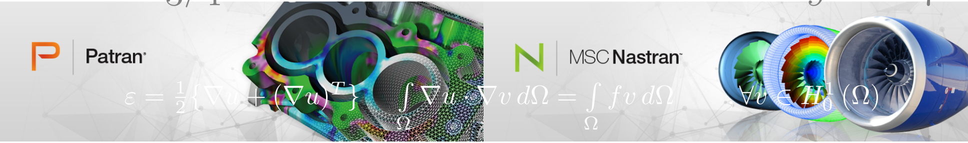
Patran / Nastran

Lecture 3/4 March 29th 2017



Insa Toulouse
Tp Génie Mécanique
Ingénierie des Systèmes

Julien Le Fanic

Lectures Scope

1. Lecture 1 deals with basics Finite Elements Method and introduces NASTRAN and PATRAN softwares. A cantilever beam is studied in linear elasticity and then with geometrical non linearity. If time left students can realize another exercise defined in appendixes §D.
2. Lecture 2 deals with plates and shells. A 2D plate with a hole is studied to assess a K_T. Then buckling modes are computed for the same plate under compressive load. Finally a GUYAN static reduction is performed.
3. Lecture 3 will let students finish Lecture 2 case studies before an assessment of a time dependent response for a beam and a contact 3D modelization.
4. Lecture 4 deals with FSM idealization.

Table of contents

Table of contents

Introduction to theory of time dependent problem

Idea

Transient analysis deals with the computation of the response of a structure submitted to loading (which can be time dependent) during elapsed time $[0, t]$.

Two numerical approach families to assess a transcient response of a structure

1. Modal analysis
   Eigenmodes are built $[\omega_n, \varphi_n]$ with ω_n eigenvalues i.e. eigenfrequencies and φ_n orthogonal eigenmodes

$$u(x,t) = \sum_n f_n(t)\varphi_n(x) \qquad (64)$$

   Time scale and space scale are uncoupled.

2. Time step by step integration
   Classical Finite Elements Method for space scale. Time scale depends upon space scale (e.g. for stability of numerical scheme as NEWMARK's ones). Of utmost importance is the knowledge of an upper bound of ω_n.

The choice of the approach often driven by the number of modes of interest.

Mpc & Rbe in Nastran

Mpc

- `MPC` means Multipoint Constraints i.e. enforced relations between degrees of freedom
- `MPC` to be called by a case control `MPC` = command
- `MPC` can be merged in a `MPCADD` Nastran card
- `MPC` useful to maintain distance between nodes
- `MPC` to be considered mainly between coincident nodes

Mpc & Rbe in Nastran

Rbe2

- `RBE2` is a element from Nastran rigid body elements family particularly suitable to dispatch an enforced displacement.
- one dependent node / multiple independent nodes
- `RBE2` is rigid

MPC & RBE in NASTRAN

RBE3

- RBE3 is a element from NASTRAN rigid body elements family particularly suitable to dispatch a force/-moment
- RBE3 works as a bolt groups : RBE3 computes a centre of gravity and dispatch resultant to master nodes
- weighting factors can be included
- RBE3 does not modify initial stiffness of the structure (RBE2 does)
- RBE3 is not rigid

```
$ Bulk Section
$ Multipoint Constraints of the Entire Model
$1      2       3       4       5       6       7       8       9       0
RBE3     1000            50      123456 1.       123456  19751   19752
         19753   19754   19755   19756   19757   19758   19759   19760
         19761   19762   19763   19764   19765   19766   19767   19768
         19769   19770   19771   19772   19773   19774   19775   19776
         19777   19778   19779   19780   19781   19782   19783   19784
         19785   19786   19787   19788   19789   19790   19791   19792
         19793   19794   19795   19796   19797   19798   19799
```

MPC & RBE in NASTRAN

Summary

- Caution often linear elements : relationship based on initial geometry
- `RBE2` & `RBE3` switch independent/dependent degrees of freedom
- independent/dependent degrees of freedom a.k.a. master/slaves degrees of freedom
- `MPC`, `RBE2`, `RBE3` introduce displacement relationship in a model
- Many others : Refer to NASTRAN Quick Reference Guide [1]

Case study # 6 - Modal Transcient Response

Theoretical Aparté

One solves

$$M\,\ddot{q} + K q = 0 \qquad q(0) \text{ and } \dot{q}(0) \text{ given} \quad (65)$$

In case of damping

$$M\,\ddot{q} + C\,\dot{q} + K q = 0 \qquad \text{and BCs} \quad (66)$$

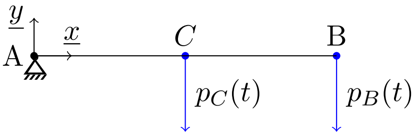
with

- M mass matrix.
- C damping matrix. As C is touchy to assess the BASILE's assumption
  $$C = \alpha M + \beta K \quad (67)$$
  is commonly done.
- K stiffness matrix.

Case study # 6 - Modal Transcient Response

Definition

The framework modal transient response from [2] p. 240. Refer to [3] for deeper theoretical insight.



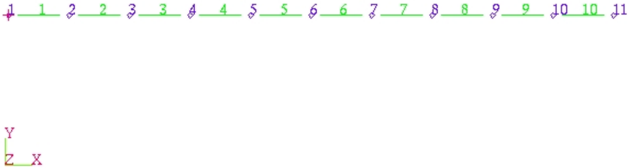
The beam is clamped in A. The beam has a circular section with $R = 14 \times 10^{-3}$ m and $l = 3$ m. Inertia worth $I_z = I_y = \frac{\pi R^4}{4}$. Torsion inertia worth $J = \frac{\pi R^4}{2}$. The beam is made of aluminium $E = 71 \times 10^9$ Pa and $\nu = 0.33$. Two transient loads are applied in B and C.

Aim of Case study # 6 : Students have to compute the modal transcient response of the beam.

Case study # 6 - Modal Transcient Response

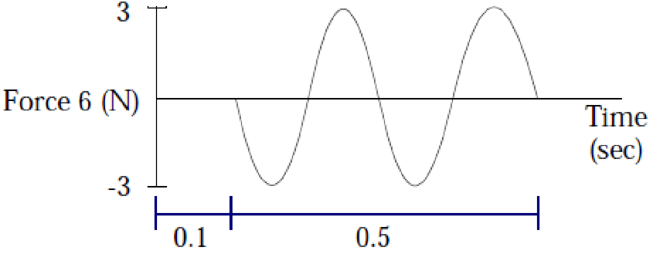
Approach for Meshing

The mesh is to be designed under PATRAN.



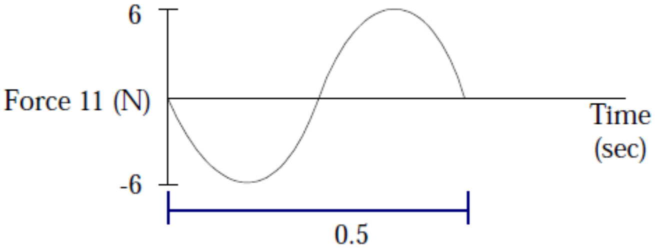
Case study # 6 - Modal Transcient Response
Load

A first transcient load $p_6(t)$ is applied at Node 6.



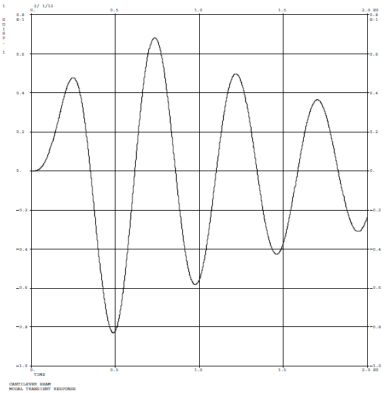
Case study # 6 - Modal Transcient Response
Load

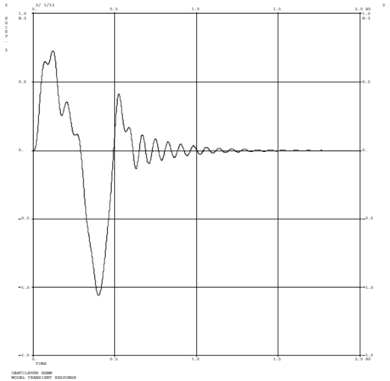
A second transcient load $p_{11}(t)$ is applied at Node 11.



Case study # 6 - Modal Transcient Response

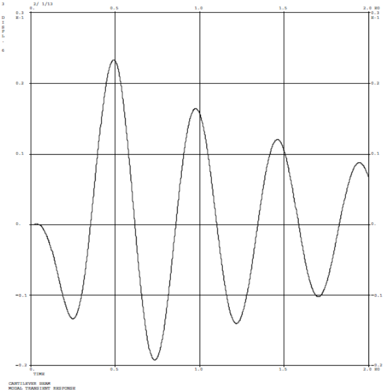
Results

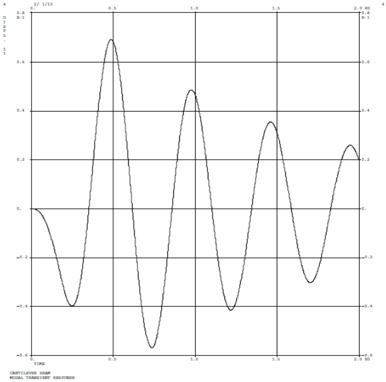




Case study # 6 - Modal Transcient Response

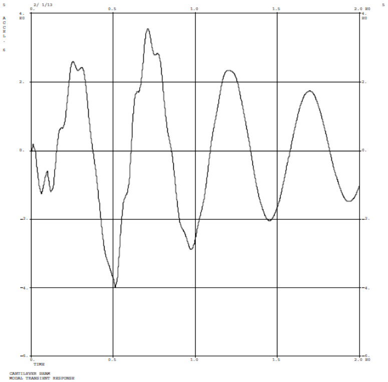
Results

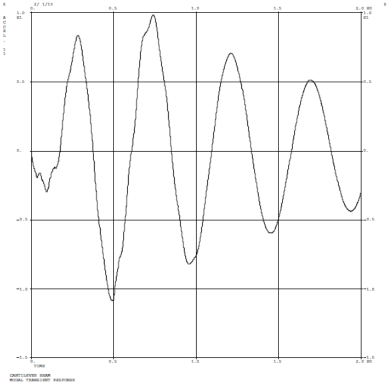




Case study # 6 - Modal Transcient Response

Results





Case study # 6 - Modal Transcient Response

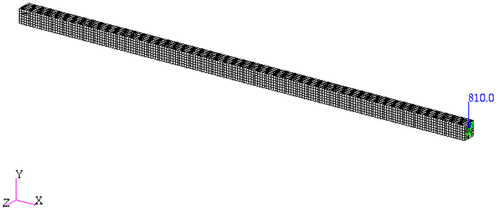
Conclusion

- Modal transcient response is an alternative to a direct transient response analysis

Case study # 7 - Cantilever 3D Beam

Definition

Nodes associated to 3D finite elements as NASTRAN **CHEXA8** own 3 translation degrees of freedom (nodes associated to a shell element as **CQUAD4** own 6 degrees of freedom).

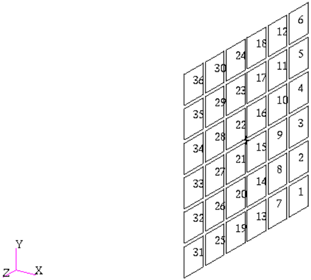


Aim of Case study # 7 : Students have to realize a linear static analysis of the cantilever beam of Lecture 1 under bending.

Case study # 7 - Cantilever 3D Beam

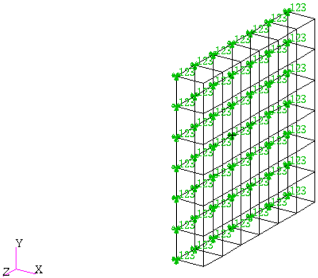
Mesh

- 3D mesh (`CHEXA8`) can be generated from the extrusion of a 2D (`CQUAD4`) slice



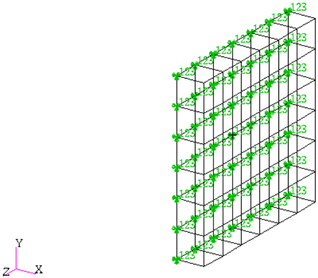
Case study # 7 - Cantilever 3D Beam

Mesh



Case study # 7 - Cantilever 3D Beam

Boundary Conditions



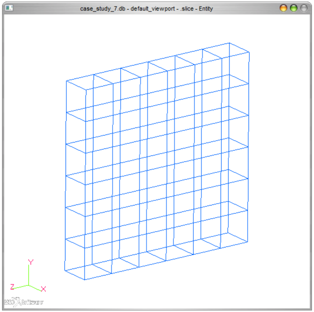
- All 3 degrees of freedom of CHEXA8 nodes are clamped at the LHS end of beam.

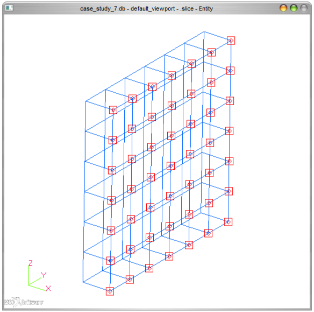
Case study # 7 - Cantilever 3D Beam

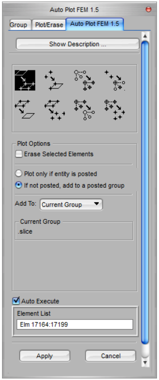
Load

Background : In order to apply load one has to install a `RBE3` element.

Technique : Go to **Mesh** > **MPC** > `RBE3` menu form.





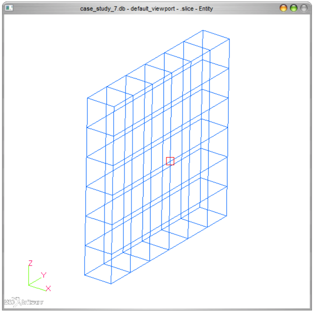


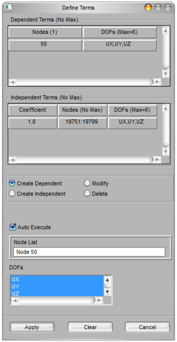
Example : The NASTRAN `CARD` to be created is exactly the one shown before.

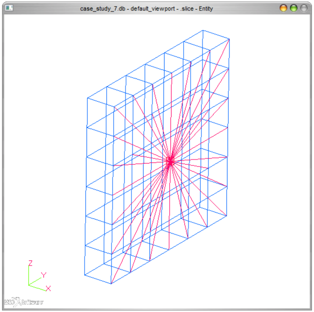
Case study # 7 - Cantilever 3D Beam

Load

GRID 50 □ is created a the center of the section.

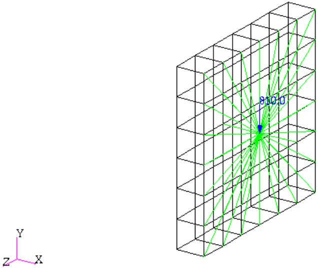






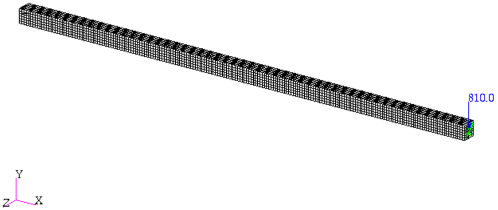
Case study # 7 - Cantilever 3D Beam

Load



Case study # 7 - Cantilever 3D Beam

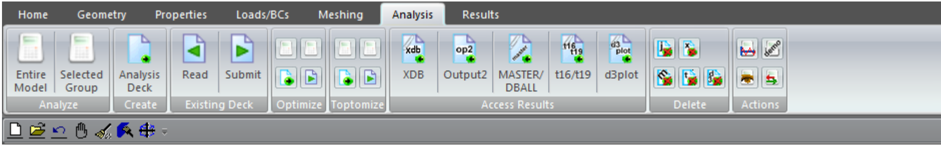
Load



Case study # 7 - Cantilever 3D Beam

NASTRAN linear run

NASTRAN .dat is generated from **Analysis** menu



Then run the analysis with NASTRAN

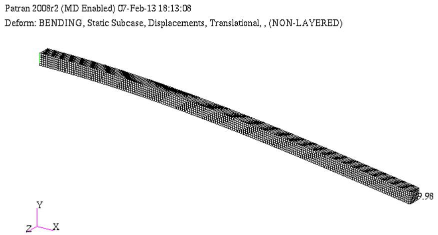
`$ nastran case_study_7.1.dat news=n old=n scr=y`

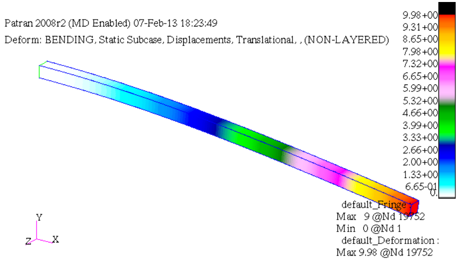
User obtains as output to NASTRAN run

- `case_study_7.1.log` : Control File
- `case_study_7.1.f04` : Execution Summary Table
- `case_study_7.1.f06` : ASCII Results file
- `case_study_7.1.op2` : Binary Results file

Case study # 7 - Cantilever 3D Beam

Results





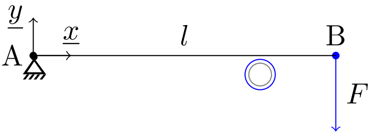
Case study # 7 - Cantilever 3D Beam

Conclusion & Outlook

- Results are in good agreement with the beam modelization lead in Case Study # 1.
- The 3D modelization of the beam bending Case Study #7 is to be modified in order to take into account a cylinder stop.

Case study # 7 - Introduction of Contact Challenge

Definition of Contact Challenge



A load F is applied in B and is tuned to $F = 2$ tons. The beam is clamped in A. The beam has a rectangular section $S = b \times h$ with $b = h = 60$ mm and $l = 2000$ mm. Inertia worth $I_z = \frac{bh^3}{12}$. The beam is made of steel $E = 200$ GPa and $\nu = 0.30$. A 200 mm wide, 5 mm thick and Ø=100 mm steel shaft is placed along $\underline{z}$ axis centered in C (1500 mm, −84 mm) below the beam. Shaft is clamped at its both ends. Realize a frictionless contact and then with $\mu = 0.3$ in a Coulomb law.

Aim of Contact Introduction : Students have to assessed the contact force reacted by the tube from the

Nastran `SOL 600` results.

Case study # 7 - Introduction of Contact Challenge

Theoretical Aparté

Closed form solution from linear elasticity : According to linear elasticity for $x = x_C$ displacement worth

$$v(x = x_C) = -\frac{27}{128}\frac{Fl^3}{E\,I_z} \quad (68)$$

As stop is closer than v contact is expected between the beam and the stop cylinder.

Unilateral contact definition : Frictionless contact is defined with the set of inequalities/equality on $(\underline{U}, \underline{F})$

$$\begin{cases} u = \underline{U}.\underline{n} \geq 0 \\ f = \underline{F}.\underline{n} \leq 0 \\ uf = 0 \end{cases} \quad (69)$$

if $\underline{U} = \underline{0}$ there is contact and if $\underline{F} = \underline{0}$ there is no contact. $(\underline{U}, \underline{F})$ may be used to write a dual version of the latter unilateral contact definition with FENCHEL inequality [4].

Bilateral contact definition : The definition introduces friction. NASTRAN allows to use various contact law.

Case study # 7 - Introduction of Contact Challenge

Numerical analysis Aparté

A common way to include contact conditions in a classical $K\,u = f$ finite element analysis problem is with multiplicateurs de LAGRANGE. If ones choose namely λ as the multiplicateurs de LAGRANGE the latter stand for the force useful to install the contact/gap conditions. One has

$$\begin{bmatrix} K & C^T \\ C & 0 \end{bmatrix} \begin{Bmatrix} u \\ \lambda \end{Bmatrix} = \begin{Bmatrix} f \\ \beta \end{Bmatrix} \qquad (70)$$

solved in most finite elements software through LU decomposition (e.g. CROUT method). If one considers equation (70) as $Ax = b$ one has $LUx = b$. Refer to literature for deeper insight [5, 6, 7].

Some users use penalty method to introduce numerical contact conditions.

Case study # 7 - Introduction of Contact Challenge

NASTRAN BCBODY & BCSURF Cards

```
$1      2       3       4       5       6       7       8       9       0
BCBODY  1       3D      DEFORM  5       0       .3
BSURF   5       ...
```

In order to run a SOL 600 in NASTRAN one has to code BCBODY & BCSURF cards [1].

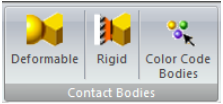
Case study # 7 - Introduction of Contact Challenge

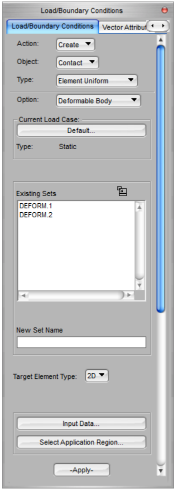
NASTRAN BCBODY & BCSURF Cards

The bodies involved in the SOL 600 have to be called by a BCONTACT card [1].

```
$ Case Control Section
BCONTACT = ALL
```

The **Menu Form** in PATRAN is :





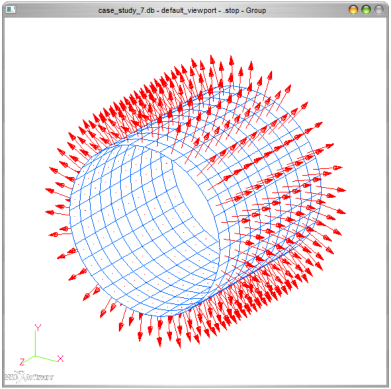
The two bodies are to be defined as 3D deformable with PATRAN.

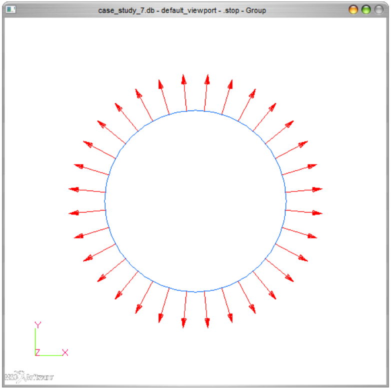
The more accurate the contact bodies are defined the faster is the NASTRAN run.

Case study # 7 - Introduction of Contact Challenge

NASTRAN BCBODY & BCSURF Cards

Cross check the normals of the element coordinate system $\underline{z}$.

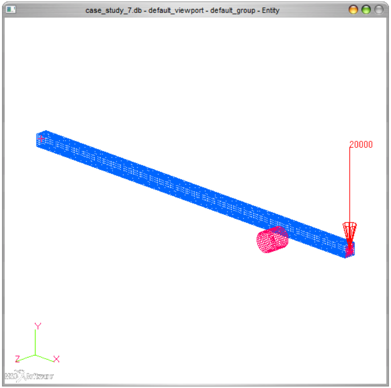


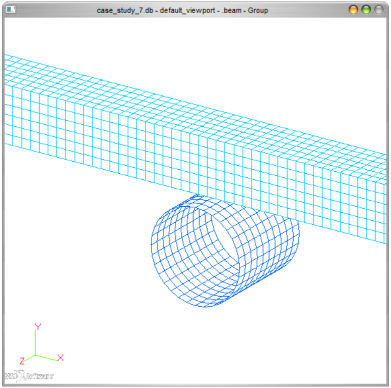


Is presented the slave body above.

Case study # 7 - Introduction of Contact Challenge

NASTRAN BCBODY & BCSURF Cards

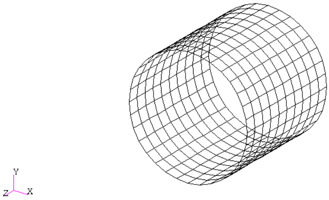




Case study # 7 - Introduction of Contact Challenge

Mesh

- The cylinder stop is meshed with the same density of the beam with CQUAD4 elements in a first step

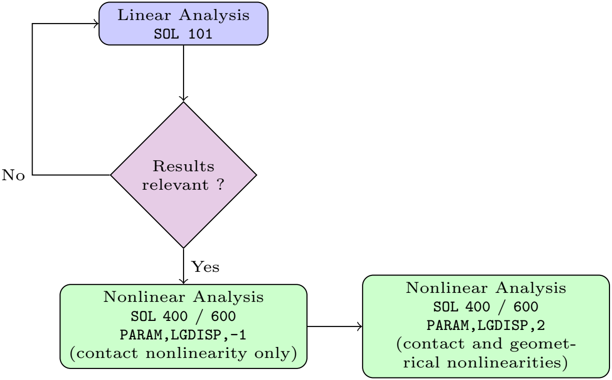


- It is worthwhile pointing out that slave part may be meshed finer than master surface (convex hull of CHEXA8 beam elements or better a relevant set of CHEXA8 only expected to enter in contact with the cylinder stop) in order to ease the convergence of the numerical algorithm that solves contact

Case study # 7 - Introduction of Contact Challenge

Flowchart

Next flowchart is suitable to all reliable non linear analysis.

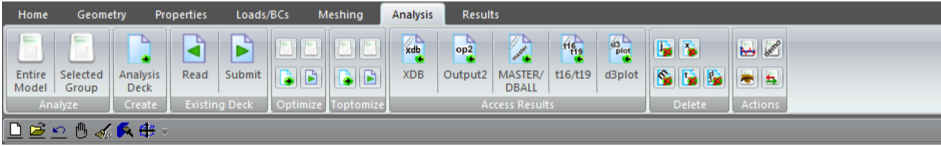


It is mandatory to understand linear analysis before non linear analysis.

Case study # 7 - Introduction of Contact Challenge

NASTRAN nonlinear run

NASTRAN .dat is generated from **Analysis** menu



Then run the analysis with NASTRAN

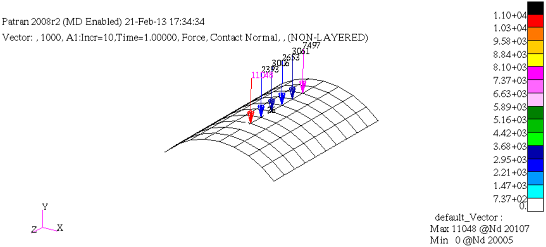
`$ nastran case_study_7.2.dat news=n old=n scr=y`

User obtains as output to NASTRAN run

- `case_study_7.2.log` : Control File
- `case_study_7.2.f04` : Execution Summary Table
- `case_study_7.2.f06` : ASCII Results file
- `case_study_7.2.op2` : Binary Results file

Case study # 7 - Introduction of Contact Challenge

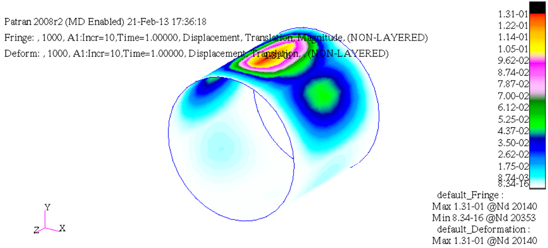
Results - Contact Forces [N]



- The contact forces [N] of the cap of the cylinder is plotted above.

Case study # 7 - Introduction of Contact Challenge

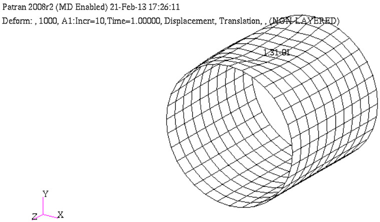
Results - Displacement [mm]



- The displacement field [mm] associated to the cylinder is plotted above.

Case study # 7 - Introduction of Contact Challenge

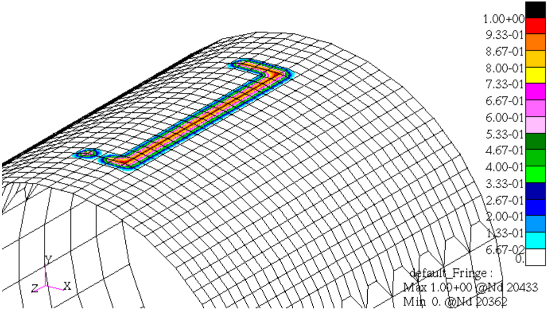
Results - Deformed shape



- The deformed shape associated to the cylinder is plotted above.

Case study # 7 - Introduction of Contact Challenge

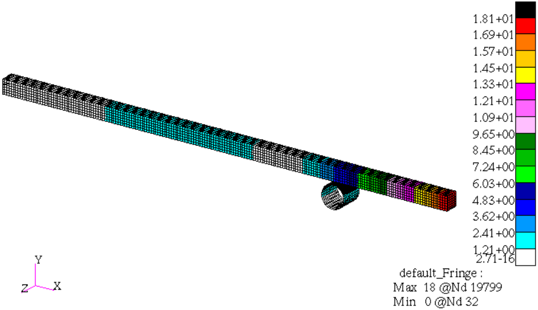
Results - Contact Status



- The contact status associated to a refined mesh for the cylinder is plotted above.

Case study # 7 - Introduction of Contact Challenge

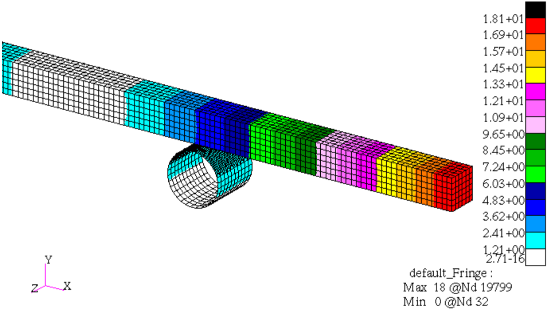
Results - Displacement



- The displacement field [mm] associated to the full structure with a refined mesh for the cylinder is plotted above.

Case study # 7 - Introduction of Contact Challenge

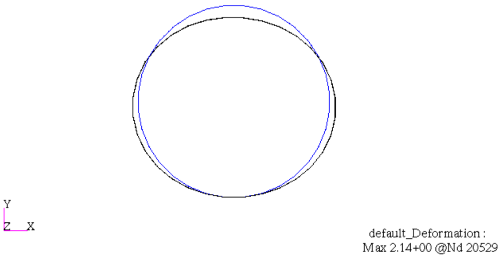
Results - Displacement [mm]



- The displacement field [mm] associated to the full structure with a refined mesh for the cylinder is plotted above.

Case study # 7 - Introduction of Contact Challenge

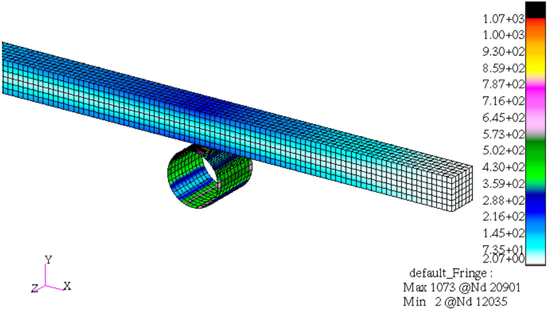
Results - Displacement [mm]



- The displacement field [mm] associated to a refined mesh for the cylinder is plotted above.

Case study # 7 - Introduction of Contact Challenge

Results - $\sigma_{\text{von Mises}}$ [MPa]

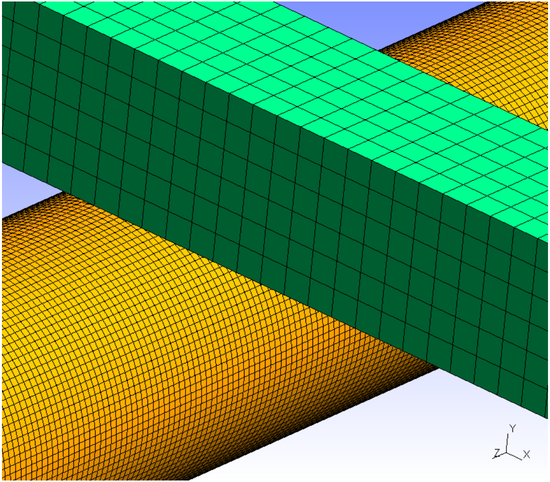


- The $\sigma_{\text{von Mises}}$ [MPa] field associated to the full structure with a refined mesh for the cylinder is plotted above.

Case study # 7 - Introduction of Contact Challenge

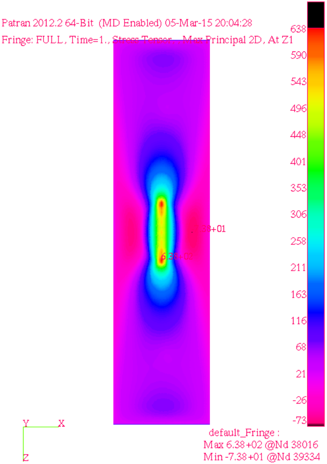
Finer Mesh for Slave Part

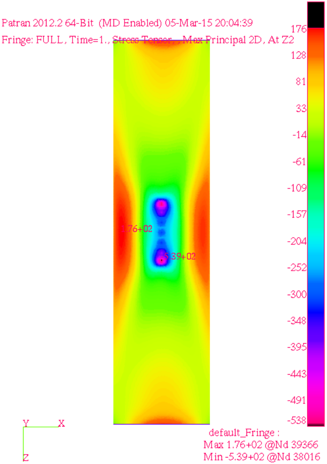
Variation : a finer mesh is to be done in order to assess the stress for the cylinder stop. Its width is twice the former value.



Case study # 7 - Introduction of Contact Challenge

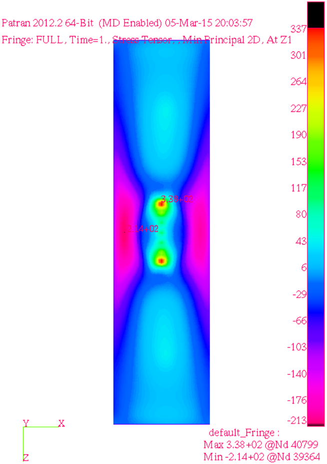
Results - σ_I [MPa]

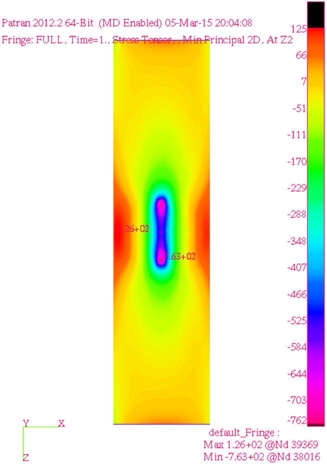




Case study # 7 - Introduction of Contact Challenge

Results - σ_{II} [MPa]

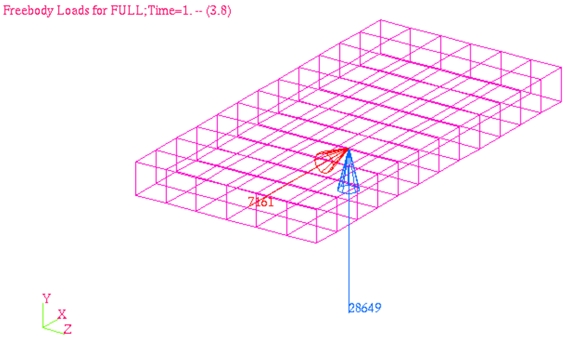




Case study # 7 - Introduction of Contact Challenge

Results - Freebody diagram

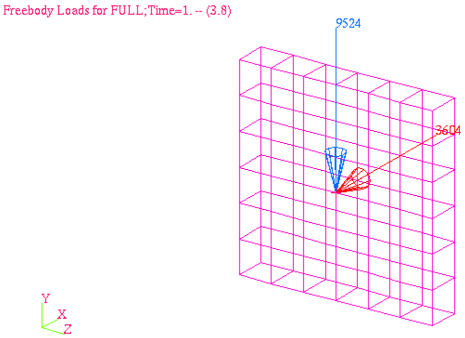
Force reaction of the cylnder stop is assessed below with the PATRAN freebody **Menu Form**. One can derive the same freebody from the `PUNCH GPFORCES` for example with a shell script.



Case study # 7 - Introduction of Contact Challenge

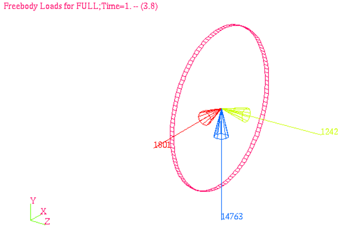
Results - Freebody diagram

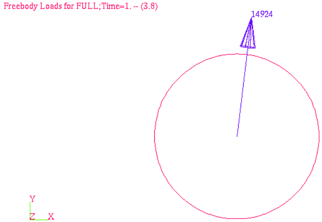
Force reaction of the clamped boundary condition of the beam is assessed below with the PATRAN freebody **Menu Form**. One has to notice that case study #7 leads to a reaction component $F_y < 0$: the cylinder stop modifies the mechanical simple cantilever beam solution and pull upward the clamped part.



Case study # 7 - Introduction of Contact Challenge

Results - Freebody diagram





Force reaction of the clamped boundary condition of the cylinder is assessed below with the Patran freebody **Menu Form**.

One has to notice that cylinder stop clamped sections reacted half the contact force of the beam on the stop with a slight hyperstatic $\underline{z}$-component and a anticlockwise 83^o angle of reaction in (x,y) plane.

Case study # 7 - Introduction of Contact Challenge

Conclusions & Outlook

- The contact condition is integrated with NASTRAN whatever in `SOL 600` or in `SOL 400`
- A relevant approach is to run first a contact nonlinear only run before coupling with geometrical nonlinearity; first :

```
$ Bulk Section Parameters
PARAM    LGDISP  -1
```

then :

```
$ Bulk Section Parameters
PARAM    LGDISP  +2
```

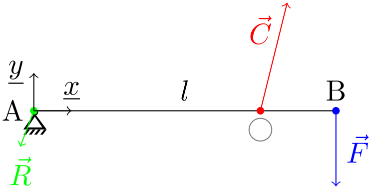
Nota : follower forces may be switched on

```
$ Bulk Section Parameters
PARAM    LGDISP  +1
```

Case study # 7 - Introduction of Contact Challenge

Conclusions & Outlook

- Beam freebody on the undeformed structure is shown below with • the contact point between the beam and the stop and the contact force $\vec{C}$. The applied load is $\vec{F}$ and the reaction at the clamped boundary condition is $\vec{R}$.



  Cylinder stop freebody is left to students as an exercise
- A cylinder made in non high strength steel would lilely exhibit plasticity for a load below F : material nonlinearity has thus to be introduced
- If boundary conditions on cylinder stop are not fully clamped and F increases cylinder stop is likely to buckle

References

[1] MSC Software.
Quick Reference Guide: MSC Nastran 2012.
MacNeal-Schwendler Corporation, 2011.

[2] MSC Software.
Dynamic Analysis User's Guide: MSC Nastran 2012.
MacNeal-Schwendler Corporation, 2011.

[3] M. Géradin and D. Rixen.
Théorie des vibrations: application à la dynamique des structures.
Recherche en mécanique. Masson, 1996.

[4] G. Duvaut and J.L. Lions.
Les inéquations en mécanique et en physique.
Travaux et Recherches Mathématiques. Dunod, 1972.

[5] A. Quarteroni, R. Sacco, and F. Saleri.
Numerical Mathematics.
Texts in applied mathematics. Springer, 2000.

[6] M. Schatzman.
Analyse numérique: cours et exercices pour la licence.
InterEditions, 1991.

[7] P.G. Ciarlet.
Introduction à l'analyse numérique matricielle et à l'optimisation.
Collection Mathématiques appliquées pour la maîtrise. Masson, 1982.