



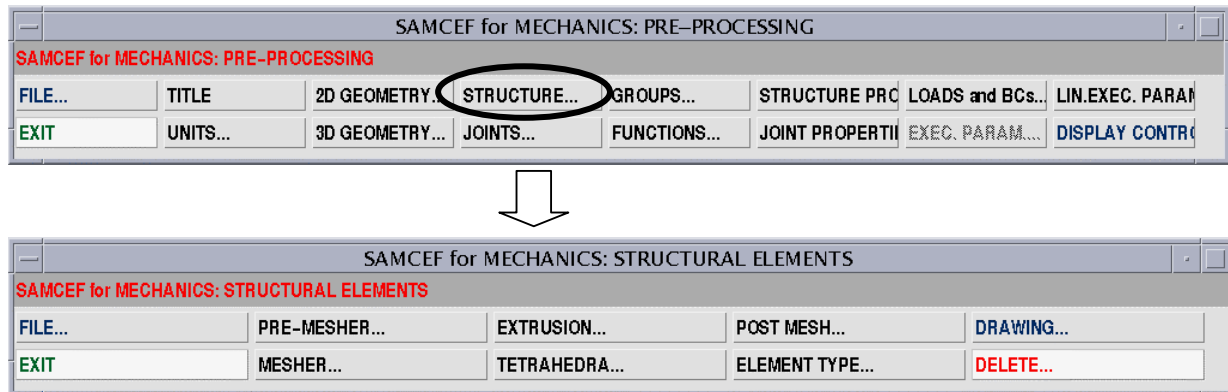
Meshing

Table of content

| | |
|--|----|
| 1. Introduction | 3 |
| 2. Mesh preparation | 4 |
| 2.1. Contours | 4 |
| 2.2. Domains | 6 |
| 3. Surfacic mesher (.GEN) | 8 |
| 3.1. Different surface mesh generators | 10 |
| 3.2. Offset | 11 |
| 3.3. Transfinite mesher | 13 |
| 3.4. LUI Mesher | 14 |
| 3.5. Triangular mesher | 14 |
| 3.6. Sadek triangular mesher | 16 |
| 3.7. Line mesher | 17 |
| 4. Choice of mesher, saving, deletion, ... | 18 |
| 4.1. Choice of mesher | 18 |
| 4.2. Numbering, elements attribute, size | 18 |
| 5. Extrusion Mesher | 20 |
| 6. Transfinite 3D Mesher | 24 |
| 7. GHS3D free Mesher | 26 |

1. Introduction

The meshing steps are distributed among different menu boxes.



- **Pre-mesher...**

Definition of the contours and domains.

- **Mesher...**

Mesher itself.

- **Post Mesh ...**

Improvement of an existing mesh including node and element modification.

- **Extrusion...**

Extrusion of an existing mesh.

- **Tetrahedra...**

Mesh with tetrahedral of a existing meshed skin with triangles.

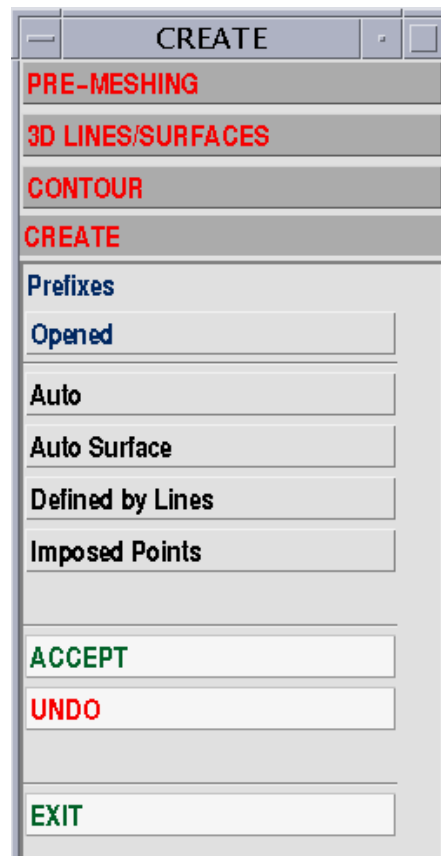
- **Element Type...**

Element behavior selection.

2. Mesh preparation

These steps consist of creating contours (outline of the mesh) and domains (part of the surface delimited by the outline to mesh)

2.1. Contours



| | |
|---|--|
| | <i>It automatically creates contours (only in $z=0$ plane)</i> |
| <code>.contour auto</code> | |
| | <i>It automatically creates contours on a surface.</i> |
| <code>.contour auto surface 2</code> | |
| | <i>Contours defined by lines.</i> |
| <code>.contour I 1 ligne 4 6 7 8</code> | |
| | <i>Contours with imposed points (imposed nodes for the mesher)</i> |
| <code>.contour I 1 point 2 3 4 ! to impose internal points inside a domain</code> | |

Remarks :



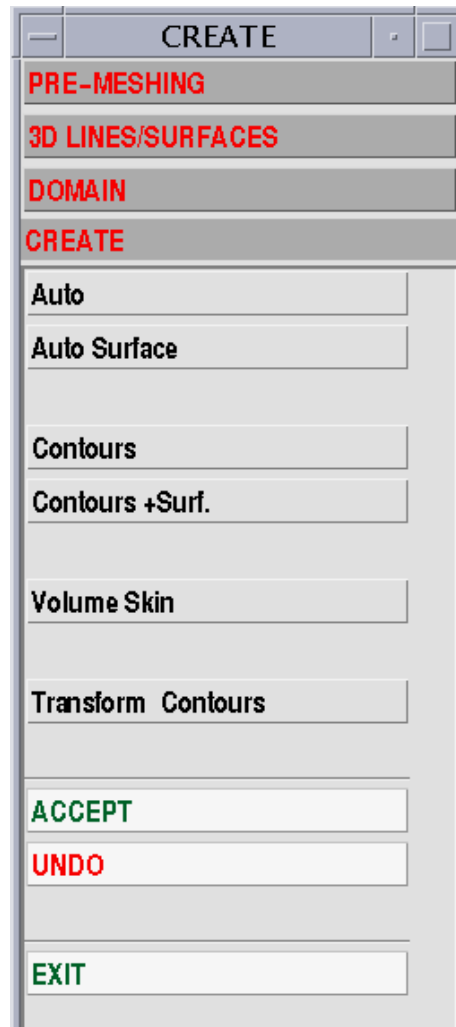
The automatic contour research does not detect contour made of 2 lines. Manually, choose **LIGNE** parameter. The sens of the first line gives the defined contour sens.

Closed contour : The contour represents a closed surface. This parameter is used with only the LINE parameter.

Opened contour : The contour is made of lines that are not closed (contour for a beam description for example). With this kind of contour, we can use the option LINE in the command .GEN to mesh a line (.GEN MAILLE Ligne....)

This command accept to create some contours only in a pre-defined surface. In 2D, the plane $z=0$ is automatically created. If the user wants to work in another plane, this plane must be created first (command .PLAN)

2.2. Domains



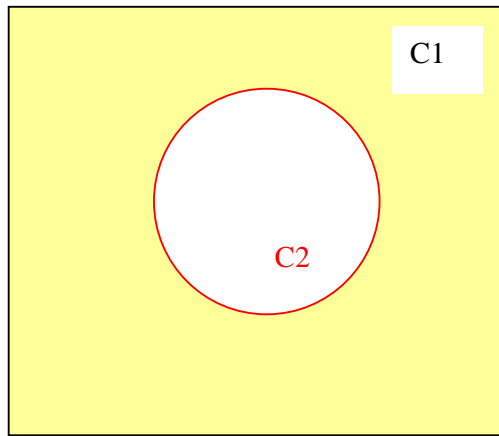
| | |
|--|--|
| | <i>It automatically creates domains (only in z=0 plane)</i> |
| <code>.domaine auto</code> | |
| | <i>It automatically creates domains on a surface.</i> |
| <code>.domaine auto surface 2</code> | |
| | <i>Domains defined by contours (the first one must be the outer contour).</i> |
| <code>.domaine I 1 contours 4 6 7 8</code> | |
| | <i>Domains defined by contours on surface 4</i> |
| <code>.domaine I 1 contours 4 6 7 surface 4</code> | |
| | <i>Domains defined by volumic skin</i> |
| <code>.domaine I 1 vpeaux</code> | |

Example: Plate with a hole

.Domaine I 1 contour 1 2

C1 is the external boundary

C2 is a closed line contour



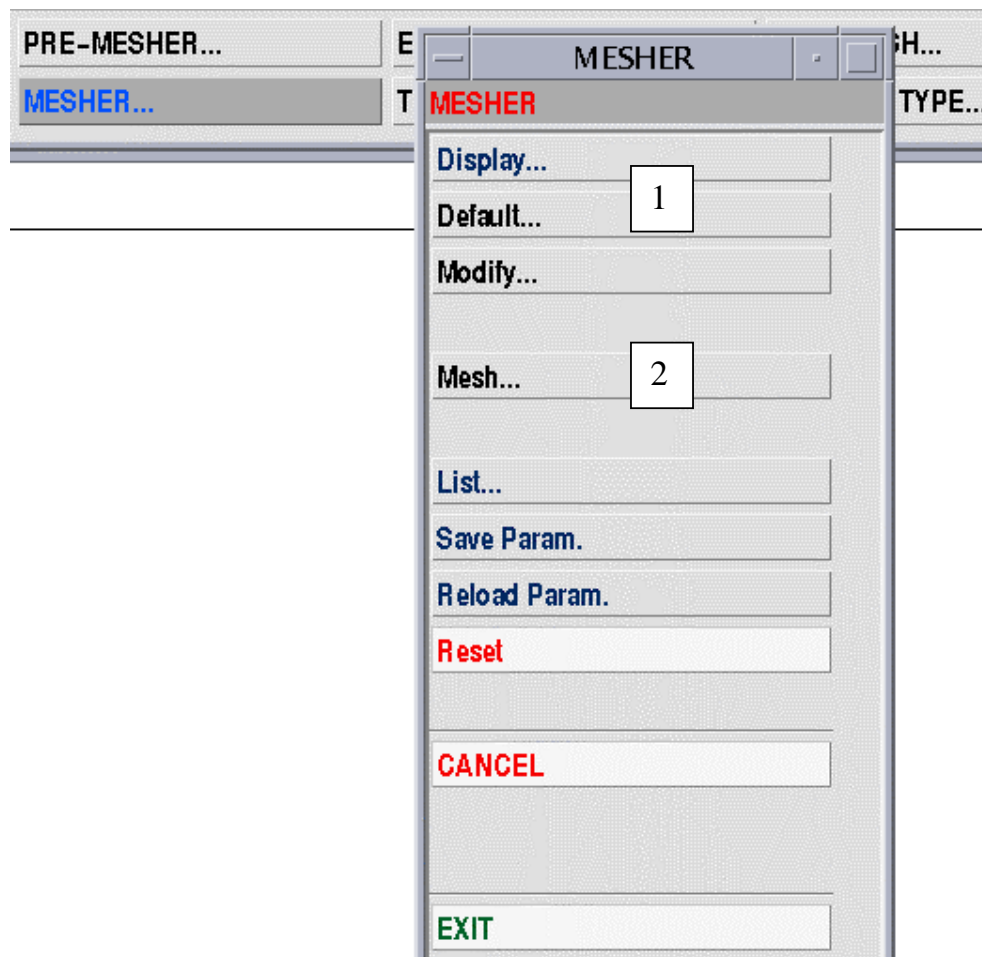
Exercise :

- Create contours and domains of 2D tool (exercice N° 7)
- Create contours and domains of 2 Materials plate (Exercise N°10)

3. Surfacic mesher (.GEN)

Meshing in 2 steps :

1. Definition of parameters (size, number of elements, ...) :
2. Meshing.



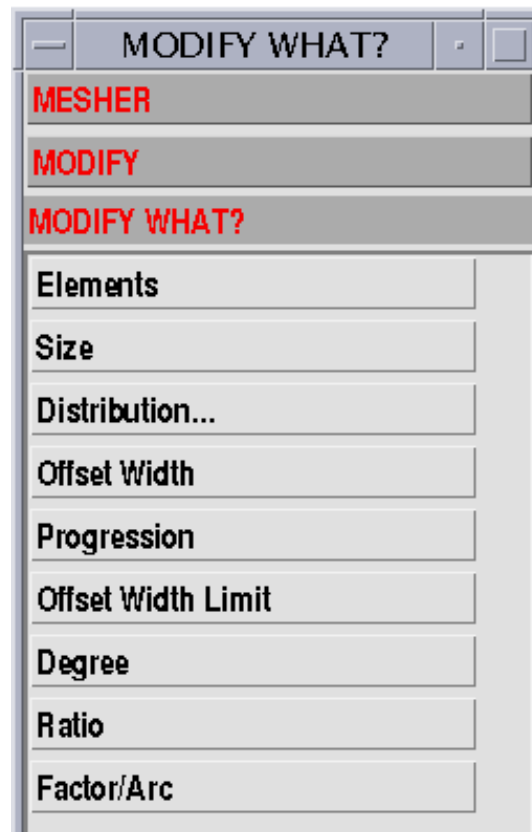
By default as preview, nodes are distributed along the lines.

Default size

1. Element size is the thirteenth of medium size of structure ;
2. Mid-Nodes are automatically created on curved lines.

The **[DEFAULT]** choice allows to modify global default values computed by program.

The discretisation can be imposed using the **Modify** menu and selecting the entities you want to affect : lines, domains or points.



| | |
|--|--|
| | <i>Number of element on lines.</i> |
| .gen modify ligne 12 13 element 5 | |
| | <i>The size on a domain.</i> |
| .gen modify domain 12 dimension 0.5 | |
| | <i>The ratio (1=uniform, 2=ratio first/last element, 3= ratio Last/first element)</i> |
| .gen modify ligne 24 distrib 2 0.5 | |
| | <i>The degre on all domains to be meshed mesh inside the command</i> |
| .gen tout degre 2 | |

Examples :

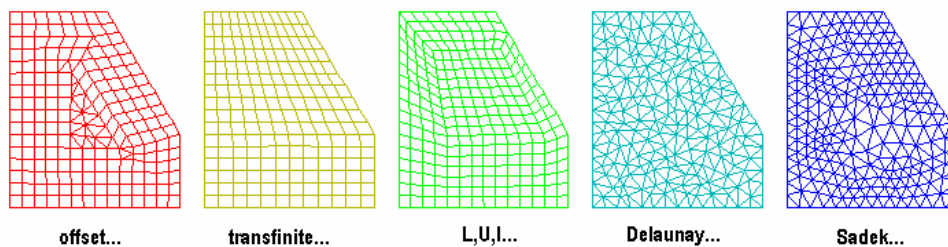
.gen modif ligne 24 distrib ri rj ! ri=1 : Uniform distribution

.gen modif ligne 24 distrib ri rj ! ri=2 and rj=r : Distribution following a geometric progression such that: **r** = size of the **last** element on the line / size of the **first** element on the line.

.gen modif ligne 24 distrib ri rj ! ri=3 and rj=r : Distribution following a geometric progression such that: **r** = size of the **first** element on the line / size of the **last** element on the line.

3.1. Different surface mesh generators

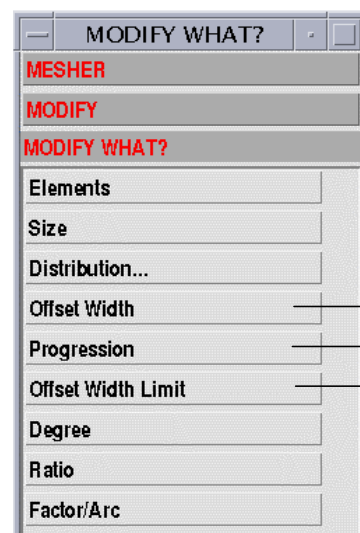
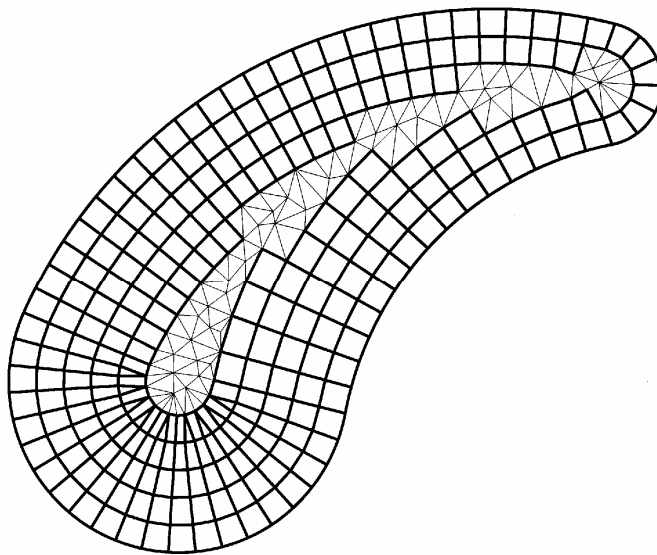
Different mesh generators are available. To activate this mesher, its name should be added as parameter in the command .GEN



3.2. Offset

For each contour of domain to be meshed, the program computes an "offset" contour, parallel to initial polygon. In corridors, quadrangular elements (or triangles if asked) are created. The last zones are filled up with triangles (Delaunay mesher).

- The domain must be closed and located on a surface (verification by **.DOM**).
- the contours of a same domain must have no contact or intersection.
- the domain can be holed.



General modification on a domain.

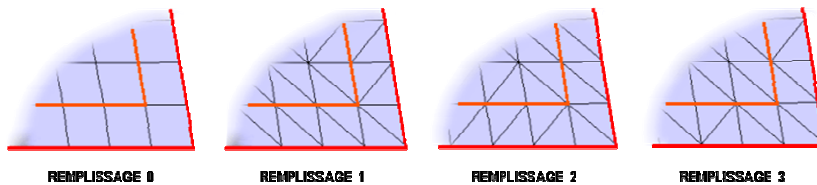
```
.gen
modif domain 1 decalage 3 progression 1.2 limit 6.
Maille 1 offset
```

Offset : Number of the offset after which the generator is stopped. On the picture above, offset is 5. If no value is specified, there is no upper limit.

Limit : Upper bound for the mesh generator

Remplissage- Determines the most suitable method of filling in the offset corridors:

- 0: the corridors are filled in with quadrangular cells (*by default*)
- 1: the corridors are filled in with triangles (method 1)
- 2: the corridors are filled in with triangles (method 2)
- 3: the corridors are filled in with triangles (method 3)



Progression r : Progression of the mesh size inside the offset width

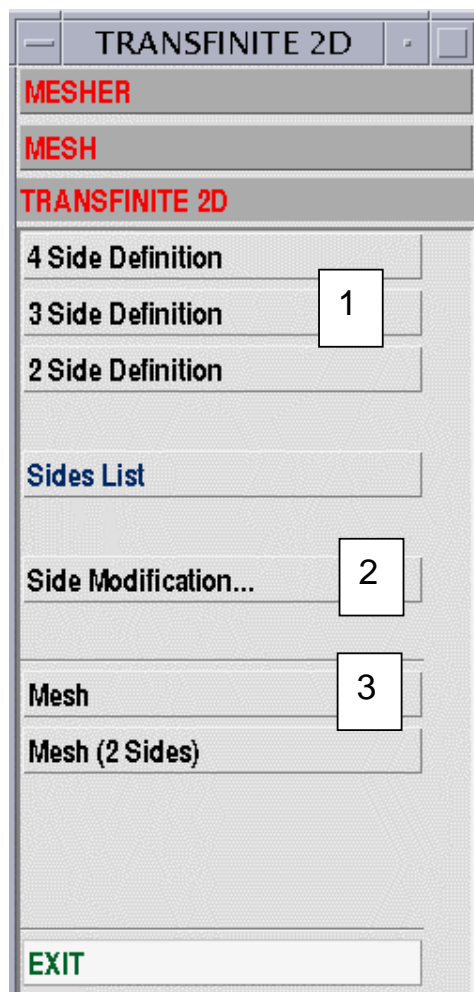
If $r > 0$, the progression is geometric,

If $r < 0$, the progression is arithmetic following $|r|$

3.3. Transfinite mesher

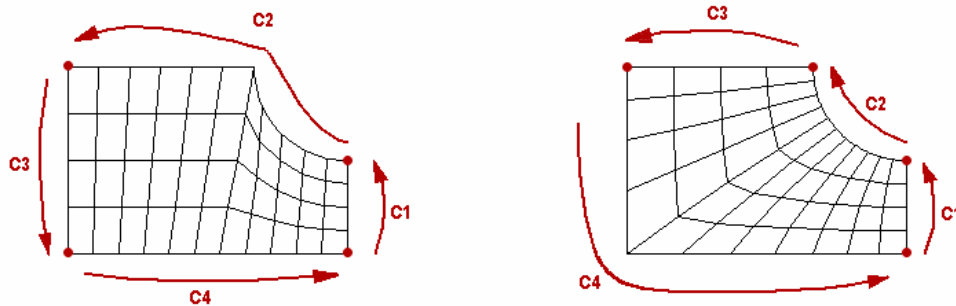
The transfinite mesher allows to only mesh domains with same number of element on opposite side. There are 3 steps :

1. Definition of the sides
2. Modification to imposed same number of element on opposite sides.
3. Meshing



| | |
|---|--|
| <i>Mesh of 4 sides domain.</i> | |
| <i>Side 1 : 4 7 9 6; Side 2 : 10; Side 3 : 23; Side 4 : 30 32 1</i> | |
| .gen | <pre> c1 4 7 9 6 c2 10 c3 23 c4 30 32 1 modif direction ligne 4 domain 5 element 5 maille 5 transfini </pre> |

Example :

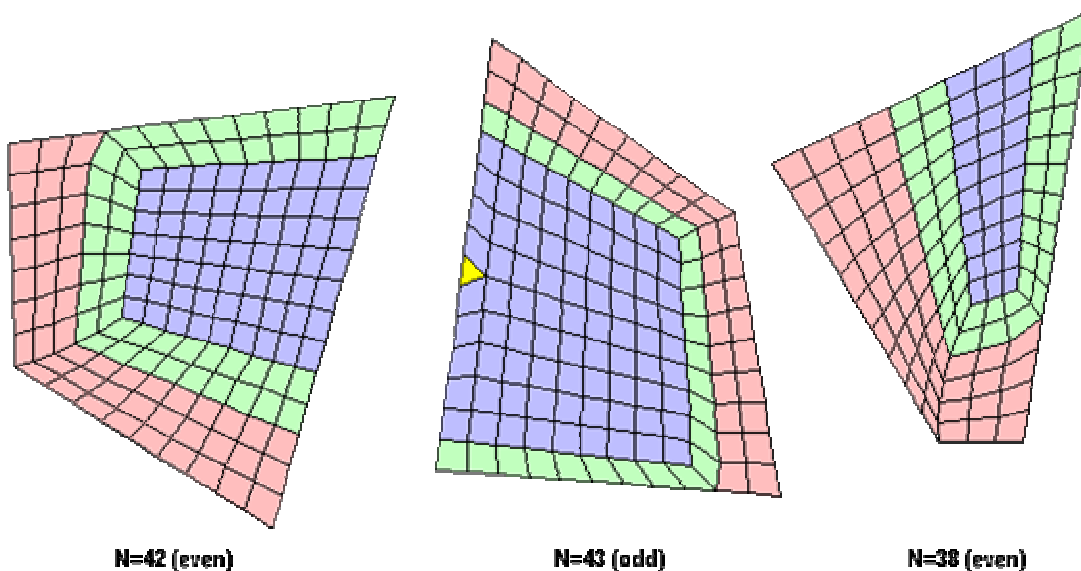


3.4. LUI Mesher

This extension of the Transfinite Mesh Generator is only to **used for 4-sided domains with any wanted number of nodes on the four sides**. The parameter **Transfini** needs only to be replaced by **LUI**. The other parameters C1, C2, C3, C4 remain available.

Remarks :

If the sum total of nodes laying on the outline of the domain is **even**, the mesh is composed of quadrangular cells only. On the opposite, if the sum total of nodes laying on the outline is **odd**, one triangular element must appear somewhere along the outside.



N= sum total of nodes laying on the outline

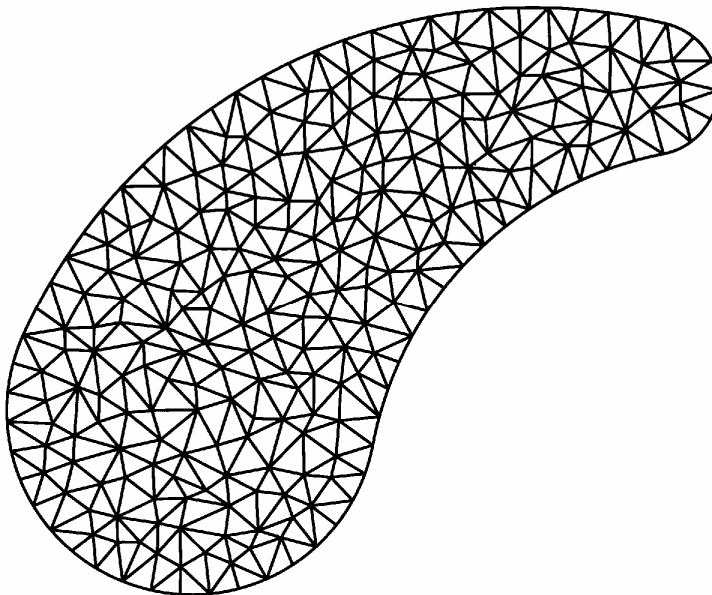
3.5. Triangular mesher

The triangular mesher is based on the Delaunay method. It works for any configuration. This mesher divides the domain into triangles and then add nodes located at the center of gravity of each triangle in order to create new triangle until the area of these triangle is smaller than a reference value.

This mesher must be used to generate the skin mesh of solids necessary for the tetraedral mesher (GHS3D).

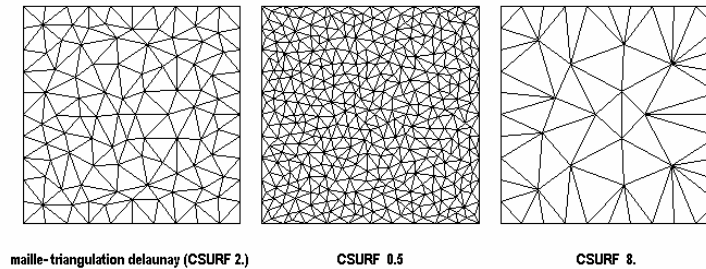
| | |
|---------------------------------------|--|
| <i>Triangular Meshing.</i> | |
| .gen maille 5 triangle | |
| .gen maille 5 triangle csurf i | |

- The domain must be closed and located on a surface



(verification by **.DOM**).

- The contours of a same domain must have no contact or intersection.
- The domain can be holed.



CSURF : Multiplicative factor of the reference surface for cells. The mesh generator will try to divide the cells until the produced cells have a surface area smaller than this reference area. The higher the multiplicative factor, the fewer cells are generated.

3.6. Sadek triangular mesher

This mesher is generally reserved for disc meshing. It systematically creates a **node** at the **center** of the meshed disc. It is only accessible in command mode.

MAILLE- - ...TRIANGLE SADEK Sadek meshing

MAILLE- - ...TRIANGLE DELAUNAY use of Delaunay meshing for domain previously meshed with Sadek mesher.
Other parameters : online documentation

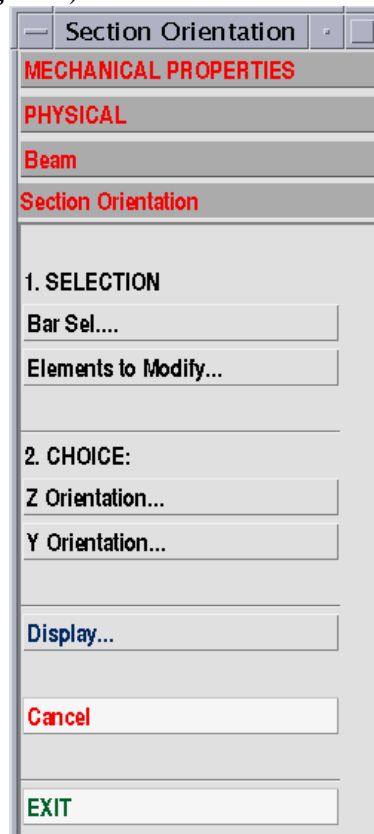
3.7. Line mesher

Line domain can be meshed with 1D Type elements, lately used as rods or beams. It is possible to choose element type at this step of the meshing.

For the beams, you have to specify a point for the definition of the local Z-axis.

| | |
|------------------------------------|--|
| Rods | |
| Beams | |
| <i>Rod mesher</i> | |
| .gen maille 1 ligne | |
| <i>Beam mesher.</i> | |
| .gen maille 1 ligne point 3 | |

It is also possible (largely **suggested**) to mesh with rods and later transform rods in beams:



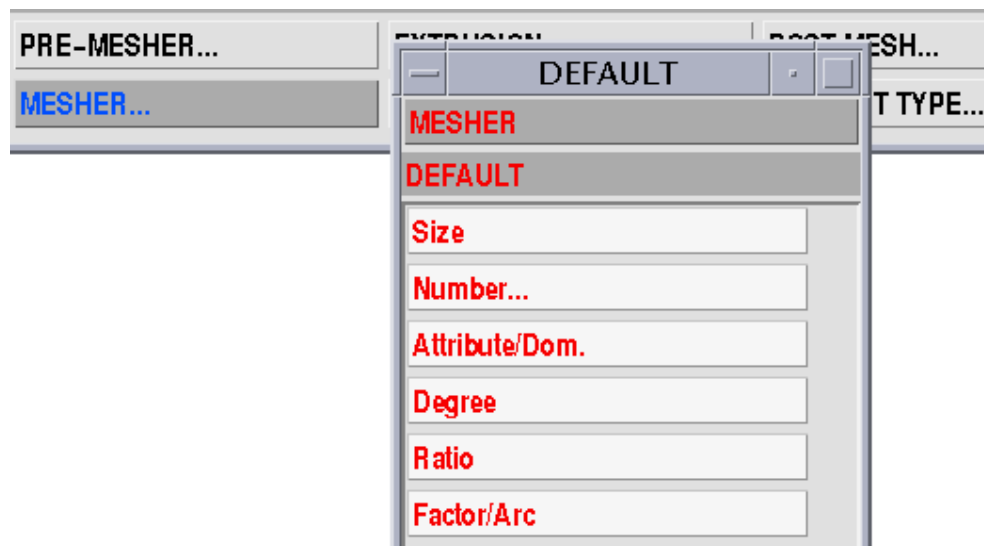
| |
|--|
| <i>Transform all rods in beams defining point 3 for the local z plane.</i> |
| Grap orient ; .beam tout planz point 3 |

4. Choice of mesher, saving, deletion, ...

4.1. Choice of mesher

- Priority is given to the specified mesher type : transfinite, offset, etc.
- If no mesher is chosen (**Free** at menu) :
 - When domain has already been meshed, the last type is used again.
 - If not, the program respectively tries :
 1. transfinite mesher ;
 2. offset mesher ;
 3. Triangular mesher.

4.2. Numbering, elements attribute, size



| | |
|---|--|
| <i>Numbering for nodes (or elements).</i> | |
| .gen numerote 1 noeud | |
| <i>Give a unique attribute to cells belonging to each domain on the same surface.</i> | |
| .gen | |
| Domaine 1 att 1 | |
| Domaine 2 att 2 | |
| Maille 1 | |
| Maille 2 | |



| <i>Size of element</i> |
|---------------------------|
| .gen dimension 2.5 |

For the size, see also in the Manual the following parameters : Repartition, rapport, degree....

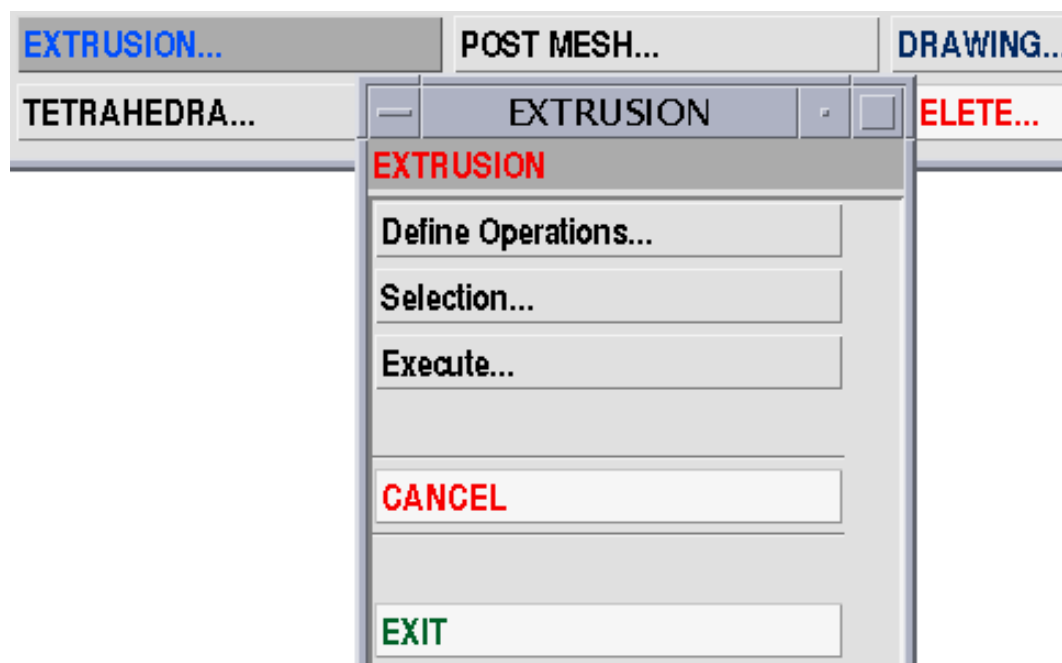
5. Extrusion Mesher

The extrusion works on an existing mesh (line, edges of 2D element, 2D element or faces of volumes). It allows to create $n+1$ dimension cells from n dimension cell, i.e. :

- $n = 1$ (rods, beams, membranes edges, shells edges) ----> membranes or shells surfacic cells from lineic cells;
- $n = 2$ (membranes, shells, volumes faces) ----> volumes volume cells from surfacic shells.

There are 3 steps :

1. Definition of the operations (translation, rotation, symmetry...)
2. Selection of the cells to be extruded by group, number, attribute..
3. Execution of the specified operations with following options :
 - With gluing or not between existing cells and newly created one (option COLLER)
 - With attribute number modification or not for newly created cells (option GTAR)
 - About keeping of initial cells (**conserver 1** or **0**).



Extrusion of a group of faces with operations 1 and 2 (old elements are removed)

```
.EXT  
OPERATION 1 TZ 2 ELEM 3 INTER 1  
OPERATION 2 TZ 3 ELEM 4 INTER 1  
ELEMENTS GROUP "SolidFaces"  
EXEC 1 2
```

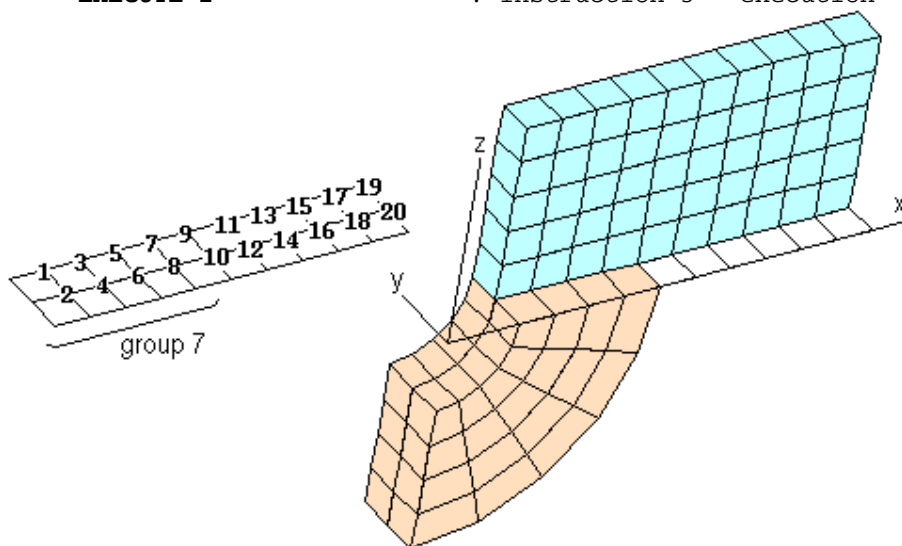
Comments :

- Operation 1 : Generation of 3 elements, along z-axis with one node of interface
- Operation 2 : Generation of 4 elements, along z-axis with one node of interface
- The operation will be applied on the group of "SolidFaces"
- Execution of the operations 1 and 2

Examples :

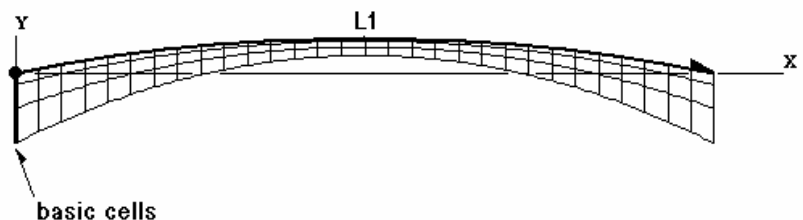
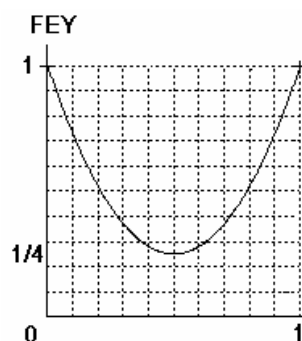
Example 1:

```
.EXT MAILLES 1 A 20 PAS 2      ! instruction 1 = selecting cells
TZ 10. ELEMENTS 5              ! instruction 2 = defining operation
EXECUTE                        ! instruction 3 = execution
MAILLE                          ! instruction 1 = selection cells and groups
GROUP 7
RY 90. ELEMENTS 6              ! instruction 2 = defining operation
EXECUTE 2                      ! instruction 3 = execution
```



Example 2: Extrusion along a curve by a function

```
.ARC I 1 CPOINTS 0. 0. 10. 1. 20. 0.
.FCT      CREE FONCTION I 1
          CREE VALEUR Y U ANALYTIQUE "3.*$u*$u -3.*$u +1." BORNES 0 1
.EXT
LIGNE 1 PARALLELE FEY 1 ELEMENTS 30 INTERFACE 1
```

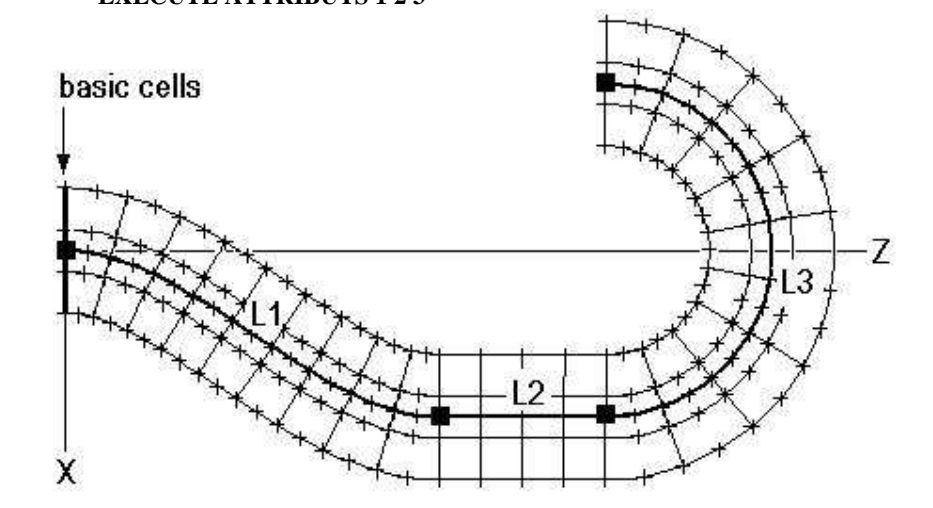


EXECUTE

Example 3:

.EXT

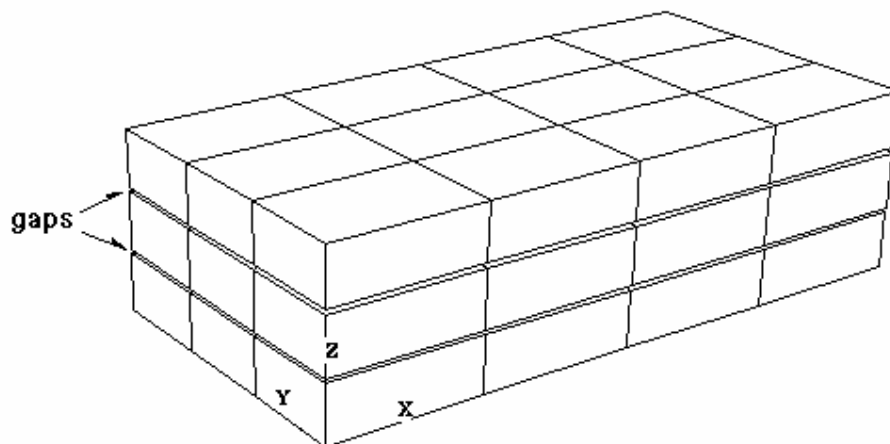
LIGNE 1 NORMALE ELEMENTS 9 INTERFACE 1
LIGNE 2 PARALLELE ELEMENTS 4
LIGNE 3 NORMALE ELEMENTS 9 INTERFACE 1
EXECUTE ATTRIBUTS 1 2 3



Example 4:

.EXT

TZ 3. ELEMENT 1 ! operation 1 = translation along Z axis with creation of one layer of volumic element
NULLE ! operation 2 = simple duplication
EXECUTE 1 2 1 2 1 ! generation of 3 layers of volumic element without commons nodes



Exemple 5:

.EXT

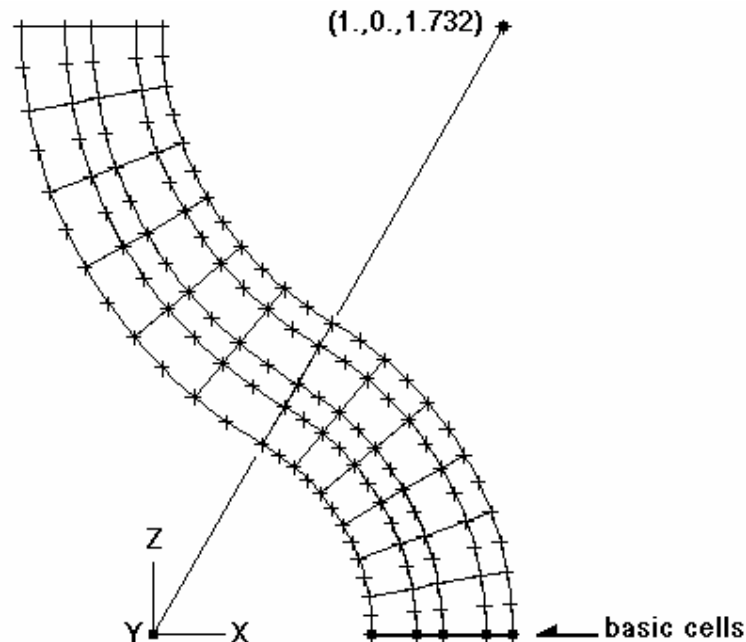
RY -60. ELEMENTS 6 INTERFACE 1

! opération 1 = rotation de -60 degrés autour de l'axe Y créant 6 couches de volumes de degré 2

RY 60. ELEMENTS 6 INTERFACE 1 CCENTRE 1. 0. (sqrt(3.))

! opération 2 = rotation de 60 degrés autour d'un axe parallèle à Y passant au point de coordonnées (1,0,1.732) et créant 6 couches de volumes de degré 2

EXECUTE 1 2



6. Transfinite 3D Mesher

The automatic mesher allows meshing 3 types of **3D domains** created with .BOX command:

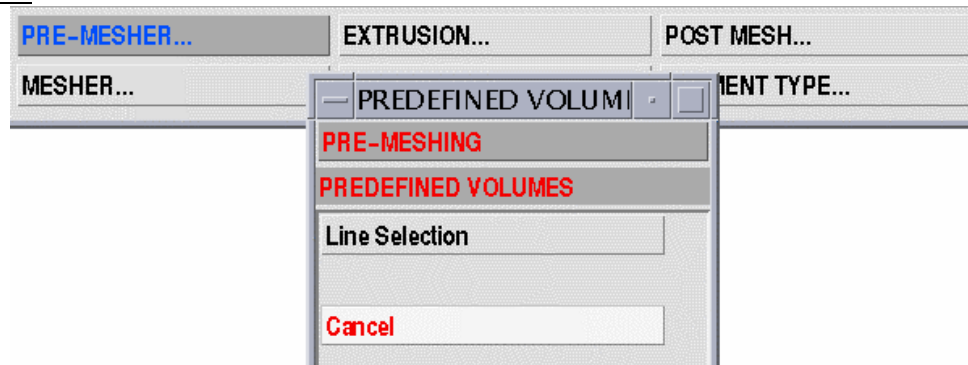
- Hexahedra (6 faces, 12 edges).
- Prism with triangular base (5 faces, 9 edges).
- Pyramid with quadrangular base (5 faces, 8 edges).

Using of the command .BOX

1. Create the lines constituting volume edges.

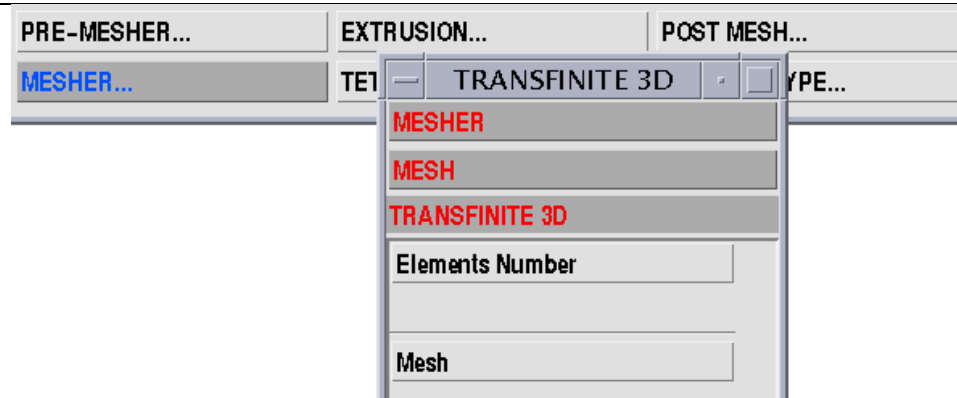
2. Select all these lines with **.BOX** command, Contours and 3D domains are automatically created by the command.
3. Mesh with the command **.GEN**

Example :



Definition of the box by selection of the lines

| .BOX | LIGNES | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 |
|------|--------|---|---|---|---|---|---|---|---|---|----|----|----|
| | | | | | | | | | | | | | |

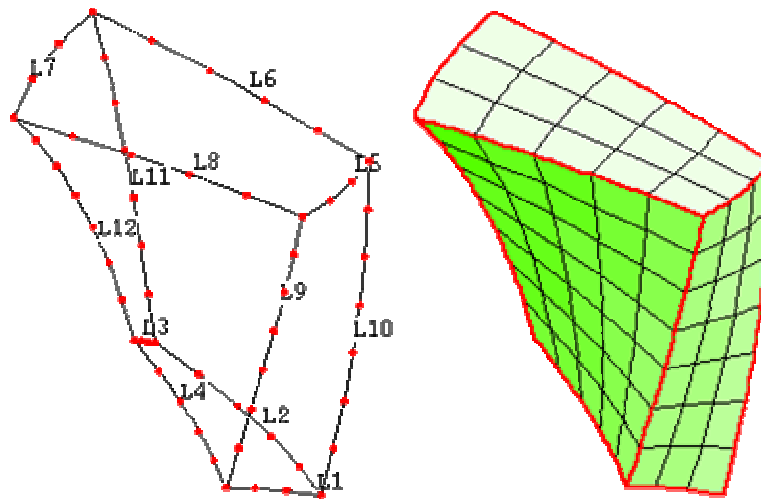


Meshing modifying the number of elements

| | |
|--|--|
| .GEN MODIF LIGNE 9 DOMAIN 2 ELEMENTS 2 MAILLE 1 | |
|--|--|

• **Example: Transfinite mesh of a hexahedron**

```
... (line definition)
.BOX LIGNES 1 A 12
.GEN
  MODIFIE DOMAINE 1 DIRECTION LIGNE 1 ELEMENTS 3
  MODIFIE DOMAINE 1 DIRECTION LIGNE 2 ELEMENTS 5
  MODIFIE DOMAINE 1 DIRECTION LIGNE 9 ELEMENTS 7
  NUMEROTE NOEUDS
  MAILLE 1 TRANSFINI
```



7. GHS3D free Mesher

- Free Mesher based on the VORONOI-DELAUNAY method
- Developed by INRIA-SIMULOG
- For any 3D domains with any shape
- In Samcef, available in option
- Launch by the command .GHS3D

Using

- From a triangular surface mesh that represents the boundaries of the domain, the mesher generates a tetrahedral mesh.
- The object can contain cavities et holes but can be only made of one domain (only one .DOM command to create this domain)
- If the structure contains several 3D domains, then the user must use the command .GHS3D as many times as there are some domains.
- It is possible to keep the initial surface mesh.

Steps

- Mesh the external surface of the structure with degree 1 triangles (command .GEN)
- Check that external surface is closed (.COL or GRAP ARETE 2).

- Select a group of triangular elements making a closed 3D domain (GROUP and ATTRIBUTE of the **.GHS 3D** command).
- Mesh directly using the command **.GHS MAILLE.....**

Example :

| |
|---|
| <i>Automatic mesh on a group (skin of the volume)</i> |
| .GHS MAILLE "filemane" GROUP "triangle_group" IATTRIBUT 12 |

.GHS MAILLE "name" group 1
Iattribut 12

Meshing of the surfacing mesh contains in the group 1. The attribute of the generated mesh is 12.

Exercise :

- Definition of the mesh (exercises 7) : use of the 2D mesher
- Definition of the mesh (Exercise N°10)
- Building a complete mesh by extrusion (exercise 13)
- Building a complete mesh by extrusion (exercise 11)