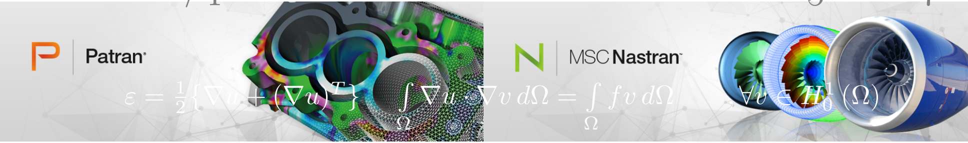
Patran / Nastran

Lecture 2/4 March 15^{th} 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

Finite Element Method's Theory

Weak form

One solves POISSON's equation :

$$\begin{cases} -\Delta u = f & \text{in} \quad \Omega \\ u = 0 & \text{on} \quad \partial\Omega \end{cases} \tag{33}$$

It is considered $f \in L^2(\Omega)$. It is assumed $u \in H^2(\Omega)$. Thus by GREEN formula

$$\int_{\Omega} \nabla u \cdot \nabla v d\Omega = \int_{\Omega} f v d\Omega + \int_{\partial\Omega} \frac{\partial u}{\partial n} v d\Gamma \tag{34}$$

If it is chosen $v \in H_0^1(\Omega)$ one has

$$\int_{\Omega} \nabla u \cdot \nabla v d\Omega = \int_{\Omega} f v d\Omega \qquad , \forall v \in H_0^1(\Omega) \tag{35}$$

The equation (35) is called the variational formulation or the weak form of the differential equation (33).

Finite Element Method's Theory

Simple axial element

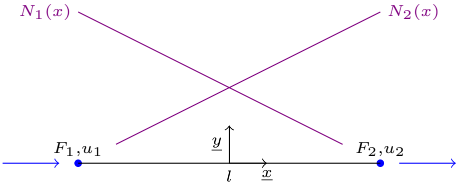


Figure 24: Simple axial element assumption sketch

Linear form function

$$\left\{ \begin{array}{l} N_1(x) = \frac{1}{l}\left(\frac{l}{2} - x\right) \\ N_2(x) = \frac{1}{l}\left(x + \frac{l}{2}\right) \end{array} \right. \qquad (36)$$

Strain worth

$$\varepsilon = B \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} \frac{\partial N_1}{\partial x}, & \frac{\partial N_2}{\partial x} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} -\frac{1}{l}, & \frac{1}{l} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} \qquad (37)$$

Finite Element Method's Theory

Simple axial element

Stiffness matrix is derived as

$$K = \int_{\Omega} B^T E B d\Omega = S \int_{-\frac{l}{2}}^{\frac{l}{2}} \frac{E}{l^2} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} dx \tag{38}$$

The simple bar element works merely as*

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \frac{ES}{l} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{39}$$

with stiffness matrix

$$K = \frac{ES}{l} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{40}$$

Nota : Mass matrix is derived as

$$M = \frac{\rho S l}{6} \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \tag{41}$$

NASTRAN's user can output K, M or any matrix in an `.op2` or an `.op4` file (cf. next Figure 27).

* ... a $F = k\,u$ spring ☺

Finite Element Method's Theory

Simple bending element

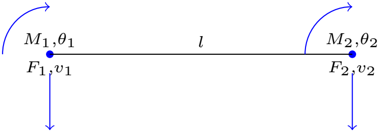


Figure 25: Simple bending element assumption sketch

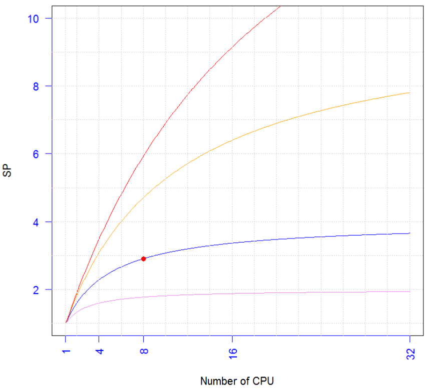
For a simple bending element one has the relationship

$$\left\{ \begin{array}{c} F_1 \\ M_1 \\ F_2 \\ M_2 \end{array} \right\} = \frac{2EI}{l^3} \left[\begin{array}{cccc} 6 & -3l & -6 & -3l \\ -3l & 2l^2 & 3l & l^2 \\ -6 & 3l & 6 & 3l \\ -3l & l^2 & 3l & 2l^2 \end{array} \right] \left\{ \begin{array}{c} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{array} \right\} \quad \ldots \quad (42)$$

Computer Performance

AMDAHL's law

Nowadays every computer comes with more than one CPU. The time elapsed for a single run (sometimes refered as wallclock time) desired to be be fine tuned will comply to AMDAHL's law (1967):



$$\mathrm{SP} = \frac{1}{S + \frac{1-S}{R}} \quad \ldots\ldots \quad (43)$$

Opposite are plotted some speed-up reachable from 50% (pink) to 95% (red) //-ed code.

For example the speed-up obtained for a program whose 75% (thus $1 - S = 75\%$) of its internal code take benefit of 8 CPU in comparison to the serial part (ratio R) is:

$$\mathrm{SP} = \frac{1}{0.25 + \frac{0.75}{8}} \sim 3 \quad \ldots \quad (44)$$

The latter is the ● opposite.

Refer to modern textbooks about parallel computing (as [1]) or higher performance computing (CPU/GPU) for deeper insight.

Computer Performance

INTEL/MSC.NASTRAN 2016 α Benchmark

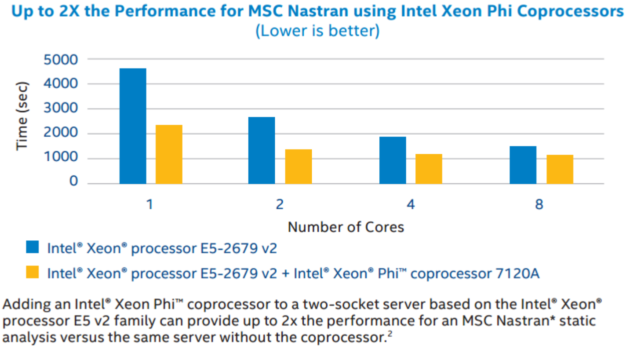
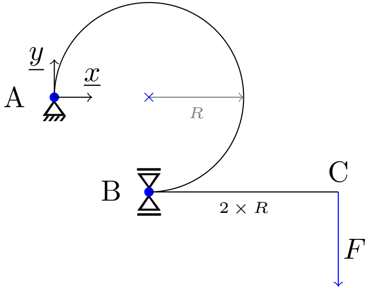


Figure 26: INTEL/MSC.NASTRAN 2016 α Benchmark [2]. This is a real world example of communication done by processors/hardware manufacturers (INTEL here). One encounters vocabulary such as MPI, MKL, SMP, DMP, Dual Core, CUDA, GPU/CPU, coprocessors... in those technical papers. Today a simulation engineer has to have a background in HPC (High Computer Performance) because it is an available standard in most companies. MSC introduced in 2017 a dedicated User's Guide [3]. Now return to real world too... Strength of Materials ☺

Extra case studies - Clamped curved beam

Definition #1

The framework is linear elasticity. A load F is applied in C. The beam has translation degrees of freedom clamped in A and is supported in B. The beam has a rectangular section $S = b \times h$ with $b = h = 10\,\text{mm}$ and $R = 1000\,\text{mm}$. Inertia worth $I_z = \frac{bh^3}{12}$. The beam is made of steel $E = 200\,\text{GPa}$ and $\nu = 0.30$. A 1000 kg mass has been hung at C.



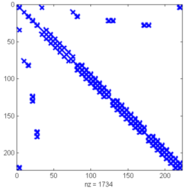


Figure 27: NASTRAN K_{GG} fill-in pattern plotted under MATLAB. Matrix is 3.52% dense (222 dof).

Aim of Extra Case study : Students have to run a NASTRAN `SOL 101` analysis and assess the deformation of the structure.

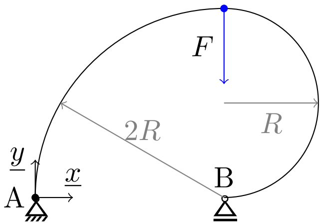
Closed form solution : At C vertical displacement worth

$$v_C = -39\frac{FR^3}{EI_z} \qquad (45)$$

Extra case studies - Clamped curved beam

Definition #2

The framework is linear elasticity.



Aim of Extra Case study : Students have to run a NASTRAN `SOL 101` analysis and assess the deformation of the structure.

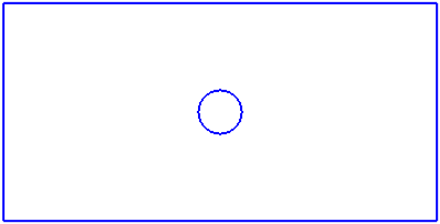
Closed form solution : At load application point vertical displacement worth

$$v = -\frac{2\pi}{5}\frac{FR^3}{EI_z} \qquad (46)$$

Case study # 3 - Plate with a hole

Definition

The framework is linear elasticity.



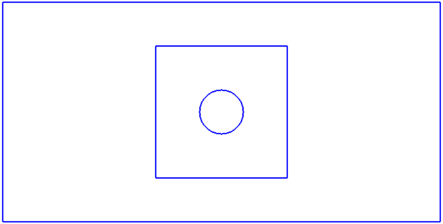
The plate is $t = 5\,\mathrm{mm}$ thick, $w = 50\,\mathrm{mm}$ wide and $L = 100\,\mathrm{mm}$ long. Structure is clamped LHS and a $\sigma_0 = 100\,\mathrm{MPa}$ tensile load is applied RHS. Hole has a diameter $D = 10\,\mathrm{mm}$. The plate is made of aluminium $E = 70\,\mathrm{GPa}$ and $\nu = 0.33$.

Aim of Case study # 3 : Students have to find the stress concentration factor K_T associated to the hole.

Case study # 3 - Plate with a hole

Meshing Technique

A square patch is to be designed under PATRAN around the hole.



This will basically allow to have a better assessment of stress around the hole.

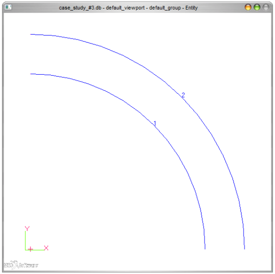
Case study # 3 - Plate with a hole

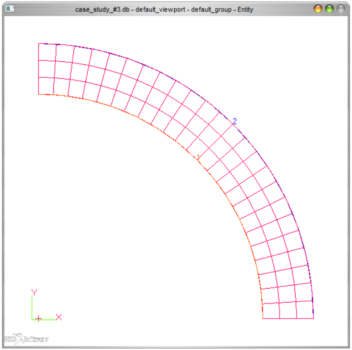
Meshing Technique

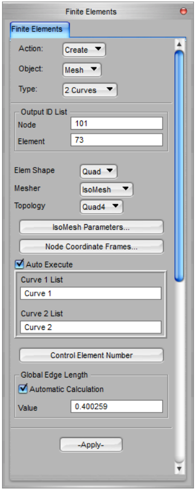
Background : In order to mesh a surface the latter may be defined. But it does exist a more efficient solution.

Technique : A surface may be meshed only drawing in PATRAN boundary curves and using the **Mesh > 2 Curves** menu form.

Example : Consider the two arcs in next PATRAN snapshots. One realizes the mesh of the surface between the 2 curves without necessity of its definition.



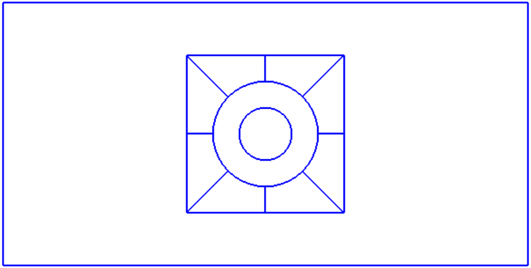




Case study # 3 - Plate with a hole

Meshing Technique

The patch is to be split under PATRAN in 8 subdomains. The mesh is to be built with NASTRAN CQUAD4 elements.



Nota : 2 circles are not shown in the above figure but do exist to build the mesh.

Case study # 3 - Plate with a hole

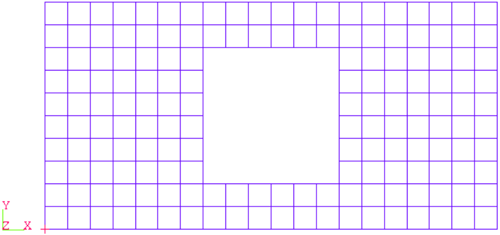
Meshing Technique

With PATRAN sketch the patches first. Assumptions are :

1. choose the number of square elements around the hole N with $N \equiv 0 \bmod [6]$ and $N \equiv 0 \bmod [4]$ e.g. $N = 72$
2. circular patch around the hole for a side length N square elements
3. circular patch around the hole for $3a \times 2a$ rectangular elements to be broken for the transition mesh
4. $2 \times R$ radius circular patch around the hole
5. square patch has a $6 \times R = 3 \times D$ side length

Case study # 3 - Plate with a hole

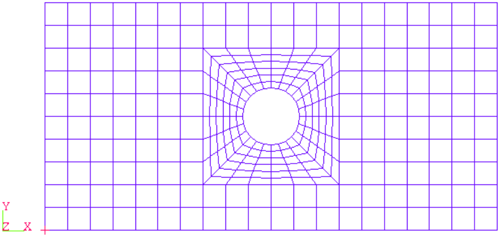
Mesh



The plate contains coarse `CQUAD4` 5×5 mm^2 elements.

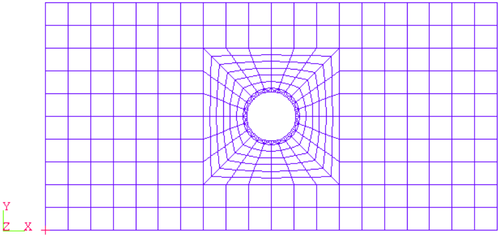
Case study # 3 - Plate with a hole

Mesh



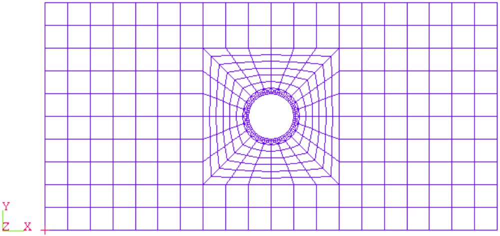
Case study # 3 - Plate with a hole

Mesh



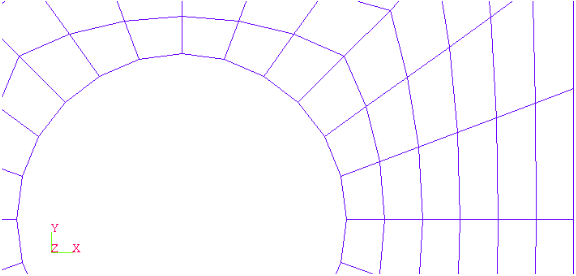
Case study # 3 - Plate with a hole

Mesh



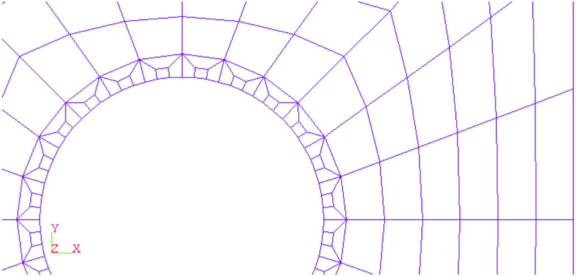
Case study # 3 - Plate with a hole

Mesh



Case study # 3 - Plate with a hole

Mesh



The plate contains an area of transition elements from 1 to 3 `CQUAD4`. The transition is done in PATRAN splitting 24 `CQUAD4` with the **Utilities** menu.

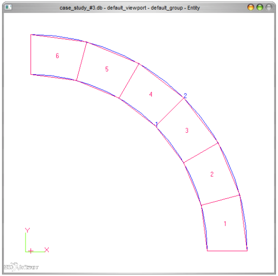
Case study # 3 - Plate with a hole

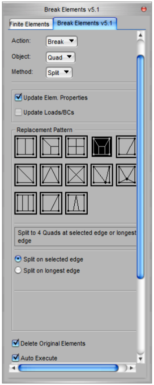
Mesh

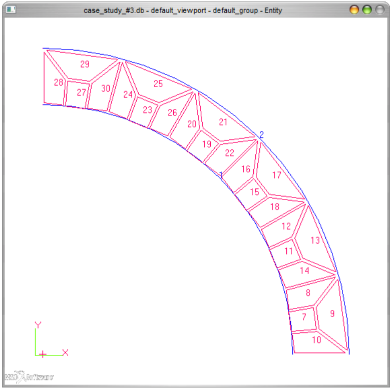
Background : In order to refine a mesh one can shape every element. But it does exist a more efficient solution to derive the fine mesh from a coarse mesh.

Technique : Enter the **Utilities**> **Fem-Elements** > **Break Elements** menu form.

Example : Consider the mesh done between the two arcs. One splits each `CQUAD4` in four smaller ones with the selection of the desired element and an edge.





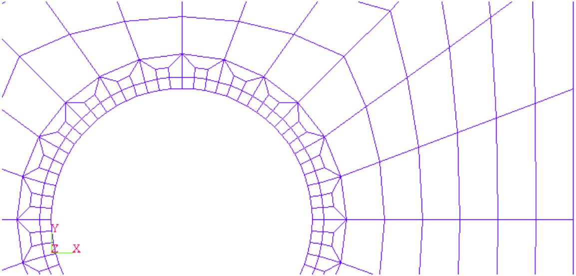


Nota : Next to the break operation the inner nodes need to be orthogonally moved onto the inner circle (From **Finite Elements** Menu Form : Action/Modify, Object/Nodes, Type/Projection).

Hint : Select inner nodes with PATRAN lasso (Ctrl + left mouse)

Case study # 3 - Plate with a hole

Mesh

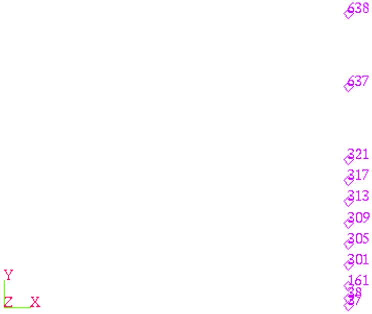


The circular boundary of the structure contains 72 `CQUAD4`.

Nota : Remove duplicate nodes with the **Equivalence** PATRAN function of the **Mesh** Menu form.

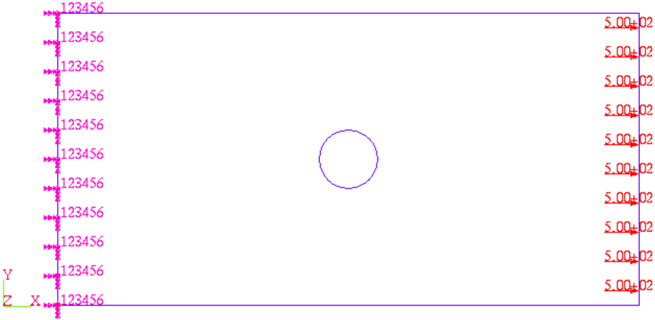
Case study # 3 - Plate with a hole

Mesh



Case study # 3 - Plate with a hole

BCs & Loading



Case study # 3 - Plate with a hole

K_T assessment

K_T is defined as

$$K_T = \frac{\sigma_{\max}}{\sigma_0} \qquad (47)$$

Nota : when the problem is easy K_T definition is straightforward ; but for complex structure σ_0 is usually particularly touchy to recover a universal definition. A reference textbook for K_T definition is the PETERSON [4].

Case study # 3 - Plate with a hole

Closed form solution

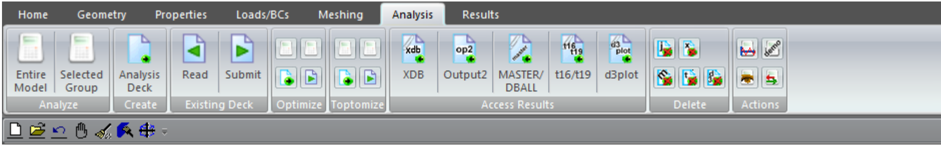
For a circular hole the mathematical theory of elasticity (refer e.g. to [5]) leads to

$$\sigma_{xx}(x = \frac{l}{2}, y) = \left\{ 1 + \frac{1}{2}\frac{R^2}{y^2} + \frac{3}{2}\frac{R^4}{y^4} \right\} \sigma_0 \quad (48)$$

Case study # 3 - Plate with a hole

Nastran linear static run

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



Then run the analysis with Nastran

`$ nastran case_study_3.dat news=n old=n scr=y`

User obtains as output to Nastran run

- `case_study_3.log` : Control File
- `case_study_3.f04` : Execution Summary Table
- `case_study_3.f06` : ASCII Results file
- `case_study_3.op2` : Binary Results file

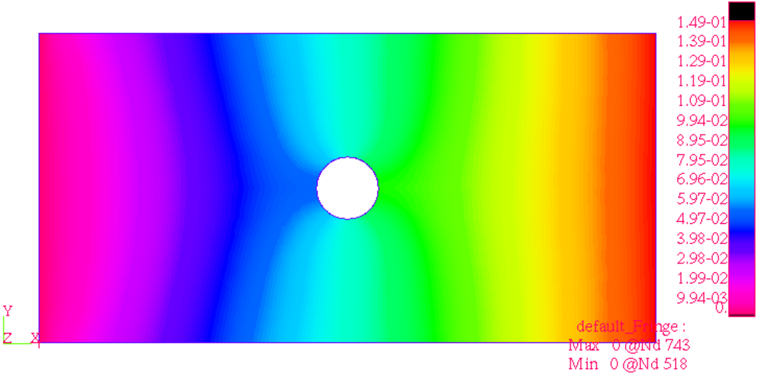
Case study # 3 - Plate with a hole

Results from NASTRAN linear static run from .op2

Students have to plot the displacement field u in PATRAN after the import of the `.op2`.

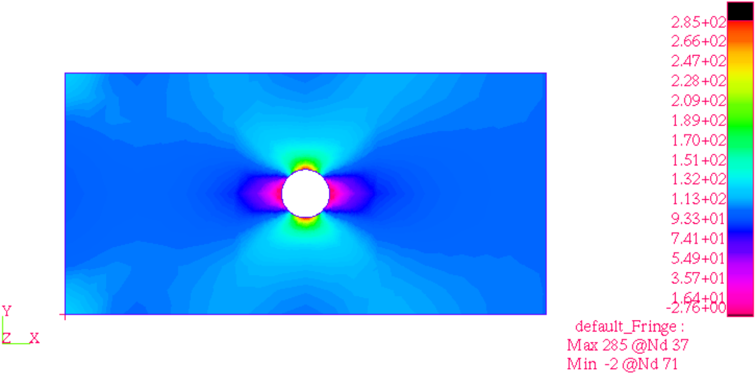
Case study # 3 - Plate with a hole

Results from NASTRAN linear static run from .op2 - u [mm]



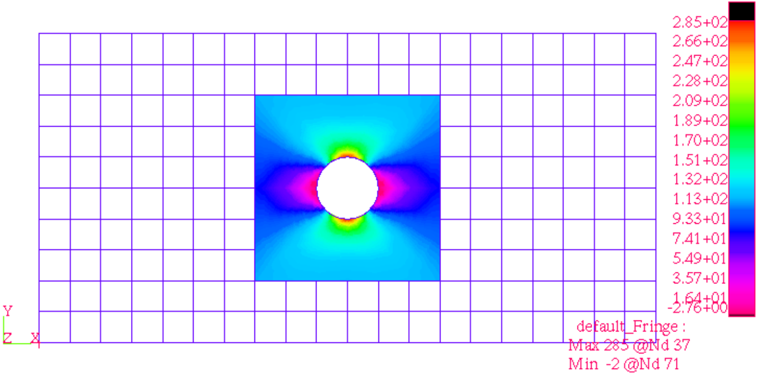
Case study # 3 - Plate with a hole

Results from NASTRAN linear static run from .op2 - σ_I [MPa]



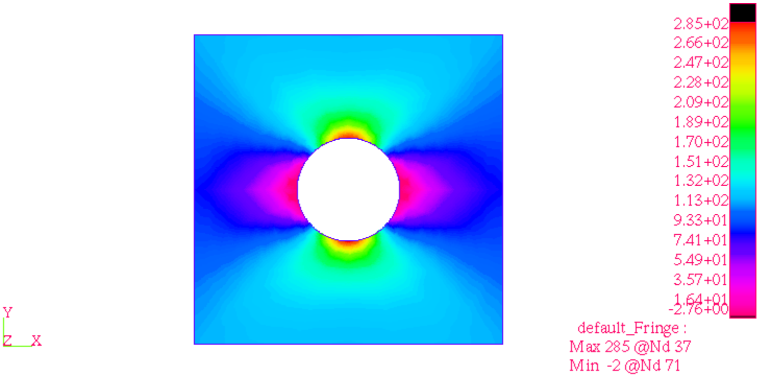
Case study # 3 - Plate with a hole

Results from NASTRAN linear static run from .op2 - σ_I [MPa]



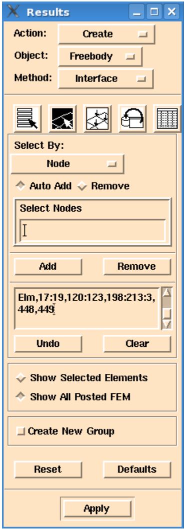
Case study # 3 - Plate with a hole

Results from NASTRAN linear static run from .op2 - σ_I [MPa]



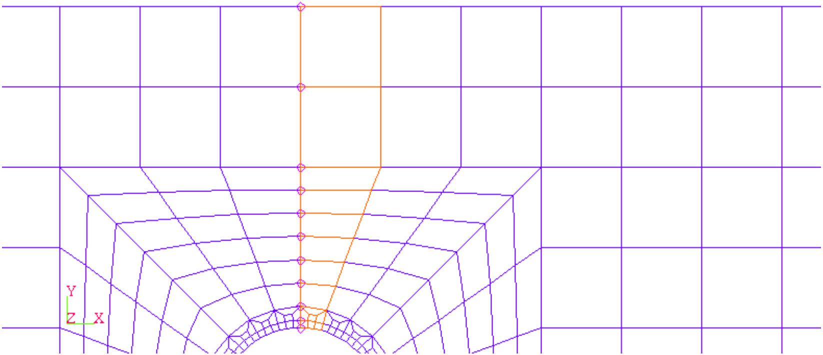
Case study # 3 - Plate with a hole

Results from NASTRAN linear static run from .op2 - freebody diagram form



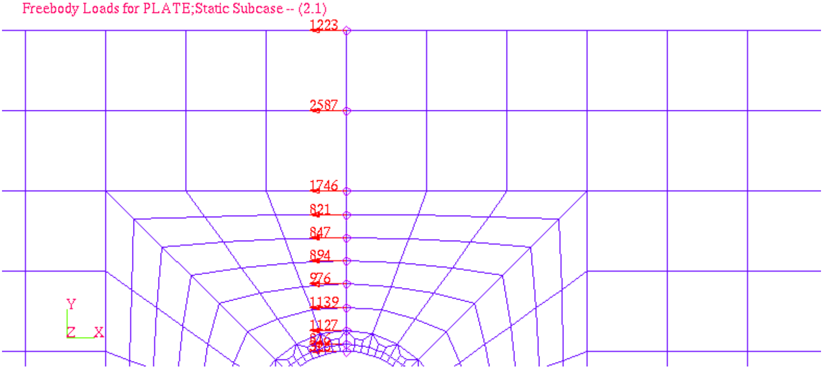
Case study # 3 - Plate with a hole

Results from NASTRAN linear static run from .op2 - elements and nodes for freebody diagram



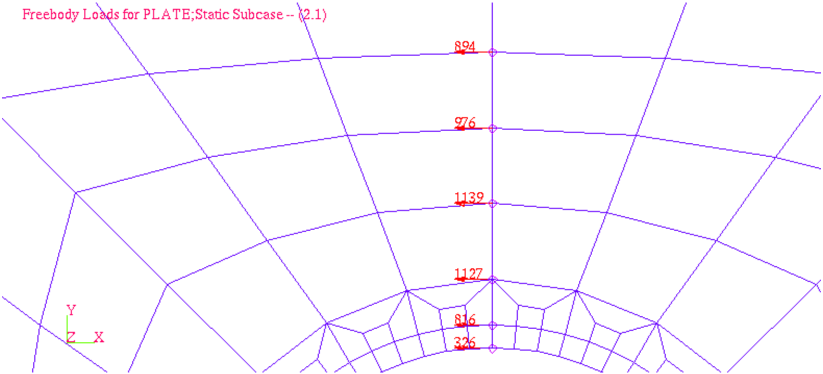
Case study # 3 - Plate with a hole

Results from NASTRAN linear static run from .op2 - freebody diagram



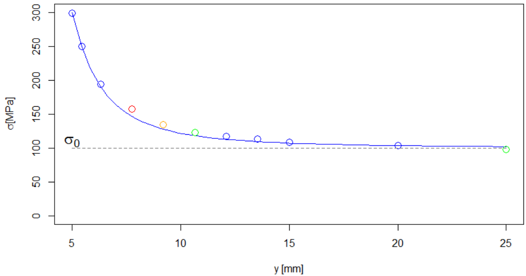
Case study # 3 - Plate with a hole

Results from NASTRAN linear static run from .op2 - freebody diagram



Case study # 3 - Plate with a hole

Results from NASTRAN linear static run from .op2 - σ_{yy} [MPa] vs. y [mm]



The error to closed form solution of NASTRAN assessed from freebody results can be assessed through color spectrum : ○ < 4.00 %≤○ < 5.00 %≤○ < 7.00 %≤○< 8.00 %. From freebody it is assessed

Case study # 3 - Plate with a hole

Results from NASTRAN linear static run from .op2 - σ_{yy} [MPa] vs. y [mm]

$$K_T = \frac{2F}{a \times t \times \sigma_0} \quad (49)$$

$$= \frac{N \times F}{\pi \times R \times t \times \sigma_0} \quad (50)$$

$$= 2.99- \quad (51)$$

From `GPSTRESS` (grid point stress) in `.f06` one derives

$$K_T = \frac{\sigma_{37}}{\sigma_0} \quad (52)$$

$$= 2.86 \pm 3.45\%- \quad (53)$$

Case study # 3 - Plate with a hole

Conclusion & Outlook

- One does not match the closed form solution because the studied plate is not an infinite continuum
- Error has been assessed after computation of a load F derived from a freebody and a surface computed from plate thickness $\times$ sum of mid adjacent quads heigth for comparison with a closed from solution
- Error in regard of closed form solution from theory of elasticty is under 3.00% but is highly mesh dependent ; stress results have been assessed on a single path at $x = 0$ in the range $y \in [\frac{D}{2}, \frac{w}{2}]$
- A more generalized methodology to assess the convergence of a mesh under Nastran is to use the `GPSTRESS` (grid point stress : the extra/interpolation of σ recovery at Gauss points to nodes) output and the associated discontinuities the `GPSDCON`

Case study # 3 - Plate with a hole

Conclusion & Outlook

For the mesh used for Case study # 3 illustration purpose at Node 37 whose coordinates are $(0, \frac{D}{2}, 0)$ one has

```
                                    S T R E S S E S   A T   G R I D   P O I N T S   - -     S U R F A C E     10
                  SURFACE X-AXIS X  NORMAL(Z-AXIS)  Z        REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CI
GRID ELEMENT                STRESSES IN SURFACE SYSTEM          PRINCIPAL STRESSES            MAX
ID      ID     FIBER    NORMAL-X    NORMAL-Y    SHEAR-XY     ANGLE       MAJOR       MINOR       SHEAR       VON MISES
  37     0     MID      2.860E+02  1.036E+01 -4.517E-03    -0.0009   2.860E+02  1.036E+01  1.378E+02   2.809E+02
```

and

```
          G R I D   P O I N T   S T R E S S   D I S C O N T I N U I T I E S   - -     S U R F A C E     10
                  SURFACE X-AXIS X  NORMAL(Z-AXIS)  Z        REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CI
GRID NORMAL STRESS DISCONTINUITY IN SURFACE SYSTEM     PRINCIPAL STRESS DISCONTINUITY              ERROR
ID             FIBER    NORMAL-X    NORMAL-Y    SHEAR-XY    MAJOR       MINOR      MAX SHEAR   VON MISES    EST.
 37            MID    1.413E-03  7.667E-04  5.977E+00  1.831E-01  1.831E-01  1.831E-01  2.696E-01    3.451E+00
```

Error formula are quoted in NASTRAN Linear Static Guide [6] ; e.g. for a tensor component

$$\delta_g = \frac{1}{N_e} \left\{ \sum_{i=1}^{N_e} (\sigma_i - \sigma_g)^2 \right\}^{\frac{1}{2}} \tag{54}$$

then one derives a mean value depending upon tensor components number.

Case study # 4 - Buckling Analysis

Theoretical Aparté

The buckling analysis is a bifurcation analysis.

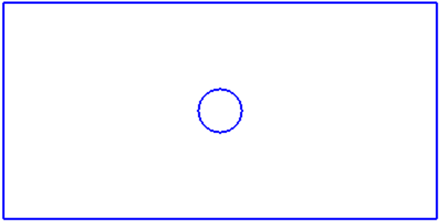
Start point is the functional W that describes the elastic energy of the solid with a parameter λ associated to the load.

The eigenvalues stem from setting equal to 0 the second derivative of the functional W.

Case study # 4 - Buckling Analysis

Definition

The framework is linear elasticity.



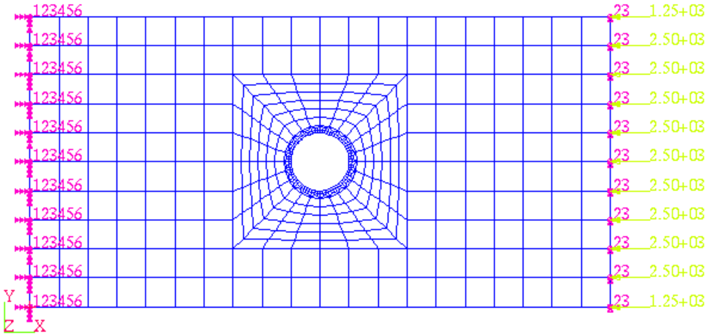
The plate is $t = 3\,\mathrm{mm}$ thick. A compressive load $\sigma_0 = -100\,\mathrm{MPa}$ is applied at one side. The beam is clamped at one side. The plate is made of aluminium $E = 70\,\mathrm{GPa}$ and $\nu = 0.33$.

Aim of Case study # 4 : Students have to find the buckling modes associated to the compressive load and assess the critical load below which the structure remains stable.

Case study # 4 - Buckling Analysis

Mesh

The mesh of case study # 3 is suitable for the buckling analysis.



The mesh is to be built with NASTRAN CQUAD4 elements.

Case study # 4 - Buckling Analysis

NASTRAN **EIGRL** Card

NASTRAN **SOL 105** is to be used.The NASTRAN **EIGRL** Card has to be called in Case Control Section after the linear **SUBCASE** associated to the compressive load by a NASTRAN **METHOD** Card.

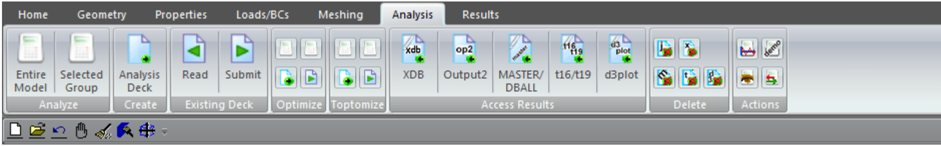
```
$ Case Control Section
...
$ 1st SUBCASE to install loading
SUBCASE 1001
   SUBTITLE=COMPRESSION
   LABEL=COMPRESSION
   LOAD = 2

$ 2nd SUBCASE to solve eigenvalue problem
SUBCASE 1002
   SUBTITLE=COMPRESSION
   LABEL=COMPRESSION
   METHOD = 1
   STATSUB(BUCKLING)=1001
$
BEGIN BULK
$ Bulk Section
$        EID     V1       V2       ND
EIGRL    1                         10
...
```

Case study # 4 - Buckling Analysis

NASTRAN buckling run

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



Then run the analysis with NASTRAN

`$ nastran case_study_4.dat news=n old=n scr=y`

User obtains as output to NASTRAN run

- `case_study_4.log` : Control File
- `case_study_4.f04` : Execution Summary Table
- `case_study_4.f06` : ASCII Results file
- `case_study_4.op2` : Binary Results file

Case study # 3 - Plate with a hole

Results from NASTRAN buckling run from .f06

Students have to find the eigenvalues associated to the computed modes in the `.f06`.

Case study # 3 - Plate with a hole

Results from NASTRAN buckling run from .f06

Students have to find the eigenvalues associated to the computed modes in the `.f06`.

R E A L   E I G E N V A L U E S

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS
1	1	2.922786E+00	1.709616E+00	2.720938E-01	1.147029E+03
2	2	8.688133E+00	2.947564E+00	4.691193E-01	2.579305E+03
3	3	1.231678E+01	3.509528E+00	5.585586E-01	4.383950E+02
4	4	1.690665E+01	4.111770E+00	6.544085E-01	3.473366E+03
5	5	1.714466E+01	4.140611E+00	6.589987E-01	8.870814E+02
6	6	2.491786E+01	4.991779E+00	7.944663E-01	1.714101E+03
7	7	2.727503E+01	5.222550E+00	8.311947E-01	6.138878E+03
8	8	3.472403E+01	5.892710E+00	9.378539E-01	2.523782E+03
9	9	3.905111E+01	6.249089E+00	9.945734E-01	8.644048E+03
10	10	4.561754E+01	6.754076E+00	1.074944E+00	3.599225E+03

Case study # 4 - Buckling Analysis

Results from NASTRAN buckling run from .op2

Students have to plot the buckling mode in PATRAN after the import of the `.op2`.

Case study # 4 - Buckling Analysis

Results from NASTRAN buckling run from .op2

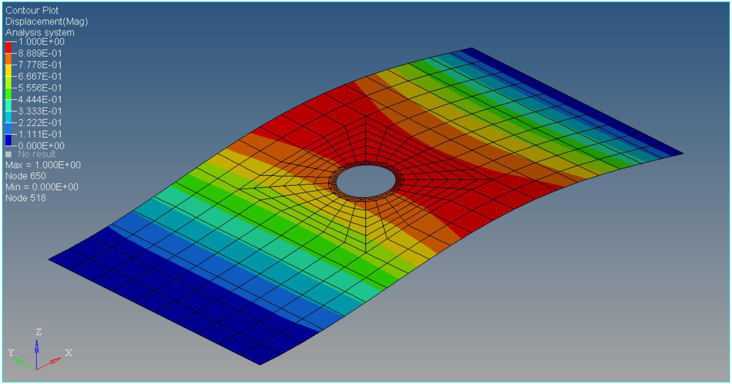


Figure 28: `SOL 105` - 1st Buckling Mode.

Case study # 4 - Buckling Analysis

Results from NASTRAN buckling run from .op2

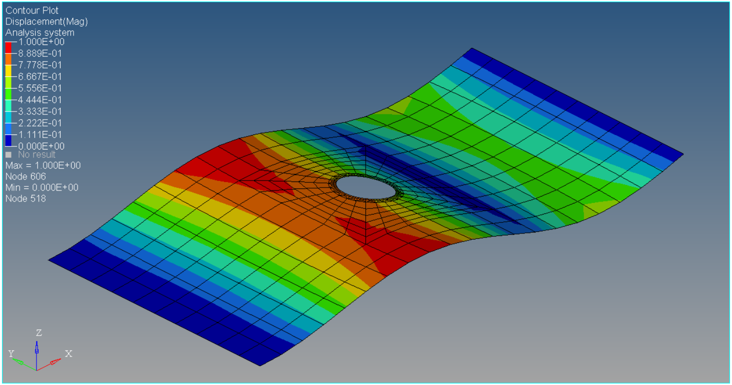


Figure 29: `SOL 105` - 2nd Buckling Mode.

Case study # 4 - Buckling Analysis

Results from NASTRAN buckling run from .op2

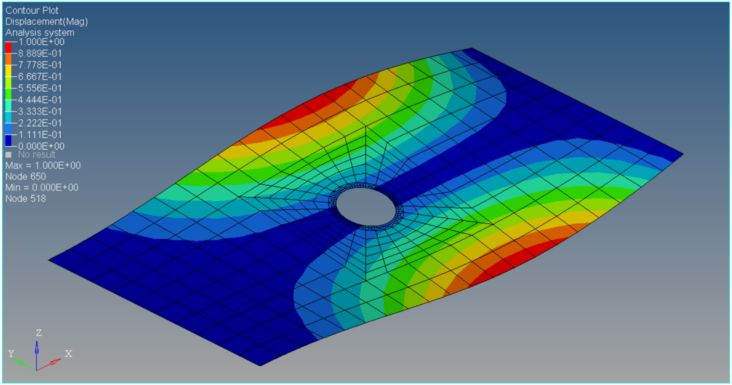


Figure 30: `SOL 105` - 3rd Buckling Mode.

Case study # 4 - Buckling Analysis

Results from NASTRAN buckling run from .op2

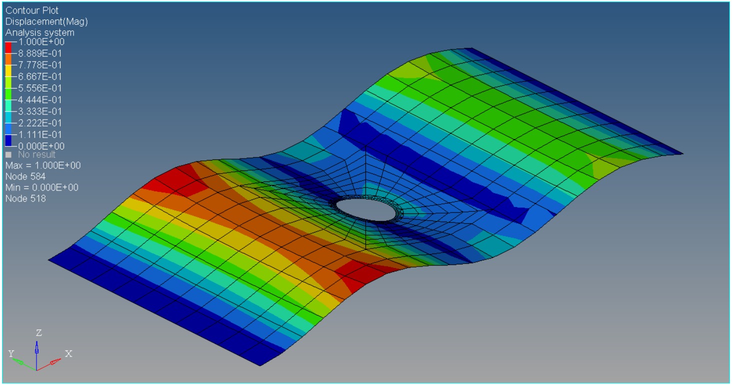


Figure 31: `SOL 105` - 4th Buckling Mode.

Case study # 4 - Buckling Analysis

Results from NASTRAN buckling run from .op2

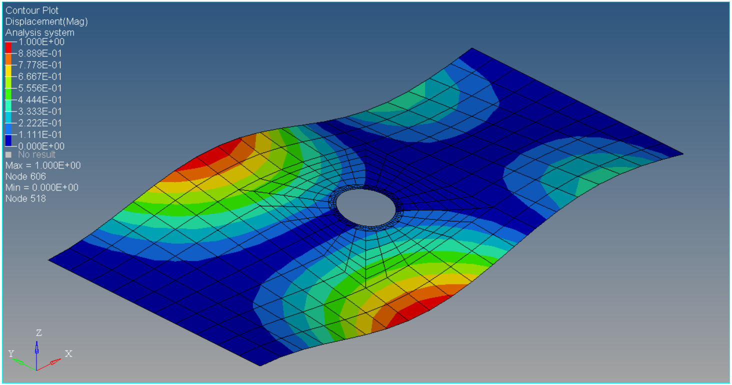


Figure 32: `SOL 105` - 5th Buckling Mode.

Case study # 4 - Buckling Analysis

Results from NASTRAN buckling run from .op2

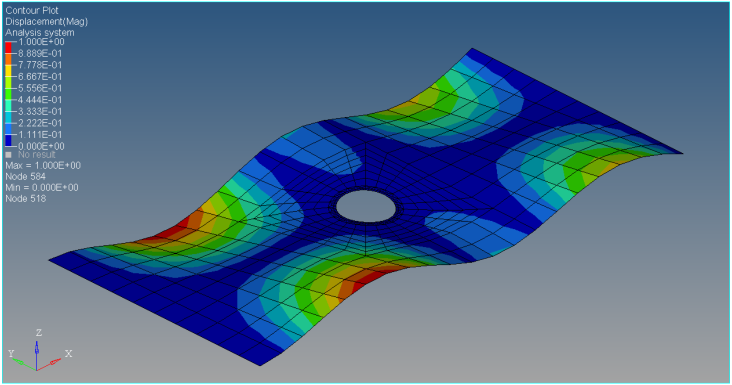


Figure 33: `SOL 105` - 6th Buckling Mode.

Case study # 4 - Buckling Analysis

Results from NASTRAN buckling run from .op2

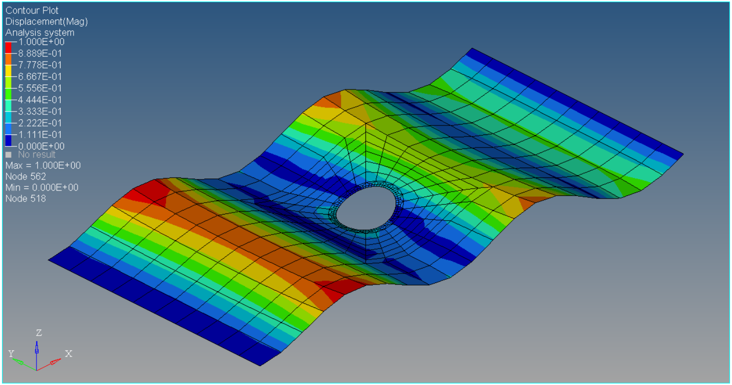


Figure 34: `SOL 105` - 7th Buckling Mode.

Case study # 4 - Buckling Analysis

Results from NASTRAN buckling run from .op2

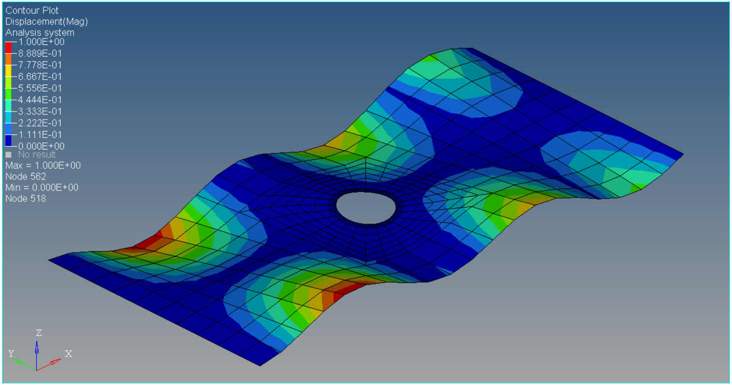


Figure 35: `SOL 105` - 8th Buckling Mode.

Case study # 4 - Buckling Analysis

Results from NASTRAN buckling run from .op2

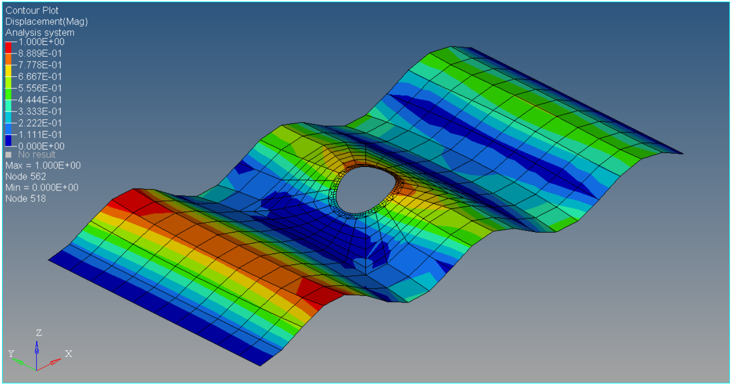


Figure 36: `SOL 105` - 9th Buckling Mode.

Case study # 4 - Buckling Analysis

Results from NASTRAN buckling run from .op2

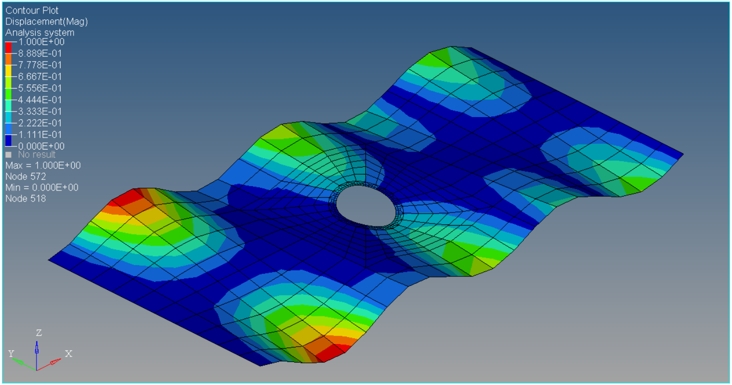


Figure 37: SOL 105 - 10th Buckling Mode.

Case study # 4 - Buckling

Conclusion & Outlook

- Eigenmodes are particularly dependent upon boundary conditions applied at the loaded plate end : eigenvalues are different if rotations are clamped or not

Case study # 5 - GUYAN Static Reduction

Theoretical Aparté

The framework is linear elasticity. One solves

$$K_{ff}u_f = F_f \qquad (55)$$

with o-set interior degrees of freedom and a-set exterior degrees of freedom one can split the equation (55)

$$\begin{bmatrix} K_{oo} & K_{oa} \\ {K^T}_{oa} & K_{aa} \end{bmatrix} \begin{Bmatrix} u_o \\ u_a \end{Bmatrix} = \begin{Bmatrix} F_o \\ F_a \end{Bmatrix} \qquad (56)$$

whose first line of (56) means

$$\begin{aligned} & K_{oo}u_o + K_{oa}u_a = F_o \\ & \Leftrightarrow K_{oo}^{-1}\left(K_{oo}u_o + K_{oa}u_a\right) = K_{oo}^{-1}F_o \\ & \Leftrightarrow u_o = K_{oo}^{-1}F_o - K_{oo}^{-1}K_{oa}u_a \\ & \Leftrightarrow u_o = \underbrace{K_{oo}^{-1}F_o}_{u_o^0} - \underbrace{K_{oo}^{-1}K_{oa}}_{-G_{oa}}u_a \end{aligned} \qquad (57)$$

or

$$u_o = u_o^0 + G_{oa}u_a \qquad (58)$$

with G_{oa} boundary transformation and u_o^0 fixed boundary displacement.

Case study # 5 - GUYAN Static Reduction

Theoretical Aparté

Thus the second line of (56) means with the definition (58) of u_o

$$
\begin{aligned}
& {K^T}_{oa} u_o + K_{aa} u_a = F_a \\
\Leftrightarrow\ & {K^T}_{oa} \left(u_o^0 + G_{oa} u_a \right) + K_{aa} u_a = F_a \\
\Leftrightarrow\ & \left(\underbrace{{K^T}_{oa} G_{oa} + K_{aa}}_{\overline{K_{aa}}} \right) u_a = \underbrace{F_a - {K^T}_{oa} u_o^0}_{\overline{F_a}}
\end{aligned}
\qquad (59)
$$

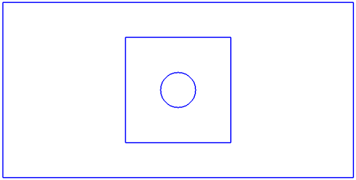
with $\overline{K_{aa}}$ boundary stiffness and $\overline{F_a}$ boundary load. Typically with a classical NASTRAN run one has $\overline{K_{aa}} \equiv$ `KAAX` and $\overline{F_a} \equiv$ `PAX`.

Note that (59) defines the boundary displacement field associated to the `ASET`.

Case study # 5 - GUYAN Static Reduction

Definition

The framework is linear elasticity.



The residual structure is selected as the square structure $30 \times 30\,\mathrm{mm}^2$ centered on the circular hole.

Aim of Case study # 5 : Students have to realize a GUYAN Static Reduction of structure associated to Case study # 2 and run a `SOL 101` for the residual structure. The fill-in of `KAAX` is to be assessed by the students and compare to a classical finite elements matrix (cf. Figure 27).

Case study # 5 - GUYAN Static Reduction

NASTRAN EXTOUT Parameter

In order to realize a GUYAN Static Reduction one has to use EXTOUT parameter. The latter is either a Case Control Section or a Bulk parameter. Hereafter used as a bulk parameter.

```
$ Bulk Data - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
BEGIN BULK
$ Bulk  Parameters- - - - - - - - - - - - - - - - - - - - - - - - - - - +
$ To write an .op2 file
PARAM,POST,-1
$ For an AUTOSPC Reminder cf. Lecture 1/4
PARAM,AUTOSPC,NO
$ To write in the .f06 a summary of maximum displacements
PARAM,PRTMAXIM,YES
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
$ Next Parameter to be 8< & Pasted to output Boundary Matrix & Boundary |
$      Load vector in a PUNCH file                                      |
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
PARAM,EXTOUT,DMIGPCH
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
```

Table 1: NASTRAN EXTOUT Parameter.

Set equal to DMIGPCH the EXTOUT parameter means a .pch file creation for output of KAAX and PAX.

Case study # 5 - GUYAN Static Reduction

NASTRAN **ASET** Card

NASTRAN offers a card called **ASET** to define the degrees of freedom to be omitted in the reduction process. Hereafter the **ASET1** is used to list the reduction nodes after the degrees of freedom selection `123456`.

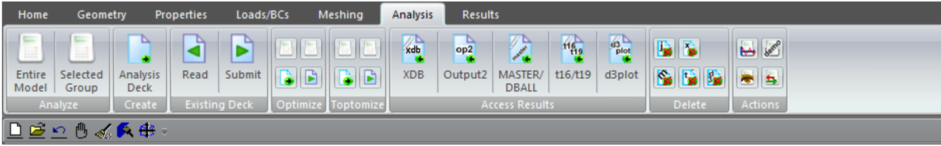
```
$ Bulk Data - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
BEGIN BULK
$ Bulk  Parameters- - - - - - - - - - - - - - - - - - - - - - - - - - - +
.
.
.
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
$ Next is sample ASET1 card to be 8<, Pasted & Adapted to target dof    |
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
$1      2       3       4       5       6       7       8       9       0
ASET1    123456 318     319     321     347     348     349     375
        376     377     403     404     405     431     432     433
        459     460     461     487     488     489     515     516
$ - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - +
.
.
.
```

Table 2: NASTRAN **ASET1** Sample.

Case study # 5 - GUYAN Static Reduction

NASTRAN linear static run for reduction

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



Note the **Selected Group** Menu Form of the **Analysis** menu is particularly suitable to export a part of a previous model (here the external part to central square of the plate).

Then run the analysis with NASTRAN

```
$ nastran case_study_5.dat news=n old=n scr=y
```

User obtains as output to NASTRAN run

- case_study_5.log : Control File
- case_study_5.f04 : Execution Summary Table
- case_study_5.f06 : ASCII Results file
- case_study_5.op2 : Binary Results file
- case_study_5.pch : PUNCH file ← Interesting output in this file

Case study # 5 - GUYAN Static Reduction

Results from NASTRAN linear static run from .pch

Students have to check `.pch` file contains a boundary stiffness matrix and an interface load vector. The former is called `KAAX` and the latter `PAX`.

Case study # 5 - GUYAN Static Reduction

Reading external element

In order to read the result of a GUYAN Static Reduction one has to use `K2GG` and `P2G` commands.

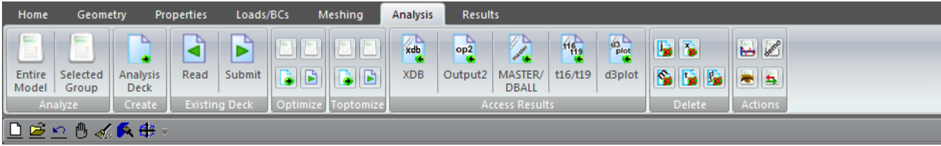
```
$ Case Control Section
$ Command for reading boundary stiffness
K2GG = ...
$ Command for reading interface load
P2G  = ...

SUBCASE 1000
$ SPC call is now useless because included
$    in the K2GG reading
$
$ LOAD call is now useless because included
$    in the P2G reading
```

Case study # 5 - GUYAN Static Reduction

NASTRAN linear static run for residual structure

NASTRAN **.dat** is generated from **Analysis** menu for the residual structure.



Note the **Selected Group** Menu Form of the **Analysis** menu is particularly suitable to export a part of a previous model (here the central square of the plate).

Case study # 5 - GUYAN Static Reduction

NASTRAN linear static run for residual structure

The NASTRAN run `case_study_5_residual.dat` has to call the result of the `SOL 101` reduction run `case_study_5.dat`.

```
$ Case Control Section
INCLUDE 'case_study_5.pch'
$ Command for reading boundary stiffness
K2GG = ...
$ Command for reading interface load
P2G  = ...
```

Then run the analysis with NASTRAN

`$ nastran case_study_5_residual.dat news=n old=n scr=y`

User obtains as output to NASTRAN run

- `case_study_5_residual.log` : Control File
- `case_study_5_residual.f04` : Execution Summary Table
- `case_study_5_residual.f06` : ASCII Results file
- `case_study_5_residual.op2` : Binary Results file

Case study # 5 - GUYAN Static Reduction

Results from NASTRAN linear static run from .op2

1. Students have to compare at reduction run the displacements output of the `ASET1` nodes (cf. Table 2) vs. the vector

$$\texttt{KAAX}^{-1} \times \texttt{PAX} \tag{60}$$

2. Students have to check with PATRAN that for the residual structure the results from the run of case study #3 and case study #5 are in accordance

Case study # 5 - GUYAN Static Reduction

Conclusion & Outlook

- The analysis performed is particularly used in dynamic analyis to reduce the degrees of freedom of a model [7].
- The GUYAN Static Reduction [8] was particularly used by the past to allow concurrent work between engineering teams.
- One can check that on the opposite of an assembly global finite elements stiffness matrix `KAAX` is not sparse.
- One can use a `DMIGOP2` output for the matrix `KAAX` and the load vector `PAX` instead of the ASCII `DMIGPCH` output ; reading the binary format in NASTRAN to run the reduced structure is a little bit more touchy. Main outcome is the binary file size is significantly smaller than the ASCII one.

References

-

[1] G.E. Karniadakis and R.M. Kirby.
Parallel Scientific Computing in C++ and MPI: A Seamless Approach to Parallel Algorithms and Their Implementation.
Cambridge University Press, 2003.

[2] Intel.
Slash your MSC NASTRAN★ *Simulation Runtime by Up to 50%.*
Solution Brief, 2016.

[3] MSC Software.
MSC NASTRAN *2017 High Performance Computing User's Guide.*
MacNeal-Schwendler Corporation, 2017.

[4] R.E. Peterson.
Stress concentration factors: charts and relations useful in making strength calculations for machine parts and structural elements.
Wiley-Interscience publication. Wiley, 1974.

[5] B.F. De Veubeke.
A Course in Elasticity.
Number vol. 29 in Applied Mathematical Sciences. Springer-Verlag, 1979.

[6] MSC Software.
Linear Static Analysis User's Guide: MSC Nastran 2012.
MacNeal-Schwendler Corporation, 2011.

[7] J.F. Imbert.
Analyse des structures par éléments finis.
Cépaduès, 1991.

[8] R.J. Guyan.
Reduction of Stiffness and Mass Matrices.
AIAA Journal, 3, February 1965.

[9] X.S. Yang.
Engineering Optimization: An Introduction with Metaheuristic Applications.
Wiley Online Library: Books. Wiley, 2010.

[10] J.M. Danskin.
The Theory of Max-Min and its Application to Weapons Allocation Problems.
Ökonometrie und Unternehmensforschung Econometrics and Operations Research. Springer Berlin Heidelberg, 2012.

§A - Relations between elastic moduli

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

$$\lambda = \frac{\nu E}{(1+\nu)(1-2\nu)} \tag{61}$$

$$\mu = G \tag{62}$$

§E - Two Springs/Three Masses Case Study

One considers the next Two Springs/Three Masses system. One is interested in the frictionless harmonic answer of the system sketched in Figure 38.

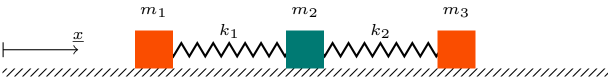


Figure 38: Two Springs/Three Masses system.

Aim of Case study (extracted from [9]): Using a NASTRAN `SOL 103` run and ISIGHT (although matricial calculations would lead to a closed form solution) students have to choose masses and spring stiffnesses to satisfy next optimization (min-max [10]) problem with S and k_0 constants:

$$\S E \left\{ \begin{array}{c} \min\limits_{p \in \{m_1, m_2, m_3, k_1, k_2\}} \omega = \max \omega_i \\ S = m_1 + m_2 + m_3 \\ m_1 = m_3 \\ k_1 = k_2 \geq k_0 \end{array} \right.$$

§E - Two Springs/Three Masses Case Study

Answer : One can reach the minimum-max ω_i

$$\min\max\omega_i = \left\{\frac{8k_0}{S}\right\}^{\frac{1}{2}} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (63)$$

with the *ad hoc* mass split (cf. complete demonstration in [9]).