Practical Recommendations for Nonlinear Structural Analysis in DIANA

Kesio Palacio, July 2013

Abstract: This paper presents some practical recommendations for nonlinear structural analysis in DIANA.

After presenting the basic convergence issues, practical recommendations for both modelling considerations and analysis procedures of a structural finite element model in DIANA are described. Also, specific recommendations for the analysis of reinforced concrete and masonry structures are given.

Introduction

"The establishment of an appropriate mathematical model for the analysis of an engineering problem is to a large degree based on sufficient understanding of the problem under consideration and a reasonable knowledge of the finite element procedures available for solution. This observation is particularly applicable in nonlinear analysis because the appropriate nonlinear kinematic formulations, material models, and solution strategies need to be selected. [K.J. Bathe (1996) - Finite element procedures]"

Nonlinear analysis poses far more challenges than solving a system of equations in linear elastic regime because there is not one unique solution procedure that is suitable for solving all nonlinear problems. For the execution of linear elastic analysis in DIANA the user can usually rely on default solution procedures. However, for nonlinear analysis, an appropriate solution procedure must be selected. When the model definition is not ok or the solution procedures are not properly chosen, convergence issues may arise in nonlinear analyses. Also, convergence issues can be raised because the iterative solution method is unable to find a solution for the nonlinear problem. Basically, three types of convergence issues can be raised in DIANA:

- In nonlinear global equilibrium iterations

```
SEVERITY   : ABORT
ERROR CODE: /DIANA/AP/LB40/0053
ERRORMSG.A:   Divergence   occured,
iteration   method   failed.   Use
smaller   load   steps,   different
iteration procedure or different
control procedure.  Restart  from  a
saved  FILOS  file  or  rerun  the
complete job.
```

- In nonlinear local stress-return mapping iterations

```
SEVERITY   : WARNING
ERROR CODE: /DIANA/NL/LB41/0210
ERRORMSG.W:   Local   crack   stress
iterations did not converge in:
Element number              : 1414
Integrationpoint number : 7
Stress in main material = 2.1621 ,
stress in crack material= 2.0712 .
Stresses   in   main   and   crack
material  should  be  equal  in  the
converged situation.  If values are
not  equal  then  stresses  do  not
follow   the   softening   curve
exactly. Use smaller load-steps to
overcome this problem.
```

- Error in the iterative solution method

```
SEVERITY   : ABORT
ERROR CODE: /DIANA/NL/DU41/0042
ERRORMSG.A:   Division   by   zero,
value of ALFA     = 0.0000  in BFGS
secant   method.      Check    your
iterative procedure.
```

The first convergence issue is related to the iterative solution scheme (specified by the user) for the global equilibrium equations,

$$[K(U)]\cdot\{\Delta U\}=\{F_{ext}\}-\{F_{\text{int}}(U)\}$$

which is defined in the degrees of freedom in the nodes, $[K(U)]$ being the global stiffness matrix in function of the displacement vector $\{U\}$, ΔU incremental displacement vector, $\{F_{ext}\}$ is the external force vector, and $\{F_{\text{int}}(U)\}$ is the internal force vector which is dependent on the displacement vector $\{U\}$.

The second convergence issue is related to the iterative solution scheme (internally specified by DIANA) for the stress increments (return mapping), which is calculated at the integration points.

Finally, internal errors in the iterative solution methods may also be raised.

When convergence difficulties are found in nonlinear structural analyses, usually the solution procedures must be updated to find a solution. Also, the model set-up shall be checked, since wrong or not appropriate modelling definitions (e.g.

elements with connectivity issues, lacking of supports, etc) may exacerbate the nonlinear analysis.

Modelling Considerations

Many of the issues (errors in boundary conditions, element coupling, irregular element shapes, etc) can be identified simply by performing a linear elastic analysis and other are exclusively related to the nonlinear analysis behavior itself.

Perform first a linear elastic analysis

A good method to check the model definition is first to perform a linear elastic analysis of the model. Check whether the analysis results (deformed shapes, stress and strain fields) are realistic, i.e., whether the structural behavior of the model, for the loading level applied, is found within the bounds of its structural response. Also, check the solution status of the linear elastic analysis in the analysis output (*.out) file and see if there are negative eigenvalues (NP) or singularity problems (SD). This information is provided right after the sum of external loads, as for example:

```
SPARSE: DIM=1658 NNZ(MAT)=15096
NNZ(LU)=124788
NR. OF NEGATIVE PIVOTS : NP=1
DECOMPOSITION EXECUTED:
DIM=1658 SD=1.50e-007 HD=3.09e+007
SOLVE: REDUCTION RES=0.46E-14
(INIT. RES=0.22E+08) NI=    1
```

Where:

DIM = dimension of the stiffness matrix
SD = smallest diagonal term in the stiffness matrix
HD = maximum diagonal term in the stiffness matrix

In this example, a negative pivot was found (NP=1) in the Sparse solver, meaning that the analysis model is unstable (existence of bifurcation points or rigid body motions). Also, the SD is approaching zero. If $(HD/SD)>10^{14}$, the system stiffness matrix in DIANA is classified as singular (existence of rigid body motion or mechanisms), in which for the example is verified.

Usually, when such problems are raised in linear elastic regime, the model is **not** properly constrained. In this case one should check the supports applied to the model.

Check input data affecting nonlinear behavior

- Mesh input
  - Mesh density

  DIANA evaluates the strain and stress fields at the integration points. Thus, mesh regions undergoing plastic deformation or cracking require a reasonable number of integration points in order to properly capture the distribution of such nonlinearities.

- Material input
  - Stress-strain curves with no stress gradient

  Usually such specification leads to numerical difficulties for the nonlinear analysis because, once the yield point is reached, there is no gradient of stress, i.e., the stresses remain constant while the strains vary, causing convergence difficulties.

  - Very different stiffness parameters in the same material

  This may lead to numerical difficulties (global stiffness matrix susceptible to ill-conditioning) because of large variations in the magnitude of the diagonal stiffness terms; for example, large stiff elements being connected to small less stiff elements.

- Geometry data input
  - Integration scheme for shell models

  The default integration point scheme for shell and beam elements may not be suitable for the nonlinear analysis, since the stress distribution across the shell thickness or beam height cannot be described properly by the definition of three integration points across the thickness. In such cases the user may explicit use more integration points across the thickness of the shell or beam.

  - Interface local axes

  The correct definition of the interface element local axes, specifically the normal axis, is very important since they set the orientation of interface element results. If the interface local axes point in the opposite direction of the continuum element local axes, then the interface element results will be oriented with opposite signs. Such condition can be a problem, especially for interfaces with material nonlinearities. For the tangential directions, unless explicitly defined, the interface behaviour in shear is independent of the numerical sign of the shear components. However, for the normal direction, usually the interface behaviour is different for tension and compression states.

- Loads input
  - Point loads

Point loads such as forces are suitable for line elements models (beams, truss, springs, etc). In solid and surface models (plane strain, plane stress, shells, etc), point loads usually cause stress singularity. Such conditions when associated with local failures, usually lead to converge difficulties. In these cases it is advised to introduce a loading plate or distribute the point load on a number of elements.

Analysis Procedures

When solving structural nonlinear analysis, the correct choice for:

- Load incrementation procedure
- Iterative solution method
- Convergence criteria

are fundamental for maintaining the solution of the analysis on the equilibrium path.

Load Incrementation Procedures

The analysis of a nonlinear finite element model falls into the solution of a system of nonlinear equations, for which an incremental-iterative solution procedure must be applied. Also, contrary to linear elastic analysis, more than one solution may exist. Thus, to guarantee the valid solution, the concept of incremental procedures is used.

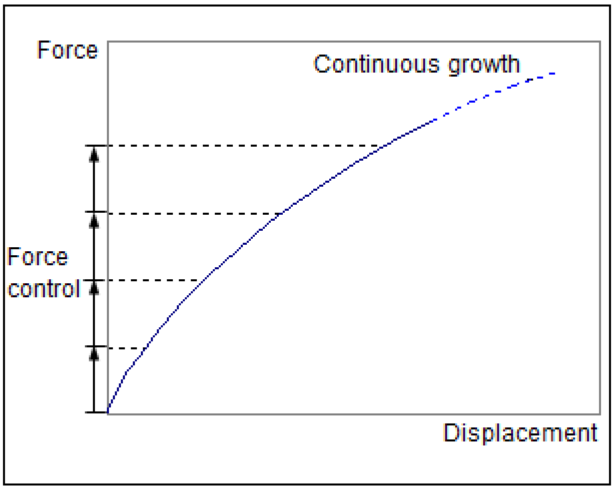
Depending on the shape of the equilibrium path, i.e., on the structure response curve, three types of load incrementation can be listed:

1) Force control

Loads (base motions, forces, thermal, gravity, etc.) are incrementally applied. Pure load control analyses are applied to models with continuous force increasing, i.e., without softening behavior and limit points. Consequently, if the analysis model experiences softening, or limit points are encountered, the pure load incremental procedure does not lead to a solution when a load is applied that is higher than the load capacity of the model.

The analysis procedures for the execution of it in DIANA are given, as for example:

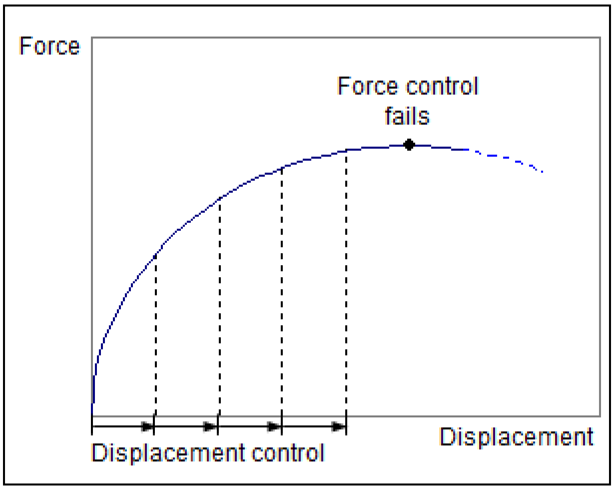
```
BEGIN LOAD
 LOADNR= loadset with load forces
 STEPS EXPLIC SIZES 0.1(5) 0.05(10)
END LOAD
```



Force control

2) Displacement control

Prescribed displacement in the direction of a degree of freedom of a reference point on the structure is incrementally applied. Displacement control analyses are suitable for models with snap-through behaviors.



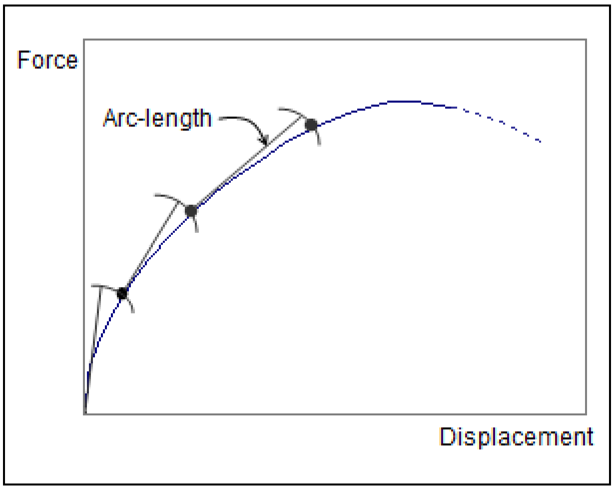
Displacement control

The analysis procedures in DIANA for the execution of models with prescribed displacements are given, as for example:

```
BEGIN LOAD
 LOADNR= loadset with prescribed
         displacements
 STEPS EXPLIC SIZES 0.1(10)
END LOAD
```

3) Arc-length control

When force control or displacement control cannot be used, arc-length control may be the solution for the load incrementation. Analyses with arc-length control covers all equilibrium paths.



Arc-length method

In the arc-length method, the norm of the incremental displacements is constrained. Depending on the type of constrain, different arc-length methods can be formulated. In DIANA two methods are available:

- Spherical path arc-length method
- Updated normal plane arc-length method

Note that in arc-length control the applied load factor vary from load steps defined by the user because step size does not apply to the applied load but to the combination of force and displacement.

As an example, below the application of the external load is controlled according to the spherical path arc-length method:

```
BEGIN LOAD
 LOADNR= loadset with force loads or
         prescribed displacements(1)
 STEPS EXPLIC SIZES my_steps
 ARCLEN SPHERI
END LOAD
```

(1) With reference to the type of external load, from DIANA 9.4.4 force or prescribed displacement is supported. For earlier versions, only force is supported.

4) Arc-length method with indirect displacement control

The arc-length method described previously includes all displacements. Such strategy is adequate for analysis models with global failure behaviors, but for structures that fail locally, which are typically from concrete structures, the arc-length method can perform better if only a part of the displacements are considered in calculating the arc-length. Within this context, DIANA offers two options:

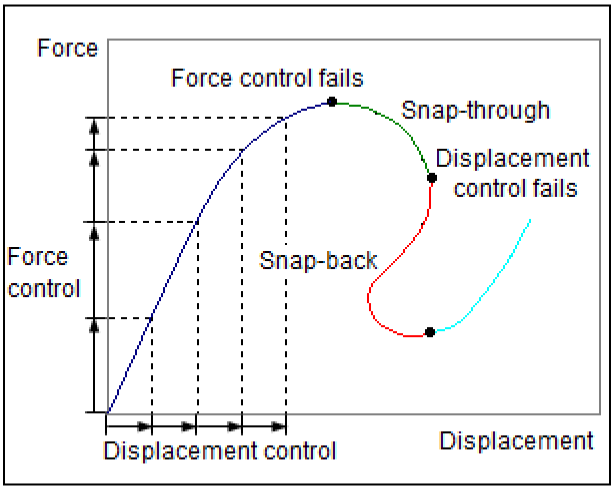
- Regular indirect displacement control
- Crack Mouth Opening Displacement (CMOD)

An example of commands with the arc-length method with regular indirect displacement control can be as follows:

```
BEGIN LOAD
 LOADNR= loadset with force loads or
         prescribed displacements(1)
 STEPS EXPLIC SIZES my_steps
 BEGIN ARCLEN
    SPHERI
    BEGIN REGULA
     BEGIN SET
      NODES 1504
      TYPE TRANSL
      DIRECT 2
      ALPHA 1.0
     END SET
    END REGULA
   END ARCLEN
END LOAD
```

(1) Available from DIANA 9.4.4

In summary, the suitable choice of the incremental procedure depends on the shape of the equilibrium path. Displacement control offers great advantage over load control, providing better conditioned tangent stiffness matrix and the capability of passing limit points where load control fails. However, the method is usually restricted to structures with one point load only and fails to trace snap-back behaviors. To overcome these drawbacks, load control with arc-length is of choice.



Failures points in load and displacement controls

5) Automatic incrementation procedures

In the incremental procedures discussed previously, the number of steps along with the step sizes are specified by the user. DIANA also offers automatic incrementation procedures, in which both the number of steps and the corresponding step sizes are automatically computed, the so called adaptive loading methods. In DIANA there are three methods:

- Iteration based automatic step size control
- Energy based automatic step size control
- Automatic step size control

In these procedures, basically an attempt is made not only to provide a more general and robust incrementation strategy but also a method aiming at a better chance for the iterative solution method to find an initial equilibrium state that will lead indeed to convergence.

Iterative Solution Methods

After the determination of the load incrementation, which is nothing more than the linearization of the nonlinear problem in the form:

$$[K_T(U_t)] \cdot \{\Delta U\} = \{F_{ext}(U_t)\} - \{F_{\text{int}}(U_t)\}$$

an iterative solution method shall be used to find a displacement increment $\{\Delta U\}$, being $[K_T(U_t)]$ the tangent stiffness matrix in function of the displacement vector $\{U_t\}$ at step *t*, so that the equilibrium condition:

$$\{F_{ext}(U_{t+\Delta t})\} - \{F_{\text{int}}(U_{t+\Delta t})\} = 0$$

is satisfied. The $U_{t+\Delta t}$ is total displacement at step $t + \Delta t$ and is defined as

$$U_{t+\Delta t} = U_{t+\Delta t} + \Delta U$$

Thus, the solution of the nonlinear system equilibrium equations is achieved using an incremental-iterative solution method.

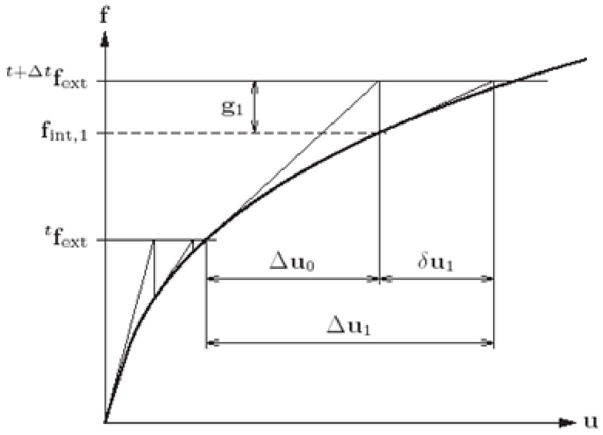
With reference to iterative solution methods, the following are available in DIANA:

- Regular (or full) Newton-Raphson
  The tangent stiffness matrix is derived at every iteration.

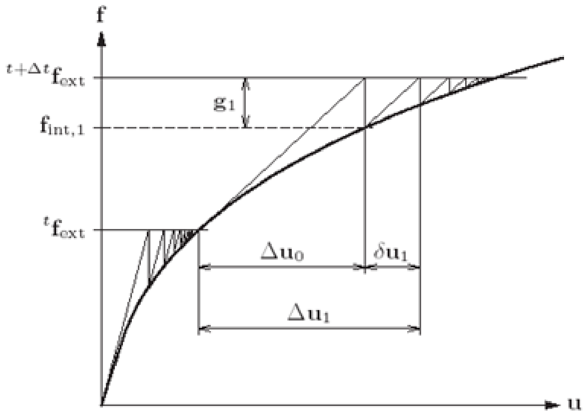
- Modified Newton-Raphson
  The tangent stiffness matrix is derived at the start of every load increment.

- Secant methods (Quasi-Newton methods)
  Secant stiffness matrix is derived.

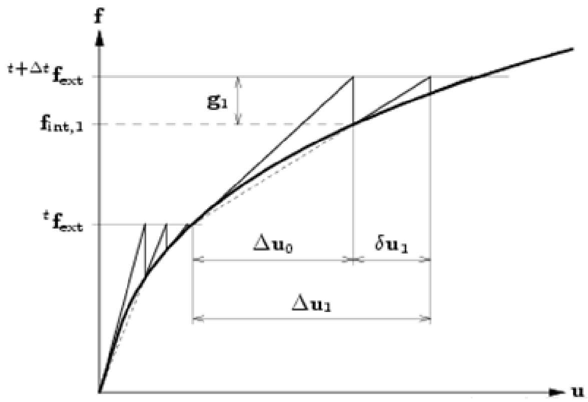
- Linear or constant stiffness iteration method
  Linear elastic stiffness matrix is used.



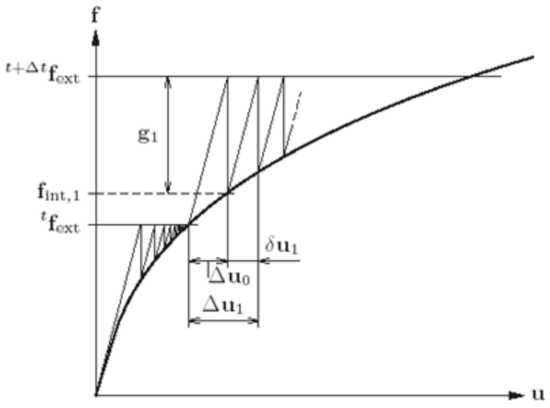
Newton-Raphson method



Modified Newton-Raphson method



Secant method



Linear Stiffness method

In general:

- Overstiff stiffness matrices lead to a slow convergence but they are stable
- Exact tangent stiffnesses will lead to a fast (quadratic) convergence but might become unstable
- Matrices which are too soft lead to divergence
- Regular Newton-Raphson requires less iterations but the computations are relatively time consuming
- Modified Newton Raphson and Linear Stiffness method require more iterations but one single iteration will be faster, due to less effort in setup and decomposition of the tangent matrix.

Convergence Criteria

The accuracy of the solution in a nonlinear analysis, i.e., how close is the solution to the equilibrium path, is measured by convergence criteria. Thus, the selection of appropriate convergence criteria and the corresponding tolerances is of high importance. Too tight tolerances may result in unnecessary iterations while a loose tolerance may provide incorrect solutions.

In DIANA there several ways to control the convergence:

- ► Force norm
  The ratio of the out-balance forces of non-constrained degrees of freedom of the previous iteration and the first predication of that step is used. It is the most demanding norm criterion.
- ► Displacement norm
  The ratio of the iterative displacement increments of the previous iteration and the first predication of that step is used.
- ► Energy norm
  The ratio of the work of the internal forces and relative displacements of the previous iteration and the first predication of that step is used.

Generally speaking, the specification of convergence criteria, and the corresponding tolerances, for the analysis of civil engineering structures depends on the experience of the analyst with the nonlinear problem. In general:

- Sensitive geometrically nonlinear analyses require a tighter tolerance than predominantly material nonlinear analyses in order to maintain the solution on the correct equilibrium path.
- Analysis models with lots of prescribed displacements make the norm displacement criterion less effective, whereas the norm force is less effective for structures that can expand freely where hardly any internal forces can be built up.
- If applicable, it is advisable to use the combination of the three norms (force, displacement, and energy) simultaneously. In DIANA this is done with the specification of SIMULT command.

Nonlinear Analysis of RC and Masonry

From our experience in dealing with analysis models that include cracking and/or interface material models, usually such analyses require some experience or good-feeling of the user. When convergence issues are found, basically the following steps can be followed to identify the convergence problem:

Step 1 - Check the nonlinear analysis results

When divergence or convergence difficulties are raised, check the results of the last converged step and see whether they are realistic. If the analysis is performed with prescribed displacements or load control without arc-length, check if the results correspond to a failure of the analysis model, i.e., the model cannot withstand more loading. If applicable, trace the load-displacement diagram and see how the structural response develops (usually, as the degradation of the stiffness matrix progresses, convergence difficulties increase).

If the analysis results do not correspond to a failure of the model and assuming that the model definition is ok, the convergence problem may be simply because the adopted incremental-iterative solution scheme cannot find a solution. In this case, check the analysis output file (*.out) to get a better understanding of the analysis history. If it is a convergence issue, adopting a different incrementation strategy and/or the iterative solution method may circumvent the convergence problem.

Step 2 - Check the analysis output file (*.out)

During the execution of the analysis, DIANA writes in the analysis output file not only information about performance and warning/error messages but also relevant information about the nonlinear analysis, as for instance:

- Stiffness matrix information (dimension, smallest and highest diagonal terms, etc)
- Information on model instability (Number of negative eigenvalues - existence of bifurcation points)
- Log summary on cracking and plasticity
- Information on convergence (number of iterations, convergence status, load factor, etc)

Step 3 – Change the solution procedures

If the adopted incremental-iterative solution scheme fails to reach convergence or simply diverge, change one or more aspects of the incremental-iterative solution scheme:

- Load incremental procedure
  In general structural models with cracking or interface material models fails locally, and thus the application of the **arc-length method with indirect displacement control** is a good load incremental procedure for such analysis problems.

  With reference to increment sizes, if explicit load increments are employed, try to apply smaller steps. If automatic incrementation procedure is used, adapt the control parameters. Also, another option is to use more than one incremental-iterative solution scheme for the same load case, which in DIANA can be achieved by adding a new EXECUT block.

- Iterative solution method
  In general, start with regular Newton-Raphson. If it fails or become unstable, apply one of the Quasi-Newton methods, starting with BFGS or Broyden. Additionally, different approaches for the evaluation of the tangential stiffness can be applied to both Newton and Quasi-Newton methods in DIANA.

  And finally, **line-search techniques** can be applied to enlarge the radius of convergence of the iterative solution method.

- Convergence criteria

  Generally speaking, the application of convergence criteria in cracking analyses may be split into parts: before and after the peak load. Tight convergence criteria may be possible to satisfy up to the peak load (or limit point) and, after that, solutions may become difficult to obtain. With reference to the norms, after the peak load, the displacement norm may become less meaningful since the displacements tend to be large.

Step 4 – Use the continuation strategy for a non-converged step

Solutions of nonlinear analysis may be difficult to be obtained for all the steps of a load case using just one single incremental-iterative solution procedure, which in DIANA is done under an EXECUT block. In some cases, it may be better to split the load incrementation process in two or more parts so that different iterative solution methods can be applied. Aiming at this solution strategy, DIANA offers the "Save/Restore" steps option.

Suppose that in an analysis the first three load steps of a load case are performed successfully and, at the fourth step, convergence is not reached.

To save time, one can perform in DIANA, with a new command file, the analysis from the last converged step (in this case, step #3). In order words, there is no need to execute the steps that were already successfully solved. To do this in DIANA we specify "SAVE CONVER" in the current command file (to save the filos file of the analysis of the last converged step) and build up a second command file with "RESTORE" steps option to execute the analysis from the non-converged step. Resuming:

Task I - Save the filos file of the last converged step

When specifying in the analysis procedures of the current command file, check on "Last converged" option under "Save steps" in Execute tab:

```
BEGIN EXECUT
   SAVE stepnumbers to be saved or
        last stepnumber, or all or
        last converged step
END EXECUT
```

Task II – Create a new command file

For this, set "Evaluate model" and "Specify nonlinear effects" off under "Model" and "Type" tabs, respectively. Finally under "Execute" tab, select "Start steps" and then under "Steps" sub-tab check on “Execute start steps” to specify the number of the last converged step:

```
*NONLIN
 MODEL OFF
 TYPE OFF
 BEGIN EXECUT
   …
   RESTORE stepnumber for restart
 END EXECUT
*END
```

Task III- Perform the analysis with the new command file

With the new analysis procedures defined, perform the analysis.