Significant Guidance to Employ the Software to Analyze and Design the Reinforced Concrete Structures: State-Of-The-Art

Kabir Sadeghi, Nivin Ghaboun

Abstract: ***Prevailing analysis and design of reinforced concrete (RC) structures is a critical stage in the construction industry to deliver the projects within the estimated construction time period and within the budget. The selection of suitable structural engineering software to perform the analysis and design tasks is not an easy matter of fact, especially with the existence of many specialized software in this domain such as Robot Structural Analysis, MIDAS Gen, SAP 2000, STAAD.PRO, Tekla Structural designer, S-Frame and many others. So a strong comparison between the existing software must be made before taking the final decision of selecting any software. The main goal of this paper is to present overall guidance for selection of the suitable software among the most common software used for the analysis and design of RC structures along with the supported design codes, analysis types and design modules incorporated in each software. The technical specifications, characteristics, application domains, incorporated structural design codes, limitation, technical popularity and capabilities of the existing most common used software were studied and compared. Based on the performed study, eleven software were selected and presented as a useful guideline for the structural engineers in the analysis and design of RC structures. In this paper, a complete guideline including the required technical information for structural engineers to choose the suitable software for analyzing and designing of the RC structures is presented.***

KEYWORDS: REINFORCED CONCRETE, SOFTWARE, MIDAS GEN, SAP2000, STAAD.PRO.

I. INTRODUCTION

Reinforced concrete structures are the most common structural type used in many different types of structures such as high-rise buildings, bridges, dams, canals, retaining walls, tunnels, and coastal, onshore and offshore structures. The main reasons behind this widespread using of RC structures in the construction industry are high strength, high durability, fire resistant, more cost-effective than other types of structures such as steel structures, weather resistant, and corrosion resistant. Optimum design of RC structures is a prerequisite for all designers to ensure safety and cost-effectiveness of the design. Despite the existence of different design methods for RC members such as working stress design (WSD) method and ultimate strength design (USD) method, limit state design (LSD) method, the most common method of computational method is the finite element method (FEM). The high competition between software companies to provide structural engineers with a robust tool for their daily work has increased the number of software available in this domain. Each software company tries to provide its customers with unique features to attract more users around the globe. Today some software companies try to specialize in the design of specific elements of the structures such as S-concrete that is used to design columns, beams, and walls, while the others aim to provide one tool to model and design any type of RC structure such as Robot structural analysis, SAP 2000, ETABS, SAFE, and many others. Also, software companies try to incorporate as much design codes in the software as possible to increase the popularity of their software. During the last five years, researchers were interested in developing new models using the artificial neural network (ANN) and fuzzy logic for the design of RC members. Chandan et al. [1] presented a MATLAB neural network model to design RC beams and columns, the data was obtained from the analysis of a four-story building using STAAD PRO software, the model predicts the percentage of steel required in the section based on the applied loads, width, depth, grade of concrete and steel to be used in the section.

Lallotra and Singhal [2] checked the reliability of the results obtained from common software such as STAAD PRO, ETABS and SAP2000 with manual calculations to find the differences between the two methods, the results indicate that the results obtained from software are close to the results obtained from hand calculations except for frames there was a significant difference. Onyebuchi et al. [3] design seven panels of a one-story building using civilsoft quickstructure software to determine the efficiency of the software, the software was used to model the structure, calculate the shear force and bending moment for beams, columns, and slabs and make all the structure detailing drawings for site work. Kumar and Saikiran [4] modeled and analyzed a ten-story RC structure building using SAP2000 to study the effect of the earthquake on different types of foundations considering soil-structure interaction. Srinivasa et al. [5] investigated the effect of blast load using 100 kg of TNT explosion on a seven-story RC structure using STAAD.PRO, the blast load was assumed to occur at 10m, 20 m, 30 m, and 40 m away from the building.

Manuscript Received on July 05, 2019.

Professor **Kabir Sadeghi,** Civil Engineering Department, Near East University, Nicosia, North Cyprus, via Mersin 10, Turkey. Email: kabir.sadeghi@neu.edu.tr.

Ph.D.Candidate **Nivin Ghaboun,** Civil Engineering Department, Near East University, Nicosia, North 10, Cyprus, via Mersin Turkey

The main objective of this paper is to look over the recent software used to design different RC structures along with the main features of the software, design codes included in the package and the capabilities of various software in design.

II. METHODOLOGY

The specifications, characteristics, application domains, covered structural design codes, limitation, technical popularity and capabilities of the existing most common used software were studied and compared. Based on the carried out study, eleven software were selected and a complete guideline including the required technical information for structural engineers to choose the suitable software for analyzing and designing of the RC structures is presented. The result of this study can be a useful guideline for the structural engineers in the analysis and design of RC structures.

In this study, for the selection of suitable software, the following five aspects were considered as main criteria:

- The ability to model RC structure of any shape and size easily,
- Flexibility in editing the created model,
- Design codes incorporated in the software,
- Types of analysis supported by the software.
- Design modules built in the software to ease the design process.

Based on the above-mentioned criteria, a complete guideline for choosing the software to analyze and design the RC structures is presented.

III. RC STRUCTURES DESIGN SOFTWARE

Traditionally, RC structures were designed manually which consume a lot of time and may be subjected to many errors. In addition, changing input data during design means repeating the design process from the beginning. Today, the robust design of the RC structure is a big challenge in the construction industry considering time, cost and safety. In this Section, the most common and new developed software for the design of RC structures are illustrated.

A. Autodesk Robot Structural Analysis

Autodesk Company [6] provides this software for the analysis and design of RC, steel and wood structures. The company is keen to improve the capabilities of the software year after year to provide their users with the best tool in design. The main features of the software are:

- Supporting multilingual design options.
- Different types of analysis including linear, non-linear, dynamic and FEM.
- Providing 30 types of design codes for RC structures and 40 design codes for steel structures.
- Supporting metric and imperial units.
- Providing 60 databases of sections and materials.
- Flexibility in displaying design results either on specific members of the structure or on the structure as a whole, also the results can be displayed in tabular form.
- The ability to analyze complex models using a powerful mesh generation.
- Wind load simulation.
- Dynamic analysis capabilities.
- Integration between Autodesk robot structural analysis and other software such as Microsoft office word and excel using an open application program interface (API).
- Three-dimensional visualization of the structure model.
- Supporting building information modeling (BIM) tool.
- Taking into consideration of users' opinions.

A.1. Design Codes

Robot structural analysis includes many codes for RC structures design but not limited to the followings [7]:

- American codes: ACI 318-08 and 11,
- French codes: BAEL 91 and modified 99,
- British code: BS 8110,
- Canadian codes: CSA 23.3-04 and 94,
- Japanese code: AIJ 1985,
- Australian code: AS 3600-2009,
- Italian codes: D.M. 2008,
- Indian code: IS 456-2000,
- Spanish code: EH 91 and EHE 99,
- Chinese code: GB 50010-2002,
- Eurocode 2: EN 1992 & 2004.

A.2. Analysis and Design Modules

Robot structural analysis is equipped with different types of analysis including but not limited to [8]:

- Linear and nonlinear static analysis,
- P-Delta analysis,
- Buckling deformation analysis,
- Elasto-plastic analysis,
- Dynamic analysis,
- Time-history analysis,
- Seismic analysis,
- Harmonic analysis,
- Spectral analysis,
- Pushover analysis,
- Model analysis.

Robot structural analysis includes many modules for the design of RC structures, in this Section, a brief description of these modules are presented:

- Building Design:

It's a template to define the shape, supports and loads for any RC structure.

- Structures Types:

This module can be used to design shells, 2D frames, 3D frames, and plates.

-Plane Deformation and axis-symmetric structures:

In-plane deformation module only 1 meter in the transverse direction is assumed and designed regardless the length of the structure, while the axis-symmetric module allows for the definition of the cross section of the structure instead of modeling the whole structure.

- Code module design:

This module allows for the design of RC members such as beams, columns, slabs, walls, and foundations.

B. Midas Design and Gen Software

MIDAS [9] package provides solutions to many engineering disciplines such as construction, industrial, mechanical and geotechnical. The complete set includes:
- Midas Civil is used for the design of bridges and culverts,
- Midas CIM is used for the modeling and management of bridge information,
- Midas FEA is used to perform finite element analysis (FEA) of complex structures,
- Midas Gen is used for the design of general purpose buildings,
- Midas Design is used for the design of all RC members,
- Midas NFX is used to perform FEA in mechanical engineering,
- Midas GTS NX is used for geotechnical analysis and soil engineering,
- Midas soilWorks is used for soil analysis,
- Midas DShop is used to provide workshop drawings and bill of quantities.

The software has many features which make it gain a superior position in the global market, these features include:
- A friendly-user interface to facilitate data input and design process.
- Providing accurate design results with AutoCAD drawings and quantity take-off calculations.
- Integration with Midas Gen software.
- Flexibility in changing input data to perform a parametric study and obtain the optimum design.
- Displaying design calculations in tabular and graphical forms.
- All types of loads can be identified including nodal, point, element, surface, dynamic, seismic, hydrostatic, wind and thermal loads.

B.1. Design Codes

The codes included in Midas Design software are [10]:

- ACI 318-08 and 11,
- Eurocode 2: EN 2004,

While the codes included in Midas Gen software are [10]:
- ACI 318-08 and 11,
- Eurocode 2: EN 2004,
- CSA 23.3,
- BS 8110,
- GB 50010,
- AIJ-WSD,
- Taiwan code: TWN-US,
- Kenyan code: KSCE,
- South Korea code: KCI-USD,
- AIK-USD, WSD,
- IS 456 and IS 13920.

B.2. Analysis and Design Modules

Midas Design can be used to design steel or RC individual structural members such as:

- Beams,
- Regular and irregular columns,
- Slabs,
- Staircases,
- Shear, combined, basement and retaining walls,
- Mat, pile and wall Foundations,
- Corbel and Buttress.

On the other hand, Midas Gen software is used to design any type of structures including but not limited
To the followings [11]:
- High rise building and towers,
- Power plant facilities,
- Schools and hospitals,
- Resort facilities,
- Airports,
- Stadiums,
- Subway stations,
- Museums.
- Marine terminal:

Different types of analysis are incorporated in MidasGen software to provide the users with a multi-purpose tool:
- Static linear and thermal analysis,
- Dynamic analysis,
- P-Delta analysis,
- Buckling analysis,
- Heat transfer analysis,
- Geometric nonlinear analysis,
- Pushover analysis,
- Boundary nonlinear analysis,
- Material nonlinear analysis,
- FEA,
- Seismic analysis,
- Material nonlinear analysis,
- Seismic analysis,
- Material nonlinear analysis.

C. SCIA Engineer Software

This software was developed by a Belgian engineering software company called SCIA. The continuous development of the software during the last few years attracts the attention of many users around the globe to use it. The software can be used to design office buildings, power plants, bridges, stadiums or any other structure type. The substantial merits of the software are [12]:
- Robust 3D modeling and visualization tools.
- Flexibility in editing model parameters to find the optimum and economical solution.
- Automatic generation of calculation sheets and drawings after any change in the model.
- Different types of loading and loading combinations.
- - Reliability in handling complex and large scale projects.
- The software can work with Autodesk Revit Structure and Autodesk Civil 3D.

C.1. Design Codes

The software provides many international codes for the design of RC structures such as [13]:
- ACI 318-2008 and 2011,
- International Building Code (IBC),
- Eurocode: EN 1990, 1991, 1992, 1993 and EN1168 for slab design,
-

- Brazilian code: NBR 6118.

In addition to the codes required to design steel, aluminum, timber structures, as well as scaffolding, and seismicity.

C.2. Analysis and Design Modules

The software contains multiple analysis options to encourage users to depend mainly on the software instead of using much software to perform a certain step in the design process. Analysis choices include [14]:

- Linear and nonlinear analysis,
- Stability analysis,
- Geometrical nonlinear analysis,
- Material nonlinear analysis,
- Dynamic analysis,
- Earthquake and seismic analysis,
- Soil interaction analysis.

RC structures' design modules in the software include the followings:

- Long term deflection of RC members,
- Fire resistance of RC members,
- Prestressed RC design,
- Composite floors,
- Foundation design,
- Bridge design.

Other modules included in SCIA Engineer software to design steel members, steel connections, timber members, aluminum members, composite columns and scaffolding.

D.CSI Software Packages

The most well-known structural and earthquake engineering software company during the last 20 years is a Canadian information technology company founded 1995 called Constellation Software Incorporation (CSI) [15] that provides much software to help engineers to speed up the analysis and design of RC, steel, aluminum, and composite structures. The company provides a variety of software for different uses such as:

* CSIBRIDGE:

It's used for modeling, analysis and design of bridge structures.

* SAP 2000:

The software is used to analyze and design the structures of transportation, industrial, sports, and other facilities.

* ETABS:

Designers use ETABS to analyze and design the buildings of different materials including towers, commercial buildings, parking garages, and healthcare facilities.

* SAFE:

The best tool to design RC foundation and floors including two-way slabs, waffle, and ribbed slabs.

* CSICOL:

The software is used to analyze and design the RC, and composite cross-section columns.

* CSIPLANTl:

It's devoted to the analysis and design of piping systems along with the framing structures required for the pipes.

* PERFORM 3D:

A displacement-based design which considers the inelastic behavior of the structure due to strong earthquake can be done using this software.

The most common software packages used by many users around the world are SAP 2000, ETABS and SAFE. The common advantages of the three-dimensional software are:

- Single-user interface for modeling, analysis, and design.
- Wide range of templates to build a new model.
- Simplicity in editing model database.
- Wide range of design codes for different materials.
- Specific tool for load optimization.
- Section designer tool to allow for creating a section of any shape.
- Powerful export and import capabilities by supporting different file formats.
- Different types of analysis are included.

D.1. Design Codes

The most advantageous feature in SAP 2000, ETABS and SAFE are the inclusion of different types of design codes, some of these codes are [16]:

- ACI 318-2008, 20011 and 2014,
- AS 3600-2001 and 2009,
- BS 8110-97,
- Eurocode 2-2004,
- CSA-A23.3-2004 and 2014,
- GB 2010,
- IS 456-2000,
- Hong Kong CoP 2004 and 2013,
- TS 500-2000,
- RCDF 2004,
- SP 63.13330.2012.

D.2. Analysis Modules

The three software contain different types of analysis to satisfy customers' needs, this Section is devoted to illustrating the function of each analysis type [17]:

- Static Analysis:

The module performs linear static and multi-step static analysis.

In the multi-step analysis, the load is applied in a sequential spatial pattern based on the location on the structure.

- Dynamic Analysis:

It is used to calculate vibration modes using Ritz or Eigenvectors, performing time-history analysis and response spectrum analysis.

- Buckling Analysis:

For any set of loads, the buckling mode of a structure can be found using nonlinear or staged construction analysis.

- P-Delta Analysis:

It is also referred to as nonlinear geometric analysis because as the deflection increases the additional forces cause this deflection must be studied using P-Delta effects.

- Pushover Analysis:

In this analysis the application of the FEMA 356 which is an American pre-standard and commentary for the seismic remedy of the buildings, this allows considering the plastic behavior of shear walls, slabs and other members in the pushover analysis.

- Staged Construction:

It's a nonlinear analysis where different stages can be defined and in these stages, different types of loads can be applied on portions of the structure also material behavior due to aging, creep and shrinkage can be studied.

- Power Spectral Density:

This analysis is used to determine the response of the structure due to cyclic loading with different frequencies such as the fatigue and earthquake analysis.

- Steady State Analysis:

The effects of multiple working machines with different frequencies on the structure are studied using this module.

E. STAAD.PRO Software

This software is provided from an American software development company called Bentley to help structural engineers analyze and design infrastructure projects including high-rise buildings, skyscraper, airports, bridges, power plants, roadways, and many others. The company provides the software in three versions [18]:

- **STAAD.PRO**: This version contains the most basic options of the software with the FEA method.
- **STAAD.PRO Advanced**: The version is devoted to solve the most complex problems with high speed than the previous version.
- **Structural Enterprise**: This software is used to analyze, design and produce calculation sheets for wastewater structures, tunnels, culverts, bridges, steel buildings, trusses, and metro stations.

The main advantages of STAAD.PRO software is:

- Wide range of material libraries to be used in the design.
- The models can be viewed and edited from mobile devices.
- Over 90 design codes are included in the software for RC, steel, timber, aluminum, and cold-formed steel structures.
- STAAD's cloud services enable users to try different design alternatives for comparisons.
- Use of CAD and DXF file format to build the structural model.
- Use of API protocol to extract data from STAAD.PRO models into other software such as Microsoft word and excel.

E.1. Design codes

STAAD.PRO supports many international codes for RC structures' design [19]:

- ACI 318-95,
- BS 8110,
- AS3600,
- CAN3-A23.3-M84,
- IS 456,
- EC2,
- German code: DIN 1045,
- BAEL,
- Japanese code: AIJ,
- GBJ10-89,
- Norwegian code: NS 3473,
- Finnish code: B4,
- Russian code: SNiP 2.03.01-84,
- Swedish code: BBK,
- Dutch code: DS 412.

E.2. Analysis and Design Modules

Analysis types built in STAAD.PRO are [20]:

- P-Delta analysis for small and large P-Delta,
- Elastic linear analysis,
- Imperfection analysis due to inaccuracy in modeling structure geometry,
- Dynamic analysis such as time-history, response spectrum, steady state, and Eigen analysis,
- Design modules for RC structures in STAAD.PRO includes:
- Pushover analysis,
- Geometric nonlinear analysis,
- Seismic analysis,
- Buckling analysis.

Design modules for RC structures in STAAD.PRO includes:

- STAAD.Foundation which is used to design any type of foundation including mat foundation, isolated footing, combined footing, and pile foundation.
- The RC structures design which is used to design:
- Short and long columns.
- Beams including rectangular, T-section, L-beams and beams with irregular cross-section.
- Slabs such as one-way, two-way, waffle, ribbed, hollow block slabs and many others.
- Frames and shear walls.

F. FEM-Design Software

A Swedish software company called STRUSOFT provides the software to facilitate the analysis and design process for engineers. The company has over 30 years of experience in software development. The company provides five applications [21]:

- FEM-Design:

It's FEA software for the analysis and design of RC, steel and timber structures based on the Eurocode.

- WIN-Static:

The software is used for normal design tasks such as columns, beams, foundation, shear walls, calculation of punching shear and settlement, in addition to masonry structures design.

- PRE-Stress:

It's a 2D FEM software for the analysis and design of prestressed RC members.

- IMPACT:

It's software for the design, management, transport, and construction of precast RC structures.

- VIP-Energy.

The energy consumed by a building during one year can be calculated and managed using this software.

FEM-Design is the most common software among the 5 packages for RC structures; the increasing numbers of users around the globe are due to many reasons:

- CAD rich environment for modeling and editing structure members easily.
- Different analysis types are incorporated into the software.
- Automatic generation of optimized FE mesh for any model.
- Graphical presentation of results either as contour lines, sections or palettes.
- Automatic generation of calculation sheets and reports using a documentation module built in the software.
- Auto-design module facilitates choosing the optimum cross-section with a suitable reinforcement arrangement.

F.1. Design Codes

The software supports Eurocode EC3 [22].

F.2. Analysis and Design Modules

Types of analysis included in the software are [23]:
- Soil interaction analysis for soil retaining structures.
- Cracking analysis
- Linear static analysis of any structure type.
- Plastic analysis of supports and connections.
- Nonlinear static analysis of supports.
- Stability analysis.
- Dynamic analysis including vibrations and seismic analysis as well as response spectra analysis.
- Buckling analysis.

Different design modules are incorporated in the software to satisfy users' needs to depend on single software for many purposes, these modules are:
- FEM-Design 3D Structure:

It's used to design any complex structure of any material or a combination of many materials.
- FEM-Design Plate:

This module is used to analyze and design any two-dimensional loaded members such as one-way, two-way, waffle, ribbed, hollow block slabs and any other type of slabs.
- FEM-Design Wall:

This module is used to design shear walls, basement walls, retaining walls and any other type of RC walls.

G. RISA 3D Software

A well-known software in the United States, developed by an American software company is called RISA. The popularity of the software in the US is due to many reasons [24]:
- User-friendly interface with powerful drawing and selection tools.
- Different materials are incorporated into the software.
- Wide range of international codes is added to the software.
- The software can be used to design any structure type including bridges, culverts, towers, stadiums, tanks, crane rails, arenas, and others. It can be applied to the design of onshore and offshore structures' elements such as vertical breakwaters [25] and the RC gravity platforms [26].
- Providing support for all users by professional expertise.
- Full integration between all RISA software packages.

G.2. Design Codes:

A wide range of codes built in the software for steel, cold-formed steel, aluminum, wood, masonry, and RC structures increases the numbers of users in the whole world, the codes for RC structures are [27]:
- ACI 318-2014, 11, 08, 05, 02 and 99,
- CSA A23.3-2014, 04 and 94,
- BS 8110-2001and 97,
- BS EN 1992, 2004 and 2014,
- EN 1992,
- IS 456-2001,
- AS 3600-2001,
- New Zeeland Standard: NZS 3101-1995,
- Saudi Building Code: SBC 304-2007,
- Mexicans Standards: NTC-DF 2004.

G.2. Analysis and Design Modules

RISA 3D can be used to conduct the followings [27]:
- 1D analysis for beams, columns,
- 2D analysis for walls and slabs,
- 3D analysis for solids,
- Dynamic analysis such as seismic analysis and vibration analysis,
- Time-history analysis,
- Response spectra analysis.

RISA 3D can be integrated with many design tools to provide complete engineering software for any structure. Design tools provided by the company include [27]:
- RISAFloor:

It's a tool to design the floor system of any material and export all design data to RISA 3D to complete the design of the model.
- RISAFoundation:

All types of foundation can be designed using this software including but not limited to mat foundation, isolated footing, combined footing, pile caps, and grade beams.
- RISA 2D:

This software is used to model and analyze 2D objects including columns, beams, shear walls, and many others.
- RISASection:

It's used to create a regular and irregular cross-section of any material and calculate all section properties such as radius of gyration, section modulus, the moment of inertia, shear strength and torsion constant. The data provided from the software can be used in RISA 3D, RISAFloor and RISA 2D to complete the model design successfully.

H. Tekla Structural Designer Software

The software was presented by a Finnish software company headed in Finland in 1966. In 2012 the software was acquired by an American business technology company called Trimble navigation. The software has many advantages such as [28]:
- One software to perform all design tasks to reduce time switching from one package to another.
- Easily creating and editing models using powerful drawing and modifying tools.
- Automatic calculation and application of wind loads on the model.
- Supporting a wide range of export options for submission.

- Providing detailed documentation including materials take-off, reinforcement drawings, and calculation reports.
- Full integration between Tekla structural designer and Tekla structures which is used to model any type of structure whether its RC, timber or steel.
- Full integration with Autodesk Revit Structures and Autodesk Civil 3D.

H.1. Design Codes

Tekla structural designer software incorporates many design codes for RC, steel, and timber structures.

The supported codes for RC structures design including loading codes are [29]:
- ACI 318-2008, 2011 and 2014,
- EC2: BS EN 1992 and 2004,
- BS 8110,
- Australian code for loading AS/NZS 1170.0 and AS 1170.1,
- IS 456-2001,
- IBC 2000, 2003 and 2006,
- AIJ,
- Uniform building code UBC 1994 and 1997.

H.2. Analysis and Design modules

Tekla structural designer software is equipped with different types of analysis such as [30]:

* 1st order analysis:
The module performs linear static analysis for any structural element.
* P-Delta:
Second order analysis is done using this module; this type of analysis is preferred for small deflection values.
* Non-linear analysis:
It's used for nonlinear analysis due to large deformations or due to elastic-plastic behavior of the material.
* Response Spectrum analysis:
The maximum displacements, forces, and stresses due to seismic effects are calculated and used in the design process of structures.

After modeling the structure using Tekla structures, it's imported to Tekla structural designer. The software can be used to design RC, steel and wood structures, the followings are the design modules for RC members:
- Beam design:
Design regular and irregular cross-section of beams.
- Column design:
For the design of short and long columns.
- Slab design:
To design one-way, two-way, hollow block, ribbed slab, and other types.
- Wall design:
To design retaining walls, basement walls, cantilever walls.
- Foundation design:
Design of strip and pad footing.
- Pile cap design,
- Seismic design.
-Design of structures due to vibrations from earthquakes and seismic effects.

I. ASDIP Structural Software

This software is released by an American structural engineering software company called ASDIP. Over two decades the software has been improved by specialists of structural engineers to help designers to perform their work more effectively and in less time. The software becomes more popular between structural engineers because of many merits such as:
- Detailed calculation sheets and reports with code references.
- Optimized design of RC members based on design provisions.
- Providing step-by-step support for users around the world.
- Modeling structural members easily and shortly.
- The beam design manager is a new tool provided to perform the reinforcement design for the beam.
- Creating and saving load combinations for future use in other projects.
- Adding a new option for the design of masonry bearing walls.

The company provides three packages for RC structures design option as mentioned below [31]:
- ASDIP Concrete,
- ASDIP Foundation,
- ASDIP Retain.

I.1. Design Codes

The software supports international codes for the design of RC, steel, cold-formed steel, and masonry members. Design codes of RC structures are [31]:
- ACI 318-2011 and 2014.
- Building code requirements and specifications for masonry structures: MSJC-2013.
- International building code (IBC).

I.2. Design Modules:

Various software of ASDIP packages consists of many modules, each one designs a specific element based on selected design code. In this Section a brief description of each module is presented based on software type [31]:

ASDIP RC Modules:
- RC Column Design:
The module is used to design rectangular and circular columns based on the slenderness ratio and generates the biaxial interaction diagram.
- Continuous Beam Design:
All cross section types of beams can be designed using this module including rectangular, T, inverted-T, spandrel and L-section beam. The module generates shear force and bending moment diagrams based on load combinations.
- Wall Design:
RC walls subjected to axial loads and weak bending moments are designed based on load combinations and slenderness ratio. The interaction diagram is drawn based on reinforcement and geometry selected.

ASDIP Foundation Modules:
- Spread Footing Design:
The footings are designed considering stability requirements against overturning, sliding and uplift pressures.

The module selects concrete dimensions based on optimized load combinations and after performing the one-way shear check, bending moment check and punching shear check.

- Strap Footing Design:

The module designs the two footings and the strap beam based on applied loads and performs the stability check requirements.

- Combined Footing Design:

The shear force and bending moment diagrams are drawn based on assigned loads then the cross-section dimensions are selected after checking the one-way, two-way shear, over-turning, sliding and uplift pressures.

ASDIP Retain Modules:

- Cantilever Retaining Wall:

The module checks the stability of the wall under service loads against sliding, overturning and bearing then the cross section dimensions of the wall components including toe, key, stem, and heel are selected.

- Basement Retaining Wall:

It's a special type of retaining walls because its restrained from the top so there is no possibility of overturning, the module checks for the sliding and settlement then the dimensions are selected for the wall based on applied loads.

- Counterfort Retaining Wall:

In this type of retaining wall, the beam acts as a cantilever beam while the stem and heel act like continuous slabs extending between the counterforts. The module firstly checks for sliding and settlement based on service loads then the dimensions are selected based on factored loads for different wall parts including stem, heel, toe, key, and counterforts.

J. S-Frame Software

The software is presented by a Canadian engineering software company called Softeck services to help civil engineers to analyze and design any type of structure within the estimated time and budget. The main merits of the software are [32]:

- Multiple views to switch between structure modeling, loading, results and tabular formats.
- Built-in templates to facilitate the creation of the model in a few minutes.
- Importing model data from Revit structure, STAAD, ETABS, Tekla Structures.
- Supporting different file formats such as Dxf, Xlsx and access files.
- Automatic generation of FE meshes for any complex structure.
- Powerful solver capabilities to analyze models of unlimited size.
- Extensive library of FEM to model any part of the structure accurately.
- Supporting torsion design for all closed and open sections.

The company has three versions of the soft:

- Standard:

This version provides a linear static analysis.

- Professional:

It's used for non-linear structural analysis.

- Enterprise:

The software devoted to making more complex nonlinear analysis such as time-history analysis, nonlinear torsional analysis, nonlinear seismic analysis, and many others.

In addition to the S-Frame software, the company has other software for specific purposes which include:

- S-CONCRETE: design of beams, columns, slabs, and walls,
- S-FOUNDATION: analysis and design of RC foundations,
- S-STEEL: for the design of 3D steel structures,
- S-PAD: design optimization of steel structures,
- S-CALC: to calculate the cross-sectional properties,
- S-TIMBER: analyze and design of wood structures,
- S-LINE: detailing of continuous beams,
- S-VIEW: model validation tool.

J.1 Design codes

This software includes different types of codes for the design of wood, steel and RC structures, the main codes used for design are [33]:

- ACI 318- 2002, 2005, 2008, 2011 and 2014,
- CSA A23.3-2004 and 2014,
- BS 8110- 1997,
- EN 1992-1-1:2004,
- Singapore Standards: CP 65-1999.

J.2. Analysis and Design Modules

S-Frame uses the FEM to perform the following types of analysis [33]:

- Linear and nonlinear static analysis,
- P-Delta analysis,
- Vibration analysis,
- Response spectrum analysis,
- Linear and nonlinear time-history analysis,
- Buckling analysis,
- Stage construction analysis,
- Moving loads analysis,
- Linear and nonlinear torsional analysis.

S-CONCRETE software contains many modules for RC structures design:

- Wall Design:

This module is used to design and preparing reinforcement drawings for rectangular, T-shaped, L-shaped, C-shaped and I-shaped sections of RC walls.

- Column Design:

It's devoted to the design and reinforcement details for rectangular and circular columns with and without holes.

- Beam Design:

Rectangular, T-shaped and L-shaped cross-section beams can be designed using this module along with preparing reinforcement details drawings.

S-Foundation software is integrated with S-frame and S-Concrete software to make the design of any foundation type including:

- Strap Footing,
- Spread Footing,
- Combined Footing,
- Continuous Footing,
- Raft Foundation,
- Wall Footing,
- Pile Foundation.

K. QuickStructure Software

The software is presented by a British development software company called QuickCivilSeries. The software works in the same way the engineers are working at an office, that is why it's gaining the attention of many engineers around the world. The unique features of the software are [34]:
- Automatic loading of the structure or user-defined loading system.
- Automatic design of any structure shape with a step-by-step guide.
- Model saving options either in PDF or DXF format.
- A complete generation of detailed drawings for the whole structure.
- Providing online training courses for new users.
- Automatic generation of the bill of quantities.
- Customers support at any spot of the world.
- Incorporating users' opinions in new versions of the software.

The company presents two types of software:
- Quick Structure:

The software uses simple drawing tools to create the model and can be viewed either as a 2D or 3D model. The software performs the analysis and design after applying the required loads.
- Quick Steel:

The software devoted to the analysis and design of steel structures.

K.1. Design Codes

The software is based on the British standards BS 8110-1997 for the design of RC structures and BS 5950-2000 for steel design.

K.2. Design Modules

Quick Structure is used to design all RC members such as:
- Beams: to design rectangular and T-section beams.
- Columns: to design circular and rectangular cross-sections.
- Slabs: one-way and two-way slabs.
- Foundations: including isolated footing and raft foundation.

IV. *CONCLUSIONS*

Based on the comparisons carried out on the existing most common used software, eleven software have been selected and technically described by giving the technical specifications, characteristics, application domains, incorporated structural design codes, limitation, technical popularity and capabilities that can be a complete guideline for structural engineers to choose the suitable software for analyzing and designing of the RC structures..

In general, five main aspects should be considered in the selection process of a software:
- The ability to model RC structure of any shape and size easily,
- Flexibility in editing the created model,
- Significant design codes incorporated in the software,
- Types of analysis supported by the software.
- Design modules built in the software to ease the design process.

In this paper, a complete guideline for choosing the software to analyze and design the RC structures is presented.

REFERENCES:

1. Chandan et al. (2017). Design of reinforced concrete structures using neural networks. *International Research Journal of Engineering and Technology (IRJET)*, vol. 4, issue 07, pp 2012-2018.
2. Lallotra B. and Singhal B. (2017). A compatibility study on design and analysis of reinforced concrete structural design using the software. *International Journal of Engineering and Technology (IJET)*, vol. 9, No.2, pp.1012-1021.
3. Onyebuchi M. et al. (2018). Determination of the efficiency of civilsoft quick series in the design of reinforced cement concrete: a case study of the design of a seven-panel single story building. International Research Journal of Engineering and Technology (IRJET), vol. 05, issue 06, pp. 680-688.
4. Kumar V. P. and K. Saikiran (2018). Non-Linear Dynamic Analysis of Multistoried Reinforced Concrete Building by Considering Soil-Structure Interaction (SSI). International Journal of Innovative Technology and Exploring Engineering, vol. 7, issue 7, pp 5-9.
5. Srinivasa P. R. et al. (2019). Study the Impact of Blast Load on G+7 Multistoried RCC Structure with Varied Distances. International Journal of Innovative Technology and Exploring Engineering, vol. 8, issue 4, pp 615-621.
6. Autodesk Incorporation (2019). Autodesk Robot Structural Analysis Professional datasheet.
7. Autodesk Incorporation (2019). Retrieved from: https://knowledge.autodesk.com/support/robot-structural-analysis-products/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/RSAPRO-UsersGuide/files/GUID-07532EC8-6712-4642-9C60-7A43EB2993F7-htm.html.
8. Autodesk Incorporation (2019). Retrieved from: https://knowledge.autodesk.com/support/robot-structural-analysisproducts/learnexplore/caas/CloudHelp/cloudhelp/2014/ENU/Robot/files/GUID-FA0255C2-587C-452B-B99C-ED44ED867AD0-htm.html.
9. MIDAS Engineering Software Incorporation, (2019). Retrieved from:https://en.midasuser.com/.
10. MIDAS Engineering Software Incorporation, (2015). Midas Gen: an integrated design system for buildings and general structures brochure.
11. MIDAS Engineering Software Incorporation, (2019). Retrieved from: https://www.midasoft.com/building/design/products/midasdesign.
12. SCIA: A NEMETSCHEK Company, (2018). SCIA Engineer Brochure of Structural analysis and design software.
13. SCIA: A NEMETSCHEK Company, (2019). Retrieved from: https://resources.scia.net/en/articles/codes/7.design_code_topic.htm#
14. SCIA: A NEMETSCHEK Company, (2019). Retrieved from: https://www.scia.net/en/software/scia-engineer/editions.
15. Computers and Structures Incorporation (CSI), (2019). Retrieved from: https://www.csiamerica.com/products.
16. 16. Computers and Structures Incorporation (CSI), (2019). Retrieved from:https://www.csiamerica.com/international-design#concrete-frame-design .
17. Computers and Structures Incorporation (CSI), (2019). Retrieved from: https://www.csiamerica.com/products/etabs/features#analysis.
18. Bentley Systems Incorporated (2019). Retrieved from: https://www.bentley.com/en/products/product-line/structural-analysis-software/staadpro.
19. Research Engineers (REI), (2019). Retrieved from:https://www.reig.de/en/our-software-solutions-at-a-glance/staadpro/international-design-codes/.
20. Bentley Systems, Incorporated (2019). Retrieved from:https://www.bentley.com/en/products/product-line/structural-analysis-software/staadpro-advanced.
21. Strusoft Software, (2019). Retrieved from: https://Strusoft.com.
22. Strusoft Software, (2019). Retrieved from: https://strusoft.com/products/fem-design/analysis.
23. Strusoft Software, (2019). Retrieved from:
24. https://strusoft.com/products/femdesign/concrete-design
25. RISA Technical Incorporated, (2018). Retrieved from: https://risa.com/p_risa3d.html.
26. 25 Sadeghi K., Abdeh A., Al-Dubai S., (2017, December) "An Overview of Construction and

Installation of Vertical Breakwaters", *International Journal of Innovative Technology and Exploring Engineering (IJITEE)* [Online]. 7(3), Available: https://www.ijitee.org/download/volume-7-issue-3/
27. 26 Sadeghi K., Al Haj Houseen Q., Abo Alsel S., (2017, December) "Gravity Platforms: Design and Construction Overview",
28. International Journal of Innovative Technology and Exploring Engineering (IJITEE) [Online]. 7(3), Available: https://www.ijitee.org/download/volume-7-issue-3/
29. 27. RISA Technical Incorporated, (2018). Retrieved from: https://risa.com/p_risa3d.html.
30. 28. Trimble Solutions Corporation, (2019). Retrieved from: https://www.tekla.com/products/tekla-structural-designer.
31. 29. Trimble Solutions Corporation, (2019). Retrieved from:https://teklastructuraldesigner.support.tekla.com/tekla-structural-designer-2019/en/bas_introducingtsd.
32. 30. Trimble Solutions Corporation, (2019). Retrieved from: https://www.tekla.com/sg/comprehensive-library-design-modules.
33. 31. ASDIP Structural Engineering Software, (2019). Retrieved from: https://www.asdipsoft.com/.
34. 32. S-Frame Software, (2019). Retrieved from:
35. https://s-frame.com/s-frame-analysis/.
36. 33. S-Frame Software, (2019). Retrieved from: https://s-frame.com/s-concrete/.
37. 34. QuickCivilSeries, Design Software, (2019). Retrieved from: https://www.quickcivilseries.com/.

AUTHORS PROFILE

Prof. Dr. Kabir Sadeghi, Kabir.sadeghi@neu.edu.tr
Near East University, Nicosia, North Cyprus Prof. Kabir Sadeghi received the Ph.D. degree in Civil (Structural) Engineering and the DEA degree in Civil (Structural) Engineering from the University of Nantes/France, in March 1995 and May 1991, respectively. He has also an MSc. degree in Structural Engineering and a BSc. degree in Civil Engineering from Amirkabir University of Technology/Iran in 1986 and 1979, respectively. He has been ranked as a top student during his Ph.D., D.E.A., MSc. and BSc. studies. He has submitted four new formulas for numerical simulation and damage evaluation of RC structures. He has had a lot of experiences and collaboration in marine, offshore and industrial RC and steel structures in international big projects as project manager, project engineering manager, senior consultant, designer, and supervisor. Over 100 articles have been published from his works in peer-reviewed SCI Journals, International Journals, and International conferences. Two technical books also have been published by him, which are mainly references for marine and offshore structural engineers, Master students and Ph.D. candidates. He is the founder of IR Marine and Offshore Structures Code of Practice, the founder of The International Conferences on Coasts, Ports and Marine Structures (ICOPMAS) (a biannual conference series) and the chairman of the International Scientific Committee of ICOPMAS90, ICOPMAS96, ICOPMAS98 and ICOPMAS2004. He is a member of the editorial board of several international journals. Since 1986 he has taught/researched as full-time/part-time professor in several high-ranked universities such as; Ecole Centrale de Nantes/University of Nantes/France, Sharif University of Technology, Amirkabir University of Technology, Iran Science and Technology University, K.N. Toosi University of Technology, Shahid Beheshti University, Girne American University and Near East University. Currently, from September 2015, he is a Full Professor of Structural Engineering at The Near East University, Civil and Environmental Engineering Faculty

Nivin Ahmed Hassan Ghaboun
ce_ng2012@yahoo.com
Education:
- Ph.D. candidate, Civil Engineering Department, Near East University, North Cyprus.
- M.Sc. Degree in Engineering Management 2015-2017 Eastern Mediterranean University (EMU), North Cyprus.
(NEU), Nicosia, North Cyprus Mersin 10, Turkey.
- M.Sc. Degree in Construction Engineering & Management 2001-2006 Arab Academy for Science & Technology – Alexandria, Egypt.
- B.Sc. Degree in Civil Engineering 1990-1994 Garyounis University, Libya.

Experience:
- Numidia Modern Oil Services company from (23/11/2013 to 23/11/2014).
- Multiple construction companies, from 01/1/2012 to 30/12/2012 (part-time).
- Al-Mutaheda for Engineering & Construction from 24/7/2010 to 7/1/2011.
- Arabic Group Company for construction (AGC) from 1/2/2008 to 30/4/2010.
- Libyan Technical Company (LTC) from 29/10/1994 to 7/4/1997.

Work experience includes:
-Design of reinforced concrete structures & foundation
-Preparing financial and technical proposals for bids
-Preparing time schedule using Primavera software.
-Estimating quantities for pricing
-Preparing drawings using AutoCAD 2D & 3D.

Publications:
Fatemeh Nouban and Nivin Ghaboun (2017). The factors affecting the methods of construction projects scheduling: A state of the art and overview. *Asian Journal of Natural & Applied Sciences Vol. 6(4)*.