A finite element program for historical stone arch bridges

T. Dede and A. Ural
Karadeniz Technical University, Department of Civil Engineering, Trabzon, Turkey

ABSTRACT: Arch forms are one of the most common used structural shapes on the historical bridges of almost all countries in the world. Besides, Turkey has also many cultural heritage structures and stone arch bridges are one of the most important constructions between them. A computer program has been developed by the writers to analyze civil structures. The program is also able to analyze ancient structures such as arch bridges. A case study has been performed with linear material behaviour and under vertical loadings on a sample bridge to checking the accuracy of the program. Analyses repeated with another finite element software package by using the same modelling data to control the results obtained from the developed program. Some useful information on programming the Finite Element Analysis is given at this study.

1 INTRODUCTION

The Finite Element Method is a computational technique used to obtain approximate solutions by discretising some equations in their space dimensions. The discretisation process is carried out locally over small areas of simple shapes. This results in matrix equations relating the input at the nodes to the output at these same points (Smith and Griffiths 1988). The method is already described in many references, such as Zienkiewicz and Taylor (2000), Hutton (2004).

Architectural heritage requires numerical implementations to keep their original form and strong behaviours. Some new approaches are possible to investigate the behaviours of these heritages in last decades such as limit analyses, linear and non-linear finite element analyses. Especially using non-linear finite element approach brings very sophisticated processes to the practitioners. This approach needs adequate knowledge on characterization of materials and large time requirements for modelling and performing the analyses. In spite of these difficulties, non-linear analyses should be done for special cases, as complex and important structures.

Limit analysis combines sufficient insight into collapse mechanisms, ultimate stress distributions (at least on critical sections) and load capacities, and, on the other, simplicity to be cast in a practical computational tool. Another appealing feature of limit analysis is the reduced number of necessary material parameters, given the difficulties in obtaining reliable data for historical masonry. (Orduna and Lourenço 2001)

Linear-elastic analysis is the most practical approach for the limited time consume and simple structures. Also, this approach gives an idea on the region of probable cracks on the structures.

Certainly, stone arch bridges are one of the most valuable heritages in many countries. These structures have an important role to create a strong link between the past and the present. They all reflect the social and architectural characteristics of the constructed period. The first examples of arches were seen in the underground tombs of the Sumerians in Mesopotamia around the year 3000 BC (Toker and Ünay 2004). The ancient Romans used stone arches for large bridges and aqueducts, and the traditional of building bridges in stone continued in medieval and Renaissance Europe.

Unfortunately, although the Turkish stone arch bridges have been widely sprouted in Turkey and surroundings, studies related these structures have been rare. The main study was being carried out by Çulpan (2002). Almost no studies in which engineering properties and structural analyses were discussed have been carried out. (Ural et al. 2007)

However, it should be well known their general behaviours to transfer them in strong conditions to the next generations. For this reason, some numerical implementations should be needed. In this study, a computer program, coded with MATLAB (2006), is introduced detailed and verified on a sample bridge. Coding process of the program has not been completed yet. It solves only linear-elastic problems in 2D and 3D workspaces.

2 FINITE ELEMENT CODE

The main objective of this research study is to develop an analysis tool suitable for the practitioners. For this aim, authors are presently developing a program to solve general finite element problems with using various element types. It is coded in MATLAB program.

MATLAB is a numerical computing environment and programming language. The program allows easy matrix manipulation, plotting of functions and data, implementation of algorithms, creation of user interfaces, and interfacing with programs in other languages. Although it specializes in numerical computing, an optional toolbox interfaces with the Maple symbolic engine, allowing it to be part of a full computer algebra system. It is now used by more than one million people in industry and academia. Boeing, Daim Chrysler, Motorola, Nasa, Texas Instruments, Toyota, Quantum and Saab companies are only few examples mostly using MATLAB. (Uzunoğlu and Onar 2002)

The developing program is able to solve 2D and 3D linear-elastic and plane stress problems with 2 main element types such as 4 nodded quadrilateral for 2D and 8 nodded solid elements for 3D workspaces (Fig. 1). Degrees of freedoms of the elements are also seen from the same figure.

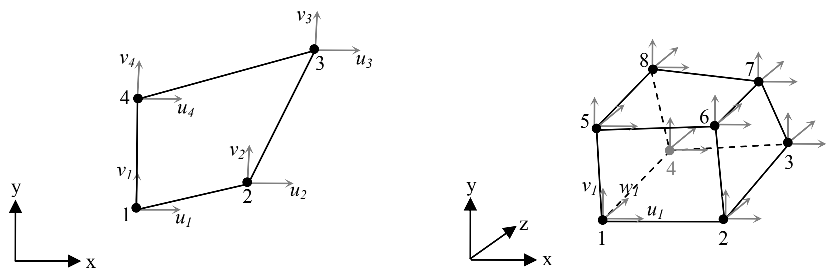


Figure 1: Finite element types used on the AKFEM.

Table 1 : The shape functions used in the code for both 2D and 3D elements.

2D quadrilateral element	3D solid element	
$N_1(r,s)=\frac{1}{4}(1-r)(1-s)$	$N_1(r,s,t)=\frac{1}{8}(1-r)(1-s)(1-t)$	$N_5(r,s,t)=\frac{1}{8}(1-r)(1-s)(1+t)$
$N_2(r,s)=\frac{1}{4}(1+r)(1-s)$	$N_2(r,s,t)=\frac{1}{8}(1+r)(1-s)(1-t)$	$N_6(r,s,t)=\frac{1}{8}(1+r)(1-s)(1+t)$
$N_3(r,s)=\frac{1}{4}(1+r)(1+s)$	$N_3(r,s,t)=\frac{1}{8}(1+r)(1+s)(1-t)$	$N_7(r,s,t)=\frac{1}{8}(1+r)(1+s)(1+t)$

$N_4(r,s)=\frac{1}{4}(1-r)(1+s)$	$N_4(r,s,t)=\frac{1}{8}(1-r)(1+s)(1-t)$	$N_8(r,s,t)=\frac{1}{8}(1-r)(1+s)(1+t)$

2.1 Basic algorithm of the program

The program is benefiting from the easy commands of Matlab. Matrices are mainly used for the input data, pre-processing and post-processing stages. The program verifies with LUSAS (2006) finite element package. This cause when preparing the codes especially for the input data, the geometrical and material properties, loading and boundary conditions of models are formatted similar with LUSAS data file as far as possible. The considered structure first modelled with LUSAS and transfer the data file to the input file of the developing program. Afterwards, the program starts to analyze the model according to the input file. Further information and basic algorithm of the program is given below (Fig. 2).

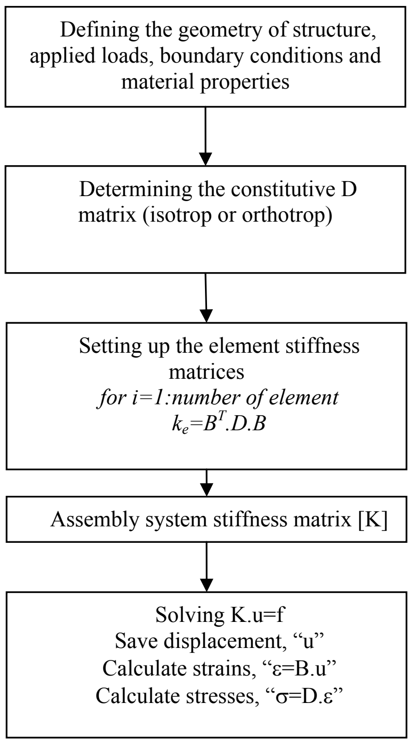


Figure 2: The basic algorithm of the program.

Determining the material characteristics of the elements, [D] elastic matrices must be constituted. These matrices surely constitute various types of materials according to such as working space and isotropy, etc. [D] matrices for 2D isotropic and orthotropic cases are given below;

$$[D]=\frac{E}{1-\upsilon^2}\begin{bmatrix}1 & \upsilon & 0\\ \upsilon & 1 & 0\\ 0 & 0 & \frac{1-\upsilon}{2}\end{bmatrix},\quad [D]=\frac{1}{1-\upsilon_x\upsilon_y}\begin{bmatrix}E_x & E_x\upsilon_y & 0\\ E_y\upsilon_x & E_y & 0\\ 0 & 0 & G_{xy}(1-\upsilon_x\upsilon_y)\end{bmatrix} \quad (1)$$

The program is able to recognize both isotropic and orthotropic elastic matrices for 2D and 3D finite element problems. Because of the limited page number, some formulations such as orthotropic elastic matrices are not given on this paper. Stiffness matrices of all elements are calculated with the assistance of the shape functions, stated above (Table 1). In this paper, only the derivation of the stiffness matrix of 2D quadrilateral elements is described for the simplicity and clearly of the topic.

Displacement functions dependent upon the shape functions are expressed with the following form;

$$u(x,y)=\sum_{i=1}^{4}N_i(r,s)u_i\,,\qquad v(x,y)=\sum_{i=1}^{4}N_i(r,s)v_i \quad (2)$$

Strain components are written in the form of displacements as;

$$\{\varepsilon\}=\begin{Bmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{Bmatrix}=\begin{Bmatrix}\frac{\partial u}{\partial x}\\ \frac{\partial v}{\partial y}\\ \frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\end{Bmatrix}=\begin{bmatrix}\frac{\partial}{\partial x} & 0\\ 0 & \frac{\partial}{\partial y}\\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x}\end{bmatrix}\begin{Bmatrix}u\\ v\end{Bmatrix} \quad (3)$$

Derivations in global coordinates from Eq. 3 are utilized in natural coordinates in matrix form;

$$\begin{Bmatrix}\frac{\partial u}{\partial r}\\ \frac{\partial u}{\partial s}\end{Bmatrix}=\begin{bmatrix}\frac{\partial x}{\partial r} & \frac{\partial y}{\partial r}\\ \frac{\partial x}{\partial s} & \frac{\partial y}{\partial s}\end{bmatrix}\begin{Bmatrix}\frac{\partial u}{\partial x}\\ \frac{\partial u}{\partial y}\end{Bmatrix}=[J]\begin{Bmatrix}\frac{\partial u}{\partial x}\\ \frac{\partial u}{\partial y}\end{Bmatrix} \quad (4)$$

Naturally, Eq. 4 can be solved for the partial derivatives of displacement component *u* with respect to *x* and *y* by multiplying by the inverse of the Jacobian matrix. Numerical methods based on Gaussian quadrature must be used for finding the inverse of the Jacobian matrix instead of using algebraic form. Invert the Jacobian matrix is being solved via Cramer's rule in the code. Application of Cramer's rule results in compact form;

$$\begin{Bmatrix}\frac{\partial u}{\partial x}\\ \frac{\partial u}{\partial y}\end{Bmatrix}=\frac{1}{|J|}\begin{bmatrix}J_{22} & -J_{12}\\ -J_{21} & J_{11}\end{bmatrix}\begin{Bmatrix}\frac{\partial u}{\partial r}\\ \frac{\partial u}{\partial s}\end{Bmatrix} \quad (5)$$

Eq. 5 is the only expanded function for *u* displacement component. Utilizing together *u* and *v* components, strain components are expressed as;

$$\{\varepsilon\}=\begin{Bmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{Bmatrix}=\begin{Bmatrix}\dfrac{\partial u}{\partial x}\\ \dfrac{\partial v}{\partial y}\\ \dfrac{\partial u}{\partial y}+\dfrac{\partial v}{\partial x}\end{Bmatrix}=\frac{1}{|J|}\begin{bmatrix}J_{22} & -J_{12} & 0 & 0\\ 0 & 0 & -J_{21} & J_{11}\\ -J_{21} & J_{11} & J_{22} & -J_{12}\end{bmatrix}\begin{Bmatrix}\dfrac{\partial u}{\partial r}\\ \dfrac{\partial u}{\partial s}\\ \dfrac{\partial v}{\partial r}\\ \dfrac{\partial v}{\partial s}\end{Bmatrix}=[G]\begin{Bmatrix}\dfrac{\partial u}{\partial r}\\ \dfrac{\partial u}{\partial s}\\ \dfrac{\partial v}{\partial r}\\ \dfrac{\partial v}{\partial s}\end{Bmatrix} \quad (6)$$

[G] called as geometric mapping matrix. We must expand the column matrix on the extreme right-hand side of Eq. 6 from the following as;

$$\begin{Bmatrix}\dfrac{\partial u}{\partial r}\\ \dfrac{\partial u}{\partial s}\\ \dfrac{\partial v}{\partial r}\\ \dfrac{\partial v}{\partial s}\end{Bmatrix}=\begin{bmatrix}\dfrac{\partial N_1}{\partial r} & 0 & \dfrac{\partial N_2}{\partial r} & 0 & \dfrac{\partial N_3}{\partial r} & 0 & \dfrac{\partial N_4}{\partial r} & 0\\ \dfrac{\partial N_1}{\partial s} & 0 & \dfrac{\partial N_2}{\partial s} & 0 & \dfrac{\partial N_3}{\partial s} & 0 & \dfrac{\partial N_4}{\partial s} & 0\\ 0 & \dfrac{\partial N_1}{\partial r} & 0 & \dfrac{\partial N_2}{\partial r} & 0 & \dfrac{\partial N_3}{\partial r} & 0 & \dfrac{\partial N_4}{\partial r}\\ 0 & \dfrac{\partial N_1}{\partial s} & 0 & \dfrac{\partial N_2}{\partial s} & 0 & \dfrac{\partial N_3}{\partial s} & 0 & \dfrac{\partial N_4}{\partial s}\end{bmatrix}\begin{Bmatrix}u_1\\ v_1\\ u_2\\ v_2\\ u_3\\ v_3\\ u_4\\ v_4\end{Bmatrix} \quad (7)$$

For shorthand notation, Eq.7 is rewritten as;

$$\begin{Bmatrix}\dfrac{\partial u}{\partial r}\\ \dfrac{\partial u}{\partial s}\\ \dfrac{\partial v}{\partial r}\\ \dfrac{\partial v}{\partial s}\end{Bmatrix}=[P]\{\delta\} \quad (8)$$

In which [P] is the matrix of partial derivatives and $\{\delta\}$ is the column matrix of nodal displacement components. Combining Eq. 6 and 8, we obtain the sought-after relation for the strain components in terms of nodal displacement components as $\{\varepsilon\}=[G][P]\{\delta\}$ and, by analogy with previous developments, matrix $[B]=[G][P]$ has been determined such that $\{\varepsilon\}=[B]\{\delta\}$. Finally the element stiffness matrix is defined by;

$$\left[k^{(e)}\right]=t\int_A [B]^T[D][B]\,dA \quad (9)$$

3 A CASE STUDY: COŞANDERE ARCH BRIDGE

In this paper, Coşandere Arch Bridge is selected as a case study. The bridge belongs to the most valuable heritages in Trabzon (Fig. 3). The Coşandere Bridge is one of the traditional bridges of the East Blacksea Region of Turkey and it was built in 19th century during the Ottoman period. It has a circular arch and constituted with yellow and green stones especially on the arch form. The bridge has dimensions about 30 m length, 8.5 height and 4 m width. Due to the cultural heritage status and being conserved, it cannot be determined the inner material properties of the bridge. For this reason, it is considered that 4 different materials used on the bridge such as parapets, infill, spandrel walls and arch. The bridge was analyzed using nonlinear material properties in another study by the authors (Ural 2005) and according to the results, the bridge exhibit good performance. In this study in the light of the LUSAS data file, an input file was prepared for the developing program. Analyses are performed only for the linear elastic materials and concentrated loadings.

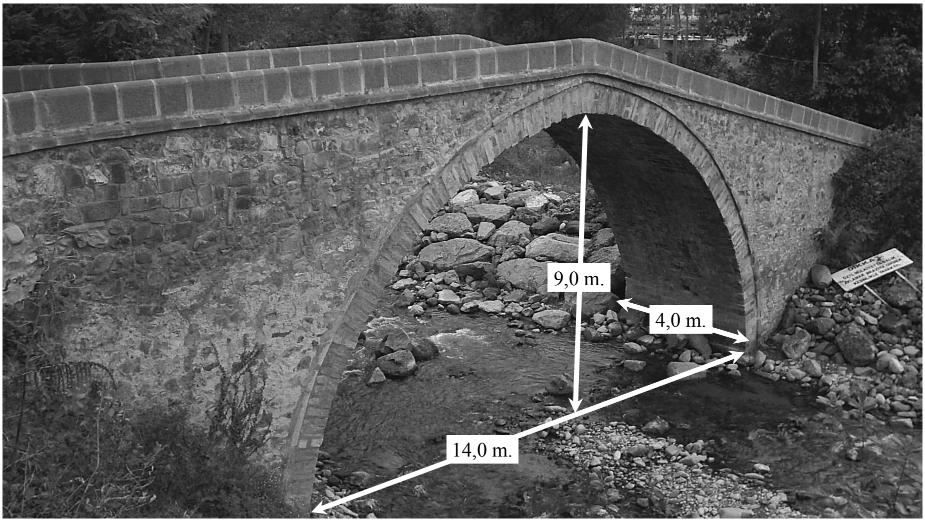
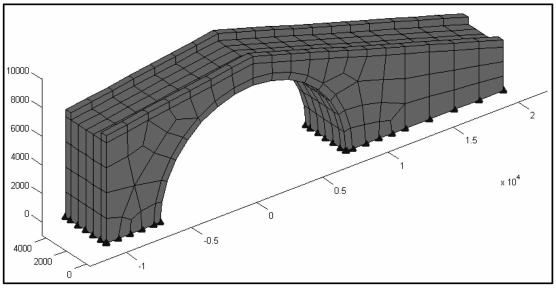
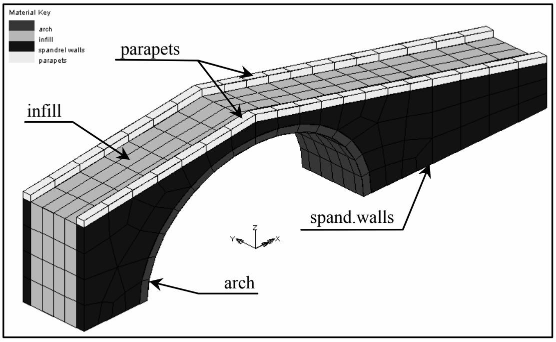


Figure 3: The Coşandere Bridge.

According to the obtained data from LUSAS, 752 nodes and 440 elements are used for the bridge. The undeformed mesh of Coşandere Arch Bridge can be seen from the following Fig. 4 constituted with both MATLAB and LUSAS graphical interfaces.



(a) MATLAB view of the model



(b) LUSAS view of the model

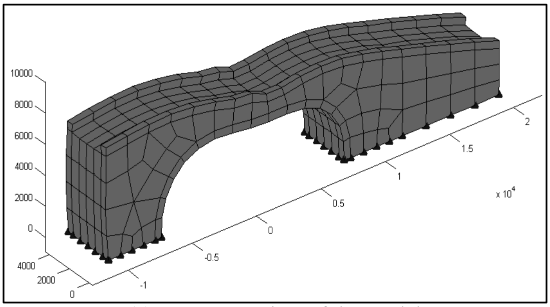
Figure 4: User-friendly meshing views of the sample bridge.

The most common difficulties on studying ancient structures are determining the properties of materials that used on the structure especially for the inner materials. Inspections on the structure are very limited due to the importance of the continuity of the cultural value. Never-theless due to the difficulty is present for this study, material properties used for the bridge is considered from literature with similar arch bridges. According to some researches on literature to determine the material properties of materials, the average properties for analyses are given below;

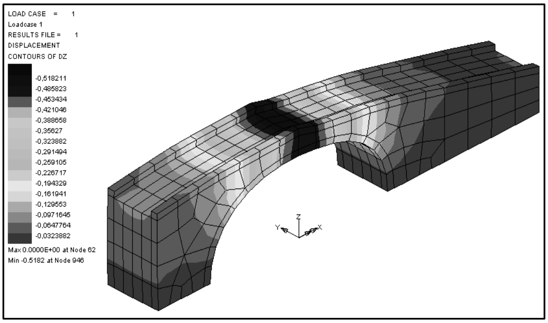
Table 2 : Mechanical properties of materials.

	Young Modulus (N/mm^2)	Poisson ratio
Arch	3,000	0.2
Spandrel walls	2,500	0.2
Infill	1,000	0.05
Parapets	1,000	0.05

The model has been considered in linear-elastic range with both developing program and LUSAS. Vertical concentrated loading are considered per node on the top of the superstructure as 10,000N. The results of the analyses, maximum vertical displacements are occurred at the mid-span of the arch as nearly 0.52 mm as shown in Fig. 5. The maximum compression and tension stresses have been determined as 0.11 MPa and 0.002 MPa respectively. These values have been occurred at the same location of the bridge. Nevertheless most of the probable cracks can be begin from these parts of the bridge.



(a) MATLAB view of the model



(b) LUSAS view with vertical displacement contours

Figure 5: Deformed mesh of the analyzed models.

The results of analyses are fortunately very similar such as displacement, strain and stress values. The bridge is in perfect condition for the static vertical loadings. Some little models such as 2D masonry walls were performed with the developing program to verify the results but this model is the biggest one that performed until now.

4 CONCLUSIONS

A finite element program has been developed using with MATLAB to perform structural analyses. It is assumed to be used for mainly concrete and masonry structures. As stated above, the coding stage has not been concluded. The authors are continued to study on coding of nonlinear analyses of the program and they plans to add some useful nonlinear material models to the program. It is undoubtedly the most suitable material model for macro modelling of masonry structures is Rankine-Hill and also Drucker-Prager criterions. If nothing unforeseen happens, these criterions being added into the program. Afterwards it is able to solve nonlinear problems for concrete and masonry structures.

In Turkish national literature, there are very few studying about stone arch bridges. Besides, architectural heritage are not given importance that they deserve and most of them are in poor condition. They should be reinvestigated and put forward the present conditions. If it necessary, restoration projects can be studied as soon as possible on damaged bridges.

From the results, maximum values of displacement and stresses are occurred on the mid-span of the arch. As known from the literature, this part of bridges should be most take care on the strengthening projects. Some other analyses must be performed for the Coşandere Bridge such as seismic and flood to determine the other responses.

REFERENCES

Çulpan, C. 2002. *Turkish stone bridges; from medieval to the end of Ottoman period.* Turkish History Foundation. (in Turkish)

Hutton, D.V. 2004. *Fundamentals of Finite Element Analysis.* McGraw-Hill Companies.

LUSAS. 2006. LUSAS finite element analysis software products, Finite Element System, FEA Ltd, United Kingdom.

MATLAB. 2006. version 7.01.

Orduna, A. and Lourenço, P. B. 2001. Limit analysis as a tool for the simplified assessment of ancient masonry structures, *Historical Constructions*, p. 511-520. Guimaraes, Portugal.

Smith, I. M. and Griffiths, D. V. 1988. *Programming the finite element method, Second Edition*, John Wiley & Sons.

Toker, S. and Ünay, A. İ. 2004. Mathematical modelling and finite element analysis of masonry arch bridges. *Gazi University Journal of Science* 17(2), p. 129-139.

Ural, A. 2005. Nonlinear analyses of historical arch bridges, *2nd Engineering Sciences Youth Researchers Congress (MBGAK)*, abstract book p. 53, Istanbul University, Turkey.

Ural, A. Oruç, Ş. Doğangün, A. Tuluk, Ö. 2007. Turkish historical arch bridges and their deteriorations and failures. *Engineering Failure Analysis,* doi: 10.1016/j.engfailanal.2007.01.006.

Uzunoğlu, M. and Onar, Ö.Ç. 2002. *Advanced Matlab 6.0-6.5 with simple expressions*. Türkme Kitabevi, İstanbul. (in Turkish)

Zienkiewicz, O.C. and Taylor, R.L. 2000. *The Finite Element Method, Fifth Edition, Vol2:Solid Mechanics.* Butterworth Heinemann.