

ANSA AS PRE-PROCESSOR FOR CODE_ASTER

PhD Roman Frątczak*, M.Eng. Maciej Czapliński

Nobo Solutions S.A., Poland

KEYWORDS –

ANSA PREPROCESSOR, CODE_ASTER

ABSTRACT –

Currently, in version 19 Ansa supports nearly all world-recognizable FE solver decks. A one more deck may be interesting for current users especially if it is an open source solver like Code_aster (CA). The very weak point of CA solver is the pre-processor where Ansa could cover a large amount of pre-processing functionalities. Especially in the structural analyses including rotordynamics. Some features in CA – common for all FE decks – are relatively “well-hidden” or introduced in a specific way. This makes a CA user non-efficient – especially when compared to a commercial FE package. We would like to share briefly our experience in CA to indicate a “bottle-neck” and present the solution already applied in Ansa based on a case study.

1. SIMULATION IN CODE_ASTER

Principles of simulation conduction in CA are the same as in most FE systems, however specific approach of definitions may be an obstacle for production use. In fact, the CA communicates with user using Python scripts and CA uses a lot of Python. The solver and available GUI is released under GPL like license under Linux OS- with availability of MS Windows systems recompilation or even delivered by 3rd parties already compiled version.



Figure 1 – Typical workflow for CA – preparation mesh & macro, run simulation and review results.

The figure above shows typical work flow – without GUI interface. The delivered GUI (being still updated) in fact realize the task from shown workflow, but with weak efficiency (compared to commercial tools). Therefore – for more experienced users - to speed up they prefer to use “old one” method for simulation runs – using batch solver runs or delivered batch-runner (ASTK tool). This is the first possible step where ANSA could help – run simulation in proper and efficient way – let’s say the middle terminal view form figure 1. Let’s focus on the first – model preparation.

As ANSA is good pre-processor, especially for FE mesh preparation and manipulation – this don’t need to be discussed at all.

The preparation of CA simulation needs also a macro file with python script containing FE simulation description. There are many different functions and keywords that may be used in such macro file (usually with extension *.comm) and may be “easily” found in the official documentation. The documentation itself (begin translated form French to English) are also

impressive as consists of thousands of pages. Let's see some example and roughly go through – see figure 2 below.

2. EXAMPLES OF USAGE

Few show-cases from FE simulations are presented in order to point out the functionalities, that are unavailable directly in CA environment, but may be found or introduced in ANSA.

```

DEBUT(LANG='EN')

mesh1 = LIRE_MALLAGE(FORMAT='MED',
                    UNITE=20)
                    Read a mesh from file to variable mesh1

mesh = CREA_MALLAGE(LINE_QUAD=_F(PREF_NOEUD='NSS',
                                TOUT='OUI'),
                  MAILLAGE=mesh1)
                  Conver mesh from linear to quadratic

mesh0 = CREA_MALLAGE(MAILLAGE=mesh,
                   MODI_MAILLE=( _F(GROUP_MA=('qua', ),
                                   OPTION='QUAD8 9',
                                   PREF_NOEUD='NSSS'),
                                   _F(GROUP_MA=('tria', ),
                                   OPTION='TRIA6 7',
                                   PREF_NOEUD='NSSSS'))),
                   Convert FE elements from incomplete 2nd
                   order to complete (additional node at middle)

model = AFFE_MODELE(AFFE=_F(GROUP_MA=('Lens', 'Housing', 'Reflector'),
                           MODELISATION=('COQUE 3D', ),
                           PHENOMENE='MECANIQUE'),
                   MAILLAGE=mesh0)
                   Define the FE element type (2nd order shell)

elemprop = AFFE_CARA_ELEM(COQUE=_F(ANGL_REP=(0.0, 1.0),
                                   EPAIS=0.003,
                                   GROUP_MA=('Lens', 'Housing', 'Reflector')),
                           MODELE=model)
                           Define properties (shell thickness)

mater = DEF1_MATERIAU(ELAS=_F(E=3000000000.0,
                              NU=0.36,
                              RHO=2000.0))
                              Define material

fieldmat = AFFE_MATERIAU(AFFE=_F(GROUP_MA=('Lens', 'Housing', 'Reflector'),
                                 MATER=(mater, )),
                        MAILLAGE=mesh0,
                        MODELE=model)
                        Assign materials to FE elements/groups

load = AFFE_CHAR_MECA(DDL_IMPO=_F(DX=0.0,
                                   DY=0.0,
                                   DZ=0.0,
                                   GROUP_NO=('fix3', 'fix2', 'fix1')),
                       MODELE=model,
                       PESANTEUR=_F(DIRECTION=(0.0, -1.0, 0.0),
                                     GRAVITE=9.81))
                       Boundary conditions & loads

ASSEMBLAGE(CARA_ELEM=elemprop,
           CHAM_MATER=fieldmat,
           CHARGE=(load, ),
           MATR_ASSE=( _F(MATRICE=CO('MASS'),
                           OPTION='MASS_MECA'),
                       _F(MATRICE=CO('STIFF'),
                           OPTION='RIGI_MECA')),
           MODELE=model,
           NUME_DDL=CO('sto'))
           Construct mass and stiffness matrixes

reslin = MECA_STATIQUE(CARA_ELEM=elemprop,
                     CHAM_MATER=fieldmat,
                     EXCIT=_F(CHARGE=load),
                     MODELE=model)
                     Solve the linear static problem

unnamed = PROJ_CHAMP(MAILLAGE_1=mesh0,
                    MAILLAGE_2=mesh1,
                    PROJECTION='OUI',
                    RESULTAT=reslin,
                    TOUT_CHAM='OUI')
                    Project results to linear FE elements

IMPR_RESU(FORMAT='MED',
          RESU=_F(RESULTAT=unnamed,
                TOUT_CHAM='OUI'),
          UNITE=80)
          Save results to MED file

modes = CALC_MODES(CALC_FREQ=_F(NMAX_FREQ=5),
                  MATR_MASS=MASS,
                  MATR_RIGI=STIFF,
                  OPTION='PLUS PETITE',
                  STOP_BANDE_VIDE='OUI',
                  TYPE_RESU='DYNAMIQUE')
                  Calculate modal frequencies

modes0 = EXTR_MODE(FILTRE_MODE=_F(MODE=modes,
                                  TOUT_ORDRE='OUI'))
                                  Extract modal shapes

modesL = PROJ_CHAMP( MAILLAGE_1=mesh0,
                   MAILLAGE_2=mesh1,
                   PROJECTION='OUI',
                   RESULTAT=modes0)
                   Project modal results to linear FE

IMPR_RESU(FORMAT='MED',
          RESU=_F(RESULTAT=modesL,
                NOM_CHAM=('DEPL', ),
                NOM_CMP=('DX', 'DY', 'DZ')),
          UNITE=81)
          Save modal results to file

FIN()

```

Figure 2 – example of comm macro in order to run modal and linear static analyses on shells

So, after even rough look on the figure 2 – one could think – that the engineer that is performing i.e. modal shapes extraction would have to be also educated in computer programming. He would have to take care about proper variable naming (mesh, mesh1, mesh0 etc.) – which one was which? Take care about proper brackets assign, distinction between lists and reals etc. The could be easily covered by ANSA deck – where the formal consistency could be prepared, checked and managed. As this is the case for shells what are element coordinate systems? Are they rotated? Why only displacement components were written from modal results (without DRX, DRY, DRZ)? So many questions, but most of them comes from FE model preparation where the FE mesh were prepared in some external tools and the results would be viewed in other tools. Keeping all together (pre & post) in ANSA & META – for example – could bring answer for such questions or even suppress them. The example below shows an automotive headlamp modal analysis (based on presented macro file) with mesh prepared in ANSA.

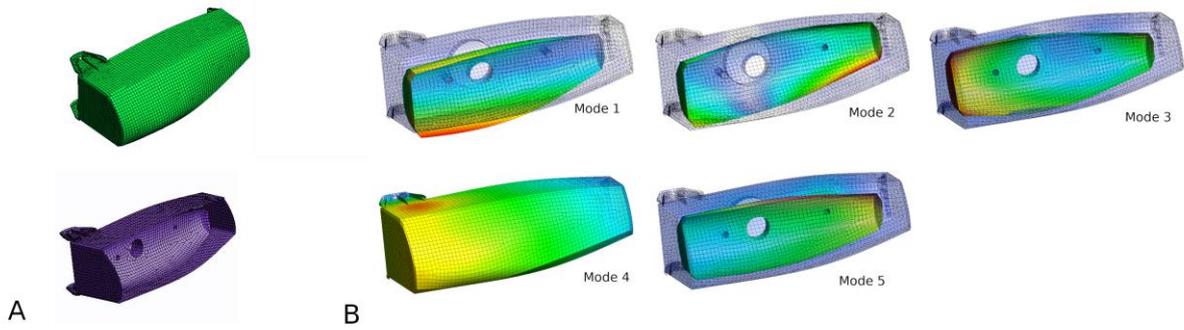


Figure 3 FE model of a lamp: A) mesh from ANSA; B) results review

In this case an additional parser was needed in order to prepare the *.comm macro automatically with recognized groups, and model definitions (see figure 4)

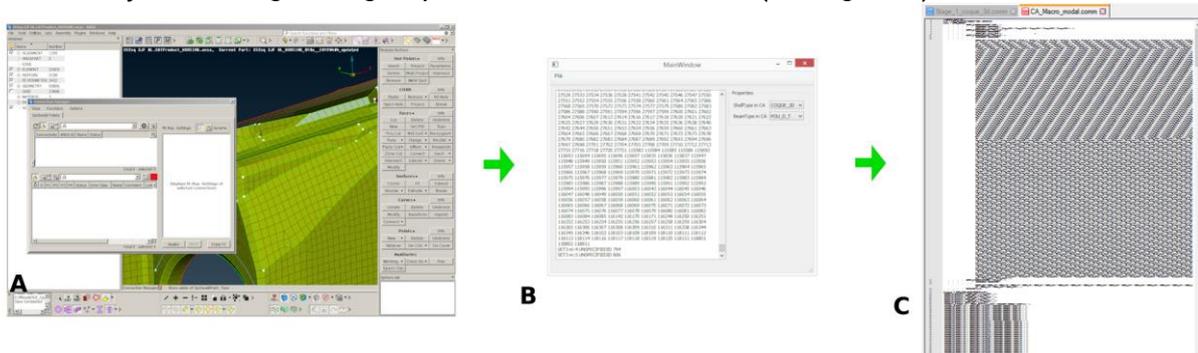


Figure 4 Model preparation to run in CA: A) ANSA pre-processor with FE model definition, mesh preparation and connection definitions using Nastran deck; B) simple NAS file parser for CA macro file preparation (auto translation); C) prepared *.comm macro – with large number of lines and variables

The same method was used in other project – for a car frame analyses. ANSA with the mid-surface capability could resolve meshing aspect as well as the connections (welds in this case). But the *.comm file – was a kind of challenge: how to deal with such big file?. When the CA deck would be available – this could be a big short-cut for simulation preparation and handling with automatic groups of elements, connections etc (see short overview on figure 5 and results on figure 6).

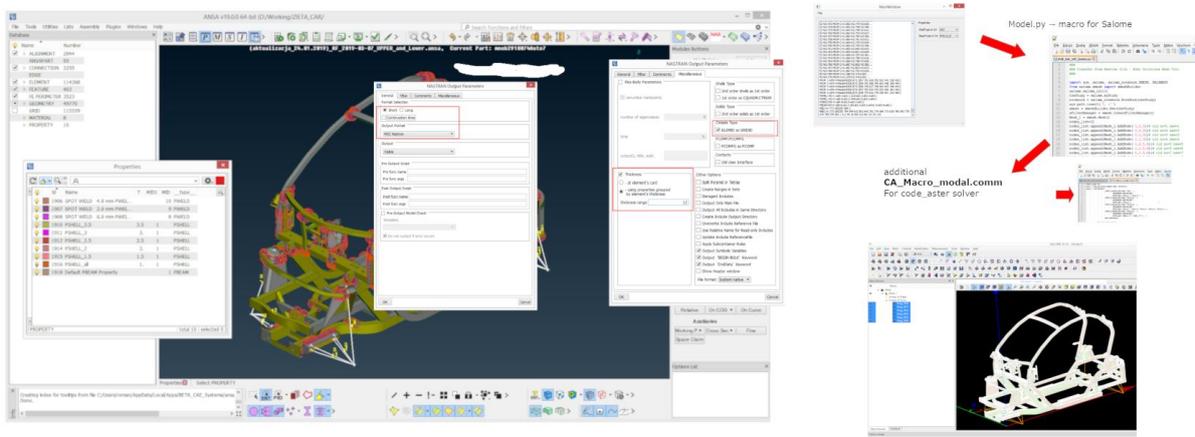


Figure 5 Example for welds including as stiff beams using PWELD, then translated to *.comm macro file

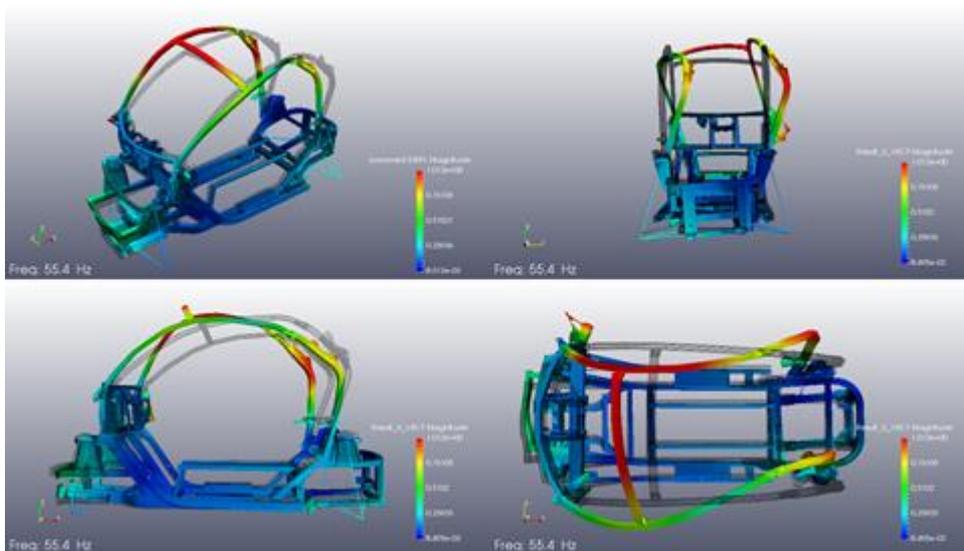


Figure 6 Obtained results from code_aster simulation

It may be noted that most of operation like FE element assignment & manipulation, boundary condition assignment, material assignment and many other – are done based on element and nodal groups. This is also pro – ANSA, where the groups could be easily constructed and maintained – especially when the mesh was updated or changed, and the nodes numbering has been changed. The problem of visibilities of a vector that is needed for solutions – i.e. shell normal, force direction or anisotropy used in material model -see example on figure 6.

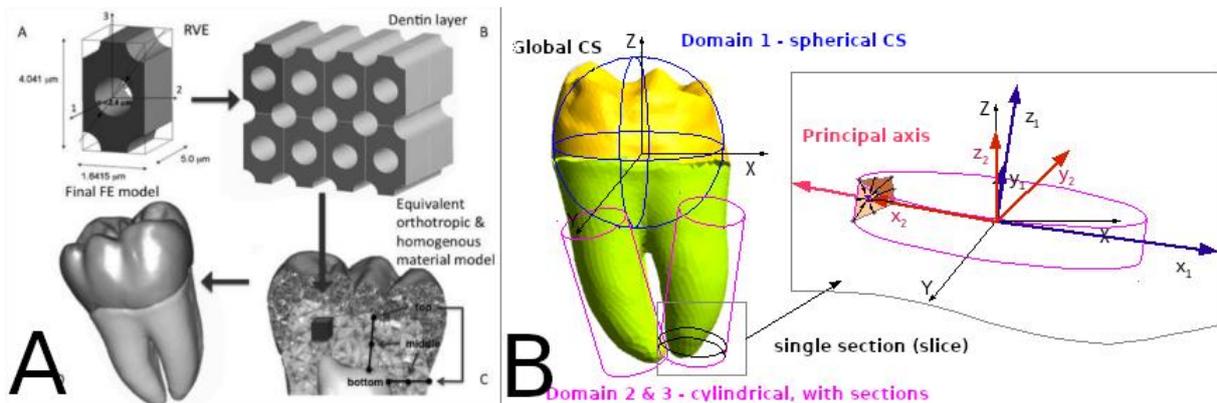


Figure 6 – Molar tooth analyses: A) inner tubules in dentin cause the anisotropy (2); B) that need to be included in FE model and somehow presented in order to check & review

Other case with usage of voxel modelling based on CT scans of human mandible. Each voxel with corresponding different material properties. In *.comm file it was required to define large number of materials and to assign it to the mesh – that means – to prepare a lot of groups, makes an assignment and the results handling (see figure 7).

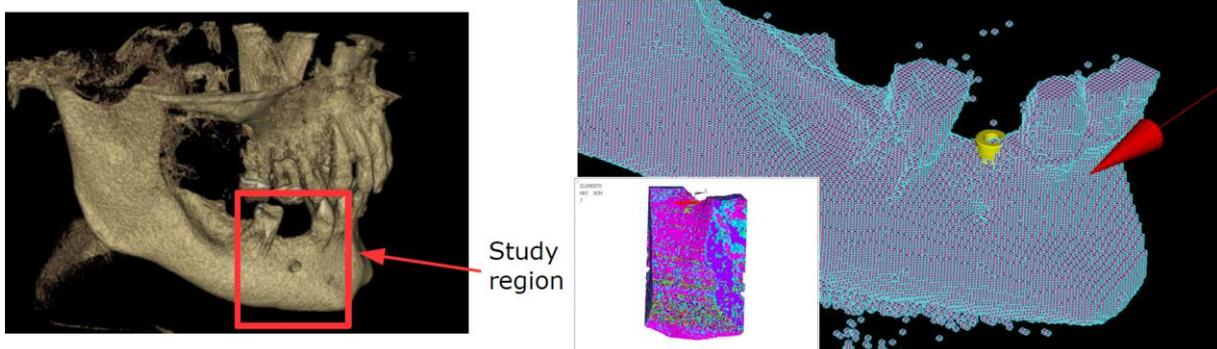


Figure 7 Voxel model with human mandible based on CT scans, problematic handling of large number of materials assigned to each voxel

Similar problem was with application of electromagnetic forces distributed around whole circumference for turbogenerator winding head (see figure 7). Once again we had to deal with hundreds of repeated lines of force vector assembly or – in the results extraction via real and imaginary part of a complex solutions.

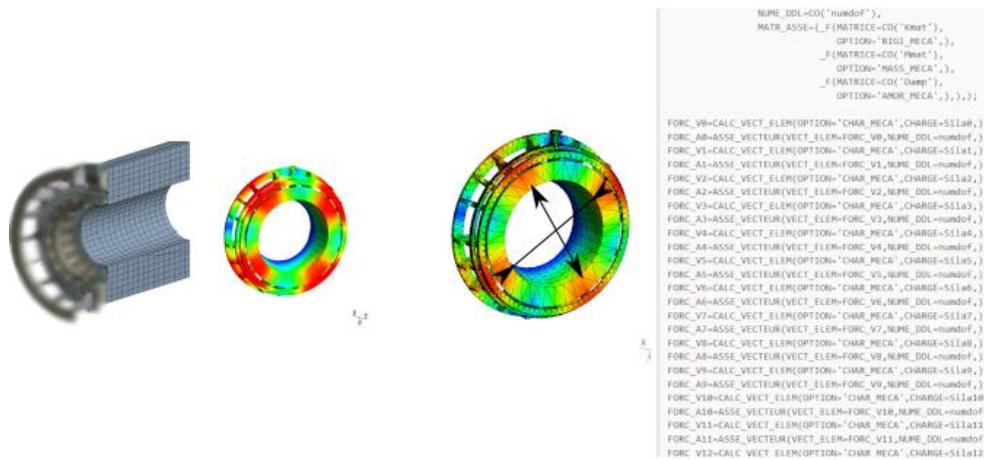


Figure 7 Assembly of force vector distributed across nodes at circumference of winding head of a turbogenerator, on the right shown also the part of *.comm file prepared for simulation.

Another example is used in rotordynamics modelling – for 1D models. As ANSA has very good algorithms for 2D mid-surface extraction – let’s say calculation, then the same could be applied for 1D beam modelling – with the automatic beam cross-section evaluation with equivalent bending and torsion stiffness as well as the mass distribution properties. Usually the input is either 2D documentation or 3D CAD model, then – how to efficiently construct an reasonable 1D model for shaft. The definition for several, or for hundreds of beam elements is somehow time consuming and often could be a source of possible mistakes. See example shown on figure 8.

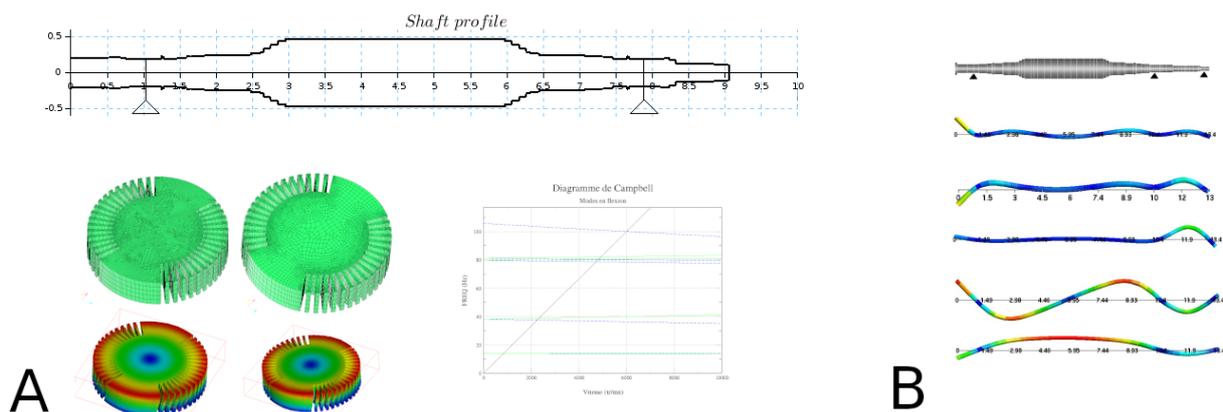


Figure 8 Application of possible functionalities for ANSA – automatic beam properties evaluation based on CAD 3D geometry; A) Used custom approach with several FE sub-models ofr each section; B) evaluated critirual speed with Campbell diagram (in CA)

3. CONCLUSIONS

Based on gathered experience in FE analyses and in CA usage – the big profit would be to have ANSA for pre-processing and META for post-processing of any model that would be calculated in CA. Non-efficient preparation of a FE simulation in CA due to macro

preparations, mesh preparation, mesh -to macro – consistency and many repeated commands seems to be easily realized with even basic ANSA functionalities. One point would be also interesting to have possibility to have interface for *.MED file type for mesh preparation as well as for post-processing.

REFERENCES

- (1) ANSA version 19.1.1 User's Guide, BETA CAE Systems, march 2019
- (2) W. Grzebieluch, R. Będziński, T. Czapliński, U. Kaczmarek, The mechanical properties of human dentin for 3-D finite element modeling: Numerical and analytical evaluation, Advances in Clinical and Experimental Medicine 2017