SSLS131 – Optimization of the radius of curvature of a bent piping

Summary:

This test of demonstration illustrates the use of the language python in *Code_Aster*.

The objective of this CAS-test is to optimize the value of the radius of curvature of a piping via a loop python. The radius of curvature is modified repeatedly. The optimal value is obtained when the maximum constraint of Von Mises is lower than a threshold.

The language python, in this CAS-test allows repeatedly:

- to modify the file describing the geometry of the elbow,
- of launching GMSH in order to net the elbow,
- to recover the maximum constraint of Von Mises,
- to evaluate a criterion of stop of the iterations,
- of launching GMSH to carry out an interactive postprocessing.

This CAS-test takes again the problem of the CAS-test forma01f: bent piping, made up of a linear elastic material, subjected to a force applied at its end, modelled by elements of hulls `DKT`.

Note:

To have the interactive functions of visualization, to put interactive = 1 at the beginning of the command file.

1 Problem of reference

1.1 Geometry

The study relates to a piping including two right pipes and an elbow [Figure 1.1-a].

The geometrical data of the problem are the following ones:

- the length L_G of the two right pipes is of $3\,m$,
- the ray Rc initial of the elbow is of $0.3\,m$,
- the angle θ elbow is of 90 degrees,
- the thickness of the right pipes and the elbow is of $0.02\,m$,
- and the external ray Re right pipes and elbow is of $0.2\,m$.

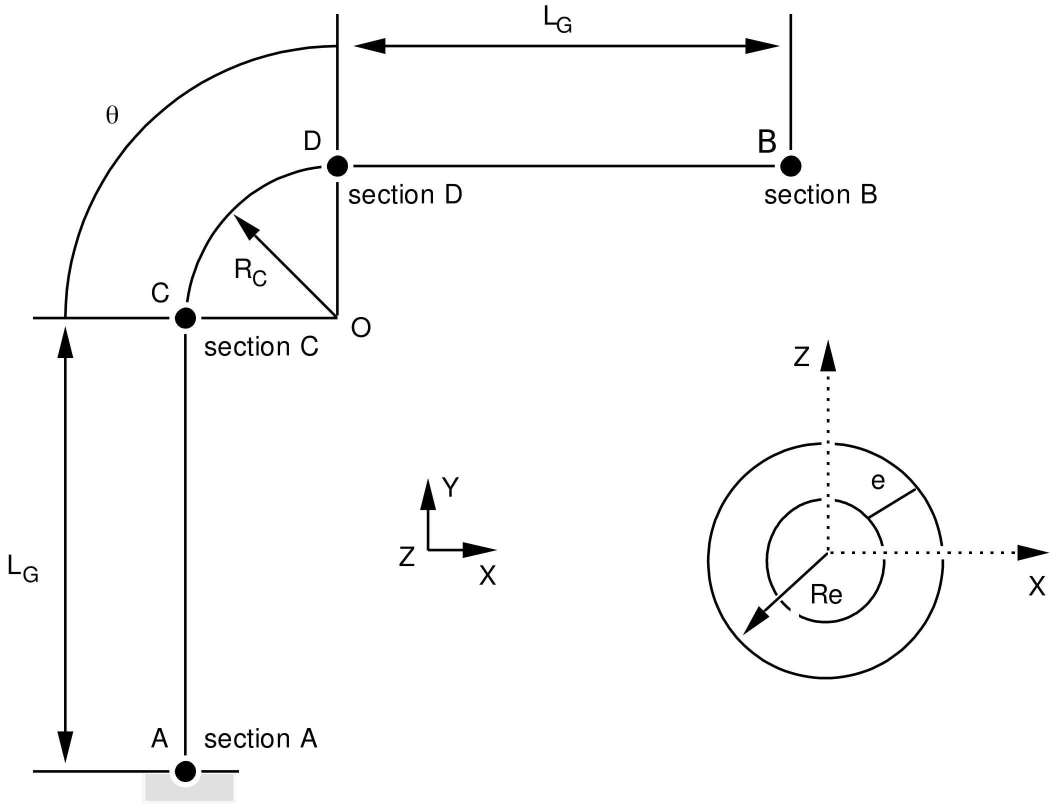

Figure 1.1-a

Note:

The geometry of the problem has a symmetry compared to the plan (A, X, Y).

1.2 Material properties

Isotropic linear elastic material. The properties of material are those of steel $A42$:

- the Young modulus: $E=1.8\,10^{11}\,Pa$
- the Poisson's ratio: $\nu=0.3$

1.3 Boundary conditions and loadings

- Boundary conditions: embedding on the level of the section A ,
- Loading: constant force FY directed according to the axis Y and applied to the section B .

The value of FY is calculated to leave:
- average radius: $RMOY=0.19$,
- total force applied: $FTOT=500000\,N/m^2$,

Its expression is the following one:
$FY=FTOT/(2\pi\,RMOY)(\simeq 418828.8)$

- The constraint of limiting Von Mises is of $2.0\mathrm{E}+09\,N/m^2$

2 Reference solution

2.1 Method of calculating used for the reference solution

The reference solution is obtained numerically. It is thus only about one test of nonregression.

2.2 Results of modeling

The optimal value of the radius of curvature, respecting the criterion $\sigma_{mises}<2.10^9 N.m^{-2}$ is of $1.1\mathrm{m}$, obtained after 5 iterations. With each iteration the radius of curvature is increased by $0.2\,m$.

Iterations	Radii of curvature	Constraint max of Von Mises
1	$0.3\,m$	$3.3315\,E+09\,N/m^2$
2	$0.5\,m$	$2.7647\,E+09\,N/m^2$
3	$0.7\,m$	$2.4256\,E+09\,N/m^2$
4	$0.9\,m$	$2.1727\,E+09\,N/m^2$
5	$1.1\,m$	$1.9670\,E+09\,N/m^2$

2.3 Uncertainty on the solution

Solution of nonregression.

3 Modeling A

3.1 Characteristics of modeling

Modeling in elements hulls (DKT).

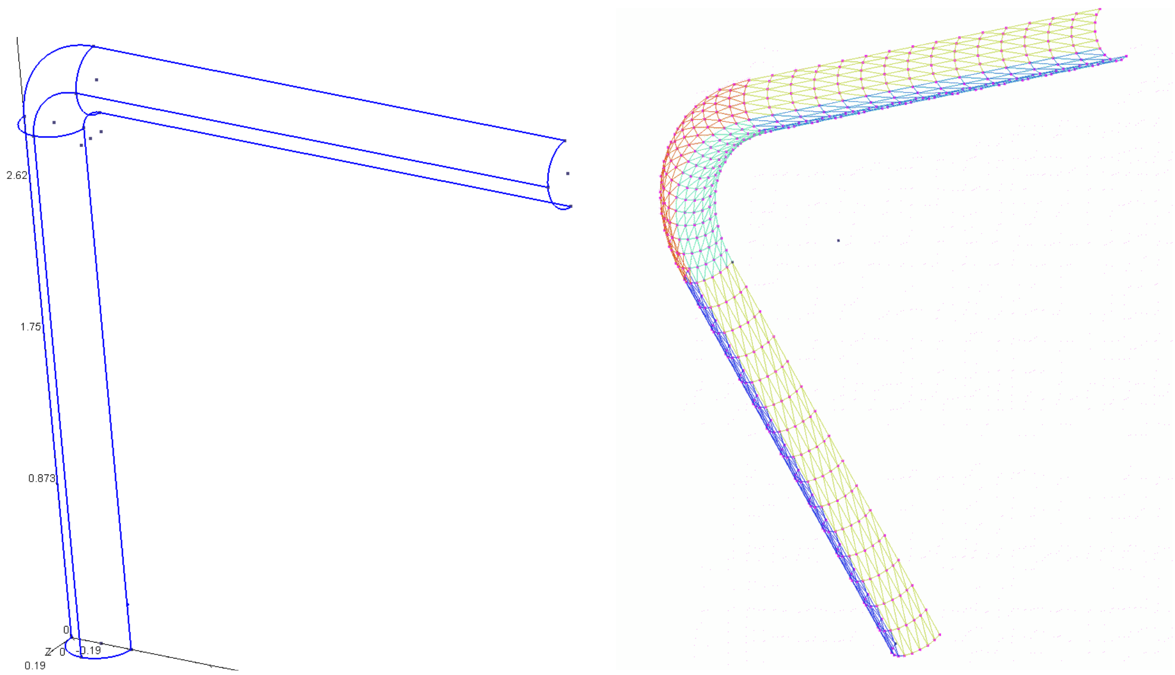

Figure 3.1-a: geometry and grid of the half roll (initial RC)

3.2 Characteristics of the grid

The grid is regenerated with each iteration because of the modification of the radius of curvature of the geometry. However the topological characteristics of the grid are unchanged:

- 1013 meshs (900 TRIA3, 110 SEG2, 3POI3)
- 507 nodes.

The groups of meshs correspond to:

- $GM30 \Leftrightarrow$ surface of $TUYAU$
- $GM28 \Leftrightarrow$ section B (effort)
- $GM31 \Leftrightarrow$ not $A1$ $(-R, 0, 0)$
- $GM27 \Leftrightarrow$ section A (embedding)
- $GM29 \Leftrightarrow SYMETRIE$

3.3 Orders Aster

This paragraph describes the algorithm used in the command file and presents the orders *Code_Aster* used.

- Initialization of certain variables:
- Radius of curvature: $Rc=0.3$
- Convergence criteria: $crit=2.0\mathrm{E}+09$ (constraint of Von Mises targets)
- One enters the loop python whose criterion of stop relates to the constraint max of Von Mises. As long as the criterion is not respected, the following instructions are carried out:

Buckle python of optimization of the radius of curvature

- Reconstruction of the grid:
  - one redefines Rc in the geometrical file .geo of GMSH
  - one launches gmsh via python to generate the file of grid .msh
- Reading of the grid (`PRE_GMSH`) and generation of the grid (`LIRE_MAILLAGE`). One uses `DEFI_GROUP` to re-elect the groups of meshs according to the correspondence:

```
# GM30 <=> PIPE # GM28 <=> EFOND
# GM31 <=> A1       # GM27 <=> ENCAST
# GM29 <=> SYMMETRY
```

- Definition of the finite elements used (`AFFE_MODELE`). The right pipes and the elbow are modelled by elements of hull (`DKT`).
- Reorientations of the normals to the elements: one uses `MODI_MAILLAGE` in the same way to direct all the elements, with a turned normal towards the interior.
- Definition and assignment of material (`DEFI_MATERIAU` and `AFFE_MATERIAU`). The mechanical characteristics are identical on all the structure.
- Assignment of the characteristics of the elements hulls (`AFFE_CARA_ELEM`): thickness, vector V defining the reference mark of examination (keyword `ANGL_REP`)
- Definition of the boundary conditions and the loading (`AFFE_CHAR_MECA`).
  - Piping is embedded in its base, on all the nodes located in the Y=0 plan. Piping presents a symmetry plane $Z=0$.
  - An effort distributed is calculated FY directed according to the axis Y and applied to the section B , (the effort distributed is such as the resultant $2\pi . RMOY.FY=FTOT$, $FTOT$ being the total force which one wishes to apply). To apply the effort to the section B, one will use `FORCE_ARETE`.
- Resolution of the linear elastic problem (`MECA_STATIQUE`).
- Calculation of the stress field by elements to the nodes for each loading case (option '`SIGM_ELNO`'). The constraints are calculated in the definite local reference mark for each element using the vector V (keyword `ANG_REP` precedent). To use `NIVE_COUCHE` to define the level of calculation in the thickness.
- Calculation of the stress field equivalent by elements to the nodes calculated starting from the stress field (option '`SIEQ_ELNO`').
- Calculation of the preceding fields to the nodes (options '`SIGM_NOEU`','`SIEQ_NOEU`')
- Impression of the results (`IMPR_RESU`).
- Determination of a table containing calculations of averages of the stress field equivalent to the nodes. ('`POST_RELEVE_T`')
- Extraction of the component `VMIS` preceding table via python.
- Test of stop:
  - If `VMIS` is higher than `CRIT`, then one reiterates with $Rc=Rc+0.2$
  - If not one leaves the loop python.

4 Results of modeling A

4.1 Values tested

Iterations	**Radii of curvature**	***Constraint max of Von Mises***
1	$0.3\,m$	$3.3315\mathrm{E}+09\,N/m^2$
2	$0.5\,m$	$2.7647\mathrm{E}+09\,N/m^2$
3	$0.7\,m$	$2.4256\mathrm{E}+09\,N/m^2$
4	$0.9\,m$	$2.1727\mathrm{E}+09\,N/m^2$
5	$1.1\,m$	$1.9670\mathrm{E}+09\,N/m^2$

5 Summary of the results

The optimal radius of curvature respecting the criterion: constraint max of Von Mises lower than $2.0\mathrm{E}+09$ is of $1.1\mathrm{m}$. It was obtained after 5 iterations, and the maximum constraint of found Von Mises is of $1.9670\mathrm{E}+09\,N/m^2$.