

COMPARISON OF PROGRAMMING ANSYS AND COSMOS/M FINITE ELEMENT SYSTEMS

FERENC JÁNOS SZABÓ

*University of Miskolc, Department of Machine and Product Design
H-3515 Miskolc-Egyetemváros
machszf@uni-miskolc.hu
<https://orcid.org/0000-0002-6694-8959>*

Abstract: A simple optimization task is programmed by using the built- in programming possibilities of two finite element systems: ANSYS APDL (Parametric Design Language) and COSMOS/M built-in macro language. The same program is written in the two different programming language and the resulting program listings is compared. The demonstration optimization problem is to find the minimum safe radius of an L-shape model, loaded by a uniform surface pressure. The program to solve this optimization task builds the model, solves the finite element problem, and iteratively improves the radius until the optimum solution. In both cases, when the program running ends, we will see the optimum structure on the screen, and we can use the menu possibilities of the finite element system to see any details of the optimized model or listing the values of the parameters.

Keywords: *programming FEM systems, ANSYS, COSMOS/M, ANSYS APDL, COSMOS Built in macro language*

1. INTRODUCTION

In this paper an optimization problem is solved by using the internal programming possibilities of two different finite element program systems (COSMOS/M and ANSYS). In COSMOS/M system, the internal programming is possible by a macro language, which is similar to FORTRAN, in the ANSYS system this is possible by the APDL. Both these programming languages give the possibility to build the 3D model, mesh it, defining material properties, boundary conditions, loads and solve the finite element task. After the finite element run, in both systems it is possible to access to the displacement and stress solutions (and many more results) and using the IF structures, loops and GOTO structures it is possible to check the availability and safety factor of the structure. If the structure is feasible for the applied load, it is possible to think about the weight reduction or improve other important characteristic of the structure, which will lead us to the optimization world. In this case it is possible to build up an optimization algorithm and by using this algorithm it is possible to define an optimization process. This complex process (build model- solve FEM task- check the feasibility of the model- change the model, iterate- optimize) can be integrated into one program inside of the finite element system, using the internal programming language and when running this complex problem, this

process will be automatically played by the system just like a film, showing the self-improving and self-optimizing process of the model. When the optimum solution is reached, the program will stop showing the optimum solution and we can see any characteristics or results more detailly (stress and displacement contours, value of the parameters used during the running of the program, etc.), by using the possibilities given by the menu of the finite element program system.

In the paper the reader can find the listing of both programs, therefore it is possible to compare the most important commands, used for the build-up of the model and for some iterative situations during the optimization process. The conclusion of this comparison could be that the programming languages, keywords and the logic of the commands are very similar, but in some details of the model building one can find some differences, which may need a time to understand or translate. The programming possibility of finite element systems is very useful during the analysis [1], [5], optimization [2], [5], and multidisciplinary optimization [3], [4] of structures.

It is necessary to comment that the CATIA Integrated CAD- FEM system bought the COSMOS/M program system, therefore in order to run the program listing given in this paper, it is necessary to use an earlier version of COSMOS/M, which is dated before this integration.

2. DESCRIPTION OF THE MODEL TO BE OPTIMIZED

The optimization task is to minimize the radius in the corner of an L-shape element (see *Figure 1*), which is loaded by a pressure of 1.15 MPa and it is supposed to be bonded to a wall at its backside surface (fixed support). The maximum permissible stress in the model can be 56 MPa, this is an implicit constraint of the optimization. 56 MPa will assure that the final result of the optimization is safely inside of the permissible region and it will be easy to test the working of the optimization algorithm. The explicit constraint is that the radius should be between 1 and 10 mm. For simplicity reasons, this optimization task is a one-variable optimization problem, in order to show by very short and easy to understand programming segments, the thinking and programming of the optimization process.

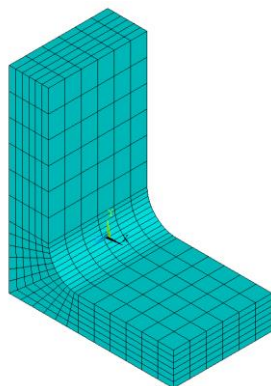


Figure 1. The model to be optimized

The model contains two brick shape blocks (a horizontal, towards x direction and a vertical, towards y direction) and in the corner a radius. The horizontal block has a length described by the parameter AA, the vertical one BB. Extrusion towards z direction is CC. The thickness of vertical block is DD and of horizontal block EE. Material of the model is structural steel, the vertical block has fixed support at the backside surface and the horizontal block is loaded in y direction. The program listing of the model in COSMOS/M macro language can be found in *Figure 2*, and in ANSYS APDL in *Figure 3*. In the program listings it is possible to identify the numerical values of the parameters.

```

PARASSIGN,AA,REAL,0.050
PARASSIGN,BB,REAL,0.060
PARASSIGN,CC,REAL,0.030
PARASSIGN,DD,REAL,0.012
PARASSIGN,EE,REAL,0.010
PARASSIGN,RR,REAL,0.009
PARASSIGN,GG,REAL,AA-DD-RR
PARASSIGN,K1,REAL,0.707107
PT,1,0,0,0,0,0
PT,2,AA-GG,0,0,0,0
PT,3,AA,0,0,0,0
PT,4,AA-GG,EE,0,0
PT,5,AA,EE,0,0
PT,6,AA-GG-RR*K1,EE+RR*(1-K1),0,0
PT,7,0,0,EE+RR,0,0
PT,8,DD,EE+RR,0,0
PT,9,0,0,BB,0,0
PT,10,DD,BB,0,0
PT,11,AA-GG,EE+RR,0,0
CRARC,1,4,6,11,RR
CRARC,2,6,8,11,RR
CRLINE,3,1,2
CRLINE,4,1,7
SF4PT,1,2,3,5,4,0
SF2CR,2,1,3,0
SF2CR,3,2,4,0
SF4PT,4,7,8,10,9,0
VLEXTRUDE,1,4,1,2,CC
M_VL,1,4,1,8,4,4,10,1,1,1
DSF,21,AL,0,21,1
PSF,8,1.15E+06,8,1.1.15E+006,1.15E+006,4
NMERGE,1,1100,1,0.0001,0,0,0
MPROP,1,EX,2.1E+11,EV,2.1E+11,EZ,2.1E+11,GXY,0.8E+11,G&XZ,0.8E+11,GYZ,0.8E+11
MPROP,1,DENS,7850
EGROUP,1,SOLID,0,1,0,0,0,0,0,0
R_CHECK,STATIC
R_STATIC

```

Figure 2. The program listing in COSMOS/M macro language (build and solve the model)

```

!!! MODEL TO BE OPTIMIZED
/RGB,INDEX,100,100,100,0
/RGB,INDEX,0,0,0,15
! Isometric view
/VIEW,1,1,1,1
/ANG,1
/REP,FAST
! Define constant parameters
! Units: m, N
ELAST=2E5
NU=0.3
P1=6
P2=8
!!SETUP: element types, material
KEYW,PR_STRUC,1 ! Pref / Struct
/PREP7
ET,1,SOLID185
MPTEMP,1,0
MPDATA,EX,1,,ELAST
MPDATA,PRXY,1,,NU
AA=0.050
BB=0.060
CC=0.030
DD=0.012
EE=0.010
RR=0.009
GG=AA-DD-RR
K1=0.707107
K,1,0,0,0,0,0,0
K,2,AA-GG,0,0,0,0
K,3,AA,0,0,0,0
K,4,AA-GG,EE,0,0
K,5,AA,EE,0,0
K,6,AA-GG-(RR*K1),EE+RR*(1-K1),0,0
K,7,0,0,EE+RR,0,0
K,8,DD,EE+RR,0,0
K,9,0,0,BB,0,0
K,10,DD,BB,0,0
K,11,AA-GG,EE+RR,0,0
LARC,4,6,11,RR
LARC,6,8,11,RR
L,1,2
L,1,7
L,1,6
L,7,8
L,2,4
L,2,3
L,3,5
L,4,5
L,7,9
L,8,10
L,9,10
AL,3,7,1,5
AL,5,2,6,4
A,2,3,5,4
A,7,8,10,9
VEXT,1,4,1,,,CC
*DO,11,3,13
LESIZE,11,,,6
*ENDDO
*DO,11,18,21
LESIZE,11,,,6
*ENDDO
LESIZE,25,,,6
LESIZE,26,,,6
LESIZE,30,,,6
LESIZE,31,,,6
LESIZE,36,,,6
VMESH,1,4,1
DA,21,ALL,0
SFA,17,1,PRES,1.15e+06
FINISH
/SOL
SOLVE
/POST1
PLNSOL,S,Y,2,1,0

```

Figure 3. Program listing in ANSYS APDL (build and solve the model)

Both of the program listings show the build- up the model, setting the material data and the boundary conditions of the finite element running (displacement constraint and load), they mesh and solve the model. After the running of these programs, in the memory of the computer there will be ready the displacement and stress results, so it will be possible to read them and check if the structure fulfils the feasibility conditions or not.

Since the optimization is a very simple, one variable minimization problem, the optimum searching algorithm is also very simple. Starting from a 'big' enough radius (for example 9 mm) which gives feasible structure so the maximum equivalent stress is under the permissible stress (56 MPa), by using a step of 1 mm we decrease the radius step by step and in each step we check the maximum

equivalent stress. If this stress is higher than the permissible stress, then the radius is too small and the previous radius was the optimum.

If the equivalent stress is smaller as the permissible stress, it means that the structure is feasible and the radius can be decreased by 1 mm as the next step. In the case when the equivalent stress is equal to the permissible stress, this means that this is the optimum radius. When the optimum is reached, the program will delete the model and build it and solve it once more for the optimum radius and it will stop showing the optimum structure. Applying the menu possibilities of the finite element system, it is possible to see the stress and displacement contours or list the values of the parameters, in order to check all the results of the running process.

In order to perform the optimization process, a main program is necessary, controlling every step of the search and setting the actual values for the parameters and calling the model building program segment (newana). This main program is shown in *Figure 4*. *Figure 5* and *Figure 6* show the stress contours in both of finite element systems.

```

ROPT=0.0
MAXITER=8
VONMIS=0.0
VONADM=0.55e+08
LEPES=0.001
RR=0.009
*DO, JJJ, 1, MAXITER
/INPUT, C:/Users/ff/Desktop/newana,txt,,,,
*GET, VONMIS, PLNSOL, 0, MAX,,
*IF, VONMIS, GT, VONADM, THEN
ROPT=RR+LEPES
RR=ROPT
*ELSEIF, VONMIS, EQ, VONADM, THEN
ROPT=RR
RR=ROPT
*ELSE
RR=RR-LEPES
*ENDIF
*ENDIF
/INPUT, C:/Users/ff/Desktop/töröl,txt,,,,
*ENDDO
RR=ROPT
/INPUT, C:/Users/ff/Desktop/newana,txt,,,,
PARASSIGN,ROPT,REAL,0.0
PARASSIGN,MAXITER,INT,8
PARASSIGN,VONMISES,REAL,0.0
PARASSIGN,VONADM,REAL,0.55E+08
PARASSIGN,LEPES,REAL,0.001
PARASSIGN,RR,REAL,0.009
#LOOP CIMKE1 MAXITER
FILE,F:\DOKUMENTUMOK\ff\NEWANA.GEO
PARASSIGN,VONMISES,REAL,VON(1|ND|0)
#IF (VONMISES > VONADM)
PARASSIGN,ROPT,REAL,RR+LEPES
#GOTO CIMKE2
#ELSE
#IF (VONMISES == VONADM)
PARASSIGN,ROPT,REAL,RR
#GOTO CIMKE2
#ELSE
PARASSIGN,RR,REAL,RR-LEPES
#ENDIF
#ENDIF
FILE,F:\DOKUMENTUMOK\ff\TOROL.GEO
#LABEL CIMKE1
#LABEL CIMKE2
FILE,F:\DOKUMENTUMOK\ff\TOROL.GEO
PARASSIGN,RR,REAL,ROPT
FILE,F:\DOKUMENTUMOK\ff\NEWANA.GEO

```

Figure 4. Program listings (main), left ANSYS APDL, right: COSMOS/M macro language

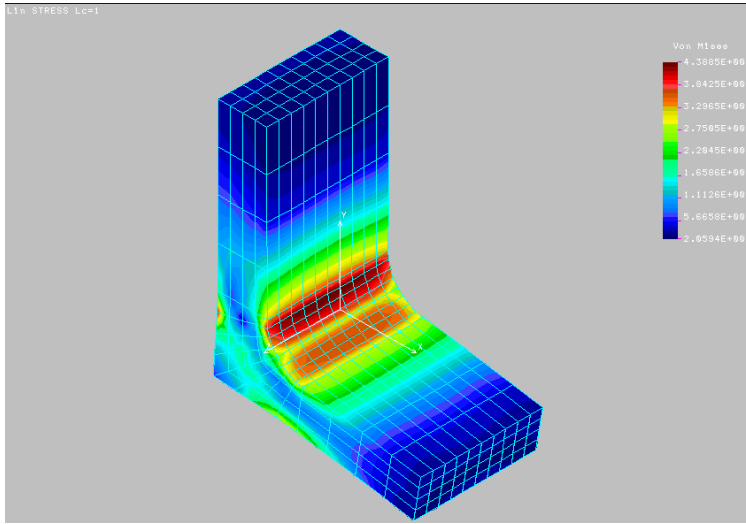


Figure 5. Stress contours in COSMOS/M

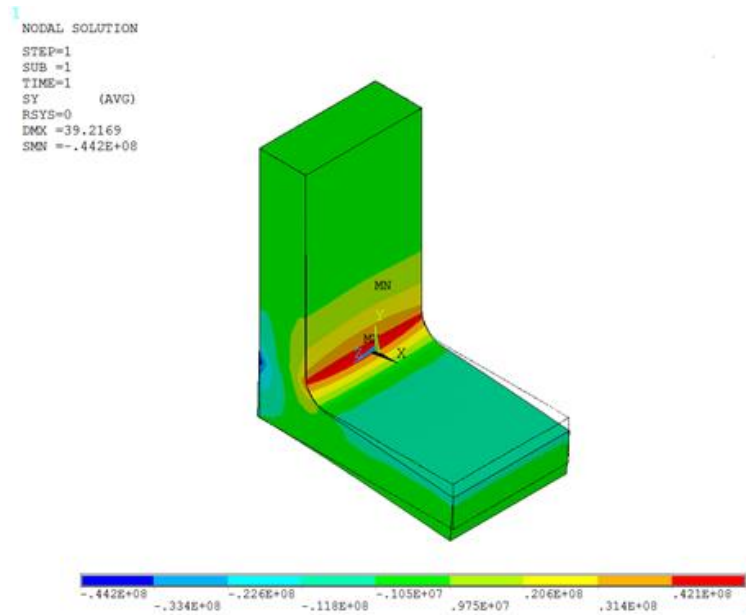


Figure 6. Stress contours in ANSYS Mechanical

Figure 5 and Figure 6 show the stress contours on the deformed shape. Figure 7 shows the displacement contours of the optimized model in ANSYS Mechanical finite element system.

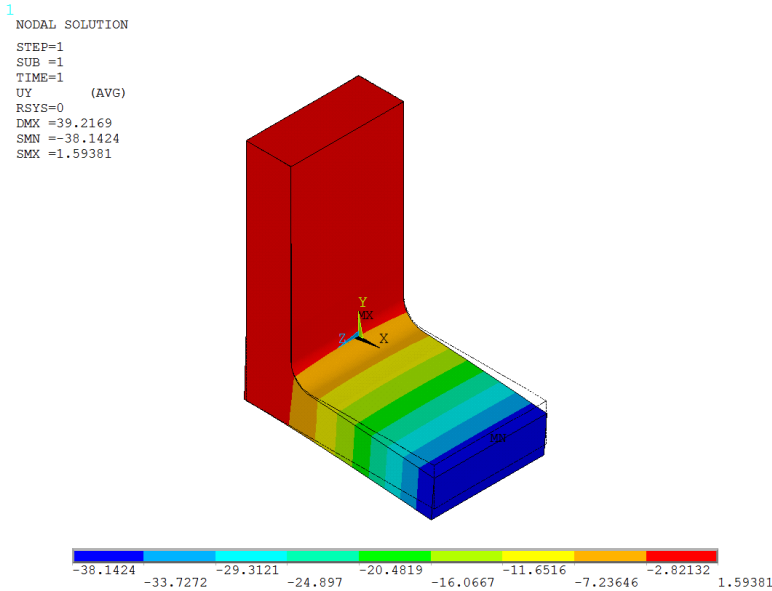


Figure 7. Displacement contours in ANSYS Mechanical

3. COMPARISON OF THE PROGRAMS

Comparing the program listings, the first thing what we can conclude is that the ANSYS APDL program is 60% longer than COSMOS/M macro language program. This is because of more comments, which can help the understanding, and there are some differences concerning the 3D model building keywords and the meshing. The length of the main program is almost the same, ANSYS APDL program is shorter. This is because the COSMOS/M program is using GOTO and LABEL statements, but in the APDL program it was easier to solve the decision-making procedure without GOTO statement. The logic and the syntax of the model building and solution keywords are very similar, easy to understand. All these characteristics make very easy to translate one program into another programming language. The history of these programs is that first the COSMOS/M program was written and later the APDL program was built by translating the macro language commands into APDL commands or into similar commands which can do the required steps. The final results of the optimization are shown in *Table 1*. These results are the optimum results in both programs, in COSMOS/M and in ANSYS, too.

Table 1
Optimum results

Param	R _{opt} [mm]	Vonadm [MPa]	Vonmis [MPa]	Step No.
Value	5	56	55.61	5

4. SUMMARY

Optimization problem and its solution is shown for a three-dimensional, L shape element, solved by programming the finite element program systems COSMOS/M and ANSYS Mechanical. In COSMOS/M the built-in macro language is used, in ANSYS the APDL (ANSYS Parametric Design Language) is applied. In both programming language, the full program listing is shown for the optimization process. This program listing contains the model building and finite element solving program segment and the optimum process controlling main program, which calls the model building segment as a subroutine. The optimum design task is a one variable problem with one explicit and one implicit constraint.

Running the programs, when they successfully finish the optimum searching, they stop showing the optimum structure and it is possible to see all the details and parameters by using the menu system of the given program system. Comparing the programs, as a conclusion it is possible to say that the logic and the build-up of the programs is very similar, however it is possible to find some differences during the model building and solution setting commands.

REFERENCES

- [1] Szabó, F. J. (2019). Finite element study of rotating elements of a ventilator. *International Review of Mechanical Engineering*, Vol. 13, No. 6. <https://doi.org/10.15866/ireme.v13i6.17145>
- [2] Kovács, B. et al. (2001). A generalized shape optimization procedure for the solution of linear partial differential equations with application to multidisciplinary optimization. *Structural and Multidisciplinary Optimization*, Vol. 21, No. 4, <https://doi.org/10.1007/PL00013280>.
- [3] Szabó, F. J. (2016). Multidisciplinary optimization of journal bearings, using a RVA evolutionary type optimization algorithm. *Acta Polytechnica Hungarica*, Vol. 13, No. 7, <https://doi.org/10.12700/APH.13.7.2016.7.10>.
- [4] Szabó, F. J. (2008). Multidisciplinary optimization of a structure with temperature dependent material characteristics, subjected to impact loading. *International Review of Mechanical Engineering*, Vol. 2, No. 3, ISSN 1970-8734.
- [5] Szabó, F. J. (2008). Finite element analysis and optimization of a car seat under impact loading. In: Jármai, K., Farkas, J. (eds.). *Design, Fabrication and Economy of Welded Structures, International Conference Proceedings*, Woodhead Publishing Ltd., Cambridge.