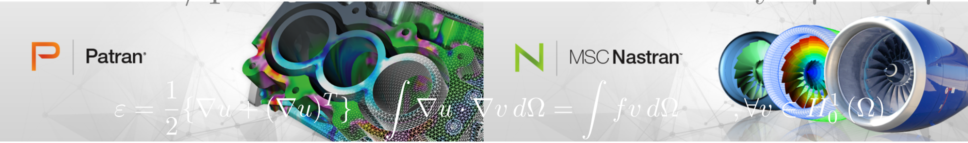
Patran / Nastran

Lecture 1/4 February 27th 2017



Insa Toulouse
Tp Génie Mécanique
Ingénierie des Systèmes

Julien Le Fanic

Lectures Scope

1. Lecture 1 deals with basics Finite Element 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

Table of contents

The Finite Elements Method

Classification of Ordinary Differential Equations

Although it is a paradox physics phenomena may be described by mathematical equations. Lectures deal with problems that are well described by ordinary differential equations. The latter may be classified as follows.

$$a\frac{\partial^2\phi}{\partial x_1^2} + 2b\frac{\partial^2\phi}{\partial x_1\partial x_2} + c\frac{\partial^2\phi}{\partial x_2^2} = 0 \quad (1)$$

$$\delta = b^2 - ac \quad (2)$$

As pointed out in Table 1 heat transfer is an elliptic problem if not time dependent. If time dependent it is an hyperbolic problem. Wave propation problems are hyperbolic problems. Steady structural analysis under a hookean strain/stress relationship deals with elliptic Ordinary Differential Equations. The latter are known as the equations of theory of elasticity.

Type	sign δ	Example	Equation
Elliptic	-	Laplace	$\Delta\phi = \frac{\partial^2\phi}{\partial x^2} + \frac{\partial^2\phi}{\partial y^2} = 0$
Hyperbolic	+	Wave	$\frac{\partial^2\phi}{\partial x^2} - \frac{\partial^2\phi}{\partial t^2} = 0$
Parabolic	0	Diffusion	$\frac{\partial\phi}{\partial t} - \frac{\partial^2\phi}{\partial x^2} = 0$

Table 1: Classification of Ordinary Differential Equations

The Finite Elements Method

Basics of continuum mechanics

A solid in static equilibrium bounded by a domain Ω and submitted to boundary conditions for $\partial\Omega$ satisfies to CAUCHY equation

$$\nabla.\sigma + f = 0 \tag{3}$$

In linear elasticity strain tensor is defined as

$$\varepsilon = \frac{1}{2}\{\nabla u + (\nabla u)^T\} \tag{4}$$

The HOOKEAN stress / strain relationship is often assumed in the course unless clearly specified (refer §A).

$$\sigma = 2\mu\,\varepsilon + \lambda\,\nabla.u\,\mathrm{I} \tag{5}$$

The assumption is valid for linear elasticity. As it is explained in Lecture 2 one sometimes prefers shell forces and moments in order to assess the loads withstand by a structure. Lecture 2 contains also a reminder about the equilibrium (3) and its weak form [1].

The Finite Elements Method

Definition

A definition of Finite Element Method quoted from [2].

> *The Finite Element Method emerged from the engineering literature of the 1950's as one of the most powerful methods ever devised for the approximate solution of boundary-value problems. [...] It is a variational method of approximation, making use of global or variational statements of physical problems and employing the Rayleigh / Ritz / Galerkin philosophy of constructing coordinate functions whose linear combinations represent the unknown solutions.*

First usage of *finite element* expression as a numerical method for solving structural analysis problems is due to Clough [3] in a 1960 paper.

The Finite Elements Method

Finite Elements Approximation

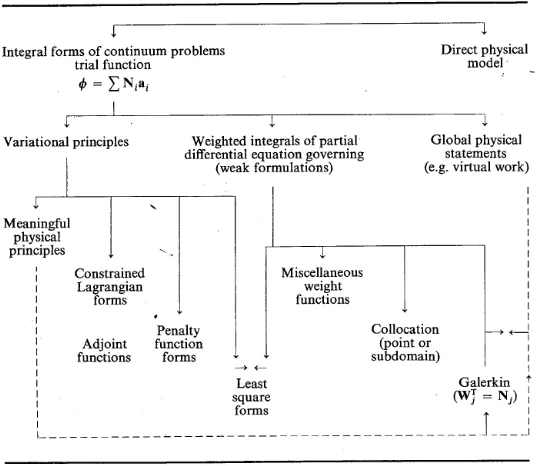


Figure 1: Finite Elements Approximation from [3] p. 90

The Finite Elements Method

Weak form

Finite elements method allows to write the equation (3) in a matricial formulation (mathematical insight are reserved for Lectures 2 & 3)

$$K\,u = F \quad (6)$$

The matrix K is symmetric, positive definite and sparse for a finite element problem for structural mechanics in linear elasticity.

The displacement field u is computed without inversion of K but through a relevant approach based on numerical analysis (e.g. conjugate gradient method).

The Finite Elements Method

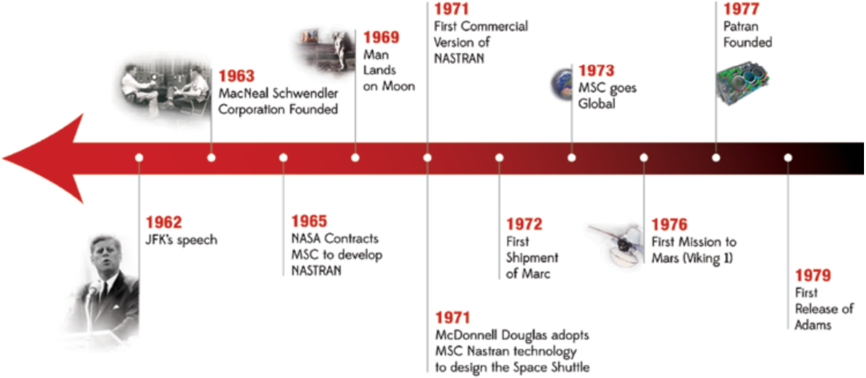
History

A short history of Finite Element Method quoted from [4]. It seems that aircraft structural engineers were at the origin of the representation of a continuum by a set of discrete elements in order to efficiently compute fuselage structure.

- 1941 - Hrennikoff : plane elastic solid as a collection of discrete bars and beams collection
- 1943 - Courant : approximation of Saint Venant torsion problem
- 1947 - Prager & Synge : hypercircle method [5] very close in the spirit of Finite Element Method
- 1954 - Argyris : start of a collection of paper about matricial structural mechanics
- 1960 - Clough : op. cit.
- 1967 - Nastran first release to Nasa
- 1970 - now : 45 years of ever growing papers about Finite Element Method, ever growing machines hardware capabilities, more & more multiphysics

The Finite Elements Method

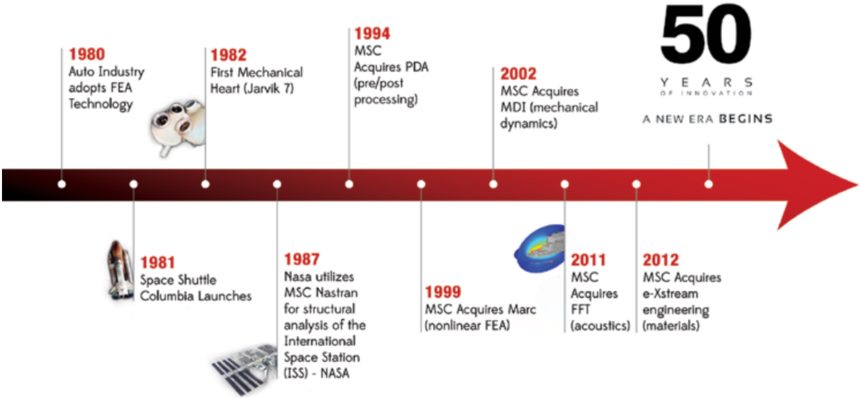
History



Chronological sketch from Msc Software Magazine [6].

The Finite Elements Method

History



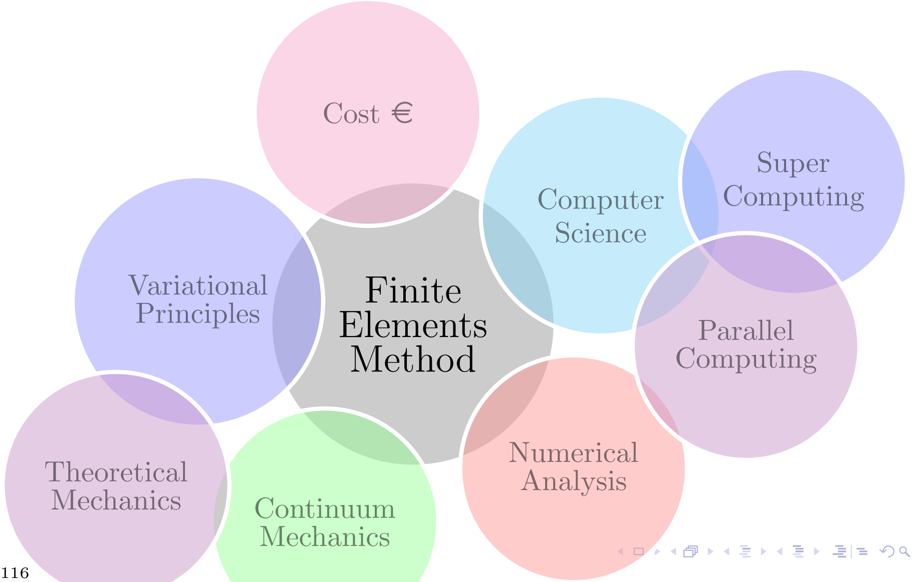
Chronological sketch from MSC Software Magazine [6].

The Finite Elements Method

MSC.Apex

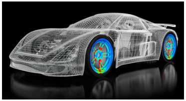
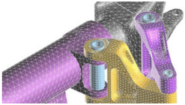
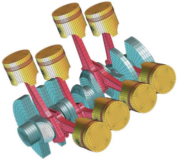
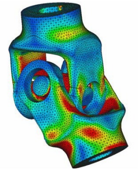
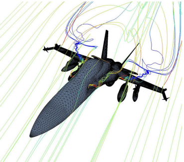
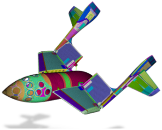
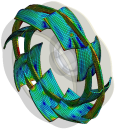
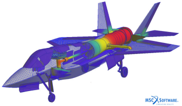
The Finite Elements Method

Bubble Summary



The Finite Elements Method

Usage within the industry

Main industries using NASTRAN are automotive, engines, agricultural machines, defense, shipbuilding, civil engineering, biomechanics, aerospace... NASTRAN is particularly suitable for structural analysis (static, dynamic, modal/transcient response) and fluid/structure interaction, aeroelasticity, thermal analysis.

The Finite Elements Method

Usage within the industry / Spacecraft Structural Analysis

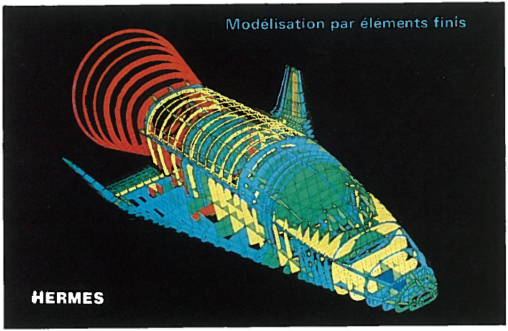


Figure 2: Spacecraft HERMÈS General Model with 6550 nodes / 17 000 degrees of freedom. Linear analysis run time < 1 min on a IBM 3090 VF [7] in ~ 1985.

The Finite Elements Method

Usage within the industry / Fighter Aircraft Eigenmodes

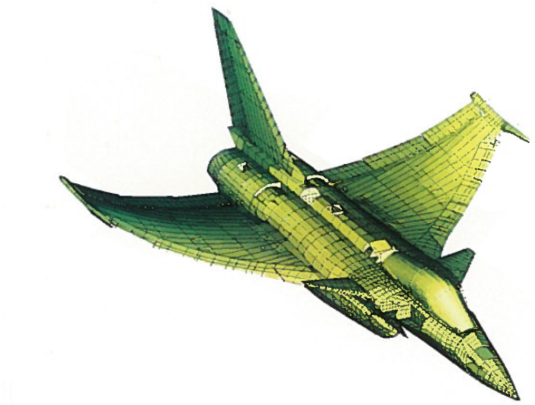


Figure 3: Fighter Aircraft : dynamic analysis. Visualization of a structural eigenmode [7] in ∼ 1985.

The Finite Elements Method

Usage within the industry / Engine Nonlinear Transcient Analysis

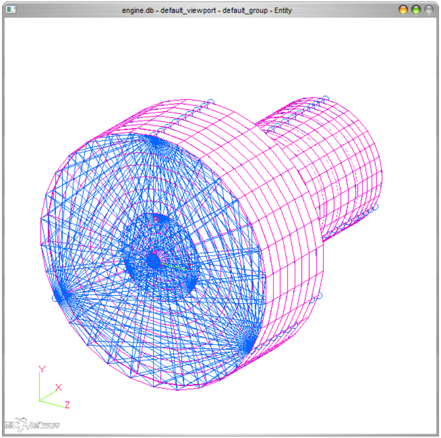


Figure 4: Aeronautical engine ∼ 5000 elements ; benchmark problem from MSC NASTRAN for `SOL 129`.

The Finite Elements Method

Usage within the industry / Curiosity Mars Approach & Landing

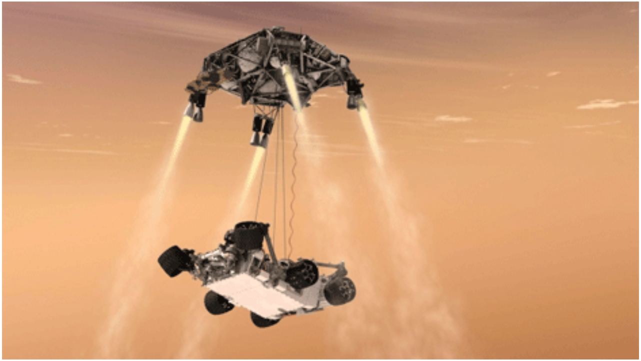


Figure 5: Curiosity landed on Mars in 2012. Simulation played a key role for this NASA achievement.

The Finite Elements Method

Usage within the industry / FBO event

The FBO simulation is a sequel of physical events which own different time scales. The past years considerable improvements have been reached in order to realize the analysis chained between dedicated solvers. The analysis can be lead with NASTRAN.

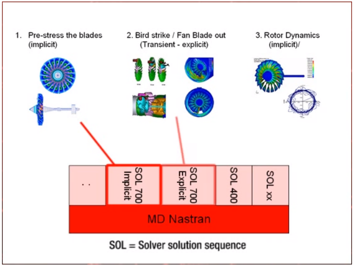


Figure 6: Simulation of an FBO event with MSC.NASTRAN- SOL used.

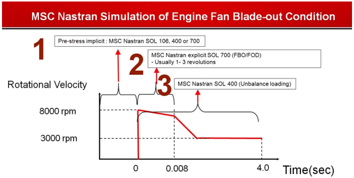


Figure 7: Simulation of an FBO event with MSC.NASTRAN- Physics timescales.

The FBO analysis benchmarked by MSC involves

- SOL 400
- SOL 700

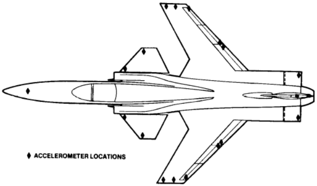
The Finite Elements Method

Check of Results Relevancy

A NASTRAN run has to be carefully check. There is no universal procedure but relevancy of the results are part of the analysis of each run.

Finite Element Method is particularly expensive within an industrial framework when one has to check the relevancy of the analysis by test on a real structure with e.g.

- strain gages for static analysis
- accelerometers for dynamic analysis (below NASA X-29)



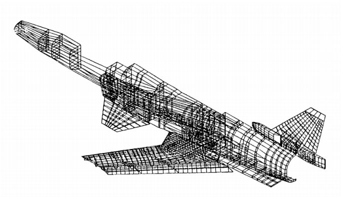


Figure 8: X-29 V&V example.

Finite Element Method owns a rich literature dedicated to error analysis ; nowadays it is covered as V & V dedicated to Finite Element Method [8].

Aparté : Numerical Verification/Criteria, Verification/Physical Meaning Check and Model Validation are different fields linked to the Finite Element Method.

Introduction to Nastran

Definition

Nastran is a solver. It is a tool for the engineer or the researcher to solve partial differential equations of mathematical physics.

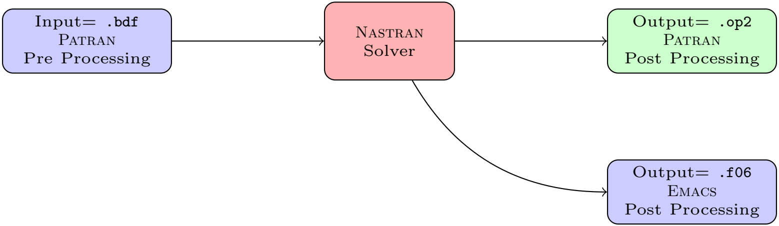


Figure 9: Basic flowchart of a Nastran run. Patran vs. Nastran usages.

Blue boxes means Ascii files (text files) and Green box means binary file.

Introduction to NASTRAN

Definition

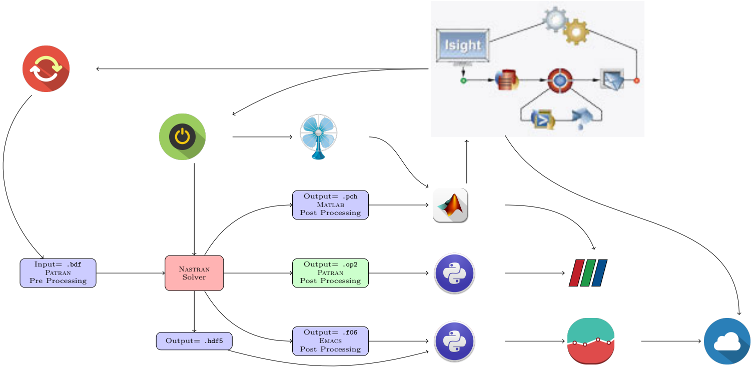


Figure 10: Advanced flowchart of a NASTRAN run. PATRAN vs. NASTRAN usages inserted in a more real world approach. are Python scripts, a CFD Code, is a MATLAB script, /// is a PARAVIEW visualization, means basic charts to monitor NASTRAN output relevancy and means a cloud storage e.g. for further big data approach. The update of input can be a solver output if the user runs a `SOL 200` or an OPTISTRUCT analysis.

Introduction to NASTRAN

History

- NASTRAN means for NAsa STRuctural ANalysis. Thomas G. BUTLER was NASA NASTRAN Project Manager.
- 1965 - MSC won NASA bid to develop a finite elements code
- MSC means MACNEAL SCHWENDLER CORPORATION. MACNEAL & SCHWENDLER two engineers who were particularly gifted in
  1. numerical analysis
  2. programmation
  3. finite elements [9]
  4. business
- The lecture focus on MSC NASTRAN (called by the past MSC/NASTRAN, MSC.NASTRAN or recently MD NASTRAN) but it exists other versions from historically COSMIC NASTRAN, UAI NASTRAN, in-house compiled aircraft manufacturers versions as NASTRAN-LCC (ca. 1975) to recent NEI NASTRAN (by 2014 AUTODESK NASTRAN) or NX NASTRAN (SIEMENS)
- NASTRAN is a registered trademark of the National Aeronautics Space Administration. MSC NASTRAN is an enhanced proprietary version developed and maintained by MSC.Software Corporation

1968

1983

1971

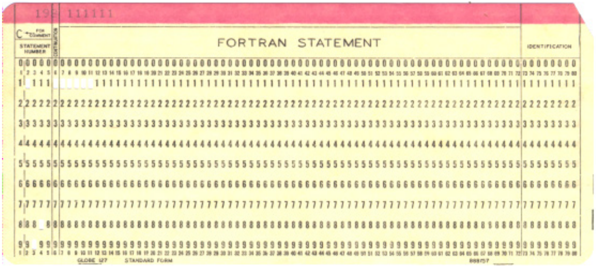
2004

Logos from MSC Software Magazine [6].

Introduction to NASTRAN

History

- The NASTRAN usage can be traced back before 1970
- NASTRAN collection of subroutines were written in FORTRAN (FORmula TRANslating SYSTEM)
- FORTRAN was developed on IBM 701 from 1953 to 1956 [10]



It explains vocabulary legacy such as `CARD` and `PUNCH` in NASTRAN.

- NASTRAN modules obey DMAP* language
- NASTRAN stores data according to NDDL† language

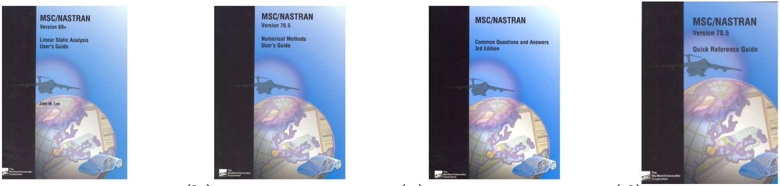
---

* Direct Matrix Abstraction Program

† Nastran Data Definition Language

Introduction to NASTRAN
Documentation

NASTRAN has a rich documentation.



(a) Linear Static Analysis (b) Numerical Methods (c) Common Questions & Answers (d) Quick Reference Guide

Figure 11: Msc NASTRAN Guides.

Its long life as a finite element standard code lead to many publications.

Introduction to NASTRAN

Documentation

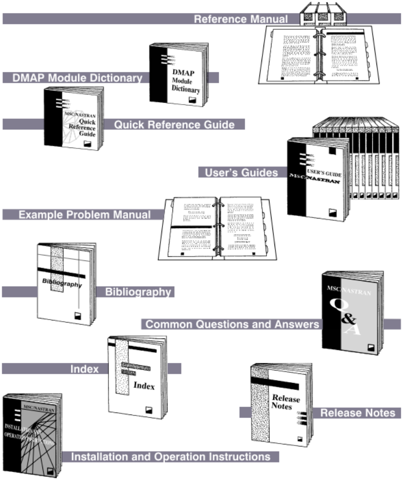


Figure 12: Full MSC NASTRAN documentation set.

Introduction to NASTRAN

Literature

NASTRAN has not many student books with tutorials apart from the following ones [11, 12, 13].

(a) (b) (c) (d)

Figure 13: Books about NASTRAN

Some are out of print. Others are either in german or chinese.

Introduction to NASTRAN

Solutions

NASTRAN offers a vast choice of solutions (aka `SOL` for Structured Solution Sequences) to solve differential equations associated to physical problems.

SOL	PHYSICS
101	Linear Static
103	Modal
105	Buckling
106	Non Linear Static
111	Modal Frequency Response
112	Modal Transient Response
129	Nonlinear Transient
144	Static Aeroelastic Analysis
145	Flutter / Aeroservoelastic analysis
146	Dynamic Aeroelastic Analysis
153	Non Linear static coupled with heat transfer
159	Nonlinear Transient coupled with Heat transfer
200	Design Optimization and Sensitivity analysis

Table 2: Sample of NASTRAN solutions sequences

Introduction to NASTRAN

Lectures vs. Solutions

LECTURE	SOL						
	101	103	105	106	112	600	200
1	●	○	○	●	○	○	●
2	●	○	●	○	○	○	○
3	●	○	●	○	●	●	○
4	○	●	○	○	○	○	○

Table 3: Lectures vs. NASTRAN solutions sequences

For Lecture 3 one can switch from `SOL 600` to `SOL 400` if NASTRAN installation is not linked to MARC solver on workstations.

Introduction to Nastran

Nastran Elements Library

Nastran offers a relevant choice of elements in its library.

In comparison with software as Abaqus, Nastran library is smaller.

Introduction to NASTRAN

Input file structure

```
$ File Management Section - - - - - - - - - - - - - - - - - - - - - - - - +
$ Default .op2 filename is as the input run
ASSIGN OUTPUT2='myfirstnastranrun.op2' UNIT=12 STATUS=NEW

$ NASTRAN Statement - - - - - - - - - - - - - - - - - - - - - - - - - - - +
$ Extended Error Messages
NASTRAN SYSTEM(319)=1

$ Executive Control Section - - - - - - - - - - - - - - - - - - - - - - - +
SOL 101
CEND

$ Case Control Section- - - - - - - - - - - - - - - - - - - - - - - - - - +
ECHO=NONE

TITLE = MY FIRST NASTRAN RUN

$ Displacement Output Request
$      - PRINT : Output to .f06
$      - PUNCH : Output to .pch
$      - PLOT  : Output to .op2
DISP(PRINT,PUNCH,PLOT)=ALL
```

Introduction to NASTRAN

Input file structure

```
$ * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
$ * BENDING                                                              *
$ * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
$
$                      y  ^
$                         |           l
$                         o - > - - - - - - - - +
$                      z      x                 |  F
$                                               |
$                                               v
SUBCASE 1000  $ - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
LABEL = CANTILEVER BEAM                                                 |
SUBTITLE = BENDING                                                      |
$ LOAD =1000      $ F                                                   |
SPC =1000         $ Boundary Conditions in o = Clamped Condition        |
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +

$ * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
$ * TENSILE FORCE                                                        *
$ * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
$
$                      y  ^
$                         |           l            T
$                         o - > - - - - - - - - +----->
$                      z      x
$
$
SUBCASE 2000  $ - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
LABEL = CANTILEVER BEAM                                                 |
SUBTITLE = TENSILE FORCE                                                |
$ LOAD =2000      $ T                                                   |
SPC =1000         $ Boundary Conditions in o = Clamped Condition        |
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
```

Introduction to NASTRAN

Input file structure

```
$ Bulk Data- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
BEGIN BULK
$ Bulk  Parameters - - - - - - - - - - - - - - - - - - - - - - - - - - +
PARAM,POST,-1
PARAM,AUTOSPC,NO
PARAM,PRTMAXIM,YES

$ NASTRAN Cards data written in 8  Characters wide field (small field)
$<  1  ><  2   ><  3   ><  4   ><  5   ><  6   ><  7   ><  8   ><  9   ><  *   >
GRID    1       0       0.      0.      0.
$
$ NASTRAN Cards data written in 16 Characters wide field (large field)
$ to allow eg. higher digits precision for GRID coordinate
$<  1  ><      2       ><      3       ><      4       ><      5       ><  *   >
GRID*   2               0               1.0000000000000 0.              *
$<  *  ><      6       ><      7       ><      8       ><      9       >
*       0.
$
GRID    3       0       1.      1.      0.
GRID    4       0       0.      1.      0.
$ Elements
$QUAD4 #1       PID     Node #1 Node #2 Node #3 Node #4
CQUAD4  1       1       1       2       3       4
$ Properties
PSHELL  1       1       5.-3    1               1
$ Materials
MAT1     1      70000.          .33
$ BCs
SPCADD   1000    1
SPC1     1       123456  1      THRU    2
$ Loads
...
ENDDATA
$- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
```

Useful NASTRAN Parameters & Best Practices

Useful NASTRAN Parameters & Best Practices

- Prefer a `.bdf` or `.dat` extension for NASTRAN deck
- Comments are inserted w/ lines beginning with a $
- Case Control command `GROUNDCHECK(SET=ALL)=YES` prints the resuts of the model moved rigidly for some NASTRAN set (cf. [14]). If the rigid body motions enforced to the NASTRAN degrees of freedom sets are not numerically strain energy free a `FAIL` flag is written in the `.f06` for each non compliant degree of freedom. Here is a 100% passed groundcheck output table :

```
 *** USER INFORMATION MESSAGE 7570 (GPWG1D)
     RESULTS OF RIGID BODY CHECKS OF MATRIX KGG      (G-SET)  FOLLOW:
     PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF   1.602612E-08
           DIRECTION        STRAIN ENERGY        PASS/FAIL
           ---------        ------------         ---------
              1              1.421085E-14          PASS
              2              5.684342E-14          PASS
              3              3.469447E-18          PASS
              4              1.301043E-18          PASS
              5              4.336809E-18          PASS
              6              3.671976E-14          PASS

     SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:
       1. CELASI ELEMENTS CONNECTING TO ONLY ONE GRID POINT;
       2. CELASI ELEMENTS CONNECTING TO NON-COINCIDENT POINTS;
       3. CELASI ELEMENTS CONNECTING TO NON-COLINEAR DOF;
       4. IMPROPERLY DEFINED DMIG MATRICES;
```

Useful NASTRAN Parameters & Best Practices

Useful NASTRAN Parameters & Best Practices

- Useful parameters to be included in each run :
  1. `PARAM,AUTOSPC,NO` in order to properly analyse singularity of a model. A dof is singular if
     $$\frac{k_{ij}}{\max k_{\mathrm{elem}}} \leq \varepsilon \qquad (7)$$
     The default value is $\varepsilon = 10^{-8}$. The latter can be modified by a `PARAM,EPZERO,` ε.

```
 *** USER INFORMATION MESSAGE 5293 (SSG3A)
    FOR DATA BLOCK KLL
    LOAD SEQ. NO.             EPSILON               EXTERNAL WORK      EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH AST
                1         -1.0426814E-13           4.0528442E+03
```

     Aparté : The latter is not to be misunderstood with the `EPSILON` printed in the `.f06` that stands with the quality of the solution found from residual vector r.
     $$\varepsilon = \frac{x^T\, r}{x^T\, b} \qquad (8)$$
     with
     $$r = f - K\,x = f - K \left\{ \begin{array}{c} u \\ \omega \end{array} \right\} \qquad (9)$$
  2. `PARAM,TINY,` • to leave out printing of underconstrained elements
  3. `PARAM,POST,-1` in order to have an `.op2` output
  4. `PARAM,PRTMAXIM,YES` in order to have a summary of maximum `SPCFORCES`, `MPCFORCES`, u (displacements) and ω (rotations) in the `.f06`
  5. `PARAM,CHECKOUT,YES` to realize some geometry/dof set check out (cf. [14])

Useful NASTRAN Parameters & Best Practices

Useful NASTRAN Parameters & Best Practices

6. PARAM,GRDPNT,0 to have the centre of gravity, inertia inertia matrix about the CG, principal inertiae about the CG, mass summary in the .f06

```
                         PARAM GRDPNT WEIGHT INERTIA MATRIX

                       OUTPUT FROM GRID POINT WEIGHT GENERATOR
                           REFERENCE POINT = 1000 }—GRID POINT OR ORIGIN OF BASIC
                                      MO             COORDINATE SYSTEM
 *  1.562092E+00 -1.669640E-17 -1.040834E-17 -4.440892E-16  5.894034E+01 -1.359722E+01 *
 * -1.767218E+17  1.562092E+00 -1.416518E-16 -5.894034E+01 -4.973032E-16  3.635694E-01 *
 * -1.040834E-17 -1.419229E-16  1.562092E+00  1.359722E+01 -3.635694E-01 -4.440892E-16 *
 *  4.440892E-16 -5.894034E+01  1.359722E+01  3.191095E+03  3.648479E+00 -2.795030E+00 *
 *  5.894034E+01  2.972035E-16 -3.635694E-01  3.648479E+00  3.283023E+03 -5.082753E+02 *
 * -1.359722E+01  3.635694E-01  2.220446E-16 -2.795030E+00 -5.082753E+02  4.852746E+02 *
                                      S
          *  1.000000E+00  0.            0.            * } TRANSFORMATION MATRIX
          *  0.            1.000000E+00  0.            * } FROM THE BASIC SYSTEM
          *  0.            0.            1.000000E+00  * } TO THE PRINCIPAL MASS AXES

       DIRECTION
   MASS AXIS SYSTEM (S)    MASS          X-C.G.         Y-C.G.         Z-C.G.
            X          1.562092E+00   0.0            8.704492E+00   3.773167E+01
            Y          1.562092E+00   2.327452E-01   0.0            3.773167E+01
            Z          1.562092E+00   2.327452E-01   8.704492E+00   0.0
                                     I(S)
          *  8.488209E+02 -6.813166E+00 -1.092305E+01  * } MOMENTS OF INERTIA IN THE
          * -6.813166E+00  1.059020E+03 -4.770467E+00  * } BASIC COORDINATE SYSTEM
          * -1.092305E+01 -4.770467E+00  3.668331E+02  * }
                                     I(Q)
          *  1.059279E+03                              * }
          *                8.488406E+02                * } PRINCIPLE MOMENTS OF INERTIA
          *                              3.665549E+02  * }
                                      Q
          * -3.274377E-02 -9.992094E-01  2.254901E-02  * } DIRECTION COSINES FROM
          * -9.994364E-01  3.290163E-02  6.665341E-03  * } BASIC COORDINATES TO
          * -7.401970E-03 -2.231805E-02 -9.997235E-01  * } PRINCIPAL INERTIA AXES

   PRINCIPLE MASSES AND ASSOCIATED CENTERS OF GRAVITY
   RELATIVE TO THE REFERENCE POINT

   RIGID BODY MASS MATRIX RELATIVE TO THE REFERENCE
   POINT IN THE BASIC COORDINATE SYSTEM
```

Table 4: GRDPNT in the .f06 [15].

Useful NASTRAN Parameters & Best Practices

Useful NASTRAN Parameters & Best Practices

7. `PARAM,MAXRATIO,•` with • a threshold value : user is interested to have a view during the decomposition (`DCMP` [16])

$$K = L\,D\,L^T \tag{10}$$

of the degrees of freedom associated to a ratio

$$\frac{k_{ii}}{d_i} \tag{11}$$

higher than • if any. The highest ratio $\max_{\text{dof}\in\text{model}} \frac{k_{ii}}{d_i} \equiv$ `NASTRAN MAXRATIO` is quoted through this message in the `.f06` (`case_study_#1` sample):

```
*** SYSTEM INFORMATION MESSAGE 4159 (DFMSA)
     THE DECOMPOSITION OF KLL      YIELDS A MAXIMUM MATRIX-TO-FACTOR-DIAGONAL RATIO OF    5.399920E+01
```

`MAXRATIO` analysis reveals the ill-conditioning of the stiffness matrix K and is an effective way to track dof that need to be deeper analyzed (eg. lack of stiffness in the definition of the model, lack of boundary conditions that leads to mechanisms) because they lead a NASTRAN run to crash (`FATAL` messages in the `.f06`). If it seems useless on a small/simple model it is priceless on a huge/touchy NASTRAN model.

8. The occurence of the word `FATAL` in the NASTRAN `.f06` means something went wrong (unless intentionally used in NASTRAN run `TITLE`, `SUBTITLE`, `SUBCASE LABEL`,...)

Useful NASTRAN Parameters & Best Practices

Useful NASTRAN Parameters & Best Practices

- Small field format is more readable so avoid large field format if not deemed necessary
- For the lectures a NASTRAN deck that begins at the executive control section (choice of the SOL) is perfectly suitable

Useful NASTRAN Parameters & Best Practices

Useful NASTRAN Parameters & Best Practices

- Here is a more complete Case Control Section description particularly for the output request (cf. [14])

```
$ Case Control Section- - - - - - - - - - - - - - - - - - - - - - - - - +

TITLE = CASE_STUDY_#1

$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
$ OUTPUT REQUEST SECTION                                                |
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
ESE(PLOT,PRINT)=ALL                                                     |
DISP(PLOT,PRINT,PUNCH)=ALL                                              |
FORCE(PLOT,PRINT)=ALL                                                   |
OLOAD(PLOT,PRINT,PUNCH)=ALL                                             |
STRESS(PLOT,PRINT)=ALL                                                  |
STRAIN(PLOT,PRINT)=ALL                                                  |
GPFORCE(PLOT,PRINT,PUNCH)=ALL                                           |
GPSTRESS(PLOT,PRINT)=ALL                                                |
SPCFORCES(PLOT,PRINT)=ALL                                               |
MPCFORCES(PLOT,PRINT)=ALL                                               |
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +

$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
SUBCASE 1000                                                            |
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
        LABEL=CLAMPED_BEAM                                              |
        SUBTITLE=CLAMPED_BEAM                                           |
        SPC=1000                                                        |
        LOAD=1000                                                       |
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +

BEGIN BULK
```

Reminder about DOS

Reminder

Students workstations are under non linux OS.

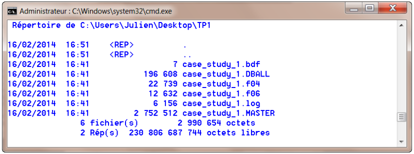


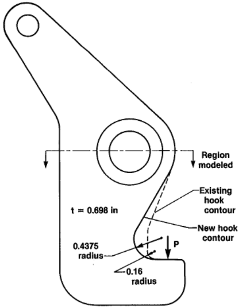
Figure 14: A DOS window.

Use `dir` to list the files in the directory.
Use `mkdir` to create a new directory.

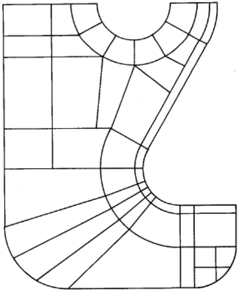
Introduction to Patran

Definition

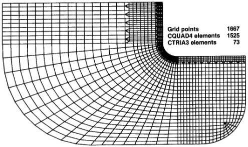
Patran is a pre and post processing software for a solver. It is a 3D Cae software with executables **patnas** and **naspat** to write and read Nastran format. Below is an example from Nasa [17].



(a) Geometry of B-52 pylon hook



(b) Patran patches for generating mesh



(c) Mesh of B-52 pylon hook

Aparté : One can also use solver algorithms to improve a mesh as shown on next slide.

Introduction to PATRAN

Definition

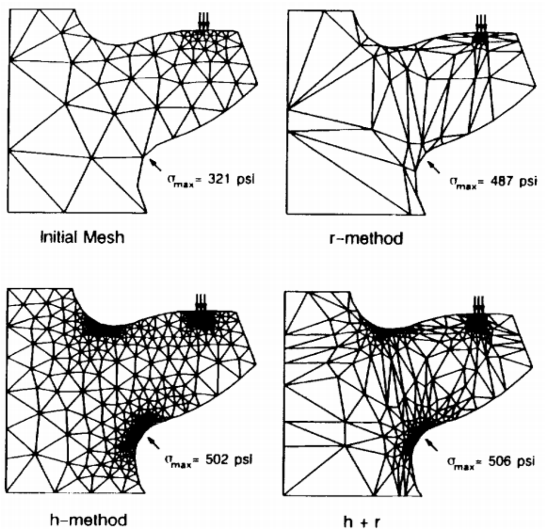


Figure 15: Adaptative mesh example for a fitting.

Introduction to PATRAN

History

- PATRAN a registered trademark of MSC.Software Corporation since they bought PDA
- PATRAN usage can be traced back before 1985 [18]
- PATRAN owns an internal progammation language called PCL

```
STRING asm_create_grid_xyz_created_ids[VIRTUAL]
asm_const_grid_xyz("1", "[0 0 0]", "Coord 0", asm_create_grid_xyz_created_ids)
```

Table 5: Example of creation of a geometrical point in PCL.

- PATRAN associated files are .db which is the binary database of command launched and data. An ASCII version called session file .ses is written in the PATRAN root directory. The .ses file contains PCL instructions.

Introduction to PATRAN

Workflow

PATRAN allows from a build or import geometry to define a mesh on a solid in order to realize a finite elements analysis with NASTRAN.

PATRAN allows to define boundary conditions, loads, materials, elements properties and launch the writing of a `.dat` file to be run with NASTRAN. In order to launch the NASTRAN run three ways of working exist :

- From PATRAN **Analysis** Menu Form with the Full run preference
- From the NASTRAN dedicated GUI on your desktop
- From the command line :

```
$ nastran input.dat app=n old=n news=n scr=y
```

The latter is the most effective way.

`.f06` is cross-checked : `GROUNDCHECK` passed, no singularities, no suspicious `WARNINGS`,...

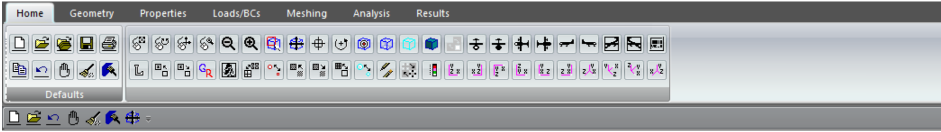
Then `.op2` result of NASTRAN run can be analyzed under PATRAN.

Introduction to PATRAN

Launch

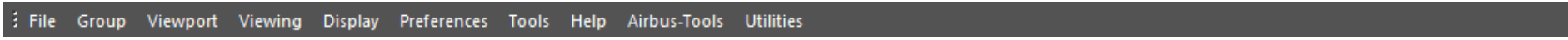
Launch PATRAN with the command line `$ patran`

PATRAN looks like this

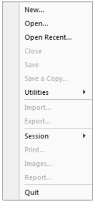


Introduction to PATRAN

Launch

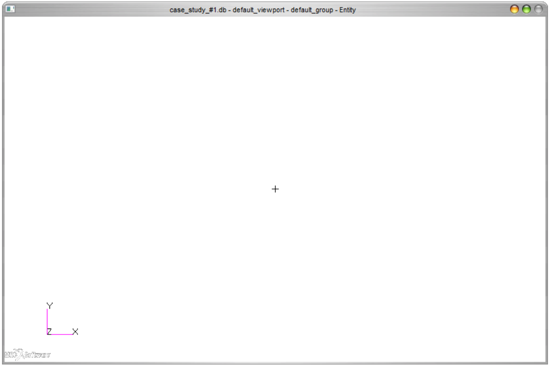


PATRAN keeps the data generated during a session in a `.db` file (database). To create a `.db`: File > New. Name your first `.db` case_study_1.



Introduction to PATRAN

Launch



A finite elements model can now be built :)

Case study # 0 - First NASTRAN run & PATRAN results visualization

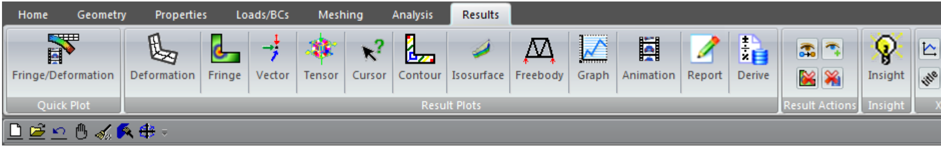
Import NASTRAN run

File > Import > MSC.NASTRAN under PATRAN

Case study # 0 - First NASTRAN run & PATRAN results visualization

Import NASTRAN results under PATRAN

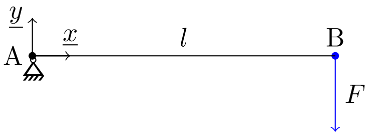
Analysis > Read Results > `.op2` and then open Results Menu Form



Case study # 1 - Clamped beam linear analysis

Definition

The framework is linear elasticity.



A load F is applied in B. The beam is clamped in A. The beam has a rectangular section $S = b \times h$ with $b = h = 60\,\mathrm{mm}$ and $l = 2000\,\mathrm{mm}$. Inertia worth $I_z = \frac{bh^3}{12}$. The beam is made of steel $E = 200\,\mathrm{GPa}$ and $\nu = 0.30$. A 81 kg mass has been hung at the beam free end.

Aim of Case study # 1 : Students have to find the closed form of the freebody diagram of the structure and cross check the NASTRAN `SOL 101` results.

Case study # 1 - Clamped beam linear analysis

Closed form solution

The displacement v satisfies

$$\frac{\partial^2 v}{\partial x^2} = \frac{M_z}{E\, I_z} \tag{12}$$

Thus the displacement worth

$$v = -\frac{F\, x^2}{6\, E\, I_z}(3l - x) \tag{13}$$

And in B displacement worth

$$v_B = -\frac{F l^3}{3\, E\, I_z} \tag{14}$$

Stress in beam section is given by

$$\sigma_{xx}(y = \pm\frac{h}{2}) = \pm\frac{F(l-x)h}{2\, I_z} \tag{15}$$

Case study # 1 - Clamped beam linear analysis

Units under PATRAN / NASTRAN

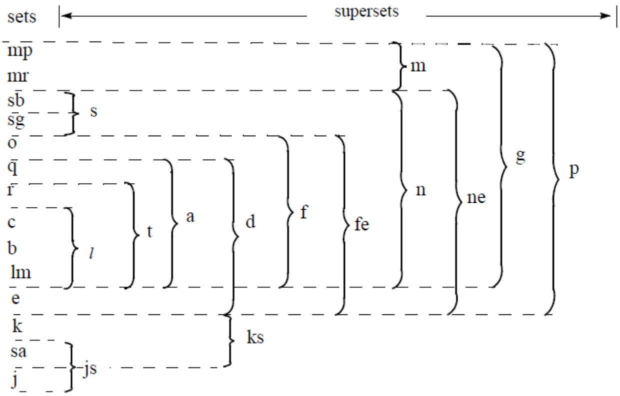
A relevant/suitable units system is to input distance in [mm]. If forces are in [N] then pressure and stresses are homogeneous to [MPa]. As every finite element solver on the market NASTRAN has no predefinite units system. The user has to take great care to it.

The same rationale applied for the mechanician sign convention. In order to assess the internal loads reacted by the structure analyzed the user has to take care the way applied loads are defined in terms of

- magnitude
- orientation

Case study # 1 - Clamped beam linear analysis

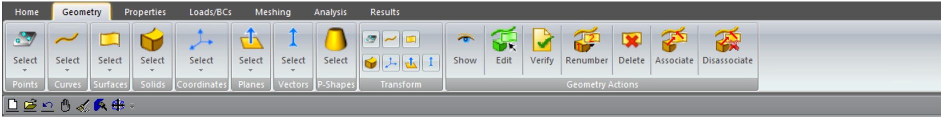
NASTRAN degrees of freedom sets



Sets of degrees of freedom are an important NASTRAN concept. Refer to NASTRAN Quick Reference Guide for deeper insight [14].

Case study # 1 - Clamped beam linear analysis

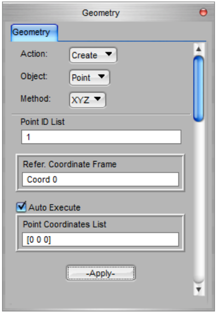
Patran Geometry Form



Case study # 1 - Clamped beam linear analysis

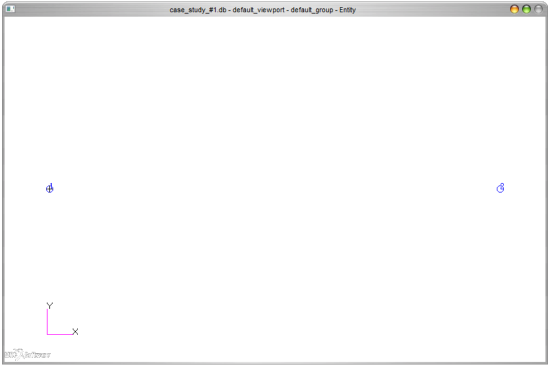
PATRAN Geometry Form

Create Point 1 at [0. 0. 0.] and Point 2 at [2000. 0. 0.].



Case study # 1 - Clamped beam linear analysis

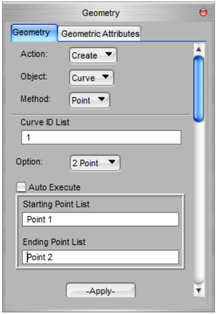
PATRAN Geometry Form



Case study # 1 - Clamped beam linear analysis

PATRAN Geometry Form

Create a curve from Point 1 to Point 2.

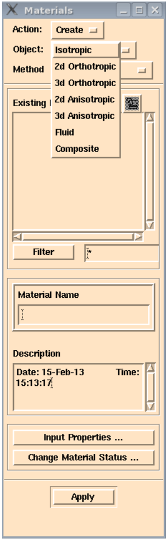


Case study # 1 - Clamped beam linear analysis

Patran Geometry Form

Case study # 1 - Clamped beam linear analysis

PATRAN Materials Form



Case study # 1 - Clamped beam linear analysis

NASTRAN PBEAM Card

Format:

1	2	3	4	5	6	7	8	9	10
PBEAM	PID	MID	A(A)	I1(A)	I2(A)	I12(A)	J(A)	NSM(A)	
	C1 (A)	C2 (A)	D1 (A)	D2 (A)	E1 (A)	E2 (A)	F1 (A)	F2 (A)	

The next two continuations are repeated for each intermediate station as described in Remark 6. and SO and X/XB must be specified.

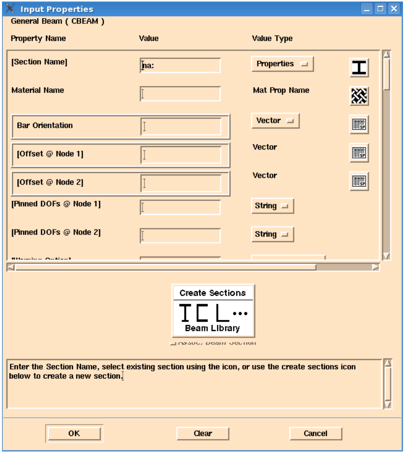
	SO	X/XB	A	I1	I2	I12	J	NSM	
	C1	C2	D1	D2	E1	E2	F1	F2	

The last two continuations are:

	K1	K2	S1	S2	NSI(A)	NSI(B)	CW(A)	CW(B)	
	M1(A)	M2(A)	M1(B)	M2(B)	N1(A)	N2(A)	N1(B)	N2(B)	

Case study # 1 - Clamped beam linear analysis

PATRAN General Beam Form



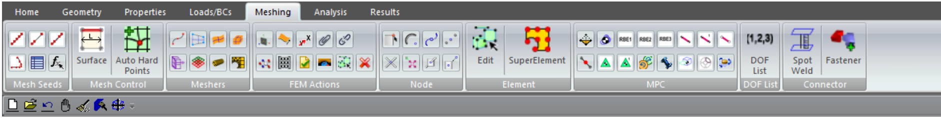
Data [•] mean Input not mandatory.
Mandatory input data in the `CBEAM` Menu Form is for `case_study_#1` the :

- Material Name to be selected through the  icon
- Area of the beam section thus $b \times h$
- `CBEAM` orientation vector `<0.,1.,0.>`
- Numerical value of $I_z = \dfrac{bh^3}{12}$ to be input as I_1
- Numerical value of $I_y = \dfrac{b^3h}{12}$ to be input as I_2

Case study # 1 - Clamped beam linear analysis

Mesh under PATRAN

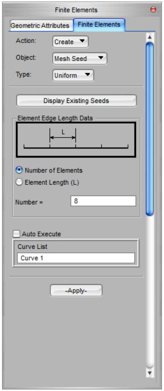
Mesh is done under **Meshing** menu



Case study # 1 - Clamped beam linear analysis

Mesh under PATRAN

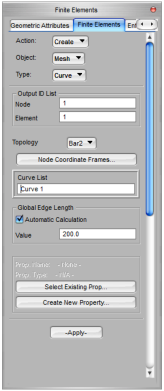
Install seeds with 8 elements for the Curve 1.



Case study # 1 - Clamped beam linear analysis

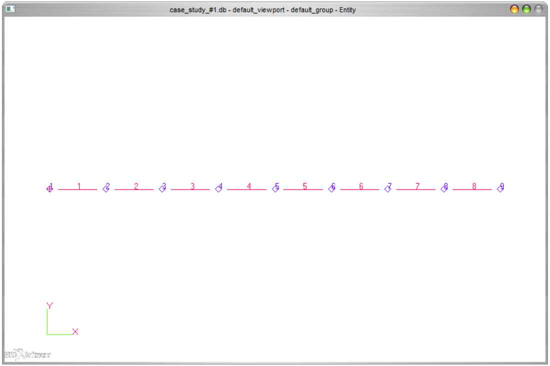
Mesh under PATRAN

Then the Curve 1 can be meshed.



Case study # 1 - Clamped beam linear analysis
Mesh under PATRAN

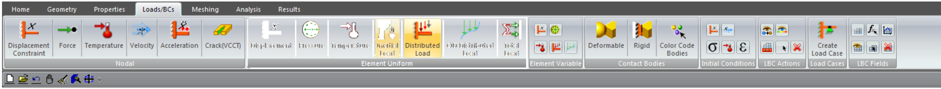
Here is a mesh proposal for case study # 1.



Case study # 1 - Clamped beam linear analysis

Boundary conditions under PATRAN

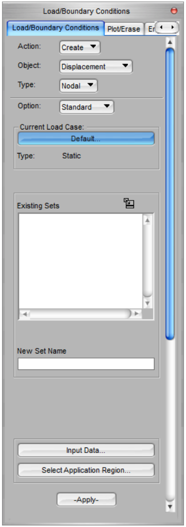
Boundary conditions are applied from **Loads / BCs** menu



Case study # 1 - Clamped beam linear analysis

Boundary conditions under PATRAN

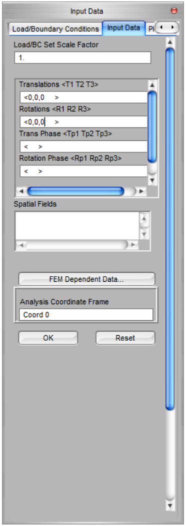
Create > Displacement > Nodal in **Loads / BCs** form



Case study # 1 - Clamped beam linear analysis

Boundary conditions under PATRAN

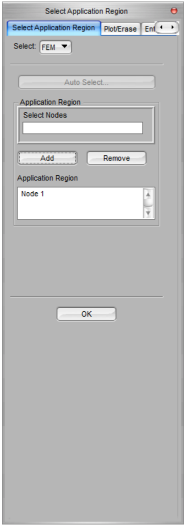
Set $< 0., 0., 0. >$ for both translation and rotation degrees of freedom



Case study # 1 - Clamped beam linear analysis

Boundary conditions under PATRAN

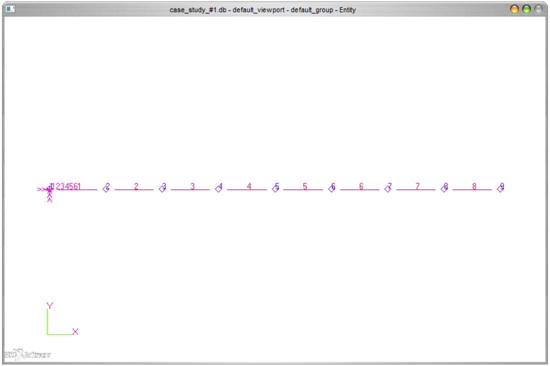
Then choose Node 1



Case study # 1 - Clamped beam linear analysis

Boundary conditions under PATRAN

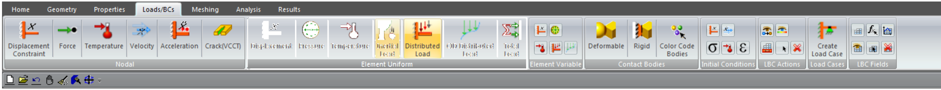
At Node 1 a flag **123456** appears : all degrees of freedom are constrained.



Case study # 1 - Clamped beam linear analysis

Loads under PATRAN

Loads are applied from **Loads / BCs** menu



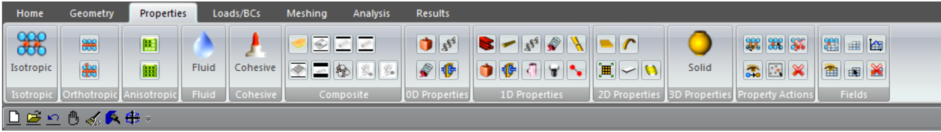
Install a NASTRAN `FORCE` under PATRAN with the → • Force Menu. It basically works as for the Boundary Conditions Menu :

- Input a vector $\vec{z}$ to support the force definition thus `<0.,-1.,0.>` and enter the magnitude of the loading desired $\vec{F} = -810\vec{y}$
- Choose Node # 9 as application location of the loading $\vec{F}$

Case study # 1 - Clamped beam linear analysis

Properties under PATRAN

Properties are applied from **Properties** menu

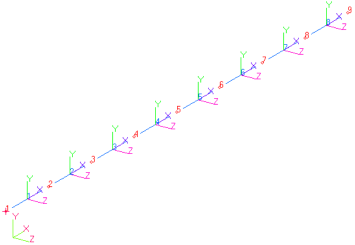


Do not forget to apply Properties (Beam I_z, Materials, ...) to the elements created.

Case study # 1 - Clamped beam linear analysis

Properties under PATRAN

Properties are applied taking into account the local coordinate system of each element. For example for NASTRAN `CBEAM` [14] element coordinate system defines the bending planes of the beam elements.

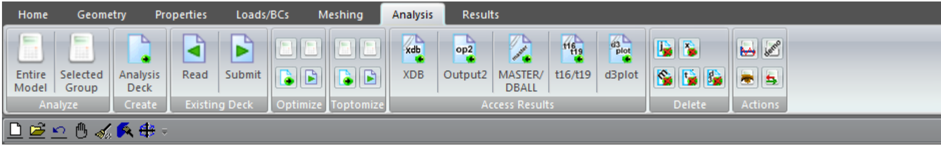


$\underline{x}$ is defined by the 2 nodes which set the topology of the beam. $\underline{y}$ is defined in the `CBEAM` card. Then $\underline{z} = \underline{x} \times \underline{y}$.

Case study # 1 - Clamped beam linear analysis

NASTRAN linear static run

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



Case study # 1 - Clamped beam linear analysis

NASTRAN Deck Cross Check

After PATRAN has written the NASTRAN Deck (a `.bdf` or `.dat` file depending upont the way PATRAN preferences are set) cross check that your NASTRAN input is relevant :

- ☐ Do I have 9 nodes defined by `GRID` cards ?
- ☐ Do I have 8 `CBEAM` elements with reference to a `PBEAM` entry ?
- ☐ Do I have 1 isotropic material defined by `MAT1` card ?
- ☐ Do I have a set of boundary conditions defined by `SPC1` card and called by a `SPC` command in Case Control Section ?
- ☐ Do I have a force defined by `FORCE` card and called by a `LOAD` command in Case Control Section ?

Case study # 1 - Clamped beam linear analysis

NASTRAN GRID Card

Format:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	

Example:

GRID	2	3	1.0	-2.0	3.0		316		

Case study # 1 - Clamped beam linear analysis

NASTRAN CBEAM Card

Format:

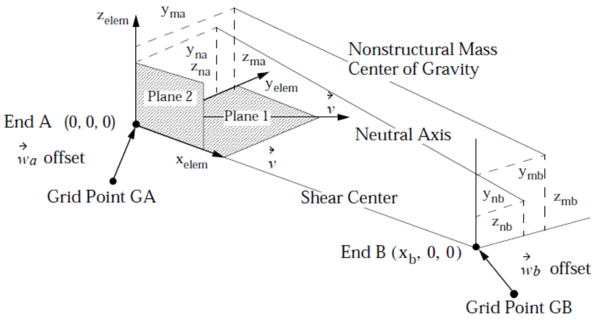
1	2	3	4	5	6	7	8	9	10
CBEAM	EID	PID	GA	GB	X1	X2	X3	OFFT/BIT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							

Example:

CBEAM	2	39	7	13	8.2	6.1	-5.6	GOG	
		513		3.0					
	8	5							

Case study # 1 - Clamped beam linear analysis

NASTRAN CBEAM Coordinate System



Case study # 1 - Clamped beam linear analysis

NASTRAN SPC Card

Format:

1	2	3	4	5	6	7	8	9	10
SPC	SID	G1	C1	D1	G2	C2	D2		

Example:

SPC	2	32	3	-2.6	5				

Case study # 1 - Clamped beam linear analysis

NASTRAN SPCADD Card

Format:

1	2	3	4	5	6	7	8	9	10
SPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc.-						

Example:

SPCADD	101	3	2	9	1				

Case study # 1 - Clamped beam linear analysis

NASTRAN FORCE Card

Format:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	N1	N2	N3		

Example:

FORCE	2	5	6	2.9	0.0	1.0	0.0		

Case study # 1 - Clamped beam linear analysis

NASTRAN LOAD Card

Format:

1	2	3	4	5	6	7	8	9	10
LOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-						

Example:

LOAD	101	-0.5	1.0	3	6.2	4			

The NASTRAN LOAD Card realizes from the forces definition F_i the linear superposition

$$F = \beta \sum \alpha_i F_i \qquad (16)$$

Case study # 1 - Clamped beam linear analysis

NASTRAN MAT1 Card

Format:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
	ST	SC	SS	MCSID					

Example:

MAT1	17	3.+7		0.33	4.28	6.5-6	5.37+2	0.23	
	20.+4	15.+4	12.+4	1003					

Case study # 1 - Clamped beam linear analysis

NASTRAN linear static run

Then run the analysis with NASTRAN

```
$ nastran case_study_1.dat news=n old=n scr=y
```

User obtains as output to NASTRAN run

- `case_study_1.log` : Control File
- `case_study_1.f04` : Execution Summary Table
- `case_study_1.f06` : ASCII Results file
- `case_study_1.op2` : Binary Results file

Case study # 1 - Clamped beam linear analysis

Results from NASTRAN linear static run from .f06

Students have to find/check the reacted force and the reacted moment in the the `.f06`.

Case study # 1 - Clamped beam linear analysis

Results from NASTRAN linear static run from .f06

Students have to find/check the reacted force and the reacted moment in the the `.f06`.

```
                       F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
      1      G      0.0            8.100000E+02   0.0            0.0            0.0            1.620000E+06
```

One has of course the balance between the applied resultant (`OLOAD` in NASTRAN) and the reacted resultant (`SPCFORCE`).

```
                                             OLOAD    RESULTANT
 SUBCASE/    LOAD
 DAREA ID    TYPE       T1            T2            T3            R1            R2            R3
            TOTALS  0.000000E+00 -8.100000E+02  0.000000E+00  0.000000E+00  0.000000E+00 -1.620000E+06

                                             SPCFORCE RESULTANT
 SUBCASE/    LOAD
 DAREA ID    TYPE       T1            T2            T3            R1            R2            R3
            TOTALS  0.000000E+00  8.100000E+02  0.000000E+00  0.000000E+00  0.000000E+00  1.620000E+06
```

Case study # 1 - Clamped beam linear analysis

Results from NASTRAN linear static run from .f06

Students have to find/check the displacement vector u in the the `.f06`.

Case study # 1 - Clamped beam linear analysis

Results from NASTRAN linear static run from .f06

Students have to find/check the displacement vector u in the the `.f06`.

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	0.0	0.0	0.0	0.0	0.0
2	G	0.0	-2.254872E-01	0.0	0.0	0.0	-1.757812E-03
3	G	0.0	-8.611307E-01	0.0	0.0	0.0	-3.281250E-03
4	G	0.0	-1.848337E+00	0.0	0.0	0.0	-4.570312E-03
5	G	0.0	-3.128511E+00	0.0	0.0	0.0	-5.625000E-03
6	G	0.0	-4.643061E+00	0.0	0.0	0.0	-6.445312E-03
7	G	0.0	-6.333392E+00	0.0	0.0	0.0	-7.031250E-03
8	G	0.0	-8.140910E+00	0.0	0.0	0.0	-7.382812E-03
9	G	0.0	-1.000702E+01	0.0	0.0	0.0	-7.500000E-03

Case study # 1 - Clamped beam linear analysis

Results from NASTRAN linear static run from .f06

Students have to find/check the displacement vector u in the the `.f06`.

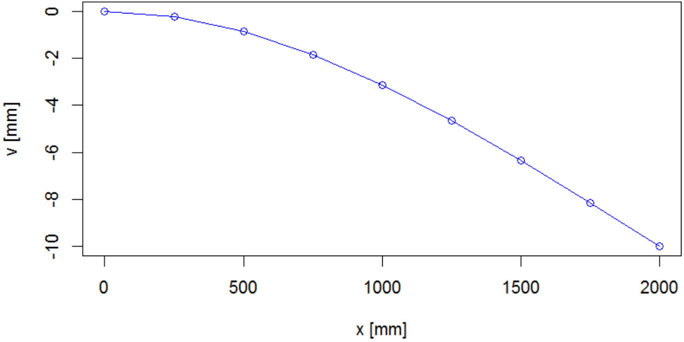


Figure 16: Plot v [mm] vs. x [mm]

Case study # 1 - Clamped beam linear analysis

Results from NASTRAN linear static run from .op2

Case study # 1 - Clamped beam linear analysis

Results from NASTRAN linear static run from .op2

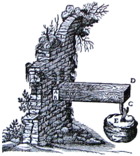
Students have to plot the reacted force and the reacted moment in PATRAN after the import of the `.op2`.

Case study # 1 - Clamped beam linear analysis

Conclusion

Nastran permits to solve a strength of materials problem easily. Great care has to be taken for the definition of

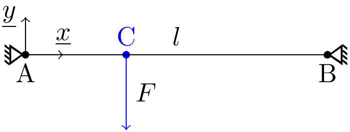
- units
- boundary conditions
- loading
- mesh
- relevant post-processing



Galileo Galilei [19]

Case study # 1
Outlook

The framework is linear elasticity.



A load F is applied in C $x = \dfrac{l}{3}$. The beam is clamped in A and B. The beam has a rectangular section $S = b \times h$ with $b = h = 60\,\text{mm}$ and $l = 2000\,\text{mm}$. Inertia worth $I_z = \dfrac{bh^3}{12}$. The beam is made of steel $E = 200\,\text{GPa}$ and $\nu = 0.30$. A 10^3 kg mass has been hung at C.

Aim of Outlook # 1 : Students have to find the freebody diagram of the structure through a NASTRAN `SOL 101` with Case Study #1 `.dat` file as a start point.

Case study # 1

Outlook

Given next parameters

$$\begin{cases} c_1 = +\dfrac{4Fl}{27EI} \\ c_2 = -\dfrac{20F}{27EI} \end{cases} \qquad (17)$$

the displacement v is derived through [20] :

$$v = \frac{1}{2}c_1 x^2 + \frac{1}{6}c_2 x^3 \qquad \text{for} \qquad x < \frac{l}{3} \qquad (18a)$$

$$v = \frac{1}{2}c_1 x^2 + \frac{1}{6}c_2 x^3 + \frac{F}{6EI}\left(x - \frac{l}{3}\right)^3 \qquad \text{for} \qquad x \geq \frac{l}{3} \qquad (18b)$$

Case study # 1

Outlook

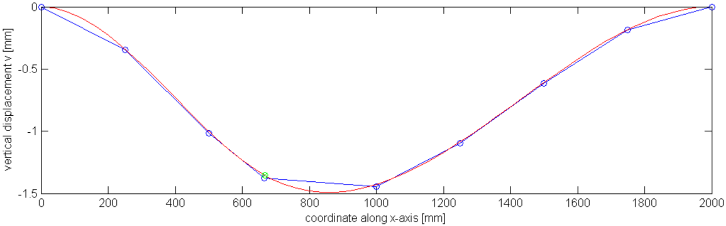


Figure 17: If the mesh is too coarse the maximum displacement of the beam is not accurately output by the finite element model (rationale is applicable also to boundary conditions locations). — NASTRAN `SOL 101` — closed form solution. Locations where $\dfrac{\partial^2 v}{\partial x^2} = 0$ are $x = \dfrac{l}{5}$ and $x = \dfrac{5l}{7}$. One encounters at $x = \dfrac{3l}{7}$ that $\min v(x) = -\dfrac{16}{3969}\dfrac{Fl^3}{EI}$. Curve plot & NASTRAN run for $E = 200\,\text{GPa}$ and $F = -10^4\,\text{N}$.

Case study # 1

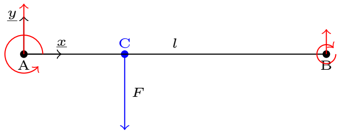
Outlook

```
$ NASTRAN SYSTEM(210)=2
$ Executive Control Section - - - - - - - - - - - - - - - - - - - - - - +
SOL 101
CEND
$ Case Control Section- - - - - - - - - - - - - - - - - - - - - - - - - +
ECHO=NONE
TITLE = CASE_STUDY_#1

DISP(PLOT,PRINT,PUNCH)=ALL
FORCE(PLOT,PRINT)=ALL
OLOAD(PLOT,PRINT,PUNCH)=ALL
STRESS(PLOT,PRINT)=ALL
GPFORCE(PLOT,PRINT,PUNCH)=ALL
SPCFORCES(PLOT,PRINT)=ALL
MPCFORCES(PLOT,PRINT)=ALL

SUBCASE 1000  $ - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
        LABEL=CLAMPED_BEAM                                              |
        SUBTITLE=CLAMPED_BEAM                                           |
        SPC=1000                                                        |
        LOAD=1000                                                       |
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +

$ Bulk Data - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
BEGIN BULK
$ Bulk Parameters - - - - - - - - - - - - - - - - - - - - - - - - - - - +
PARAM,AUTOSPC,YES
PARAM,POST,-1
$ 9 Nodes
$1      2       3       4       5       6       7       8       9       0
GRID    1       0       0.00    0.00    0.00
GRID    2       0       250.00  0.00    0.00
GRID    3       0       500.00  0.00    0.00
GRID    4       0       666.66  0.00    0.00
GRID    5       0       1000.00 0.00    0.00
GRID    6       0       1250.00 0.00    0.00
GRID    7       0       1500.00 0.00    0.00
GRID    8       0       1750.00 0.00    0.00
GRID    9       0       2000.00 0.00    0.00
$ 8 Elements
$1      2       3       4       5       6       7       8       9       0
CBEAM   1       1       1       2       0.      1.      0.
CBEAM   2       1       2       3       0.      1.      0.
CBEAM   3       1       3       4       0.      1.      0.
CBEAM   4       1       4       5       0.      1.      0.
CBEAM   5       1       5       6       0.      1.      0.
CBEAM   6       1       6       7       0.      1.      0.
CBEAM   7       1       7       8       0.      1.      0.
CBEAM   8       1       8       9       0.      1.      0.
$ Properties
$1      2       3       4       5       6       7       8       9       0
$ S =3600 mm2
$ Iz=1080000 mm2 x mm2
PBEAM   1       1       3600.   1.080E+6100.            1822500.
+       30.00   0.000   -30.00  0.000
        0.833   0.833
$ Materials
$1      2       3       4       5       6       7       8       9       0
MAT1    1       200000. 76923.  0.30    7.8E-6
$ B.C.
$1      2       3       4       5       6       7       8       9       0
SPCADD  1000    1
SPC     1       1       123456
SPC     1       9       123456
$ Loading
$1      2       3       4       5       6       7       8       9       0
LOAD    1000    1.      1.      1
FORCE   1       4       0       10000.  0.      -1.     0.
ENDDATA
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
```

Answer #1 : The proposal NASTRAN answer has been built from Case Study #1 run file.

The node at $x = 750.$ mm was moved at $x = \dfrac{l}{3}$ in order to have a node suitable to code NASTRAN `FORCE` Card.

Boundary conditions is extended vs. Case Study #1 to include the node at B in the `SPC1`(Id.=1) Set. Freebody diagram to find is drawn hereafter from numerical values found in the `SPCFORCES` Table within the `.f06` :



NASTRAN output in the `.f06` the applied loads as `OLOAD RESULTANT` Table at the origin of the basic coordinate system. The latter Table is balanced by the `SPCFORCE RESULTANT` Table. But it may not be enough to write the freebody diagram of the structure that's why :

1. Nodal applied loads and moments are output in the `.f06` through the `OLOAD(PRINT)=ALL` Case Control command
2. Nodal reaction loads and moments are output in the `.f06` through the `SPCFORCES(PRINT)=ALL` Case Control command

This approach allows hand calculations cross check.

Case study # 2 - Clamped beam nonlinear analysis

Theoretical Aparté

Non linear continuum mechanics allows to account e.g.

- materials non linearity i.e. loss of linearity in the stress/strain relationship. Typical cases are plasticity or finite elasticity for hyperelastic materials like rubber or polymer.
- geometrical non linearity stem from large rotations, large deformations or large displacements. Second order term or cross term in equilibrium equation have a numerical impact on the results of the calculation. Typical case is structural non linearity associated to shells as described e.g. by Föppl - von Kármán equation.
- follower forces
- contact

Case study # 2 - Clamped beam nonlinear analysis

Theoretical Aparté

Important ideas :

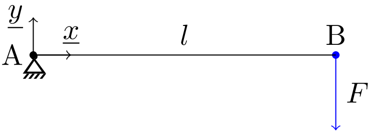
1. non linearity may be considered as the general case and not the exception
2. numerical methods associated to the linear problems are unefficient and new tools are necessary
3. start with a linear analysis first remains a good practice

Case study # 2 offers a flavour of geometrical non linearity.

Case study # 2 - Clamped beam nonlinear analysis

Definition

The framework is geometrical non linearity applied to the Case study # 1.



The beam is clamped in A. The beam has the same materials and geometrical properties than for case study # 1. A force F is applied at the free end and then a moment M is applied at the free end.

Aim of Case study # 2 : Students have to run a non linear analysis of the structure and cross check the NASTRAN `SOL 106` results. The bending inertia of the beam can be decreased in order to submit the structure to large rotations without applying a huge M.

Case study # 2 - Clamped beam nonlinear analysis

Closed form solution

Closed form solution is seldom available for a non linear problem. The displacement v for non linear geometrical analysis case study # 2 has to account for the reduction of moment lever arm as well as second order term in curvature definition.

The article offers an historical treatment of the cantilever beam with a prescribed moment. Refer to [4, 21] for more complete approach to Finite Element Method applied to non linear continuum mechanics.

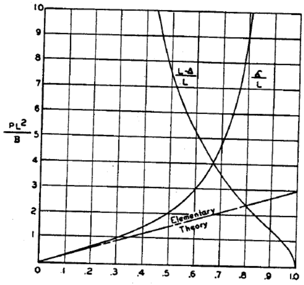


Figure 18: Plot from [22] with $B = EI_z$

Case study # 2 - Clamped beam nonlinear analysis

NASTRAN NLPARM Card

In order to run a SOL 106 in NASTRAN one has to code a NLPARM Card.

```
$-----------------------------------------------------------------------+
$                                   NLPARM  CARD                        |
$-----------------------------------------------------------------------+
$1      2       3       4       5       6       7       8       9       0
$       Id.     NINC    DT      KMETHOD KSTEP   MAXITER CONV    INOUT
NLPARM  1000    10              AUTO    1                       NO      +
$       EPSU    EPSP    EPSW
+               1.0E-3  1.0E-7
$-----------------------------------------------------------------------+
```

Table 6: NASTRAN NLPARM Card to be inserted in the bulk section and to be called in the Case Control Section by a NLPARM=1000 Command.

The NLPARM card allows to implement an incremental load history compatible with SOL 106 iterative approach. Therefore user drives the update of e.g. tangent stiffness matrix or follower forces.

Case study # 2 - Clamped beam nonlinear analysis

NASTRAN LGDISP Parameter

In order to run a texttttSOL 106 in NASTRAN with large displacement (geometrical non linearity) one has to use `LGDISP` parameter.

```
.
.
.
BEGIN BULK
$ Bulk Section Parameters
PARAM     LGDISP  1
.
.
.
```

Set to -1 the `LGDISP` parameter means no geometrical non linearity.

Set to 2 the `LGDISP` parameter means geometrical non linearity without follower forces.

Case study # 2 - Clamped beam nonlinear analysis

Conclusion & Outlook

- ☑ NASTRAN `SOL 106` permits to solve a non linear problem efficiently
- ✍ Analyze deeper `SOL 106` through implemention of a non linear stress / strain relationship to account for a material non linearity

References

-

[1] J.T. Oden and J.N. Reddy.
Variational methods in theoretical mechanics.
Universitext Series. Springer-Verlag, 1983.

[2] J.T. Oden and J.N. Reddy.
An introduction to the mathematical theory of finite elements.
Wiley, 1976.

[3] O.C. Zienkiewicz.
The Finite Element Method in engineering science.
McGraw-Hill, 1977.

[4] J.T. Oden.
Finite elements of nonlinear continua.
McGraw-Hill, 1971.

[5] J.L. Synge.
The Hypercircle in Mathematical Physics: A Method for the Approximate Solution of Boundary Value Problems.
University Press, 1957.

[6] MSC Software.
50 Years of MSC Software - When Simulation Met Reality.
Simulating Reality Magazine, III, Spring 2013.

[7] B. Drouin and J.M. Senicourt.
De la mécanique vibratoire classique à la méthode des éléments finis.
Number vol. 1. AFNOR, 1993.

[8] Oberkampf & Roy.
Verification and Validation in Scientific Computing.
Cambridge University Press, 2010.

[9] R. MacNeal.
Finite Elements: Their Design and Performance.
Dekker, 1994.

[10] P. Breton.
Une histoire de l'informatique.
Collection Points. Série Sciences. Éd. du Seuil, 1990.

[11] A.O. Cifuentes.
Using MSC/NASTRAN*: statics and dynamics*.
Springer-Verlag, 1989.

[12] H.G. Schaeffer.
MSC/NASTRAN *Primer: Static and Normal Modes Analysis*.
Wallace Press, 1988.

[13] R. Heim.
FEM mit NASTRAN*: Einführung und Umsetzung mit Lernprogramm UNA*.
Hanser Fachbuchverlag, 2005.

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

[15] Jet Propulsion Laboratory / Pasadena.
MSC/NASTRAN Model Checkout.
MSC/NASTRAN *Users's Conference*, March 1986.

[16] MSC Software / Louis Komzsik.
Numerical Methods User's Guide.
MacNeal-Schwendler Corporation, 2004.

[17] William L. Ko and Lawrence S. Schuster.
Stress analyses of B-52 pylon hooks.
NASA *Technical Memorandum 84924*, oct 1985.

[18] PDA ENGINEERING.
PAT/MSC-NASTRAN Application Interface.
PDA Engineering, Software Products Division, 1985.

[19] Galileo Galilei.
Dialogues Concerning Two New Sciences.
1638.

[20] M.R. Spiegel.
Transformadas de Laplace.
Schaum's outline series. McGraw-Hill, 1991.

[21] J. Bonet and R.D. Wood.
Nonlinear Continuum Mechanics for Finite Element Analysis.
Cambridge University Press, 1997.

[22] K. E. Bisshopp and D. C. Drucker (Armour Research Foundation).
Large deflection of cantilever beams.
Quarterly of Applied Math, 3(3):272 – 275, 1945.

[23] MSC Software.
MSC Nastran 2016 Reference Manual.
MacNeal-Schwendler Corporation, 2016.

[24] K.B. Broberg.
Cracks and Fracture.
Elsevier Science, 1999.

[25] D.R.J. Owen and A.J. Fawkes.
Engineering Fracture Mechanics: Numerical Methods and Applications.
Pineridge Press Limited, 1983.

[26] R. Krueger.
The Virtual Crack Closure Technique: History, Approach and Applications.
NASA CR 211628, Avril 2002.

[27] A. C. Orifici and R. Krueger.
Assessment of Static Delamination Propagation Capabilities in Commercial Finite Element Codes Using Benchmark Analysis.
NASA CR 216709, June 2010.

[28] J. Schijve.
Fatigue of Structures and Materials.
Springer Netherlands, 2008.

§A - Relations between elastic moduli

The LAMÉ parameters can be written as functions of YOUNG's modulus E and POISSON's cœfficient ν

$$\lambda = \frac{\nu E}{(1+\nu)(1-2\nu)} \qquad (19)$$

$$\mu = G \qquad (20)$$

§B - List of symbols

$\boldsymbol{\sigma}$ stands for the stress tensor.

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_{xx} & \sigma_{yx} & \sigma_{zx} \\ \sigma_{xy} & \sigma_{yy} & \sigma_{zy} \\ \sigma_{xz} & \sigma_{yz} & \sigma_{zz} \end{bmatrix} \tag{21}$$

$\boldsymbol{\varepsilon}$ stands for the small strain tensor.

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{xx} & \varepsilon_{yx} & \varepsilon_{zx} \\ \varepsilon_{xy} & \varepsilon_{yy} & \varepsilon_{zy} \\ \varepsilon_{xz} & \varepsilon_{yz} & \varepsilon_{zz} \end{bmatrix} \tag{22}$$

$\boldsymbol{E}$ stands for the Green strain tensor.

$$\boldsymbol{E} = \boldsymbol{\varepsilon} + \frac{1}{2}\nabla u(\nabla u)^T \tag{23}$$

§C - Nastran vs. Abaqus

Hereafter it a mixed meshed solid $a \times a \times t = 50 \times 50 \times 5\,\mathrm{mm}^3$ with a circular hole $\varnothing = \frac{3}{8}$ in analyzed under Nastran and Abaqus clamped at one side and withstanding a tensile load $\sigma_0 = 100\,\mathrm{MPa}$ on the opposite side (along an axis oriented at $\theta = +\frac{\pi}{6}$ in regard of basic coordinate system with $\underline{x} \equiv \underline{e}_r$ and $\underline{y} \equiv \underline{e}_\theta$).

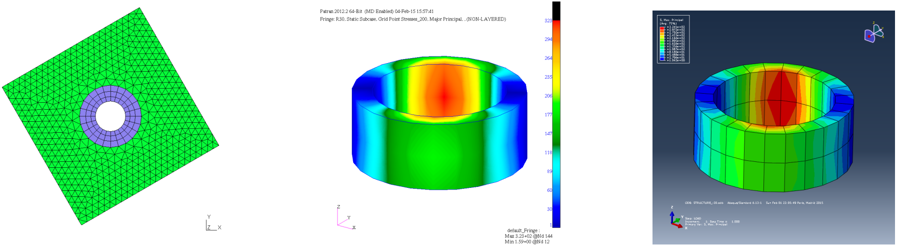


Figure 19: Nastran vs. Abaqus.

Mesh includes linear hexahedral (8 nodes) and parabolic tetrahedral elements (10 nodes). The hybrid mesh has thus a boundary steered through `MPC` under Nastran and with an `*TIE` under Abaqus.

§C - Nastran vs. Abaqus

The mean of the discrepancies of the maximum σ_I around the hole at z = MID is 1%. Nastran leads to $\sigma_I = 323\,\text{MPa}$ (`GRID POINT STRESS`) and Abaqus leads to $\sigma_I = 324\,\text{MPa}$ (`*EL PRINT, POSITION=AVERAGED AT NODES`) thus for the node $\arg\max \sigma_I$ a 0.20% discrepancy at $\theta = -\dfrac{\pi}{3}$ in Nastran basic coordinate system $\underline{x} \equiv \underline{e}_r$ and $\underline{y} \equiv \underline{e}_\theta$.

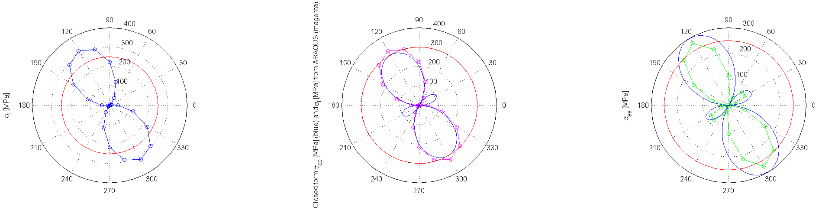
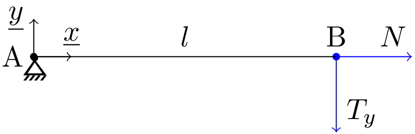


Figure 20: Abaqus σ_I vs. θ and $\sigma_{\theta\theta}$ vs. θ polar plots. Notice for the benchmark the unconverged $\sigma_{\theta\theta}$ in regard of closed form solution better assessed by σ_I.

§D - Challenges

Beam Theory Reminder

The framework is linear elasticity.



A shearing force T_y and a normal force N are applied in B. $T_y = N = 1000\,\text{N}$.The beam is clamped in A. The beam has a rectangular section $S = b \times h$ with $b = h = 60\,\text{mm}$ and $l = 2000\,\text{mm}$. Inertia worth $I_z = \dfrac{bh^3}{12}$. The beam is made of steel $E = 200\,\text{GPa}$ and $\nu = 0.30$.

Aim of Case study : Students have to Check if NASTRAN has a beam element formulation that takes into account a displacement due to the warping of the section Ω

$$\frac{\partial v_{T_y}}{\partial x} = \frac{T_y}{2K_y G} \qquad (24)$$

with

$$\frac{1}{K_y} = \frac{S}{I_z^2}\int_{h_\Omega} \frac{\mathcal{A}_z^2(\mathcal{D})}{b}\,dy = \frac{6}{5} \qquad (25)$$

§D - Challenges

Beam Theory Reminder

Assessment of the normal stress σ_{xx} in a section Ω at the upper and the lower station $(x, y = \pm\frac{h}{2}, z = 0)$ in local coordinate system *as per* slide 75/116

- σ_{xx} Tensile Contribution

$$\sigma_{xx} = \frac{N}{S} + \frac{M_z}{I_z} \times y = \frac{N}{S} \pm \frac{6F(l-x)}{bh^2}$$

- σ_{xx} Bending Contribution
- τ_{xy}

$$\tau_{xy} = \frac{T_y}{b \times I_z}\mathcal{A}_z(\mathcal{D}) = \frac{T_y}{2I_z}\left(\left(\frac{h}{2}\right)^2 - y^2\right)$$

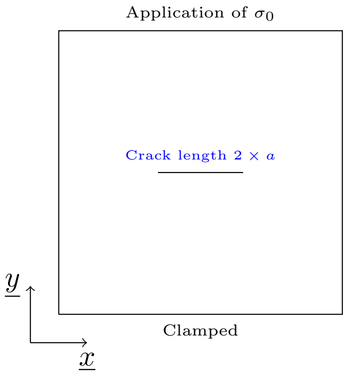
with

$$I_z = \int_\Omega y^2 dS = b \times \int_{-\frac{h}{2}}^{+\frac{h}{2}} y^2 dy = \frac{bh^3}{12} \quad \ldots\ldots\ldots\ldots \quad (26)$$

§D - Challenges

Linear Fracture Mechanics

The framework is linear fracture mechanics.



An aluminium $t = 3\,\text{mm}$ thick plate $b \times b = 100 \times 100\,\text{mm}^2$ with a $2a = 30\,\text{mm}$ long centered crack widstands a tensile stress $\sigma_0 = 100\,\text{MPa}$ along y-axis.

Aim of Case study : Students have to assess the stress intensity K_{I} factor associated to the crack tip with a NASTRAN `SOL 101` run.

Hint : Use NASTRAN `CRAC2D` element and the associated `PRAC2D` & `ADUM8` cards [23]. The former is not part of the standard library but is a dummy element.

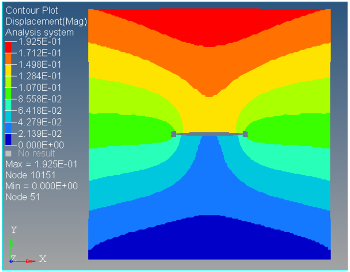
Answer : NASTRAN computes stress intensity factors K. One has for Mode I a stress intensity factor given by the formula

$$K_I = \sqrt{\frac{1}{\cos\frac{\pi a}{b}}}\,\sigma\sqrt{\pi a} \qquad (27)$$

The coefficient $\left\{\sec\dfrac{\pi a}{b}\right\}^{\frac{1}{2}}$ takes into account non finite dimensions of the plate (in the open literature often called β for the exotic configurations ; $\beta = f\left\{\dfrac{a}{b}\right\}$). Students can refer to [24] to have closed form expression of stress intensity factors or ASTM STP for practical engineering formulae.

§D - Challenges

Linear Fracture Mechanics



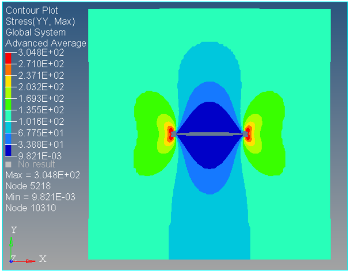
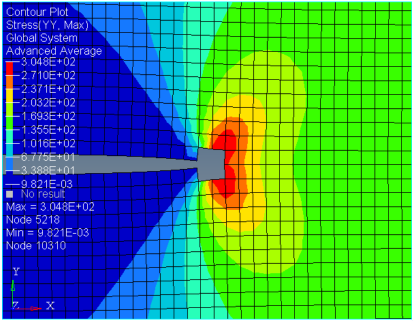


Figure 21: σ_{yy} [MPa] - The element CRAC2D is not written in the .op2 GEOM2 table and thus the element does not participate to the visualization field. The CRAC2D allows through its formulation to have components of stress tensor $\propto \dfrac{1}{\sqrt{r}}$ from crack tip (if user only uses classical elements results of the analysis as σ are mesh dependent : the usage of the appropriate element at crack tip allow to have a numerical solution with linear fracture mechanics main assumptions embedded [25]).



§D - Challenges

Linear Fracture Mechanics

Table 7: Output of NASTRAN CRAC2D in .f06 [23].

S1	S2	S3	S4	S5	S6	S7
x	y	σ_{xx}	σ_{yy}	τ_{xy}	K_{I}	K_{II}

```
                        S T R E S S E S   I N   U S E R   E L E M E N T S (CDUM8)

EL-ID       S1            S2            S3            S4            S5             S6            S7
    9  4.99995E-01  8.62500E-07  2.60449E+02  4.78151E+02 -4.15613E+00  7.33890E+02 -8.02078E+00
   10  5.00000E-01  8.62500E-07  2.60449E+02  4.78149E+02 -4.15613E+00  7.33889E+02 -8.02079E+00
```

Table 8: Extract of NASTRAN CRAC2D in .f06. Output of $K_{\mathrm{I}} = 730\,\mathrm{MPa.mm}^{\frac{1}{2}}$ is in quite good agreement with $1.05 \times 100\sqrt{\pi \times 15}$ (direct application of Formula (27)). Output of σ_{yy} in .f06 allows to derive a radius r inline with linear fracture mechanics formula $\dfrac{K_{\mathrm{I}}}{\sqrt{2\pi r}}$: here one has $r \sim 0.40\,\mathrm{mm} < \dfrac{h}{2}$ with h the mesh size 1.00 mm (in fact formulation of CRAC2D may use $r = \dfrac{1}{2\pi}\left\{\dfrac{K_{\mathrm{I}}}{\sigma_{yy}}\right\}^2$ and thus here $r \sim \dfrac{3h}{8}$).

§D - Challenges

Linear Fracture Mechanics

Nota : In the aeronautical open literature [26] a numerical trick to compute strain energy release rate associated to a flaw has been documenting for more than a decade as VCCT : it states for a simple flaw configuration that G can be derived from a coarse mesh (here square as `CQUAD4` 1.00 mm×1.00 mm) computing mechanical work done at crack tip. The latter is particularly practical to deal with composite materials. But let's derive from the configuration hereabove (isotropic material) the value of K_{I} by VCCT. One needs at crack tip :

- $[\![v]\!] = v_+ - v_-$
- Freebody F_y (upper elements action force on crack tip node)

```
$ Nodes to compute displacement discontinuity [[v]]...
SET 1=10324,10354
$ Freebody Contributions of elements 5065 & 5066 at node...
SET 2=10325
$ Output print to .f06
DISP(PRINT)=1
GPFO(PRINT)=2
```

Table 9: NASTRAN Case Control Output to derive G from VCCT : displacement field at 2 nodes and a single freebody (NASTRAN run without `CRAC2D` elements).

§D - Challenges

Linear Fracture Mechanics

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
10324	G	-2.178510E-02	6.744803E-02	0.0	0.0	0.0	9.065988E
10354	G	-2.207356E-02	9.888387E-02	0.0	0.0	0.0	-9.169104E

G R I D   P O I N T   F O R C E   B A L A N C E

POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2
10325	4965	QUAD4	-3.488973E+02	-7.262230E+02	0.0	0.0	0.0
10325	4966	QUAD4	3.674865E+02	-7.600061E+02	0.0	0.0	0.0
10325	5065	QUAD4	-3.784830E+02	7.367823E+02	0.0	0.0	0.0
10325	5066	QUAD4	3.598938E+02	7.494468E+02	0.0	0.0	0.0
10325		*TOTALS*	-3.637979E-12	2.296474E-11	0.0	0.0	0.0

Table 10: Output of NASTRAN in .f06 to allow a VCCT computation.

§D - Challenges

Linear Fracture Mechanics

By VCCT applied to a coarse simple mesh :

$$G_{\mathrm{I}} = \frac{1}{2 \times a \times t} [\![v]\!] \cdot F_y \quad (28)$$

$$= \frac{1}{2 \times 1.00 \times 3.00} (0.0989 - 0.06745) \times (760 + 726) \quad (29)$$

$$= 7.790 \cdot 10^{-3} \text{ J.mm}^{-2} \quad (30)$$

And under plane strain assumption one derives :

$$K_{\mathrm{I}} = \left\{ \frac{E \times G_{\mathrm{I}}}{1 - \nu^2} \right\}^{\frac{1}{2}} \quad (31)$$

$$= \left\{ \frac{70 \cdot 10^3}{1 - 0.3^2} \times G_{\mathrm{I}} \right\}^{\frac{1}{2}} = 774 \,\text{MPa.mm}^{\frac{1}{2}} \quad (32)$$

Nota : VCCT is in the order of magnitude of the direct application of Formula (27) or `CRAC2D` output printed in Table (8).

Outcome : VCCT is more straightforward through `SOL 400` with dedicated cards and simple crack propagation functionality [14, 27] (mixed mode propagation today open problems may not be fully solved by numerical simulations without physical tests to demonstrate e.g. relevancy of delamination/propagation law [8]).

Strength of Materials Aparté : the loading σ_0 is chosen sufficiently high to raise of course the possibility of leaving the theory of elasticity framework particularly at crack tip : it would depend upon the aluminium alloy chosen (max σ_{I} and K_{I} are above poor aluminium alloys yield strength and K_{Ic}). After setting the properties of an aluminium alloy and mechanical criterion one could run a NASTRAN run with material nonlinearity (plasticity). A vast literature exists between the aforementioned mechanical quantities and the integrity of the structure (fatigue & damage tolerance analysis [28]).

§D - Challenges

Predator Drone Q1

The framework is linear elasticity.

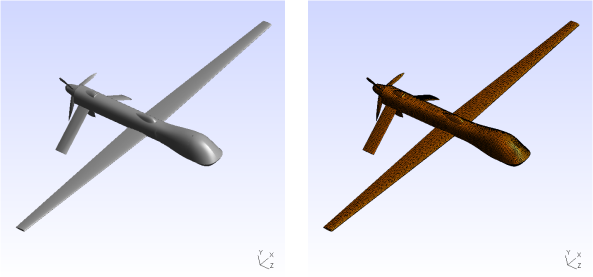


Figure 22: Master Surface definition ans a mesh proposal for the drone.

Shape of the drone : Students have to download a shape from https://grabcad.com/library.

Aim of Case study : Students have to study the effect of wing bending on the body structure of the drone through NASTRAN `SOL 101`. If time left run a geometrical nonlinear analysis has to be run.

§D - Challenges

Stability of a Spacecraft

In case of extra time assess the stability of the next spacecraft.

Figure 23: LEGO® Jedi-Interceptor™ 75038.

Flight demonstation may be watched here. May the force be with you.