Linear Static Analysis : ASEF

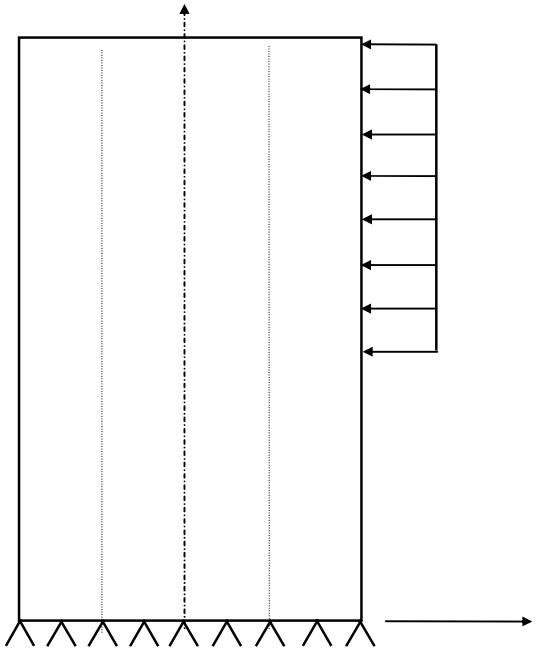


Table of content

1. Generalities

1.1. Output results of Asef

- Displacements;
- Reactions;
- Stresses or efforts;
- Strains;
- For « illimited » number of load cases.

1.2. Materials

- Isotropic;
- Orthotropic;
- Anisotropic;
- Composite laminate or wound filament.

1.3. Physical properties

- Thickness **(.PHP,.CAP)**
- Characteristics of beams/rods **(.BPR,.CAP)**

1.4. Boundary conditions

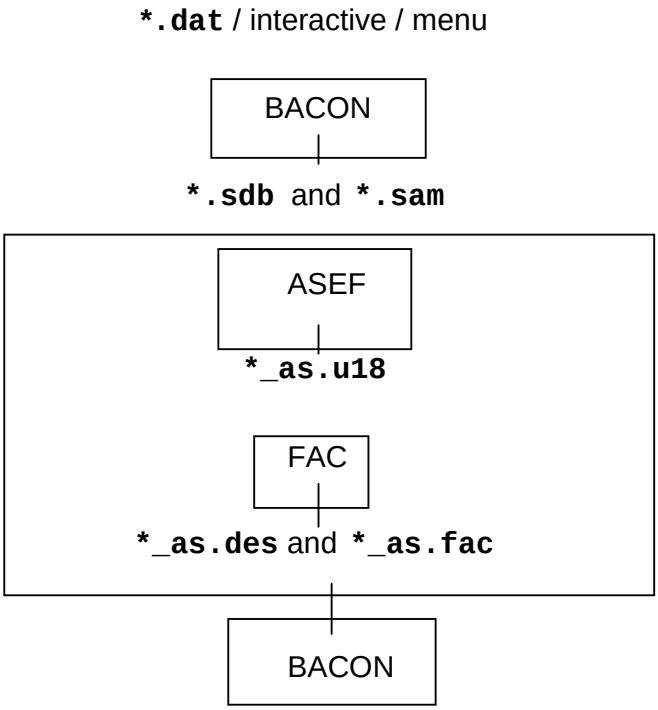
- Imposed displacements. **(.CLM DEP** and **FIX)**
- Contact structure – rigid foundation **(.JEU)**
- Contact **(.JER, .CPS)**
- Links and linear constraints **(.LIA, .CLI)**
- Rigid bodies **(.RBE)**

1.5. Loadings

- Nodal loads **(.CLM CHA)**
- Acceleration **(.CLM ACCEL)**
- Pressure **(.CLM PRESS)**
- Temperature **(.CLT TFX)**
- Speed rotation **(.CLM ROTA)**

2. Organisation of linear static computation

2.1. Organisation chart



Listing / plot

Notes:

- ***.sdb** can be output as an ***.fdb**, portable on all stations or PC's.
- ***_as.fac** is rewritable in ***.fmt,** portable on all stations or PC's
- ***_as.u18** contains Gauss points results.
- **_as.fac._as.des** contain visible results.
- Printing formats is Postscript, HPGL or options.

2.2. Steps of ASEF analysis

- Data analysis;
- Element generation;
- Inspection of the diagonal;
- Assembly and resolution;
- Computation of displacements;
- Computation of stresses.

2.3. Advantages

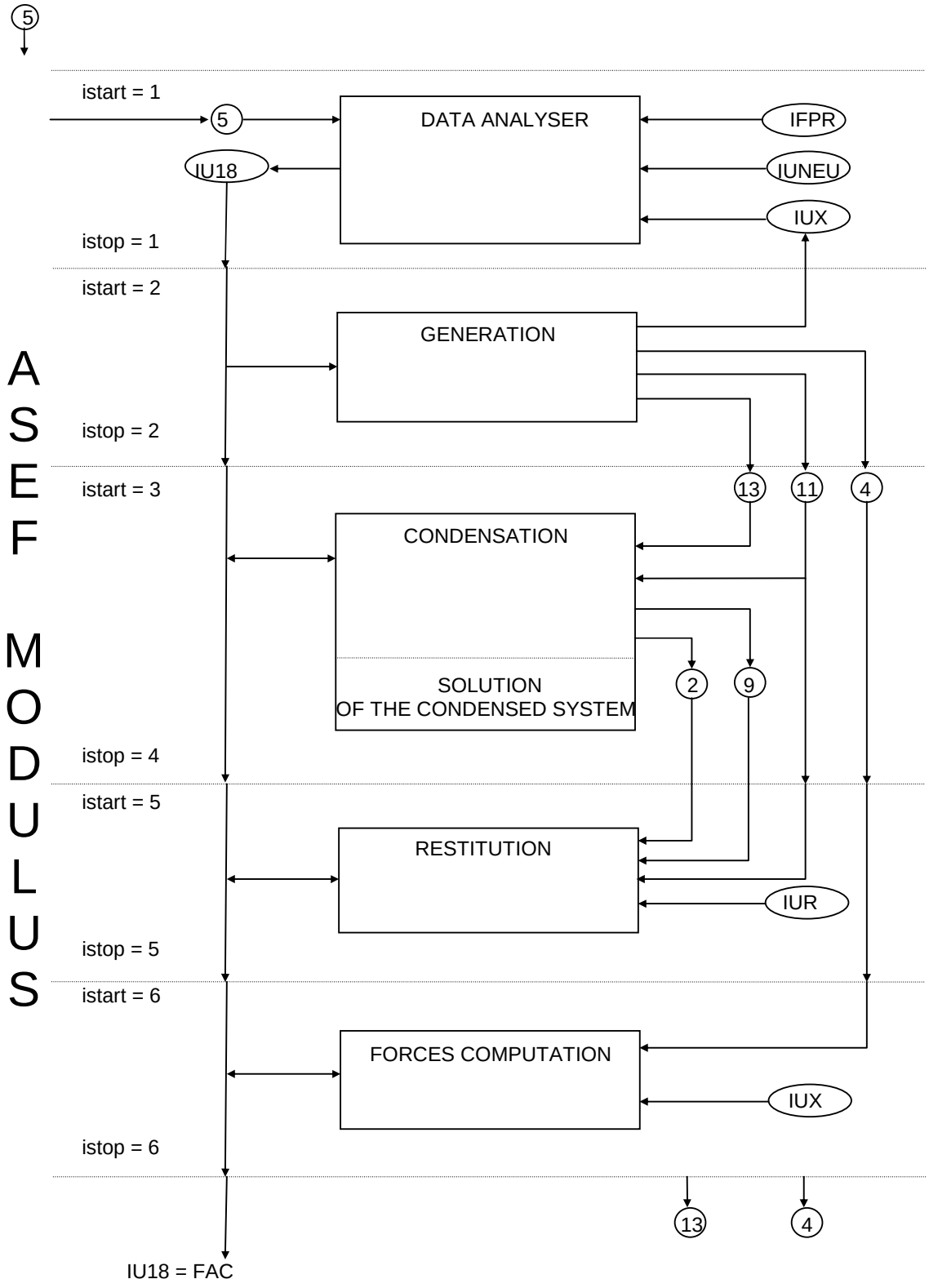
- Easy restart;
- Data transfer between modules.

Command **(.SAM) :**

- Stops **(ISTO)** at step n (between 1 and 6) and restarts **(ISTA)**

- Printing (NOPi) j ; i varying between 1 and 6
  j = - 1 : minimum printing
  0 : standard
  1 : More printing

2.4. ASEF Flowchart



3. Data analyser

3.1. Function

- Data consistency check;
- Specific operations
- Store file creation **(*_as.u18).**

3.1.1. Checking

- Errors (no materials, ...) ;
- Size of elements;
- Type of elements;
- Elements connectivity;
- Mass and inertia computation.

3.1.2. Specific operations

- Determination of local axes, of fixations;
- Memory space and work space;
- Nodal definition of elements replaced by numbered dof's;
- Division of matrix in sub-structures.

3.1.3. Creation

- Communication and store file **(U18)** containing all the informations used by computation modules.

3.2. Numbering of DOF's

3.2.1. Numbering

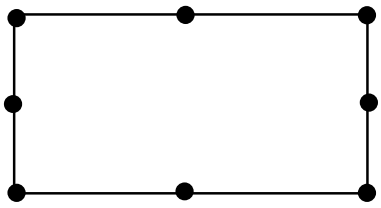
The numbering of the nodes and elements is condensed. This means there is no penalty using high numbers (internal renumbering).

3.2.2. Edge, Face Notion (Computation of nodal loads)

An edge is a side of a 2D element; a Face is a side of a 3D element.

In Samcef, the geometrical degree of an element (linked to the presence or not of nodes on element side) can be lower than discretized field degree (polynomial degree).

Example: Second degree with straight edges.



Usual

Samcef

In usual 8 nodes definition of a solid, the dof's are assigned to mid-nodes.

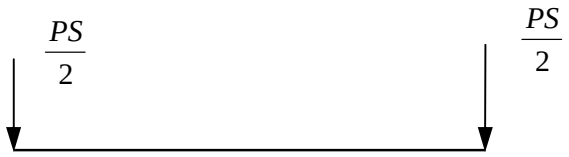
Advantage:

Automatic reduction of degree between different degrees of meshes.
Easy modification of discretized field degree.

.GEN DEGRE n	The geometrical degree of generated cells is forced to **n**.
.NIN DEGRE n FORCE	Addition (n=2) ou suppression (n=1) of nodes on sides
.SAM DEGREE n	The mechanical degree of whole structure is forced to degree n, when geometrical degree is < n.
.AEL (selection) DEGREE n	Selected elements are forced at degree n.

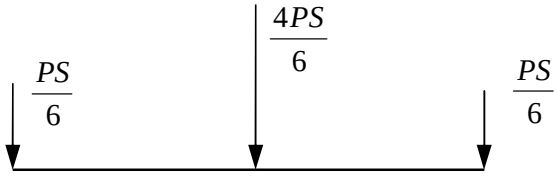
This notion has an influence on loading: a part of the stiffness is transmitted to interface DOF, so a different distribution of nodal loads which are energetic equivalent loads.

Example: Element at degree 1:



Same element at degree 2, submitted to pressure- the computation of the loads nodal repartition T= $\int \rho(s)\omega(s)ds$. ρ is the force value, ω is the shape function.

Gives:



3.3. Element loads

3.3.1. Problem

Only nodal loads are directly taken into account in Kq=g.
It means that an additional operation consists in computing nodal loads energetically equivalent to the defined element loads.

Note that these element loads depend on the element type and the material distribution.

3.3.2. Type of loads

- Pressure
- Acceleration
- Centrifugal forces
- Temperature field.

3.3.3. Processing

- Creation of these loads during the « generation » step and storage on unit 13.
- Element loads are replaced by nodal energetically equivalent nodal loads (depend on element shape function).

3.4. The files

***** is the name of the problem.

3.4.1. Input Data for Asef

BACON file: ***.sam**
Data base: ***.sdb**

3.4.2. Results

... from ASEF
Listing: ***_as.res**
Storage file: ***_as.u18**

... from FAC
Description File: ***_as.des**
File FAC: ***_as.fac**

3.4.3. Optional Files

Tension matrix: ***_as.u04**
Stiffness matrix: ***_as.u11**
Implicit loads ***_as.u13**
Direct access file: ***.u52**

Files needed for a computation by step or for supplementary post-processing with POSTFAC, OPTI, DYNAM …

3.4.4. Temporary files

1, 2, 3, 8, 9, 10, 14, 20, 40, 41, 42, 43, 60, 61, 62, 63, 64

These files sizes are printed by data analyser in the file ***_as.res.**

4. System resolution

4.1. Frontal Method .SAM MF 0

The solver must resolve the following problem :

Kq = g

Due to the large size of the complete matrix, its storage is very often impossible to be achieve. In Samcef, the frontal method is used and the structure is divided in sub-structures. This method consists in a Gauss elimination, sub-structure by sub-structure.
The dof's are processed by blocks and are divided in 2 categories:

- the NCC condensed degrees, qc
- the NCR kept degrees qr.

NCC+NCR is the width of the front and the size of a sub-structure. The degrees qc, not coupled with other sub-structure, are eliminated.

The elimination process goes from first to last sub-structure.

At each step of condensation, the matrix Kcc, Krr and Kcr are assembled:
Each sub-structure is processed like this:

a) Inversion of Kcc;

b) Computation of product Kcc^{-1} Kcr;

c) Computation of the matrix = Krr - Krc (Kcc^{-1} Kcr).

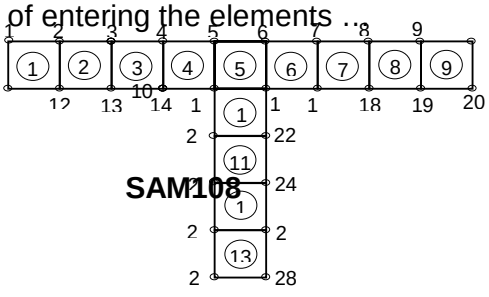
$$\begin{bmatrix} KccKcr \\ KrcKrr \end{bmatrix} \bullet \begin{bmatrix} qc \\ qr \end{bmatrix} = \begin{bmatrix} gc \\ gr \end{bmatrix}$$

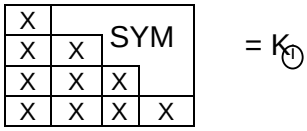
qc = Kcc^{-1} (gc-Kcr qr) Elimination of the first line

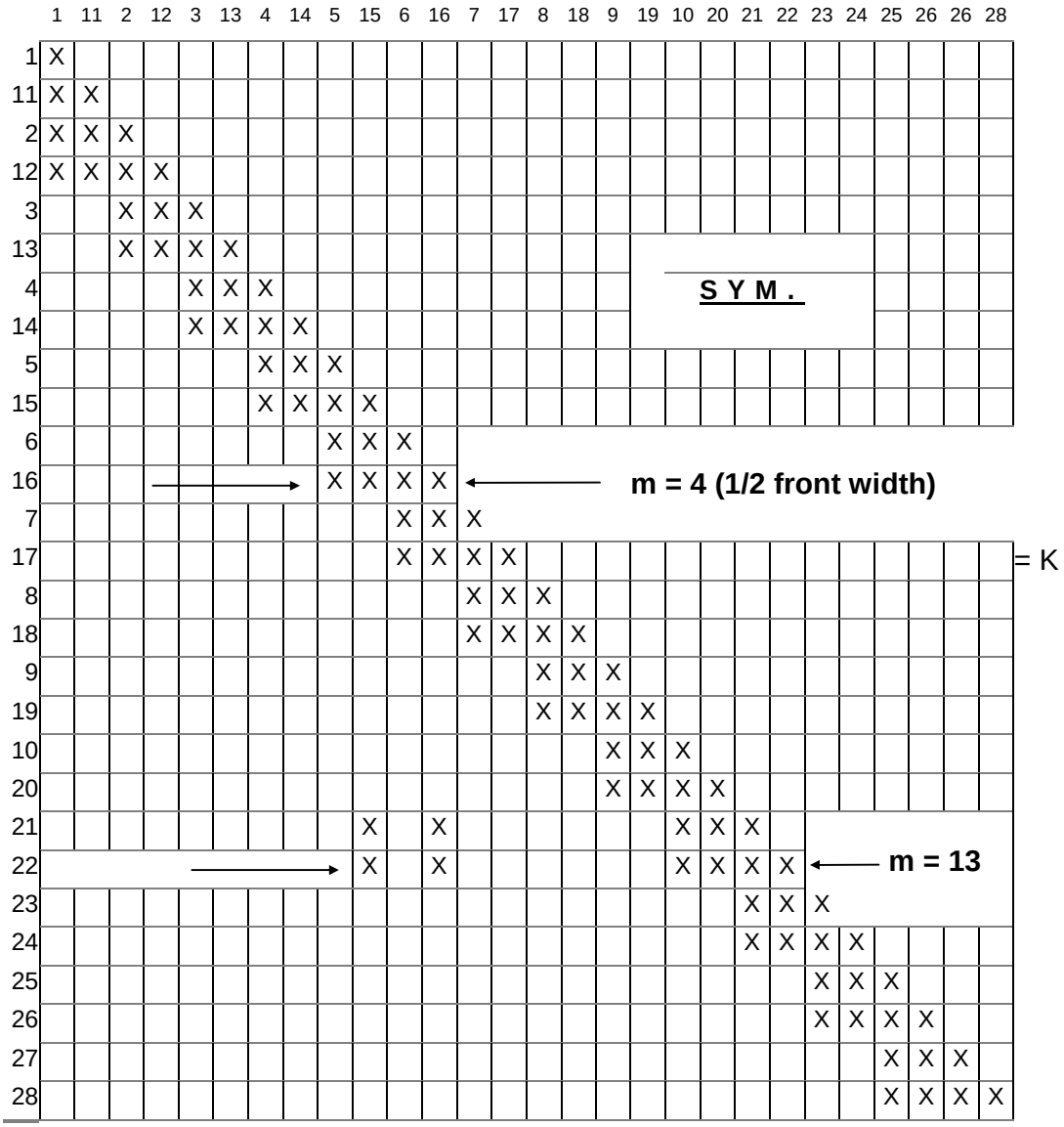
$$Krr^* = Krr - Krc\ Kcc^{-1}\ Kcr$$
$$gr^* = gr - Krc\ Kcc^{-1}\ gc$$

qr = Krr^{*-1} gr*

Front width depends on order of entering the elements ...







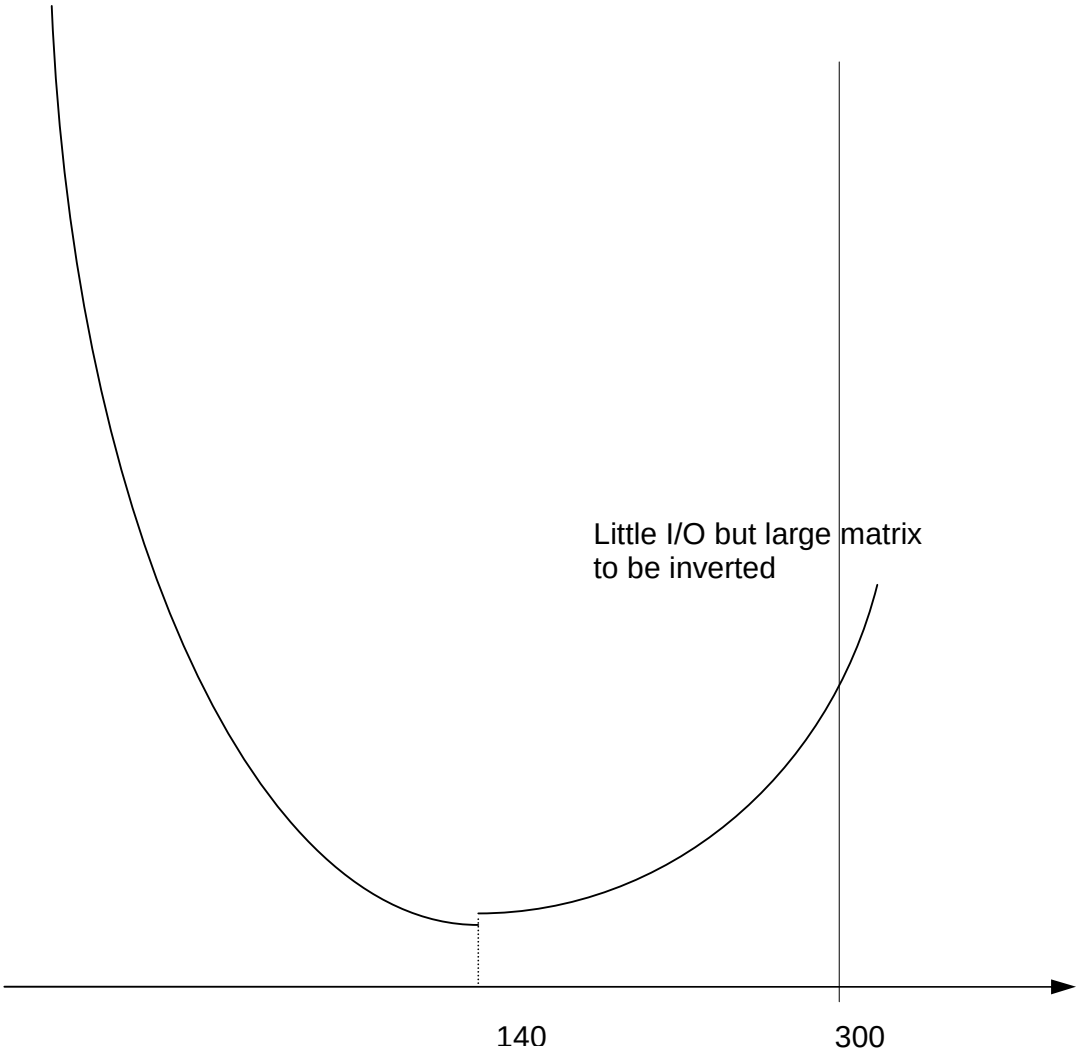
Population of assembled stiffness matrix

Command : .SAM NOPT <>

The NOPT parameter allows choosing the maximal number of condensed degree for each sub-structure.

The mean number of arithmetical operations increases as the NOPT square. If the Krr matrix can be assembled in memory, the NOPT optimal value is around 100, default value.

If the work' area and if these is small matrices have to be stored on disk, many peripheral operations are needed to assemble sub-structures. Thus it will be necessary to reduce the assemblies number by increasing NOPT until 500 or more.



4.2. Wave Front optimisation

4.2.1. Goal

Wave front optimisation consists in renumbering the degrees of freedom in order to reduce the wave front.

4.2.2. Reasons

Wave Front depends on element input order and has impact on the size of the stored matrices. Computation duration is proportional to (Front width)$^{2 \text{ or } 3}$, according to hardware.

4.2.3. Command .OPT

METH 0 Geometric Method

Method for element sorting based on the structure's geometry. If the structure's maximum sizes are measured along the 3 structural axes, the elements are arranged in order by progressing successively in three-dimensional space, starting from the axe corresponding to the smallest size and ending with that corresponding to the largest size. This method fails when the densities of elements along the three axes are not in the same ratio as the geometric sizes. That is why the user has the possibility to specify the progression axes via the ***AXE*** parameter.

METH 1 Interconnected Nodes Method

Method for element sorting based on node-interconnections. Two nodes are connected if they belong to the same element. If the complete list of nodes is examined, the gap between two connected nodes is defined as the difference between these nodes' positions on the list. The method aims to minimize the maximum gap between two connected nodes. After grading the nodes, element order is established by first taking elements whose nodes have the lowest order number. It is possible to improve grading by setting the ***NITE*** parameter to 2 or 3 and carrying out the grading process 2 or 3 times.

METH 2 Interconnected Elements Method

Method for element sorting based on element-interconnections. Two elements are connected if they have at least one node in common. The algorithm is similar to that used for connected nodes, but instead of examining a list of nodes a list of elements is examined. The ***NITE*** parameter is also available. This method sometimes fails when some parts of the structure are interconnected by linear constraints or contact conditions and gaps.

METH 3 Sloan Method

The Sloan method. This is the method used by default when there is no .OPT command. This method is based on the theory of graphs. All the nodes and cells make up an undirected graph with the cells' edges becoming the graph's connections. The distance between two nodes is defined as the shortest path between these two nodes. The graph's diameter is defined as the maximum distance between every node pair in the graph. The algorithm consists of three stages. In the beginning, the aim is to find an approximation of the diameter, called a

pseudodiameter. The second stage consists in renumbering the nodes beginning at one end of the pseudodiameter and ending at the other end. Nodes are graded in the descending order of the distance they are from the other end. The third stage consists in grading the elements. This is done by giving priority to the elements whose nodes have the lowest order number. The Sloan method is **often** the one that gives the best results.
The wavefront versus substructure number plot is shown in bar chart form in the execution listing (.res file). It shows each substructure's NCS size, number of NCC condensed degrees of freedom and the number of elements it contains. The wave front's arithmetical and quadratic means are indicated at the bottom of the bar chart. The maximum wavefront corresponds to the substructure's maximum size.

`.OPT METH 3` Sloan method

Compare the different methods for the large models (stop at end of data analyser with **`.SAM ISTOP 1`**).

4.3. Multi Frontal Method (Sparse Solver) .SAM MF 1

This algorithm is well adapted to hollowed matrices. Installed in central unit, it minimises transfer of data on disk. It is coupled to contact algorithms.

According to the performance of this algorithm, this solver is strongly recommended and is used by default.

We can note that defining .OPT with .SAM MF 1 is useless.

5. Assembly

5.1. Conditioning

Some verifications have to be realized :

- Diagonal inspection => K de-conditioning verification ;
- Null pivots fixations => loads not taken into account ;
- Negative Pivots => a priori known.

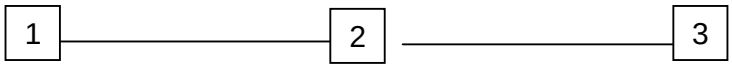
5.2. De-conditioning

5.2.1. Problem

A matrix is badly conditioned when its determinant is around 0. The matrix is invertible but its inverse is « uncertain ».

5.2.2. Example

As example, 2 aligned rods :



Node 1 is Fixed -> the system has form :

$$\begin{vmatrix} 2K & 0 & -K & 0 \\ 0 & 0 & 0 & 0 \\ -K & 0 & K & 0 \\ 0 & 0 & 0 & 0 \end{vmatrix} \begin{vmatrix} u2 \\ v2 \\ u3 \\ v3 \end{vmatrix} = \begin{vmatrix} 0 \\ 0 \\ F \\ 0 \end{vmatrix}$$

Singular System : some dof has no stiffness. Asef solver detects the situation during diagonal examination.

If there is some zero on the diagonal, ASEF fixes these degrees of freedom. It is possible to read this information in the listing file. The user can then check if there is a reaction on the fixed dof.

5.2.3. Rule

If $\frac{K\max}{K\min} > 10^8$ > (listing) => badly conditioned matrix : result can be instable.

5.3. Null Pivots illustration

Fixations by processor = fixation defined in the data file.

Adding fixations are due to :

- Either insufficient boundary condition ;
- Either a bad numerical conditioning.

Example to illustrate the second case with a system identical to the previous case but at 45 degrees :

K'=Kcos45=Ksin45

$$\begin{vmatrix} 2K' & 2K' & -K' & -K' \\ 2K' & 2K' & -K' & -K' \\ -K' & -K' & K' & K' \\ -K' & -K' & K' & K' \end{vmatrix} \begin{vmatrix} u2 \\ v2 \\ u3 \\ v3 \end{vmatrix} = \begin{vmatrix} 0 \\ 0 \\ F' \\ F' \end{vmatrix}$$

There is no zero on the diagonal but the system can not be inversed. A singular system detected by Asef during solving. The Solver fixes a dof (u or v) : this fixation is not physically acceptable.

5.4. Null Pivots (PN)

PN = MR - FIX - LIA + DDLNAR + MCP + E

MR Rigid body motions
FIX Rigid modes eliminated by fixations
LIA Rigid modes eliminated by links
DDLNAR DOF's without stiffness
MCP cinematical parasites modes
E errors:

- mechanisms ;
- independent parts of structure;
- Conditioning.

MR

3D	6	3 translations and 3 rotations
2D	3	2 translations and the rotation in the plan
Axi	1	1 translation along revolution axis
Fourier	6	3 translations and 3 rotations distributed on many harmonics

(MR - FIX - LIA) must be equal to zero (equals to 1 to 6 in eigenvalues computation).

5.5. Negative Pivots

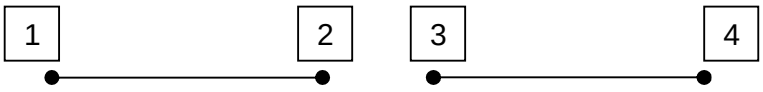
The number of negative pivots = number of generated linear constraints.

They are clearly

- generated by user with **.CLI** command. **.CLI** command generates a number of negative pivots equal to number of constraints ;
- created by rigid bodies command (**.RBE**);
- created by contact command (**.JER**).

The constraints are printed in listing for checking.

Example :



with the following condition : q2=q3.
Then we have the following system :

$$\begin{bmatrix} k & 0 & 0 & 1 \\ 0 & k & -k & -1 \\ 0 & -k & k & 0 \\ 1 & -1 & 0 & 0 \end{bmatrix}$$

Then $K_{44}^{*} = K_{44} - K_{CC}^{T} K_{CC}^{-1} K_{CC}$ but K_{cc} is positively defined because it is the stiffness of the rod and $K_{44}=0$ so finally $K_{44}^{*} < 0$.