can be accessed by other components and subsystems.

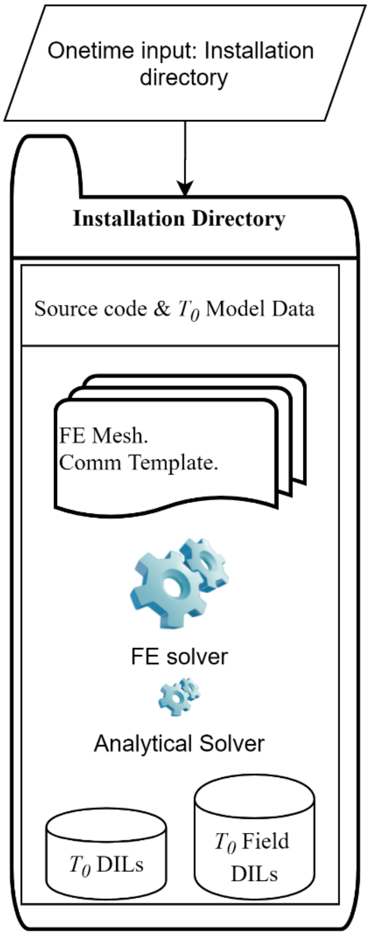
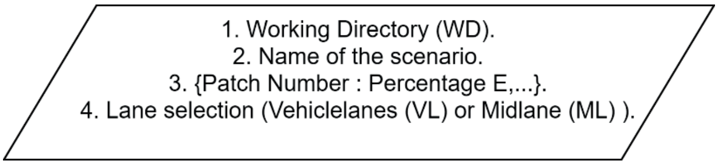


Figure C.1 Installation subsystem.

C.3 User inputs

After installation, the user starts the application by opening the spreadsheet. The user provides inputs required for the scenario analysis using the spreadsheet GUI. The inputs are shown in Figure C.2.



1. Working Directory (WD).
2. Name of the scenario.
3. {Patch Number : Percentage E,...}.
4. Lane selection (Vehiclelanes (VL) or Midlane (ML)).

Figure C.2 User input and GUI elements of the tool.

The bridge layout GUI represented in the spreadsheet is shown in Figure C.3. Inputs and user actions in the analysis steps are initiated through the GUI.

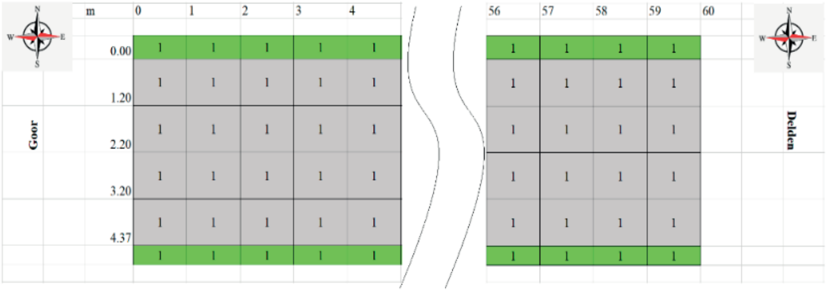


Figure C.3 GUI of the bridge deck and main girder representation.

The GUI layout shows mesh group patches with a scaled grid. The patches are linked with VDSs placed in the middle. The scaled grid around the patches and the VDS number link enables the user to model damage at a particular location and extract the influence line of that patch from the node. User inputs in Figure C.2 are explained below:

1. Working directory (WD): The working directory must be specified to store the scenario setup files and the results. The working directory should be new for every capacity parameter changing scenario. The space (ASCII 32) character is not allowed in the naming.
2. Naming the scenario: The name of the scenario is used to name the files. The space (ASCII 32) character is not allowed in the naming. If the same scenario name is repeated in the current working directory, the files will be overwritten. So name scenarios uniquely to avoid losing older files.
3. {Patch Number: Percentage E,…}: Multiple damages or structural modifications can be modelled using this input. Using the GUI bridge layout (Figure C.3), inputs are gathered from the user. The green highlighted sections represent the steel beam girders and the grey highlighted are the concrete decks. The user then specifies the factor of stiffness reduction in the corresponding deck or the girder patch. The T_0 state is set as factor 1. From the reference T_0 state, the damage factor can be modified. For example, to reduce 20% stiffness in the north beam girder between 1 to 2 m from the west end, change the Green color patch's value in 1st row 2nd column to 0.8. To strengthen it by 10 % from T_0 use 1.1 as the factor. Multiple damages can be modelled by changing the factors in different patches.
4. The user selects Mid lane (ML) or Vehicle lane (VL), and based on this decision point the next steps will follow. The mid lane load option is given to get the DIL by moving the 1 kN load on it. If the vehicle lane is selected, the unit axle load will be moved, and the field DILs are extracted. Field DILs are extracted by applying 1000 N (1 kN) axle loads in the vehicle lanes. The 1 kN axle load is divided equally and a 0.5 kN load is applied on each vehicle lane. The 0.5 kN loads are moved in parallel on the vehicle lanes. Distance between the vehicle

lanes is fixed at 2 meters considering the width of the bridge. At every instance a new damage scenario is created by the user; the FE simulation subsystem has to be rerun to calculate and extract the new DILs. These field DILs are extrapolated using the tandem load user information in the analytical section. The tandem load calculation using the vehicle information is deactivated if the mid lane is selected. Applying 1 kN load and extracting DILs is convenient for the extrapolation. The principle of superposition can be applied easily by multiplying increasing or decreasing factor to the DILs. The midlane option (ML) is given as an option envisaging that it might be useful in the future to standardize the DILs based performance assessment when this concept is scaled. The vehicle lanes dimensions and number may change for different bridges but there must be a longitudinal midline in every bridge to move the load.

5. [(Axle Number, Axle load (*AL*), Axle distance (*D*) from the first axle),...]: To consider the tandem loads on the bridge the user input in this format is used. For example, in Figure C.4, the first three axles loads and distances are shown along with its input format. An axle number is a number starting from one and is used to identify and name it. The axle distance is given in meters. At axle number one, the distance value must be zero since it is the reference. In the first axle (Axle number, Axle load in Newtons, zero (0) in meter) are specified. The tandem load format for the vehicle in Figure C.4 is [(1, AL1 ,0),(2, AL2, D1),(3, AL3, D1+D2),…]. This can be extended further for multiple axels.

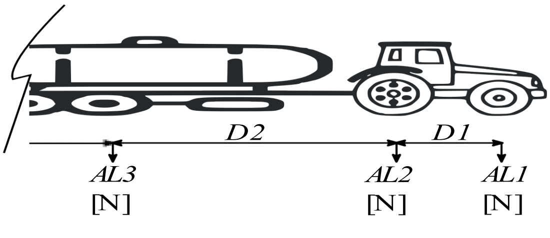


Figure C.4 Vehicle and its tandem load user input format.

C.4 Physics-based model

From the user inputs as listed in C.3, numbers 1, 2, 3 and 4 are used in this subsystem. Based on input number 4, additional inputs are requested and the process follows separate steps to create the DIL or the field DIL output. The FE section steps and their connections with the installation directory are shown in Figure C.5. The detailed process flow is shown in in Figure C.6. Three steps are involved in this section where the user has to trigger each step after completing the previous step.

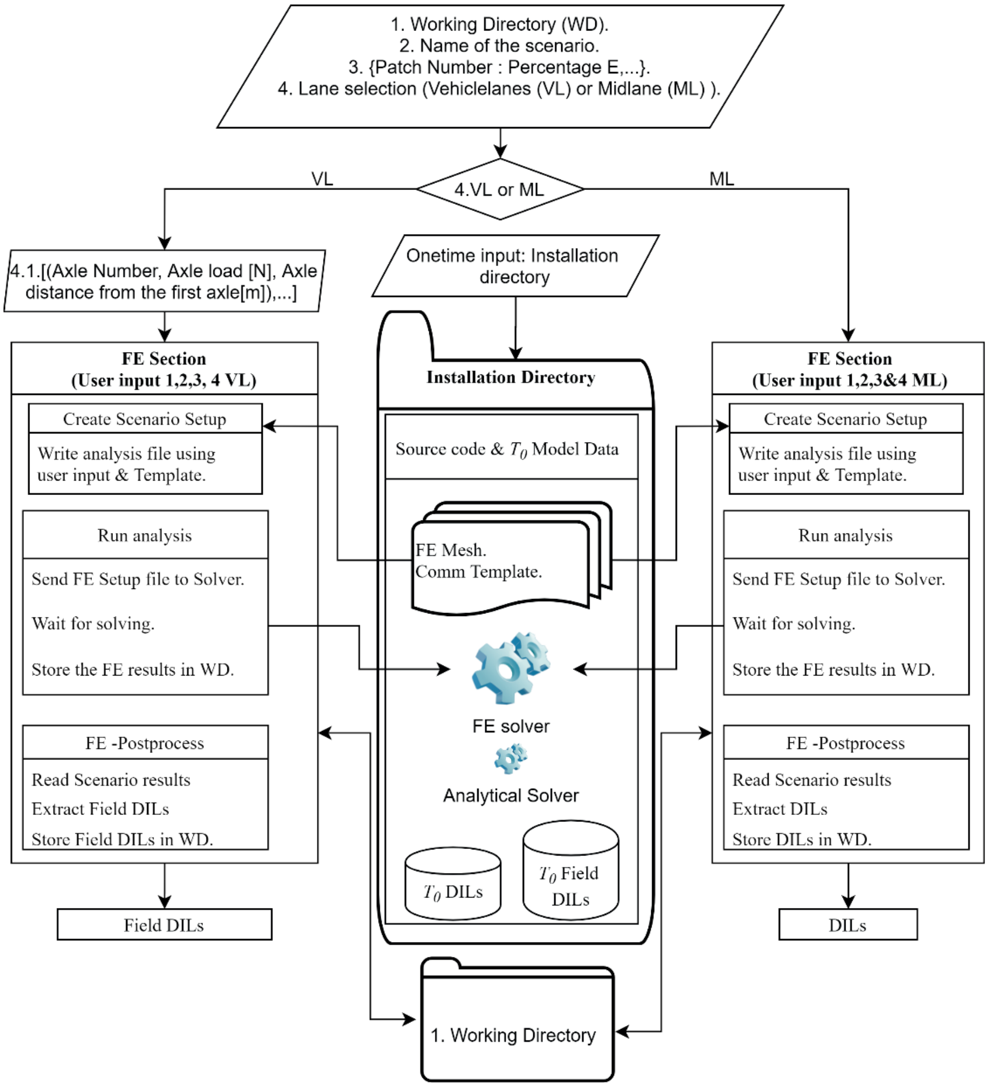


Figure C.5 FE subsystem layout.

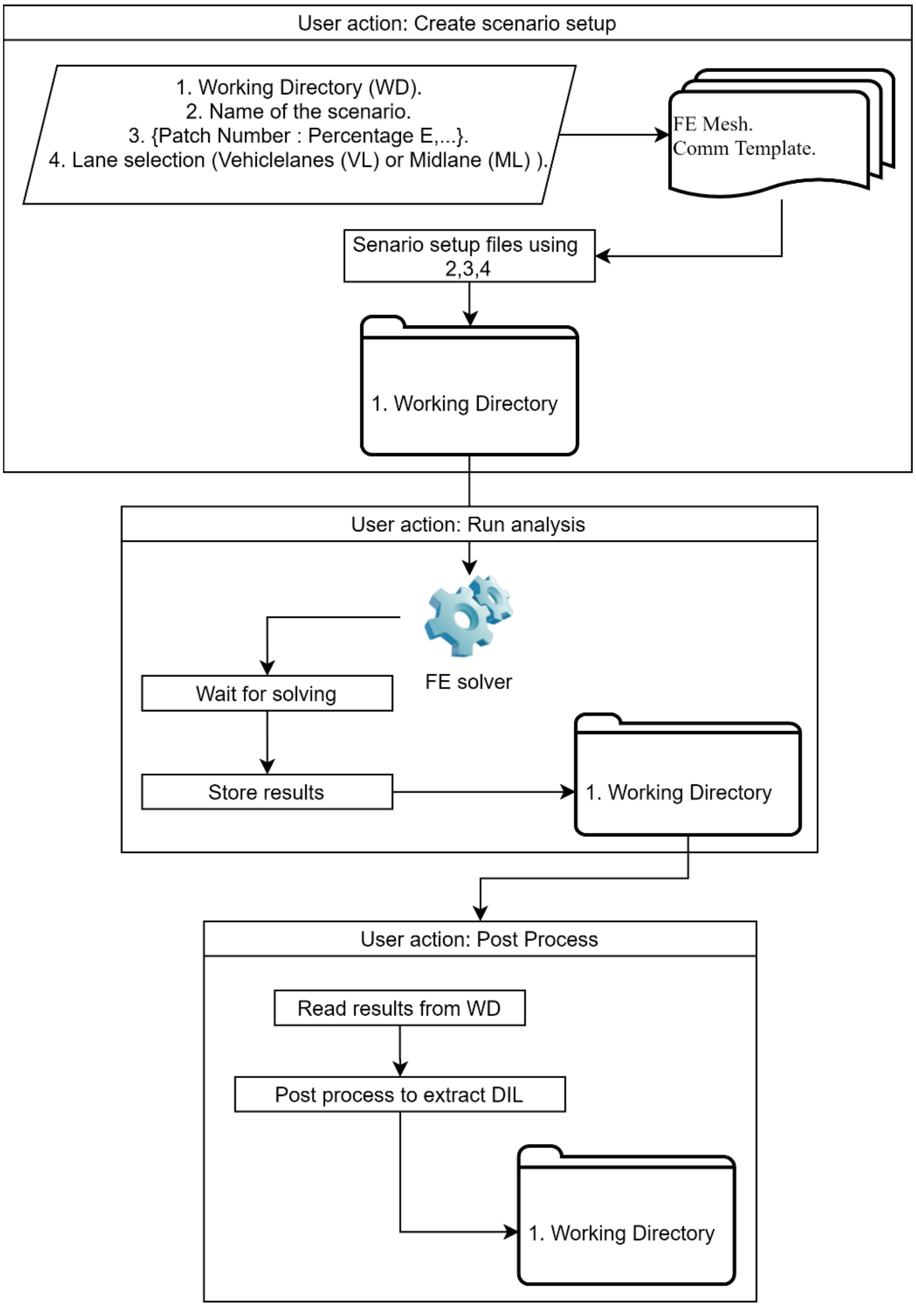


Figure C.6 FE section subsystem detail.

As depicted in Figure C.6, the user initiates the following actions,

- Create scenario - the input set [1,2,3, and 4] or [1,2,3,4 and 4.1] are implemented in the template file and the scenario setup files are stored in the working directory. Input numbers 1 and 2 are used in the export template to define the working directory and file names. Inputs 3 and 4 are used in the comm template to assign the structural properties to the mesh. A notification will be shown to the user once the scenario setup files are created and stored in the working directory.
- Run analysis – The scenarios setup files are sent to the solver once the user initiates the analysis using the GUI. The user has to wait for the FE calculations to be finished. The results will be written in the working directory. A notification will indicate the user to proceed to the next step. The result files have a nodal displacement of all the VDS points for each load steps.
- FE Post-Process – Initiating post-process will read the FE result file and extract DILs or field DILs at every VDS points. The nodal displacements are reassembled to create influence lines, a VDS displacement value at every instance for the applied load is tracked and plotted against the location of the load on the bridge. The extracted data are stored in the working directory. The field DIL or DIL from all the VDS points is the output of these subsystems. The output will be used in the analytical section based on the user input number 4 (vehicle lane or mid lane).

C.5 Analytical section and KPI

DIL or field DIL from the FE section is stored in the working directory. If the mid lane was selected by the user, the T_0 DIL of the reference state and T_d DIL from current scenario are used in the analytical section to calculate the percentage difference. The outputs T_0 DIL, T_d DIL and their percentage difference curves are stored in the working directory for the user. If the vehicle lane is selected, the tandem load analytical section is activated. The field DIL is used to calculate the tandem load DIL using the principle of superposition. The principle of superposition states that on a linear elastic structure, the combined effect of several loads acting on the structure simultaneously is equal to the algebraic sum of the effects of each loads acting individually on the structure [49]. For the same damage scenario, the effect of different load magnitudes and tandem axles on the same lane can be calculated by superposition the DILs in the analytical section. For a VDS point, T_0 tandem field DIL - The tandem load effect at T_0 state is calculated by superposition field T_0 DILs under different axle loads. T_d tandem field DIL - The tandem load effect at T_d state is calculated by superposition field T_d DILs under different axle loads. T_0 and T_d tandem field DILs percentage difference is calculated. The analytical subsystem can be rerun by changing the tandem load parameters. There is no need to run the FE analysis section if there is no change in damage factors. If the damage factors change in a scenario, the tool's FE section is run to extract new DILs. Compared to the FE subsystem calculation time the analytical section calculations will be faster. This approach enables quick computation of different vehicle loading configurations crossing the bridge for a damage scenario.

The output DILs is shown to the user. The DILs are extracted from VDS points of altered patches. The comparisons between the T_0 and T_d state DILs are done in the analytical section and the KPI is the user output.

C.6 Summary

This appendix explained the design layout, the process flow following tool installation and how the user inputs lead to the outputs. Figure C.7 shows the complete layout of the tool.

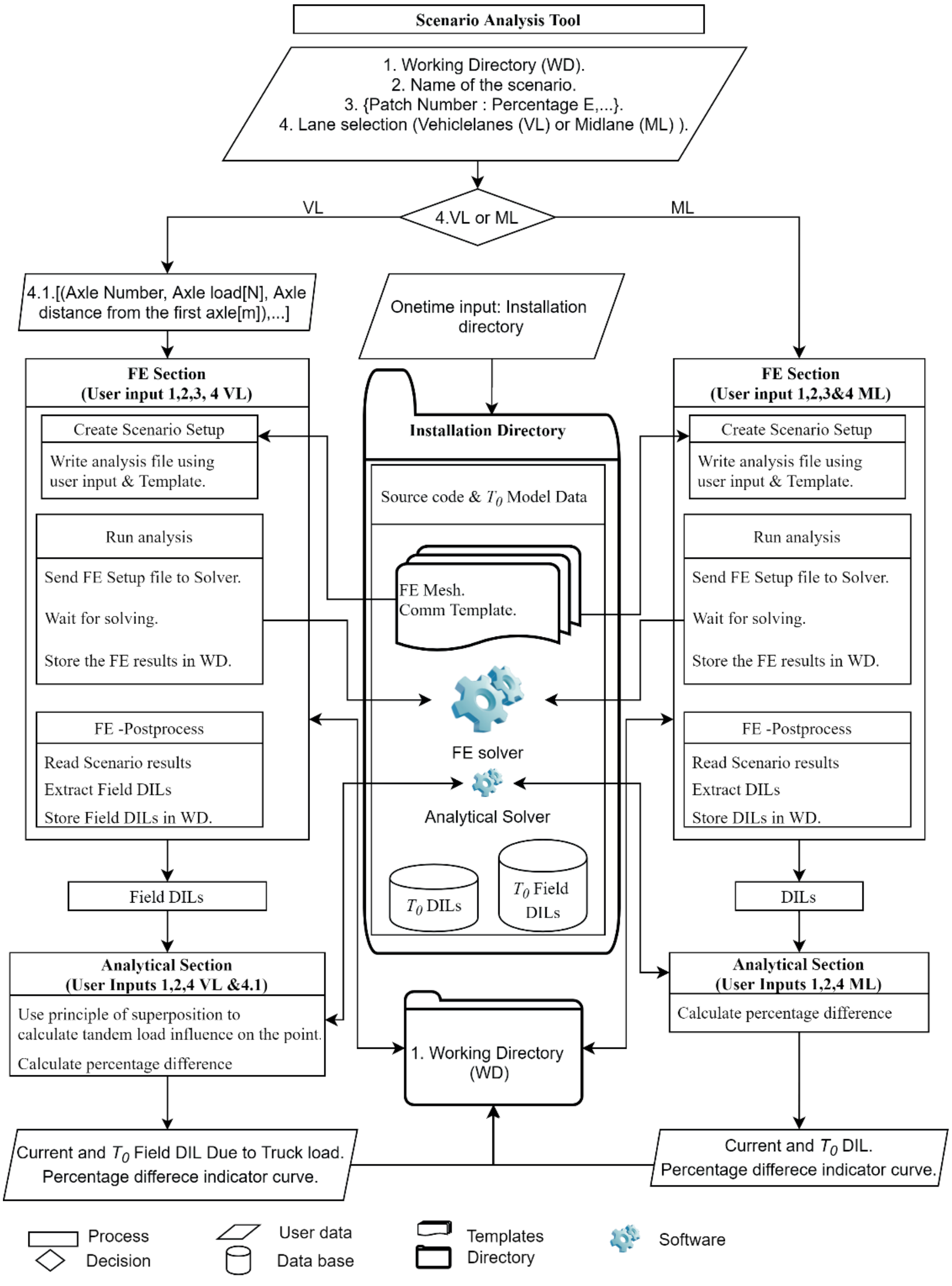


Figure C.7 Scenario analysis tool design layout.

D Opensource tools

D.1 code_aster

The code_aster is an open-source thermo-mechanical solver developed and published by Electricite de France (EDF). The name ASTER stands for “Analyses des Structures et Thermomecanique pour des Etudes et des Recherches”, French for “Structural Analysis and Thermo-mechanics for Studies and Research”. The code_aster has more than 3500 verification test cases covering all the available features in the solver. The code_aster solver is configured by a command file (.comm) as shown in Figure D.1. The code_aster takes the pre-processing data (e.g. CAD, meshes) combined with the data settings of the mechanical problem (e.g. constitutive models, behaviors, material parameters), creates a finite element model and solves it. The results are usually displacement fields, which can be post-processed into other fields of interest (e.g. stress, strain). The code_aster is well supported by EDF. According to the official website, once a year, the stable version is updated and qualified to use in engineering studies. The stable version is the preferred version. The current stable version is 14.4. It is officially available for the Linux/Unix based operating systems. In this project, the Windows operating system based code_aster 14.4 stable version is used, which is ported by a company called SimulEase and released as opensource software.

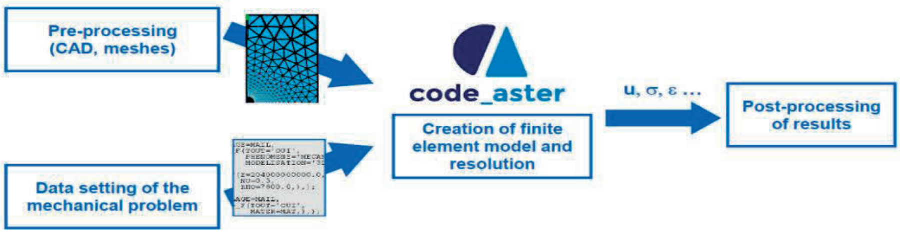


Figure D.1 code_aster general working principle.

D.2 Salome-Meca

Salome is an open-source software that provides generic pre and post-processing modules for numerical simulations. Salome has flexible software architecture and provides a graphical user interface (GUI), see Figure D.2. This is a generic framework where multiple solvers can be combined to perform a numerical study. The code_aster solver with its GUI integrated version of Salome, is called Salome-Meca.

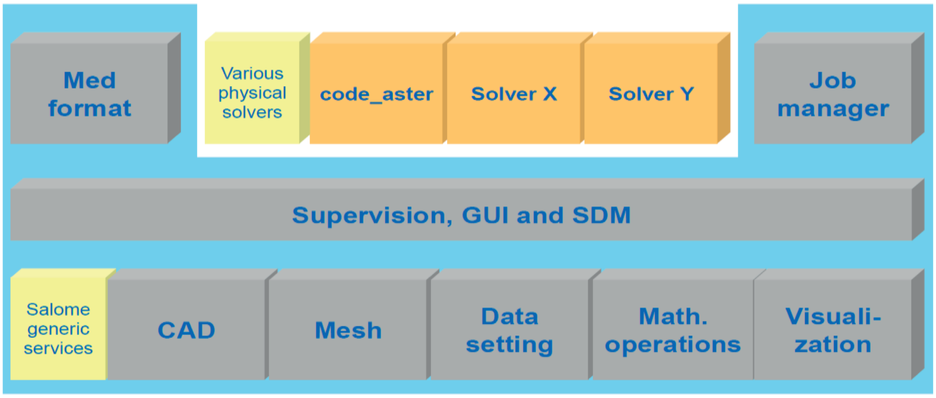


Figure D.2 Salome generic framework for pre and post-processing.

The code_aster solver GUI interface, AsterStudy is one of the modules available in Salome-Meca supported by EDF. Salome-Meca has Geometry, Mesh, Visualization and other modules. These modules ease the process of FE model preparation from geometry modeling to result from visualization in one package, see Figure D.3. It is also possible to choose different tools for each step as in Table D.1.

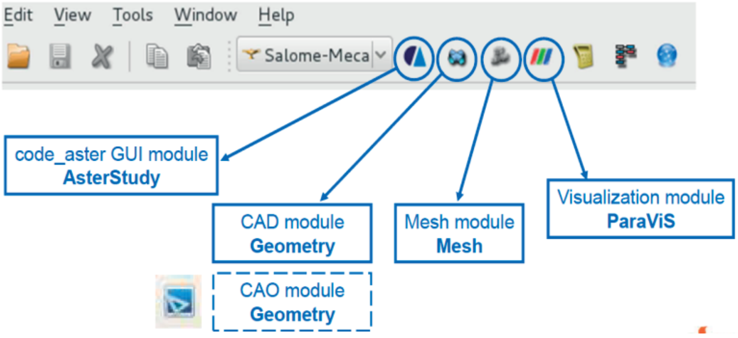


Figure D.3 Salome-Meca modules.

Table D.1 Options to perform steps in FE analysis.

Step	**Salome-Meca**	**Code_aster standalone**
Geometry definition	Salome - Geometry	Any CAD modeller
Mesh generation	Salome - Mesh	Any Mesh tool
Data settings	Salome - AserStudy	Text editor
Result Analysis	Salome - ParaVis	Visualization applications, spreadsheet, etc.

The code_aster can also be used as a standalone solver. This provides an option to customize the pre-post processing steps to match the user's needs. This option is utilized to create a customized user interface. More details, tutorials, documentation etc., can be obtained from official websites. Code_aster - https://www.code-aster.org/ Salome - https://www.salome-platform.org/.

Acknowledgements

First, I would like to thank the individuals, communities and organizations who contribute to the open-source projects.

I would like to thank Prof.dr.ir. Tiedo Tinga and dr.ir. Richard Loendersloot for giving the opportunity to work in this PDEng project. Your guidance and support helped me a lot. I would like to thank dr.ir. Annemieke Meghoe for her support in shaping this report and the ISMA conference paper. I thank Mr. Mark Bosveld, RWS Hengelo, for providing details on RWS maintenance practices.

I would like to convey my gratitude to Debbie Zimmerman Van Woesik, Axel Lok, Neda Mostafa and all my colleagues at the dynamics based maintenance department for their support. I thank Dennis, Thijs, Faizan, Keerthi, Sathyanarayanan, Balan and all my friends for making the last two years memorable with a lot of fun moments.

Last but not least, I would like to thank my wife, Sathya Prabha for her patience and immense support during this work.

தமிழ் வாழ்க!
(Long live Tamil !)

Hemanand Kalyanasundaram,
Dec, 2020.