DeepLines
Training Course

PRINCIPIA

ifp Energies nouvelles

Convergence Hints

Introduction

- The main purpose of this presentation is to provide some practical guidelines to help solving usual convergence troubles

- Solving convergence troubles requires at least basic knowledge of solution algorithms used for static and dynamic analyses

- There is unfortunately no magical modeling technique which would ensure convergence in all situations but rather series of modeling hints based on experience

Contents

Introduction

- What does “convergence” mean ?

- Convergence is reached when the system has reached the minimum level of internal energy

- This translates into the virtual works principle as follows:

$$\delta W = \vec{F}\cdot\vec{\delta V} + \vec{M}\cdot\vec{\delta\theta} = 0$$

Permissible translation (↑ $\vec{\delta V}$)

Permissible rotation (↑ $\vec{\delta\theta}$)

Introduction

- The convergence criteria in DEEPLINES ensures that the previous requirement is met
  - Convergence on forces and moments is first looked at : comparison of out-of-balance forces and moments with maximum force and moments
  - Convergence on displacements and rotation is looked at once convergence on forces and moments is achieved
- Convergence of both forces and displacements ensures that the minimum energy of the system was found

Introduction

- The solver aims at solving something like

$$\overline{\overline{K}} \cdot \vec{X} = \vec{0}$$

where X denotes the vector of ALL nodes coordinates and K the global stiffness matrix

- The system is solved through iterations using a “gradient” type method called Newton-Raphson : X0, X1, … , Xi , Xi+1…

- Forces and moments increments are checked at each iteration until the convergence criteria are satisfied

Line segmentation

- There is a widespread practice to use very refined meshing from the early stage of the analysis to properly capture high curvature levels and get accurate results
- But this is rather a widespread mistake…
- Reasonable number of elements (coarser mesh) must be used at the start of your projects and refined depending on the accuracy of your preliminary results
- Refine the segmentation only once convergence is achieved

Line segmentation

- Using too many beam elements significantly increases the calculation time

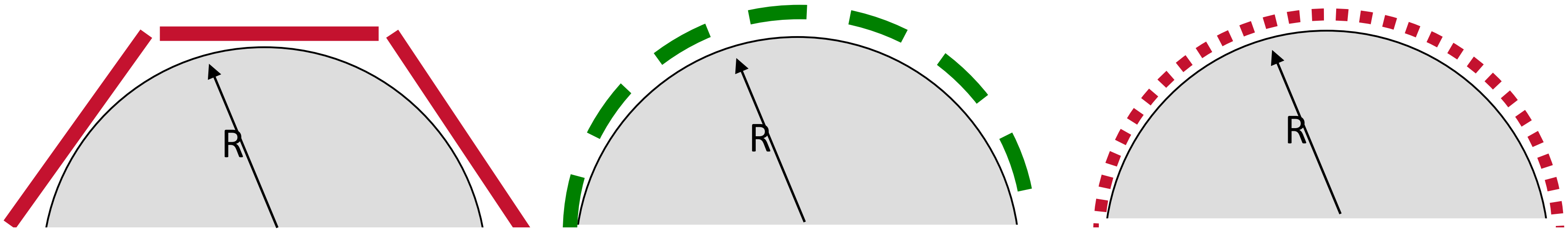
- Too much refined segmentation may also jeopardize convergence of your calculation

- Very fine elements “don’t like” large rotations and solving associated displacements requires more iterations in the algorithm

- Transition zones are required between elements having large lengths ratio

Line segmentation

- "Reasonable" meshing depends on target curvature radius (at least)

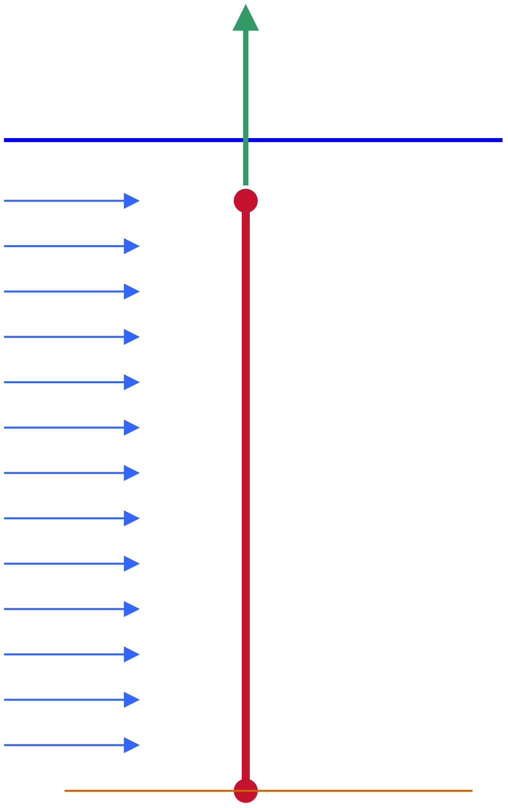


- About 8 beam elements per half circumference is generally enough

$$L \approx \frac{\pi . R}{8} \approx 0{,}4 \times R$$

Line segmentation

- Appropriate line segmentation also depends on tension gradient along the line, and external loads along the line

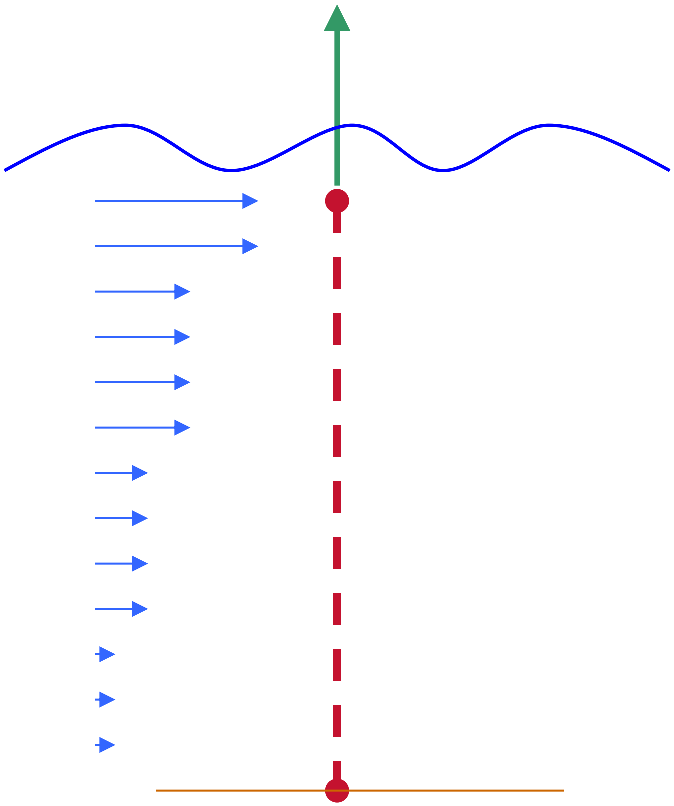


- Top tensioned riser
  - Limited deflection
  - Constant top tension
  - Constant current
  - Static analysis

> 1 beam element

Line segmentation

- Appropriate line segmentation also depends on tension gradient along the line, and external loads along the line



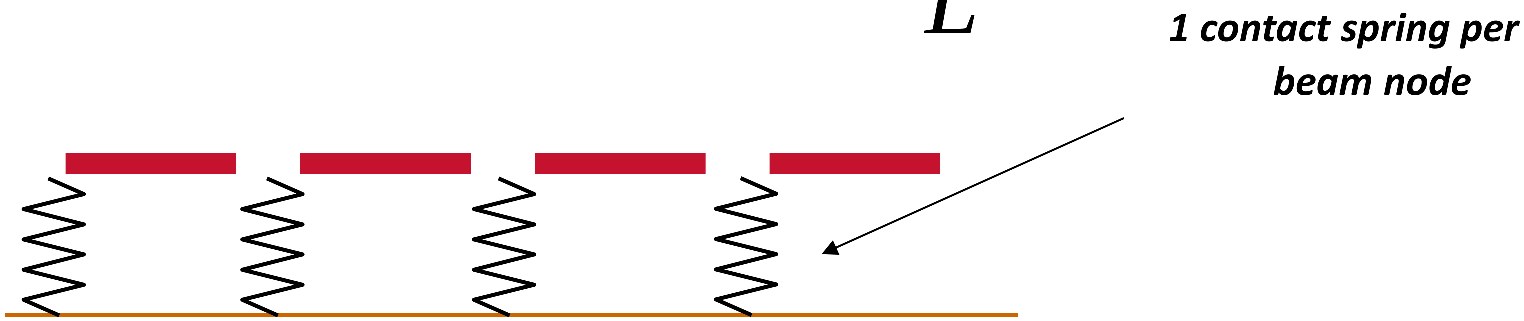
- Top tensioned riser
  - Significant deflection
  - Variable loads
  - Sheared current
  - Dynamic analysis

> Several beam elements

Line segmentation

- Line segmentation must also be set to properly catch the contact with surfaces when needed
- The density of contact elements can be defined as a starting point equal to :

$$D \approx \frac{1}{L}$$



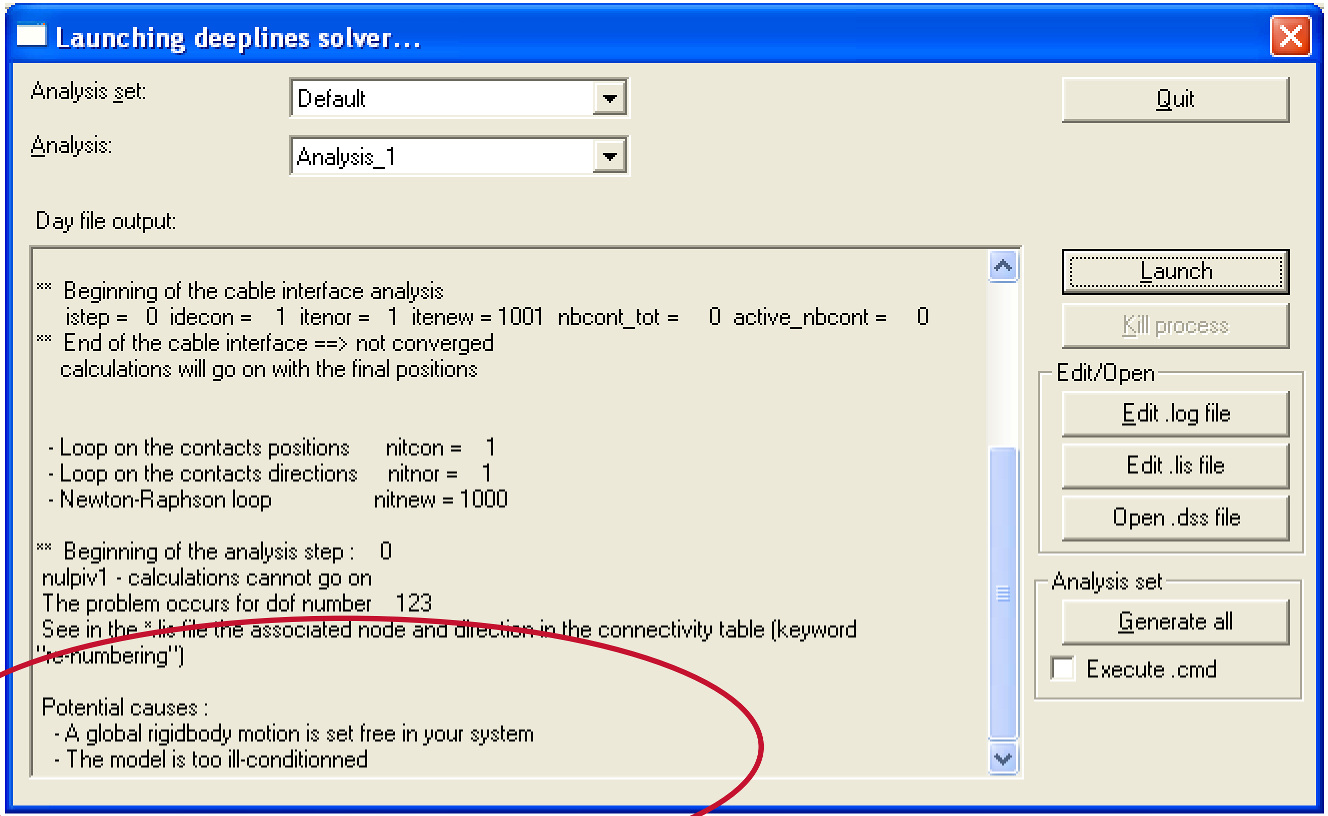
- Large elements (10m and up to 30m) can be used on flat seabeds with limited dynamics

Line segmentation

- Refine the segmentation where needed only
- Avoid elements with lengths of 0.01m at the end nodes
- Elements length of 1m is generally uselessly refined
- Use about 5 / 6 elements along bend stiffeners as a starting point
- Typical number of elements for a free hanging line is about 100 - 150

« Ill conditioned » models

- The solver may sometimes report “Ill conditioned” messages

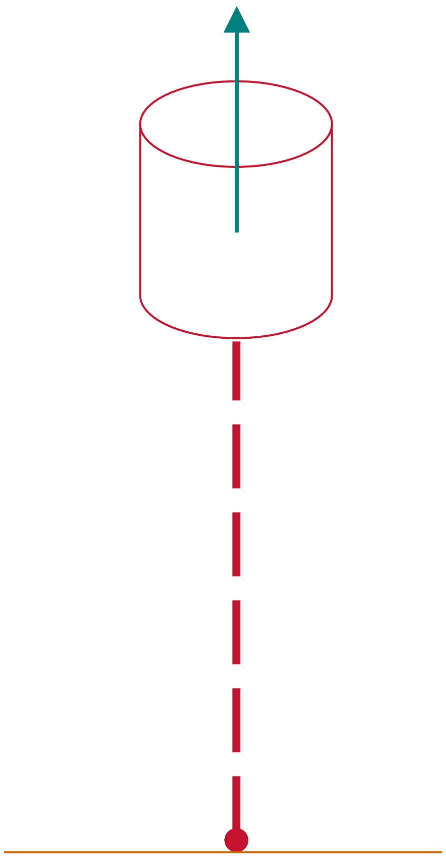


« Ill conditioned » models

- This message indicates that the static equilibrium cannot be found for your system
- The FE solver reports it is unable to invert the stiffness matrix
- This generally corresponds to situations where an infinite number of static solutions is possible due to insufficient boundary conditions
- This may also happen with systems with very low stiffness for some connections

« Ill conditioned » models

- Typical example



- A Buoy object (6 DOFs) is connected to a straight cable with a pin
- The cable is modeled through Cable/Chain type elements (3 DOFs per node)
- The bottom node of the cable is anchored with a pin
- Insufficient constraints at the connection node : the buoy is left free to rotate about the Z axis, i.e. infinite number of solutions

« Ill conditioned » models

- Workaround solutions :

- Check that all boundary conditions are correctly defined and consistent to avoid infinite number of solutions

- Even with all boundary conditions correctly defined, some systems with very low stiffness at the connection nodes may however still lead to “ill conditioned” messages

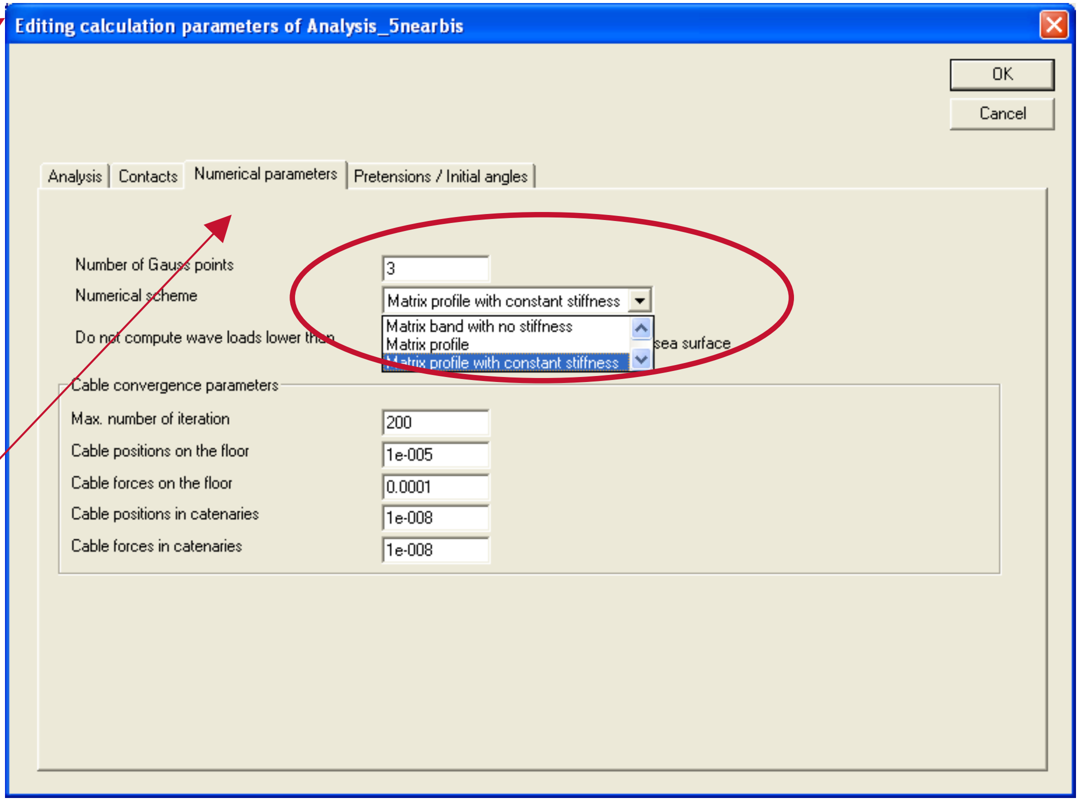
- The next workaround is to add artificial stiffness in the stiffness matrix to help the solver find the solution

« Ill conditioned » models

- Use “matrix profile with constant stiffness” instead of “matrix profile” to add artificial stiffness in the stiffness matrix

Calculation parameters window

Numerical parameters tab



Hints for static analyses

- Numerical parameters for static analysis

Cable interface

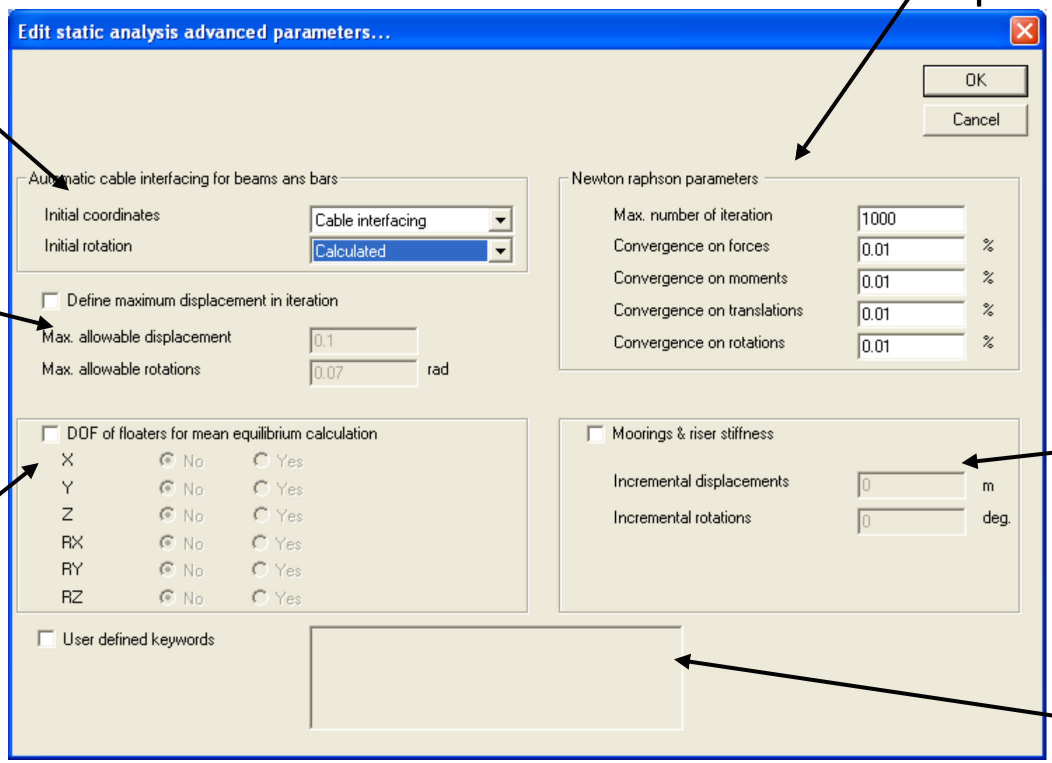
Bound increments

Coupled analysis for vessels

Newton Raphson parameters

Output of mooring file

User defined keywords



Hints for static analyses

- The calculation is converged when all convergence criteria on forces, moments, displacements and rotations are met

$$\frac{F_{i+1} - F_i}{F_{Max}} \leq \varepsilon_F \qquad \frac{M_{i+1} - M_i}{M_{Max}} \leq \varepsilon_M$$

$$\frac{X_{i+1} - X_i}{X_{Max}} \leq \varepsilon_X \qquad \frac{\theta_{i+1} - \theta_i}{\theta_{Max}} \leq \varepsilon_\theta$$

- The calculation stops when the iteration number reaches the maximum allowed value
- Default value for ε is 0.01%. Acceptable upper bound is about 0.5%

Hints for static analyses

- Convergence status echoed in the *.LIS files

F_{Max} $\quad$ $M_{i+1} - M_i$ $\quad$ M_{Max} $\quad$ $F_{i+1} - F_i$

Edit C:\Affaires\PRD_G64541004_Hotline\TECHNIP\2008-09-23\Analysis_5nearbis_Dynamic\Analysis_5nearbi...

Fast keyword access: HEADER — Cancel

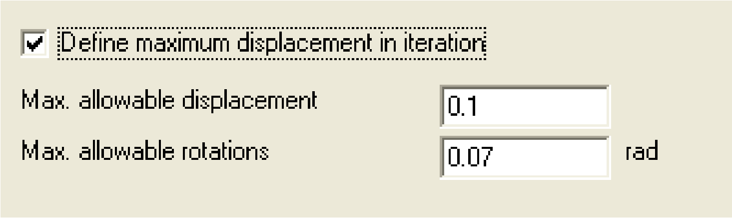
** Beginning of the quasi-static analysis **

load step nr. 0

itera	qmax1	fmax1	noeud	dir.	qmax2	fmax2	noeud	dir.
1	0.878E+08	0.161E+09	112	1	0.586E+08	0.697E+08	189	5
2	0.861E+08	0.158E+09	275	1	0.575E+08	0.692E+08	189	5
3	0.848E+08	0.156E+09	275	1	0.565E+08	0.688E+08	189	5
4	0.848E+08	0.156E+09	275	1	0.565E+08	0.688E+08	189	5
5	0.824E+08	0.151E+09	275	1	0.549E+08	0.682E+08	352	5
6	0.823E+08	0.151E+09	275	1	0.548E+08	0.682E+08	352	5
7	0.822E+08	0.151E+09	275	1	0.548E+08	0.681E+08	352	5
8	0.817E+08	0.150E+09	275	1	0.543E+08	0.680E+08	352	5
9	0.789E+08	0.144E+09	112	1	0.523E+08	0.673E+08	352	5
10	0.767E+08	0.140E+09	112	1	0.508E+08	0.669E+08	352	5
11	0.764E+08	0.139E+09	275	1	0.505E+08	0.668E+08	352	5
12	0.757E+08	0.138E+09	275	1	0.501E+08	0.666E+08	352	5
13	0.756E+08	0.138E+09	275	1	0.500E+08	0.666E+08	352	5
14	0.730E+08	0.133E+09	112	1	0.482E+08	0.661E+08	352	5
15	0.729E+08	0.133E+09	112	1	0.481E+08	0.661E+08	352	5
16	0.719E+08	0.131E+09	112	1	0.473E+08	0.658E+08	352	5
17	0.700E+08	0.127E+09	112	1	0.461E+08	0.654E+08	352	5

Hints for static analyses

- Preventing numerical oscillations can be done by setting upper bounds for incremental displacements and rotations



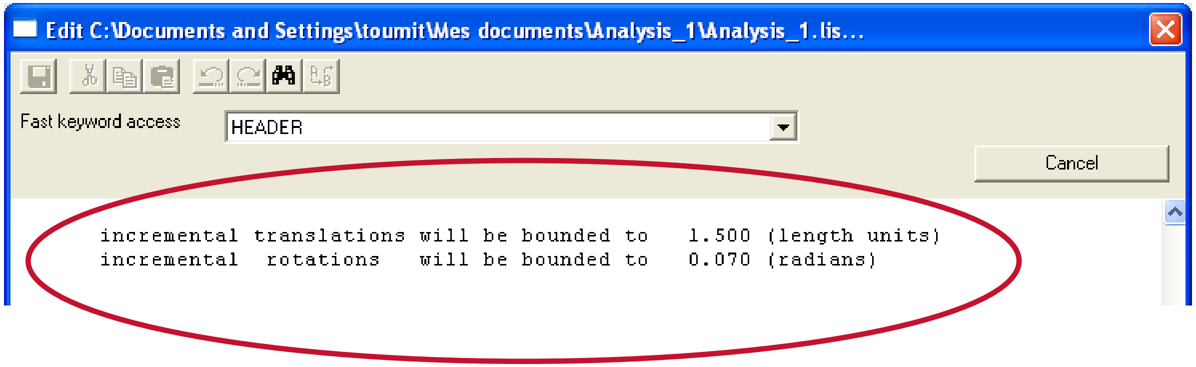
$$\left\| \overrightarrow{X_i X_{i+1}} \right\| \leq 0.1m$$

$$\left\| \overrightarrow{\theta_i \theta_{i+1}} \right\| \leq 0.07rad$$

- Setting bounds will act as a damping and help stabilize oscillations
- The maximum number of NR iterations could be increased to ensure that the total displacement necessary to reach the static solution is still achievable

Hints for static analyses

- Reference values can be found in the *.LIS file by searching for “bounded”



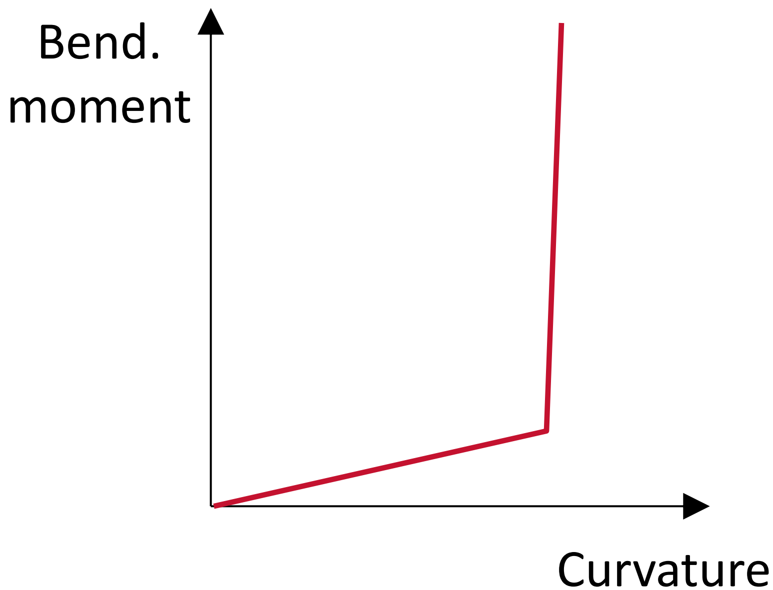
- Numerical oscillations can be observed by looking at the *.LIS file : the incremental force oscillates between successive values without decreasing significantly

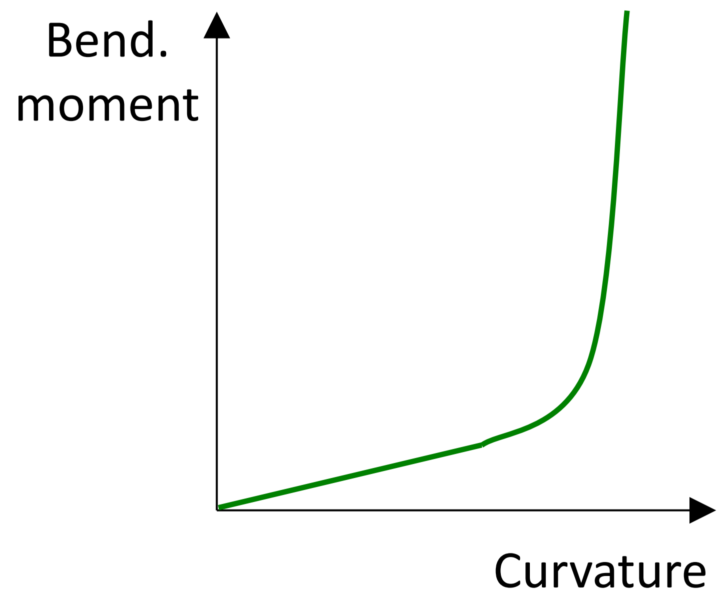
Hints for static analyses

- Numerical oscillations can also be seen with the GUI
- Non converged solutions can be displayed in the 3D View window
- This requires to save these non converged solutions :
- Add the keyword “ *DATABASE
  1 ”
- to create a *DTBI file similar to the *DTBS files

Hints for static analyses

- Avoid as much as possible sharp changes of stiffness along the lines to avoid “stiff” points
- Avoid sharp changes of stiffness within non-linear curvature – bending moment relationships (e.g. bend restrictors)





Hints for static analyses

- Use the Cable/Chain segment type when the bend stiffness is negligible instead of setting very low values in the Flexible segment type

Hints for quasi-static analyses

- Quasi-static analyses are series of static analyses solved one after the other

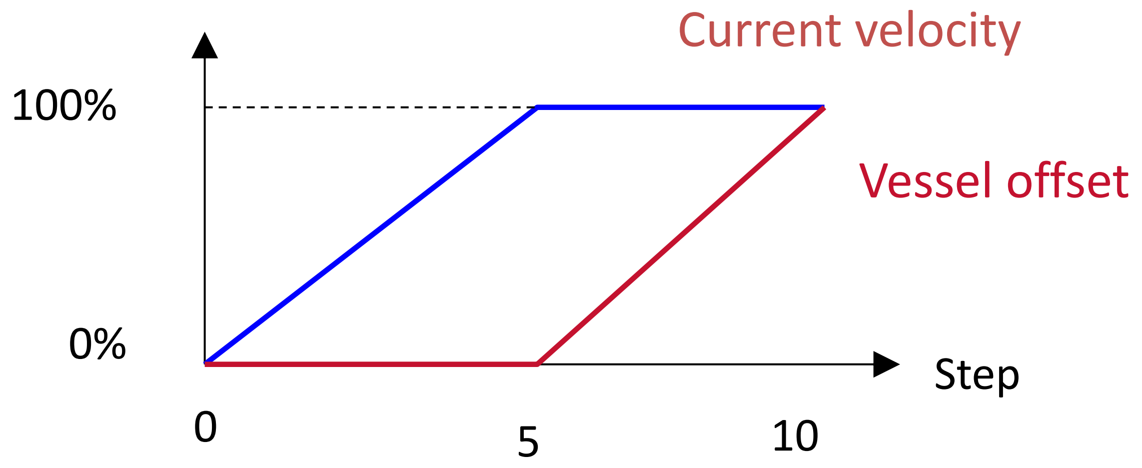
- Quasi-static analysis involves variable loads or displacements without inertia effects

- Quasi-static analyses are characterized by the number of static steps (number of problems to solve) and loading tables (percentage of loads or displacements as a function of step number)

- The same convergence hints than with “pure” static analysis apply

Hints for quasi-static analyses

- Incremental loads due to currents or incremental displacements must generally be applied progressively to ease convergence
- Use 10 steps if the solver fails finding the solution within 5 steps,…
- Incremental current and displacements may be applied using separate loading tables :

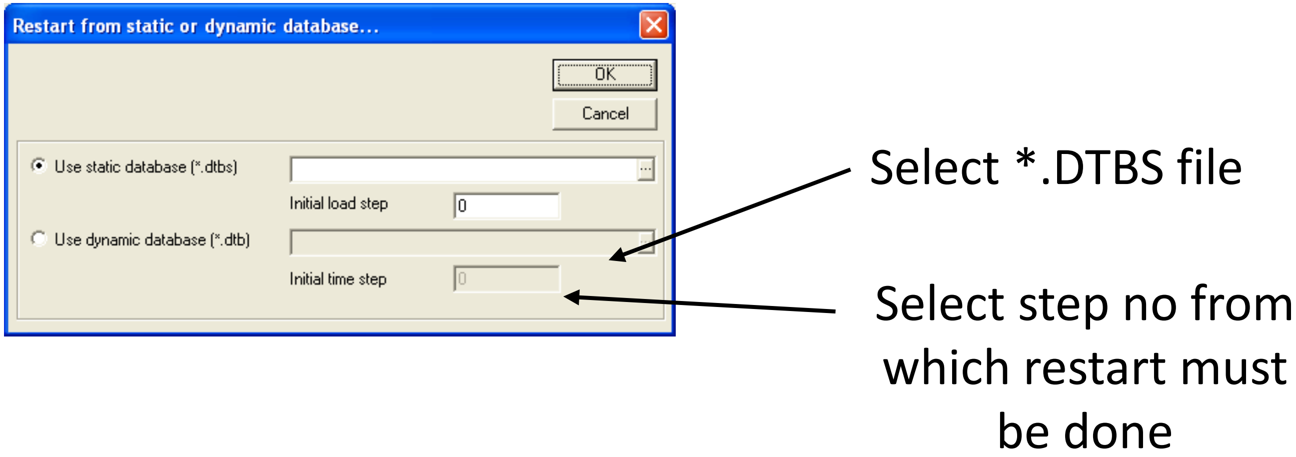


Hints for quasi-static analysis

- You could fix unstable objects to help static analysis convergence and then release the objects at a further step
- Change the boundary conditions as a function of step with the *QCDCSTAT keyword
- For instance : maintain a MWA fixed for step 0 to help the riser finding their equilibrium and then set the MWA arch free at step 1

Hints for quasi-static analysis

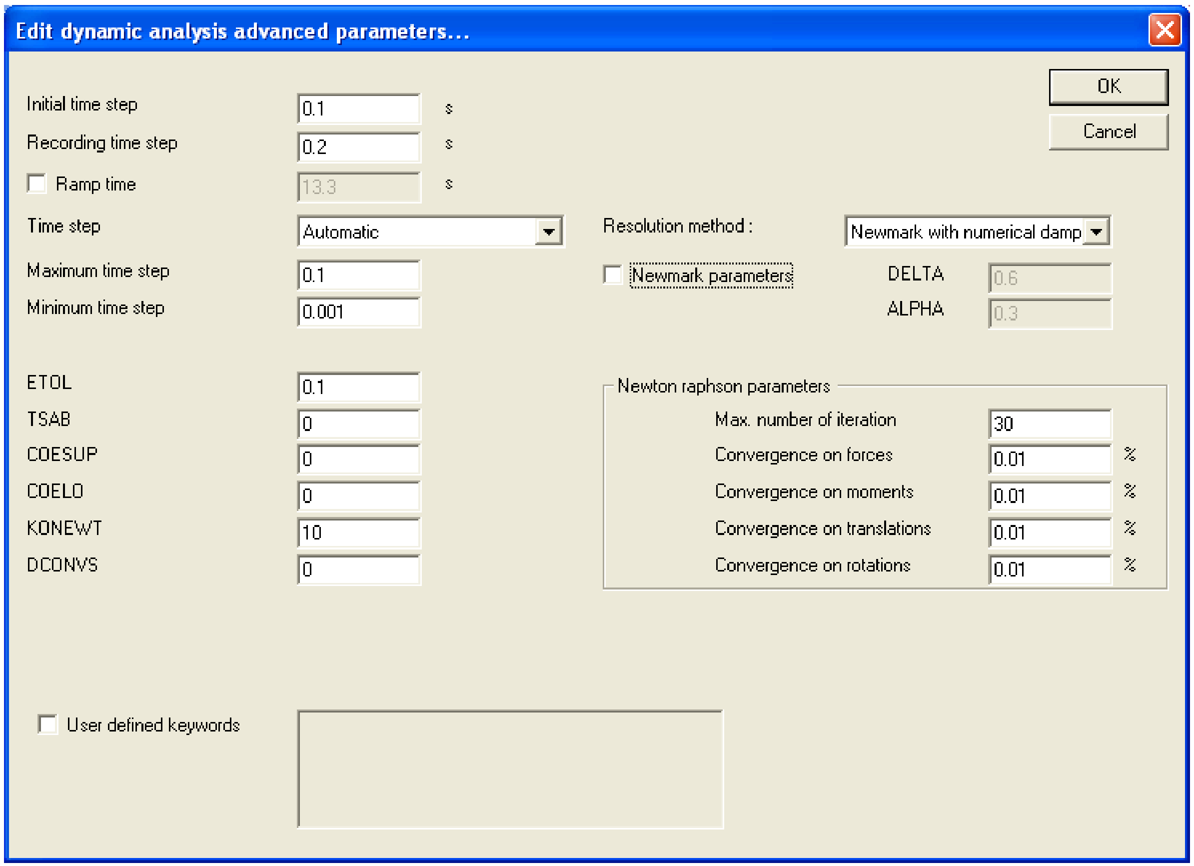
- Use the restart facility to focus on non-converged steps within the current trial and save calculation time



- For example :
  - The quasi-static analysis goes well until step 7 and then fails to converge
  - Copy the analysis and tune the numerical parameters adequately
  - Run this new analysis while using the restart facility at step 7 on the previous analysis

Hints for time-domain dynamic analyses

- Numerical parameters



Hints for time-domain dynamic analyses

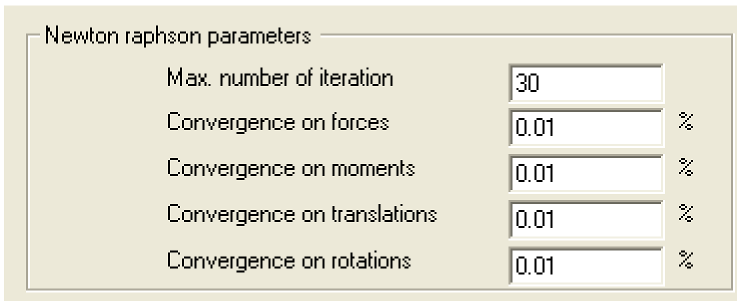
- Time-domain solutions are calculated with the implicit Newmark integration scheme

- Convergence at every time-step is ensured through iterations based on similar principles than those for static analyses (Newton Raphson)

- Error tolerances must be consistent with the error tolerances used for the static part (use similar of higher values for the dynamic part of the simulation)

Hints for time-domain dynamic analyses

- Default Newton Raphson parameters are
  - Max No of iterations : 30
  - Error tolerances : 0.01% (can be increased up to 0.5%)

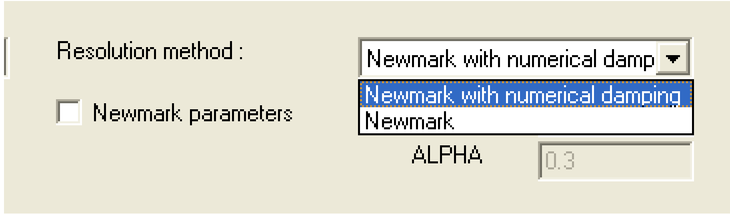


- The effect of line segmentation is even more noticeable for dynamic analyses

- The use of proper time-steps and integration scheme may have a significant impact on convergence and calculation time

Hints for time-domain dynamic analyses

- Default Newmark integration scheme is suitable for most applications

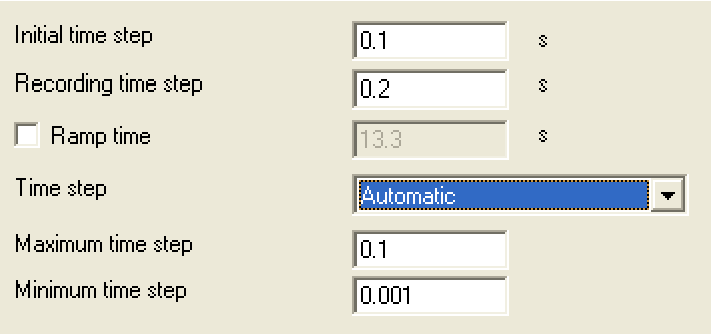


- This integration scheme introduces numerical damping (negligible for most situations)

- Newmark without numerical damping must be used for decay tests analyses (set also Delta & Alpha values 0.5 and 0.25 to cancel residual damping)

Hints for time-domain dynamic analyses

- Both constant and adaptative time-stepping options are available



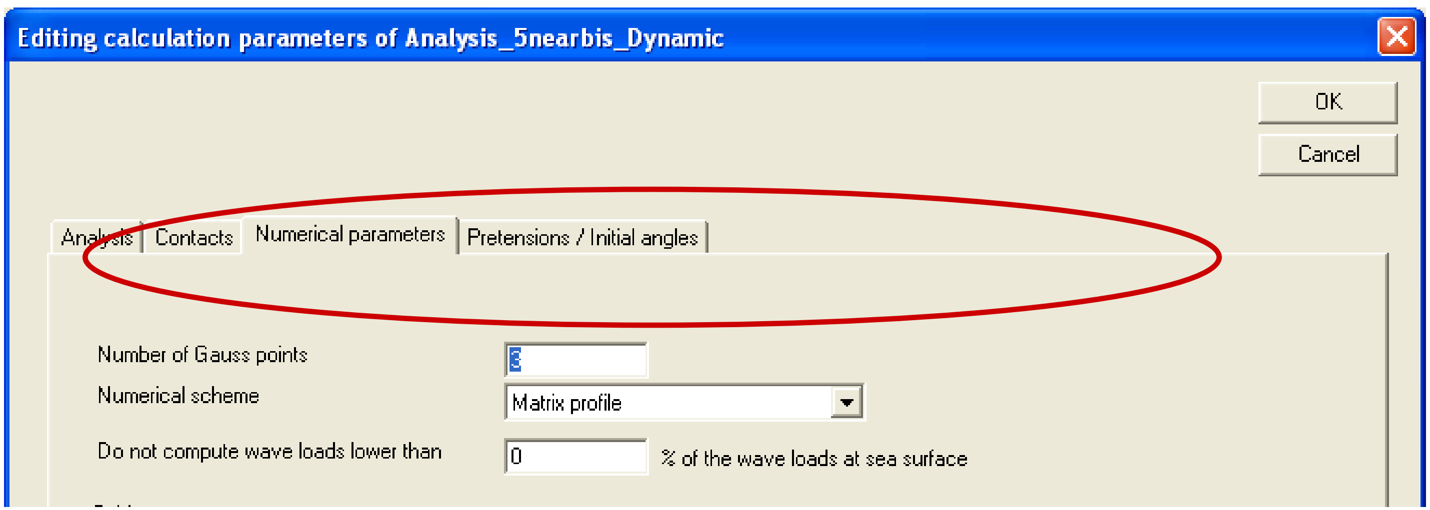
- Always use the default parameters as a starting point

- Reduce the default time-step if required
- Use a constant time-step too avoid too many time-steps changes during the simulation

- Increase the ramptime with very high waves

Hints for time-domain dynamic analyses

- Simulations with irregular waves often results in large calculation time
- This can be improved by reducing the number of wave components from 200 to 100
- Wave kinetics may also be neglected far from the sea-level



Hints for time-domain dynamic analyses

- Mechanical damping set along flexible risers may sometimes alter convergence

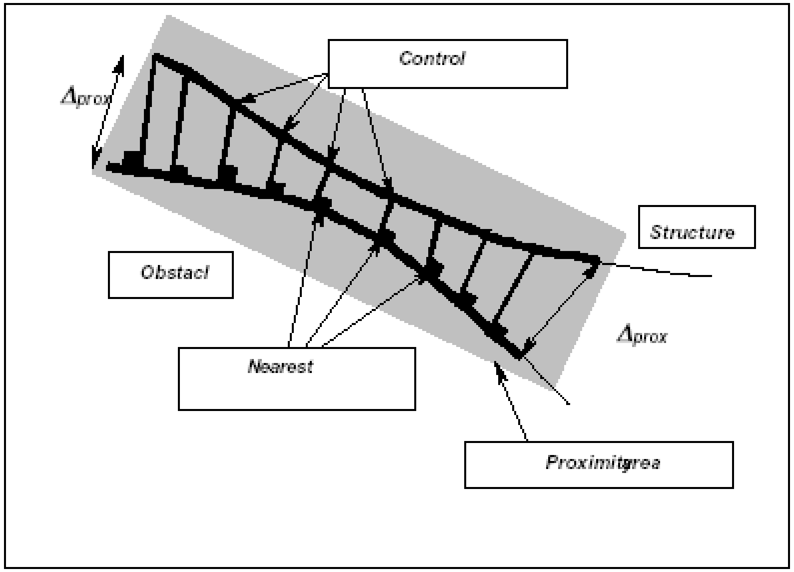
- Dynamic analyses may fail when damping ratio varies too much along the same line (typical case is when the damping is left to zero under the bend stiffener whereas it is non zero along the rest of the line)

Hints for time-domain dynamic analyses

- Flexible risers may exhibit high compression in the TDP area due to seabed friction forces
- This compression may result in buckling i.e. unstable configuration and large displacements that make convergence more difficult
- It is advised to increase the friction force mobilization distance (from 0.0001m to 0.001m) to smooth the friction loads
- Use the restart facility when trying to solve dynamic convergence troubles

Contact convergence issues

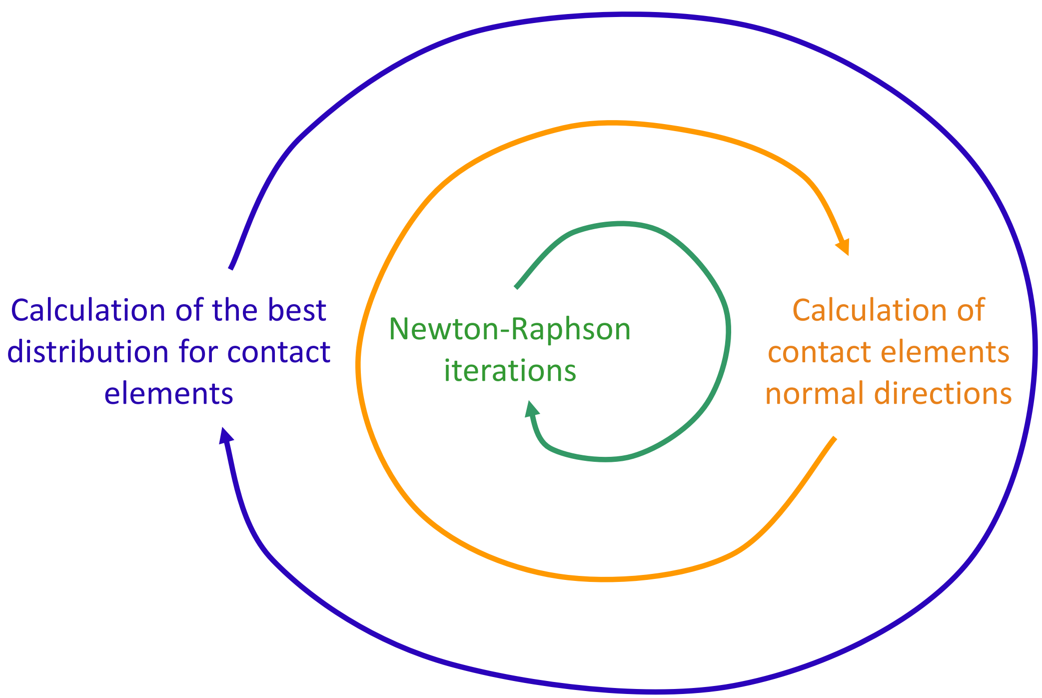
- Automatic detection of contact zones (criteria on proximity zone)
- Several contact elements are distributed along the contact zone
- An iterative algorithm is used to find out the best position for contact elements during the simulation



- Note: more details can be found in OTC paper ref. 14157 (2002)

Contact convergence issues

- An iterative algorithm with 3 loops to find out the equilibrium position



Without any contact, std value for Max number of iterations is 200 in static (*NEWTONSTAT keyword)

With contact, std values are:
Max iterations = about 10
Max iterations = 1< * <5
Max iterations = 20< * <50

Finally total nb of NR iterations is between 10*1*20 and 10*5*50

Iterations and sub-iterations are visible in the .DAY file.

Contact convergence issues

- Calculation progress is output in the .DAY file
- Check convergence from the .DAY file
- Example from a quasi-static analysis :

```
**  Beginning of the analysis step :    0
     istep =    0  idecon =    1  itenor =    1  itenew =   16  nbcont = 2222
     istep =    0  idecon =    2  itenor =    1  itenew =    1  nbcont = 2222
**  End of the analysis step :    0 ==> converged -   17 iterations

**  Beginning of the analysis step :    1
     istep =    1  idecon =    1  itenor =    1  itenew =    2  nbcont = 2222
**  End of the analysis step :    1 ==> converged -    2 iterations

**  Beginning of the analysis step :    2
     istep =    2  idecon =    1  itenor =    1  itenew =   18  nbcont = 2222
     istep =    2  idecon =    2  itenor =    1  itenew =    1  nbcont = 2280
**  End of the analysis step :    2 ==> converged -   19 iterations
```

Qstatic step nb

Iterations on contact position (external loop)

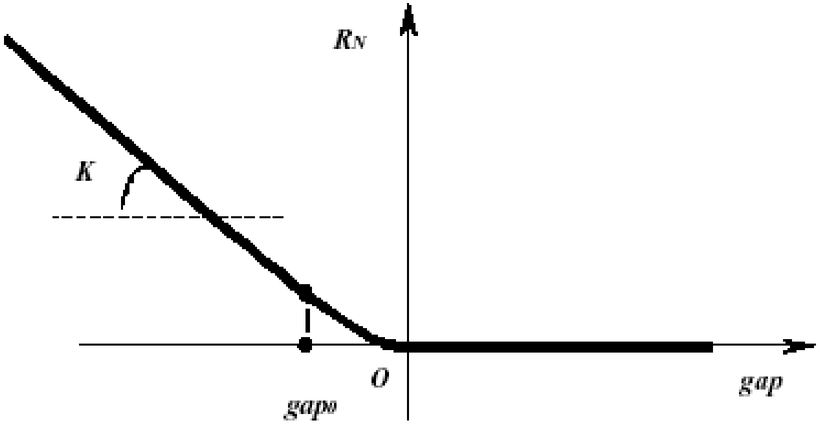
Iterations on contact normal directions (intermediate loop)

Newton-Raphson iterations (internal loop)

Number of contact elements currently used

Contact convergence issues

- Normal reaction force (Rn)



- Rn is defined as

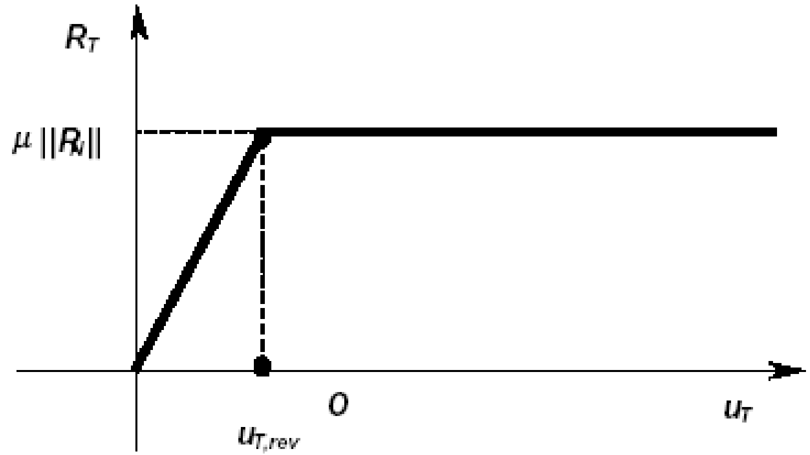
$$Rn = K.d.Dx$$

with Dx is the normal penetration, d the density of contact elements, and K the user-defined stiffness in N/m (a stiffness k = K/d is actually used)

- The use of k instead of K ensures the penetration of the pipe in the seabed Dx does not depend on the density of contact elements d

Contact convergence issues

- Tangential reaction force (Rt)



- Axial and lateral friction forces follow a regularized Coulomb law
- The friction mobilization distance is function of the pipe/soil interface (max values are about 1-3 mm for very soft clay)
- User is given full control on the friction mobilization distance

Contact convergence issues

- Contact modeling optimization depends on the configuration There is no practical hints that works for all cases, however :

- Use ‘compatible’ meshes for the objects that would experience contact (with +/- identical sizes for beams and triangles elements).
  - If triangles size >> beams size:
    contact will tend to concentrate on sharp edges (highly localized normal reaction forces, and poor resolution for friction forces).
  - If beams size >> triangles size:
    useless computations (projection algorithms) due to too much refined surface mesh.

Contact convergence issues

- Avoiding too much large displacements could sometimes help to ensure a correct ‘Contact Zones’ detection

- Generally, you should reduce the max number of iterations in the Newton-Raphson loop (contact elements are not updated in this loop) or limit the max allowed displacement

- With highly dynamic configurations, update your contact elements more often (reduce max iteration in the intermediate and inner loops), and increase the criteria on contact zones detection

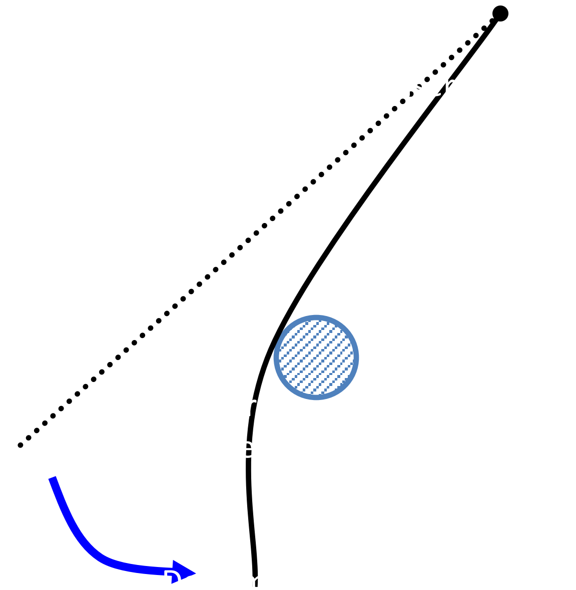
Contact convergence issues

- Use the default values for contact parameters as a starting point
- With friction, the lower UTrev the more difficult it is to reach convergence (as tangential reaction forces become more sensitive to low initial displacements)
- Stiff soil results in low values for UT, rev
- Increased density of contact elements results in improved accuracy but require more CPU time.
- With dynamic configuration, increase the minimum length (axial) for contact zones, to spread contact elements over a larger zone

Contact convergence issues

- Particular case of 'perpendicular' external riser/riser contact

- Highly localized contact supposes:
  - Use high CRITPROX value to detect the contact zone in case the lines dynamics is important
  - Increase the value for the minimum contact zone length
  - High density of elements is recommended to get the correct deformed shape (radius of curvature in the contact zone)



Contact convergence issues

- Initial line configuration in the GUI
- If the Bottom node is linked to a ‘Ground_connect’ on the seabed, contact is not perfect in the vicinity of the Bottom node: OD of the pipe must be taken into account
- Contact elements normal reaction force depends on the distance between the seabed surface and the pipe external surface
- Example of contact using a ‘Ground_connect’ (‘Ground_connect’ nodes always belong to the seabed). Artificial bending moment and curvature are induced at the extremity