

Appendix C

DESCRIPTION OF THE DEVELOPED PROGRAM FOR THE DESIGN OPTIMIZATION OF STEEL FRAME STRUCTURES

The computer program is developed for the design optimization of 2D and 3D steel frame structures. The program is linked to the finite element package ANSYS in order to compute the internal forces and moments of the structural members and also to calculate the elastic critical buckling load of the structure under consideration.

The main program performs two tasks:

1. It calls the input data files (29 files). These can be describes as follows:
 - FILE.LOG contains the finite element model, loading cases, etc.
 - GROUPING.DAT comprises the member grouping of the structural members.
 - ANS.DAT contains the numbers of the representative members for the design variables.
 - INGA.DAT contains the input genetic algorithm parameters.
 - TABLE1, TABLE 2 and TABLE3.DAT holds the cross sectional properties of universal columns, universal beams and circular hollow sections respectively,

- DATA.DAT comprises the description of each member, i.e., whether it is beam or column,
 - METHOD.DAT contains the desired approach to tackle the determination of effective buckling length of columns.
 - A27, B27, C27, D27, T11, T14 and T21 are look up tables and are taken from BS 5950.
 - UBEAMS, UCOLUMN and UTUBES.DAT contains a list of the name of used sections,
 - K1K2.DAT contains the full description of each member connectivity,
 - XDISP, YDISP and ZDISP.DAT holds a list of the node numbering corresponding to the nodes, which should be checked,
 - SWAY and NSWAY.DAT contains the values digitised from Figure 23 and 24 of BS 5950 for the determination of the effective length factor of a column in sway and non-sway framework,
 - PRACTC.DAT comprises the numbering of columns, which have practical constraints imposed on,
 - PRACTB.DAT holds the numbering of beams, which have practical constraints imposed on,
 - DECLARE.DAT is a declaration file for the used notations and
 - PARAM.DAT contains parameters.
2. The program also calls 27 subroutines. These subroutines can be described as follows:
- GA performs the developed elitist strategy incorporating the three basic steps (reproduction, crossover and mutation).
 - RAN generates a random number between 0 and 1.

- SELECT models a biased roulette wheel for the selections of parents.
- DISPLACX performs the displacement check in the X direction.
- DISPLACY performs the displacement check in the Y direction.
- DISPLACZ performs the displacement check in the Z direction.
- SWAYCHECK determines whether the structure is sway or non-way.
- CON210 converts the decimal numbers to binary numbers.
- CON102 converts the binary numbers to decimal numbers.
- SORTS sorts the population in ascending order.
- EVEN calculates the nearest even number.
- ansysi prepares the cross-sectional properties of the structural members.
- TORSION checks the lateral torsional buckling.
- OBJECT calls the finite element analysis package ANSYS.
- DELAY delays the run according to the size of the problem.
- OPENRES reads the value of the critical buckling load.
- GROUP is associated with the linking of the design variables.
- PRACTCO checks the practical constraints imposed on columns.
- PRACTBE checks the practical constraints imposed on beams.
- OBJBE checks the slenderness, strength constraints on beams.
- OBJCO checks the slenderness, strength constraints on columns.
- OBJBR checks the slenderness, strength constraints on bracing members.
- ELEMENT calculates the end coefficient constraints (k_1 and k_2).
- CODEBUCK determines the value of the effective length factor for columns.
- SHEARCH checks the shear in X and Y directions.
- FITNESS performs fitness function evaluation and writes the data file named FILE.CON that will be used by ANSYS for each design set.

- OBJECT2 computes the internal forces and binding moments of 2D frame structures and 2D trusses. This subroutine is taken from Segerlind (1984). This subroutine was designed to analyse a structure having 50 elements. It is therefore modified to suit the design optimization task as well as the allowance for the analysis of structures having a number of elements more than 50.