

Version 2.0

Technical documentation

FEM Toolbox for Solid Mechanics: Technical Documentation

Author: Anton Zaicenco

`<a.zaicenco@codedevelopment.net>`

`www.codedevelopment.net`

March 20, 2014

Table of Contents

Nomenclature	3
1 Copyright	4
2 Introduction	5
3 Installation	6
4 Main GUI	7
4.1 GUI functionality	7
5 Structure of the Toolbox	9
5.1 Toolbox file structure	9
6 Input File Structure	10
7 Library of the finite elements	11
8 Input file examples	13
8.1 2D beam (FF_BEAM)	13
8.2 2D quadrilateral element (D2_SCQ)	17
8.3 2D triangular element (D2_CST)	21
8.4 3D brick element (D3_LB)	25
8.5 3D tetrahedral element (D3_TETRAH)	29
8.6 3D beam element (D3_BEAM)	33
8.7 2D heat transfer (D2_CST_temp)	37

Nomenclature

BC	Boundary Condition
DBC	Displacement Boundary Conditions
DOF	Degree of Freedom
FEM	Finite Element Method
GUI	Graphical User Interface
MFC	Multi-freedom Constraints

1 Copyright

Copyright 2010-2032 Zaicenco Computational Solutions. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

2 Introduction

The FEM Toolbox was conceived as a teaching tool for students of structural dynamics in 2004 and gradually grew up into a versatile tool for static, transient and modal analysis of linear problems in solid mechanics based on a library of basic 2D and 3D finite elements.

This document describes the structure, functionality and the main concepts of the software package. There is no dependency on other Matlab toolboxes. The main GUI runs from the Matlab command window.

The software manual is divided into several sections:

1. Section 4 discusses the basic GUI functionality. It outlines the operations controlled through the main GUI (buttons and check-boxes).
2. Section 5 discusses the file structure of the Toolbox: directory structure and list of files.
3. Section 6 provides the content of the input files. The output of the input files is a data structure `in_data`, which stores all information about the geometry, element properties, loads, and constraints of the system.
4. Section 7 outlines the library of the finite elements used in the Toolbox.
5. Section 8 provides examples of several input files, discussing the input parameters and results.

I hope the Toolbox will be useful for students, engineers, and researchers, who use FEM in design and development of their products, and perform structural simulations of various systems.

Anton Zaicenco