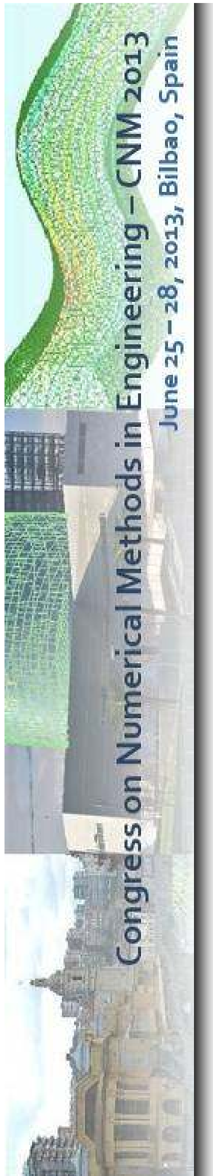
Single Point Incremental Forming using Adaptive Remeshing Technique with Solid-Shell Finite Elements

J. Sena*, R. Sousa, A. M. Habraken, R. A. F. Valente, L. Duchêne

Department of Mechanical Engineering
University of Aveiro
*sena.ilidio@ua.pt

GRIDS



Outline

1. Introduction

2. SPIF Numerical simulation

3. Adaptive Remeshing

4. Benchmark example

    4.1. Solid-Shell Element: RESS (Reduced Enhanced Solid-Shell)

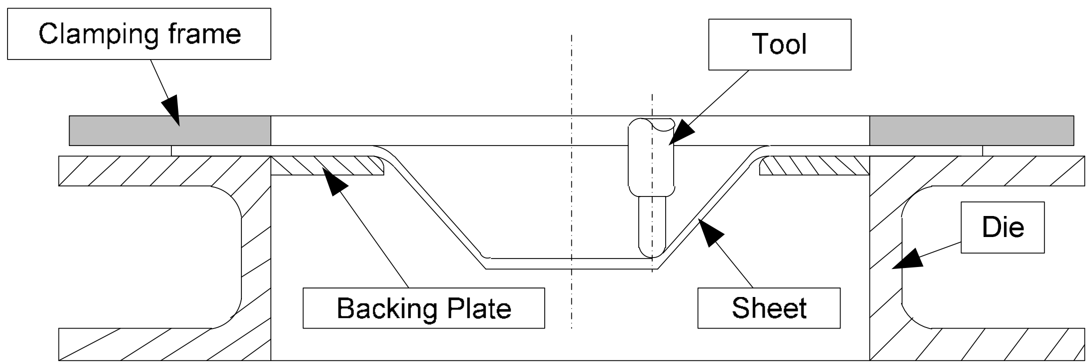
5. Final Considerations

GRIDS

Introduction

- Single point incremental forming (SPIF) is a sheet metal forming process that is appropriate for rapid prototyping.

- In SPIF, the lower surface of the metal sheet doesn’t require any dedicated dies or punches to form a complex shape.



- The tool is guided by numerical control system, which defines the trajectory where the forming tool follows and progressively deforms a clamped sheet into its desired shape

- The zone where high deformations occur is always close to the current location of the tool.

- PROBLEM:

  - The constantly changing contact between the tool and the metal sheet during the process.
  - The nonlinearities involved causes a huge computational time.

- The nonlinear finite element code used is the LAGAMINE developed in FORTRAN by MSM team of ArGEnCo department in University of Liège since 80's.

- In LAGAMINE was developed the Adaptive Remeshing available only for Shell finite elements*.

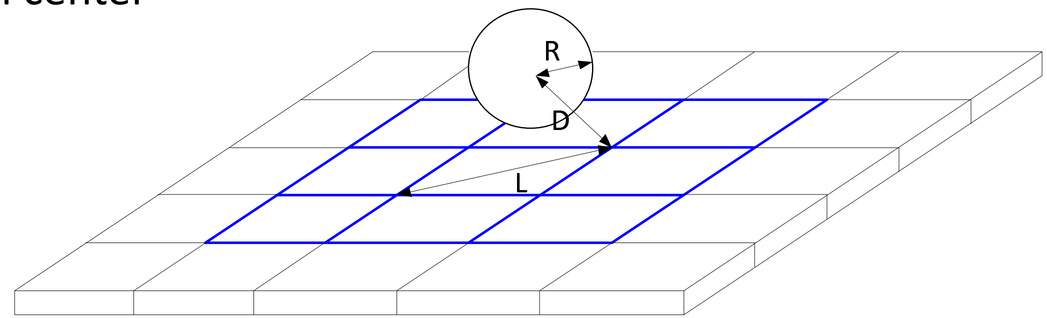
- The Adaptive Remeshing Technique was extended to use with 8 nodes solid element.

- Objective:
  - Use a 3D constitutive law
  - Prediction of the sheet thickness strain
  - Avoid the initial refined mesh with 8 nodes finite element

*see Cedric Lequesne *et al.*, Numisheet 2008, Switzerland, September 1 - 5, 2008.

Remeshing Criterion

- Selection of a neighborhood around the position of the tool center



- Proximity condition :

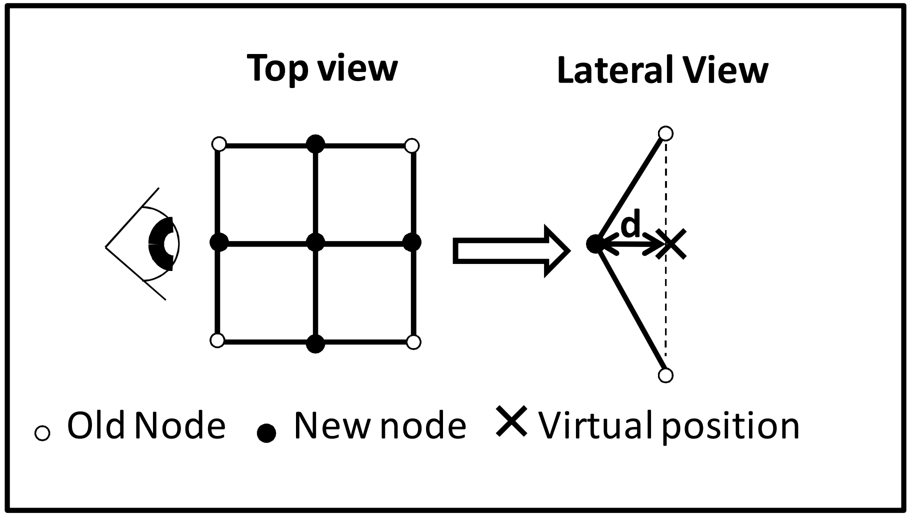
$$D^2 \leq \alpha \ (L^2+R^2)$$

- D: minimum distance between the tool center and the four nodes of the contact element
- L: length of the longest diagonal of the element
- R: radius of the tool
- α: coefficient adjusting the size of the neighborhood chosen by the user

Derefinement Criterion

- Computation of the initial relative position:

$$\underline{X}_v = \sum_{i=1,4} H_i(\xi,\eta)\underline{X}_i$$



- Computation of the distance

$$d = \left|\underline{X}_c - \underline{X}_v\right|$$

- Criterion of reaction of coarse element

$$d \leq d_{max}$$

–H_i: interpolation function
–x, h: initial relative positions of the node in the coarse element
–X_i: nodes positions of the coarse element
–X_c: Current position of the node
–d_{max}: maximal admissible distance chosen by the user

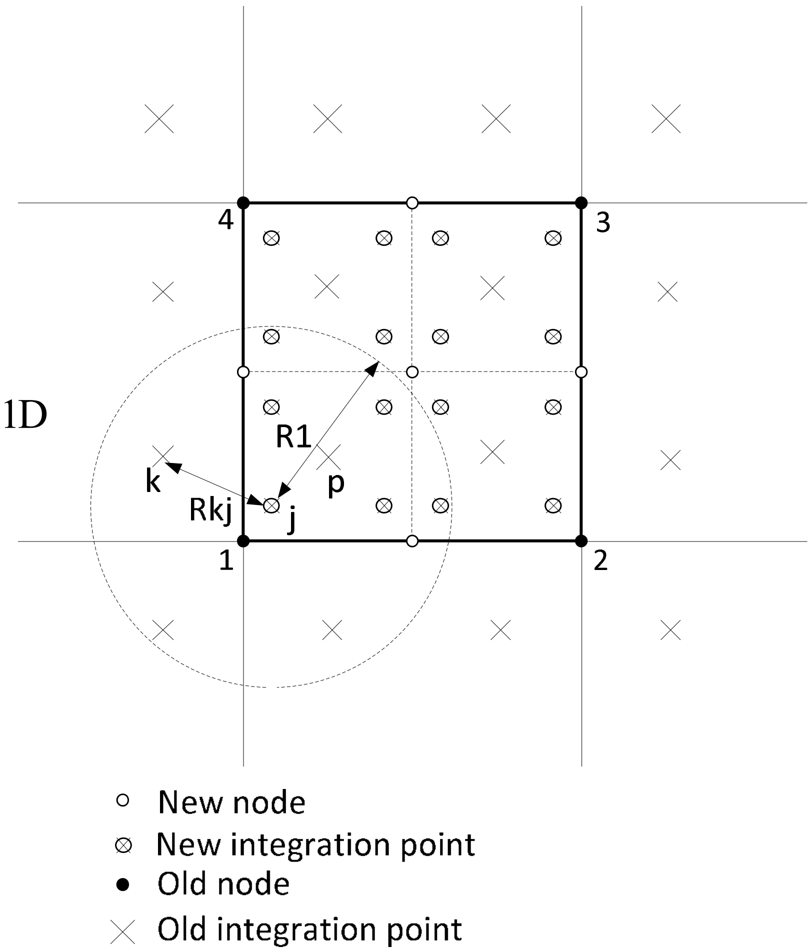
Interpolation of state variables and stress

$$Z_j = \begin{cases} \dfrac{\sum_k \dfrac{Z_k}{R_{kJ}^n} + \dfrac{CZ_p}{R_{pj}^n}}{\sum_k \dfrac{1}{R_{kJ}^n} + \dfrac{C}{R_{pj}^n}} & \text{if } R_{pj} > R_{min} \\ Z_p & \text{if } R_{pj} \le R_{min} \end{cases}$$

With :

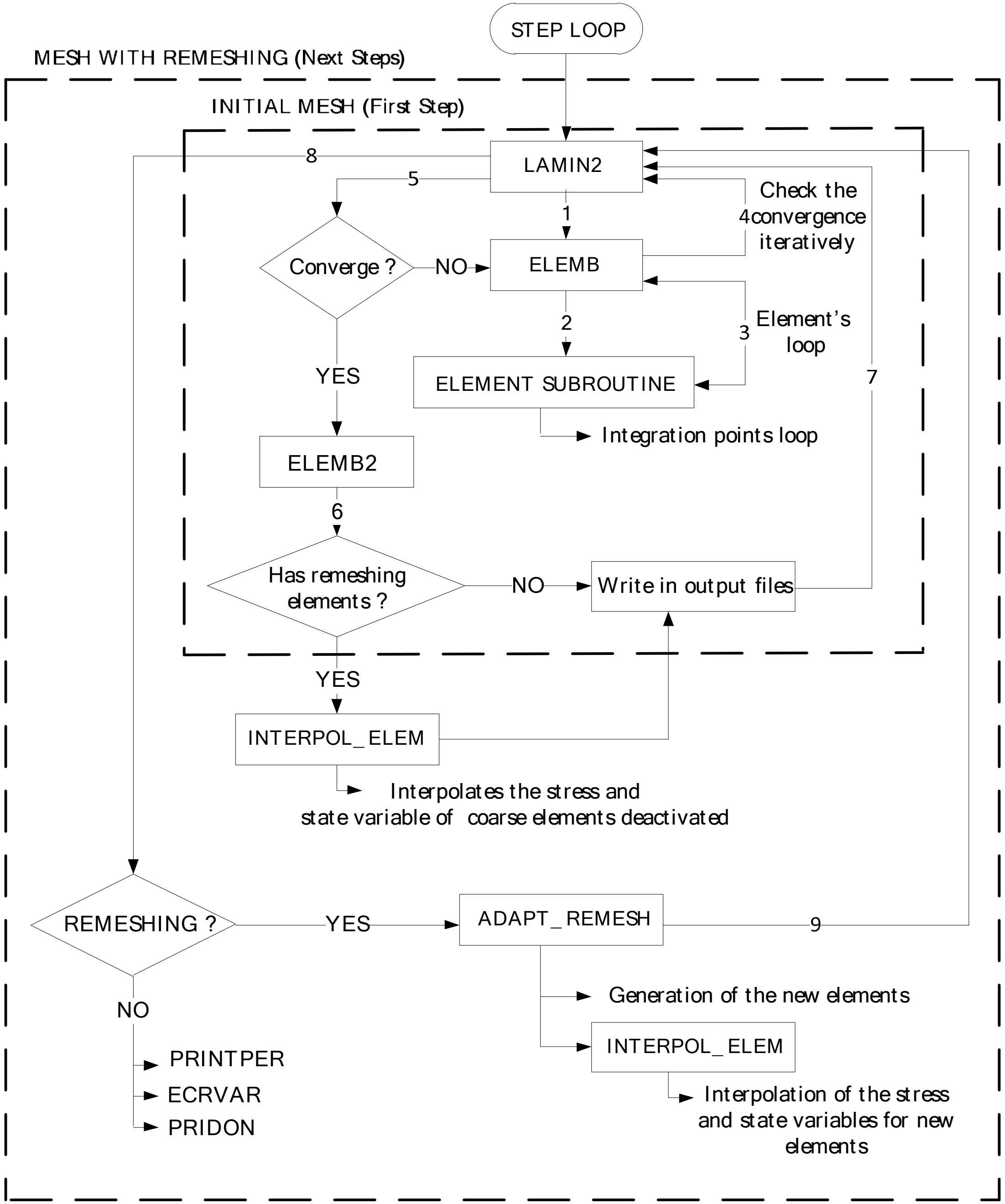
$R_1 = 1.5d$

$R_{min} = 0.0001D$

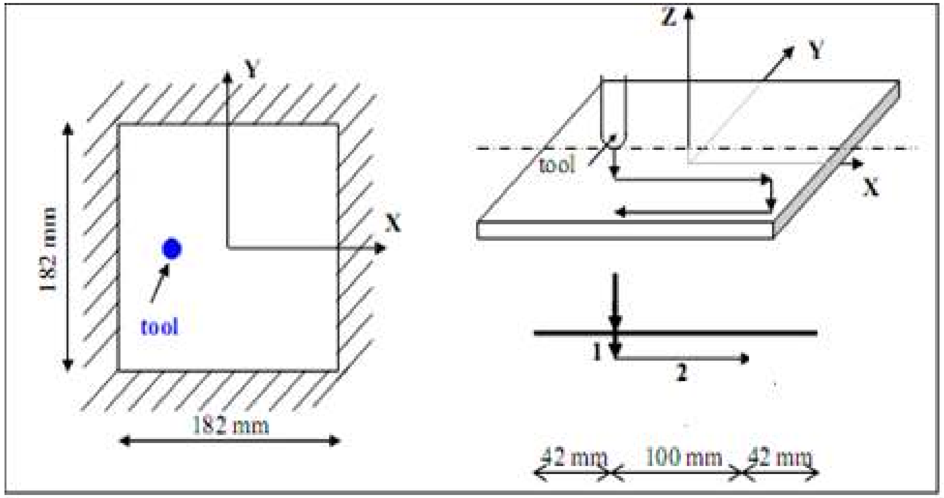


- j: is the index of the new integration point
- k: is the index of the integration point of another element
- p: is the index of the closest integration point
- Zi: can be the stress or variables of state components at the integration point j
- Rkj: distance between the integration point k and j

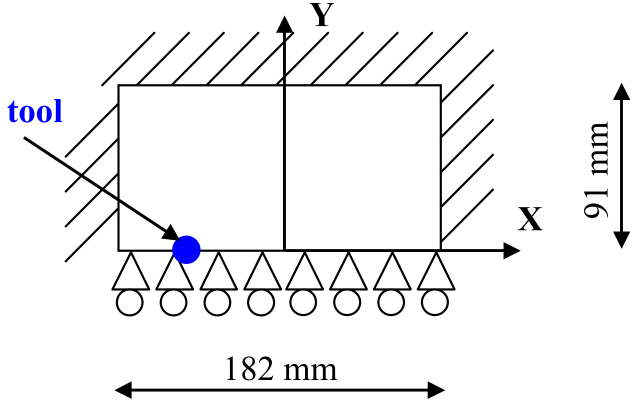
- C: coefficient defined by the user
- n: degree of interpolation
- d: highest diagonal of the element
- D: highest diagonal of the structure


STEP LOOP
MESH WITH REMESHING (Next Steps)
INITIAL MESH (First Step)
LAMIN2
8
5
1
ELEMB
Check the
4 convergence
iteratively
Converge ?
NO
YES
2
3
Element's
loop
7
ELEMENT SUBROUTINE
Integration points loop
ELEMB2
6
Has remeshing
elements ?
NO
Write in output files
YES
INTERPOL_ELEM
Interpolates the stress and
state variable of coarse elements deactivated
REMESHING ?
YES
ADAPT_REMESH
9
Generation of the new elements
INTERPOL_ELEM
Interpolation of the stress
and state variables for new
elements
NO
PRINTPER
ECRVAR
PRIDON

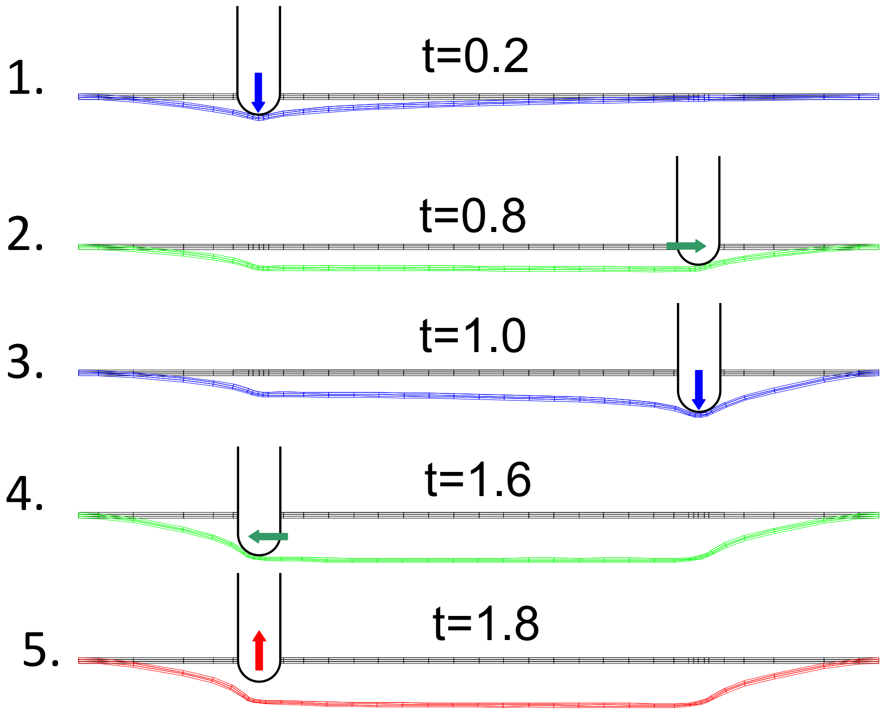
Line test description



- Simple SPIF process:
  - Material : An aluminium alloy AA3003-O
  - thickness: 1.2 mm
  - spherical tool radius: 5 mm
- Boundary conditions:



Line test description



- Composed by 5 steps :
  1. Indentation of 5 mm
  2. Line movement at the same depth along the X axis
  3. Second indention to the depth of 10 mm
  4. Line at the same depth along the X axis
  5. Unloading

GRIDS

Material

- Material : An aluminium alloy AA3003-O

- The constitutive law: Hill (Isotropic behaviour law)

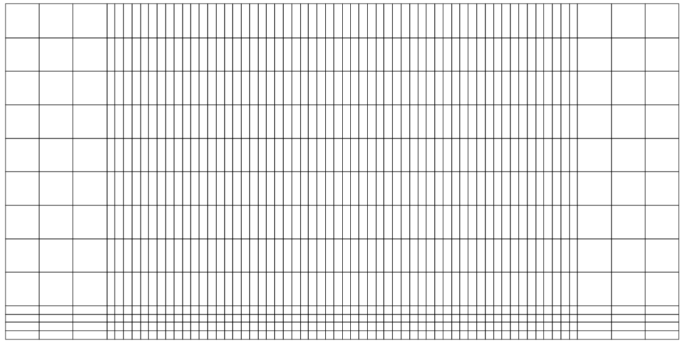
- Parameters:

  Young modulus: $E_1 = E_2 = E_3 = 72600$ MPa
  Poisson ratio: $\nu_1 = \nu_2 = \nu_3 = 0.36$
  Coulomb modulus: $G_1 = G_2 = G_3 = 26691.18$ MPa
  Lankford coefficients defining the yield locus:
  F= G= H= 1.0; N= L= M= 3.0

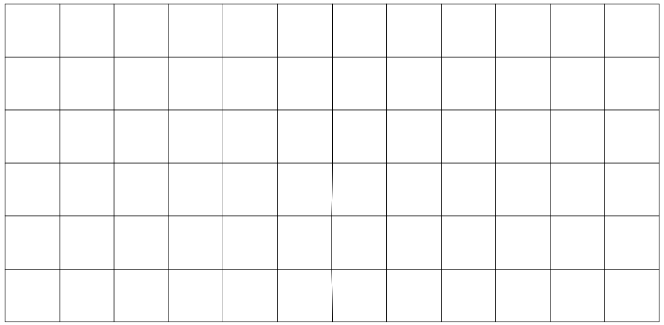
- Hardening Swift law: $\sigma_{eq} = K (\varepsilon_0 + \varepsilon_{pl})n$ with K= 180.0,
  $\varepsilon_0 = 0.00057$,
  n= 0.229

Meshes

- 4 types of Elements :
  - 8 node solid-shell finite element RESS with 3GP
  - Contact element CFI3D with 4GP
  - Shell element COQJ4 with 4GP
  - Contact element CLCOQ
- 2 types of mesh :
  - Coarse with 72x2 elements with one element in thickness direction (RESS+CFI3D)
  - Coarse with 72 elements COQJ4
  - Reference without remeshing 806X2 elements (RESS+CFI3D)



Reference: Fine mesh for without remeshing simulation



Coarse Mesh for with adaptive remeshing simulation

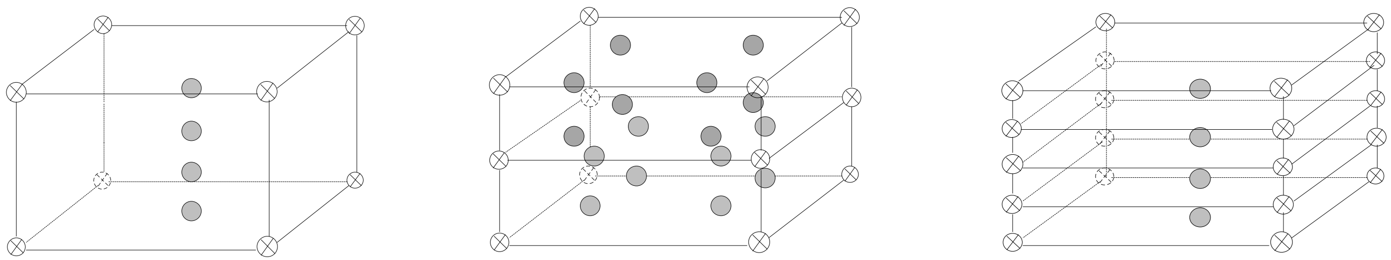
RESS (Reduced Enhanced Solid Shell)*

- Solid-Shell Element specially design to use in metal forming applications
- Implemented in LAGAMINE code
- Integration scheme (a) advantages:

  - Reduced integration in plane

  - Arbitrary number of integration points in one single layer in thickness direction

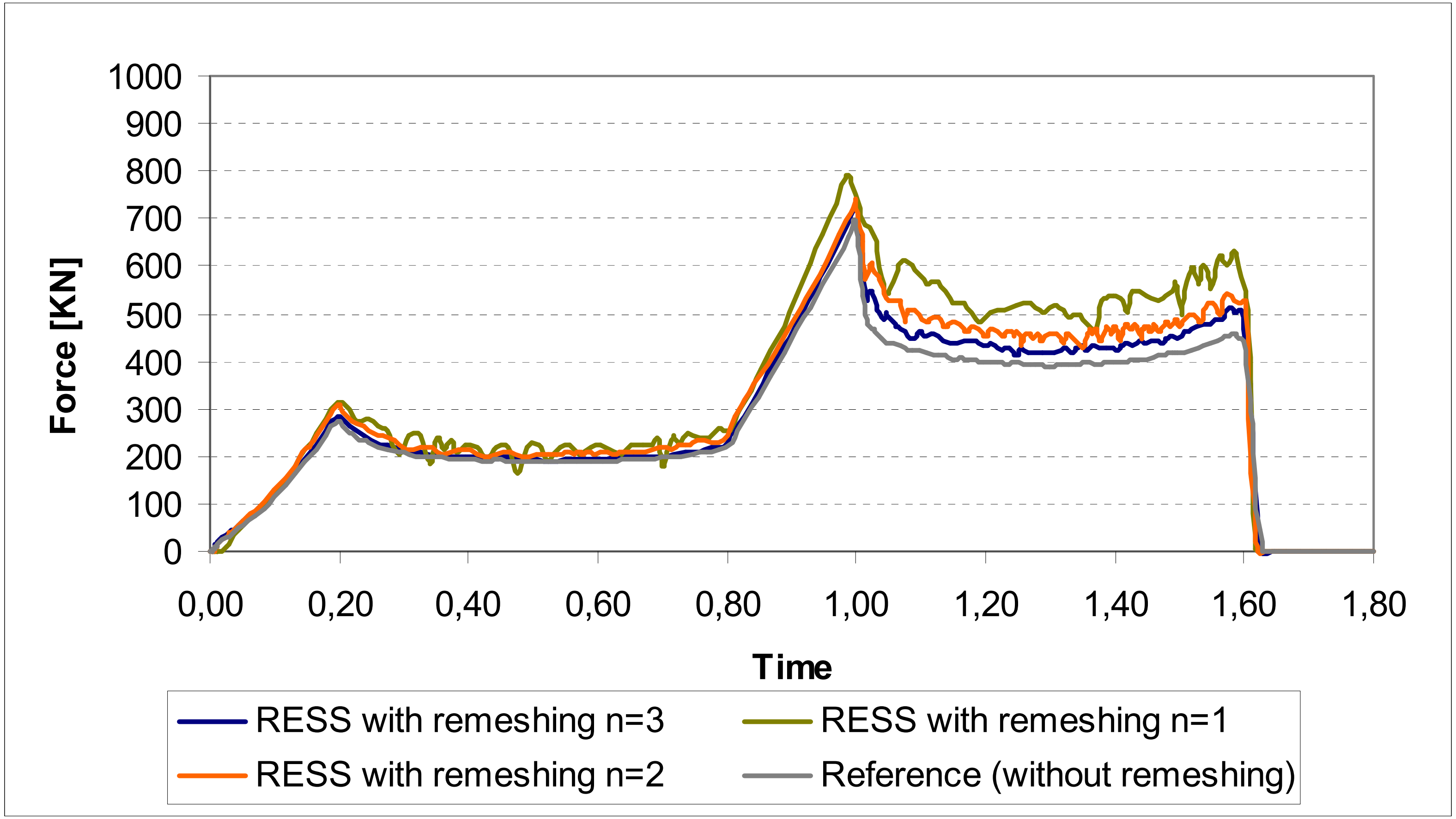
- Combination between displacement strain and enhanced strain components interpolated by enhanced matrix with only enhanced mode
- Stabilization technique



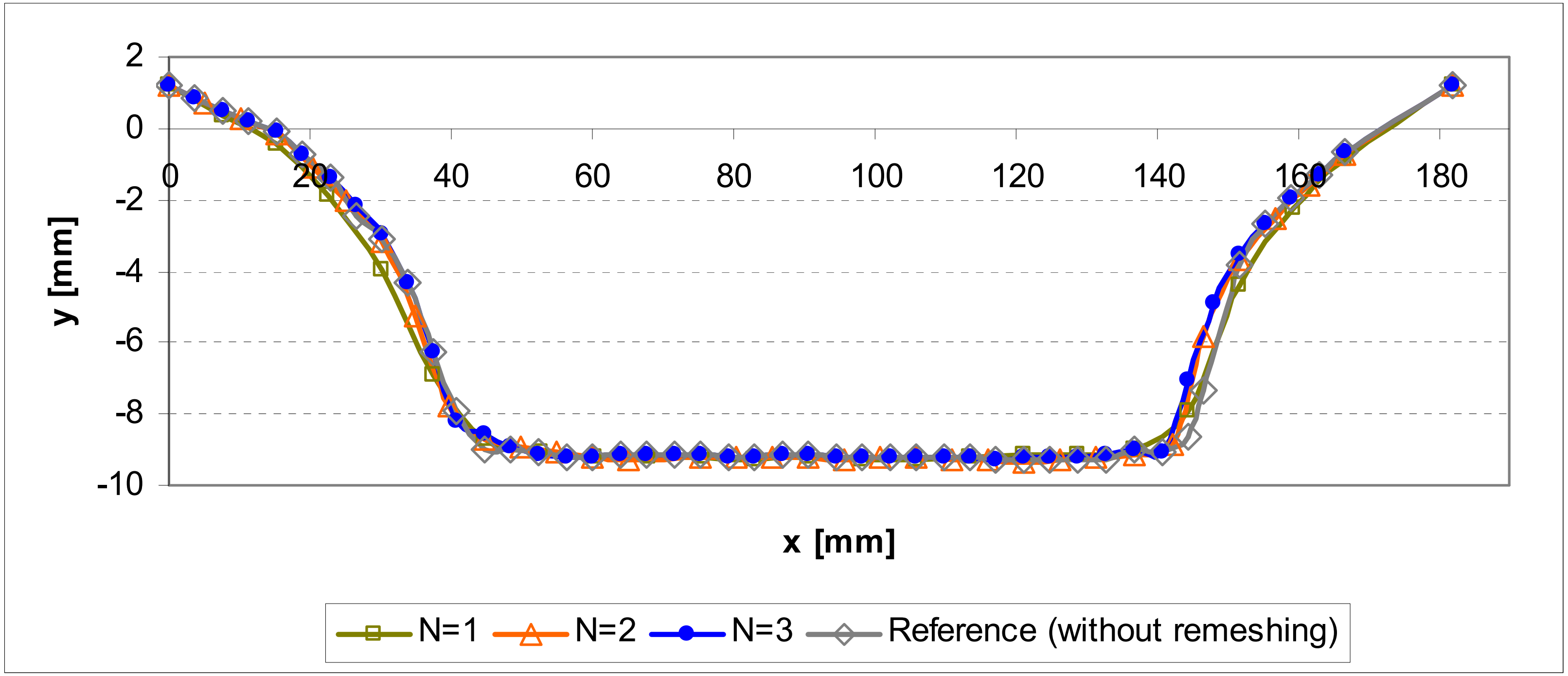
(a) (b) (c)

*See in Alves de Sousa R.J. *et al.* (2007), I. J. P., Vol. 23, pp. 490-515.

Evolution of tool during the line test with RESS

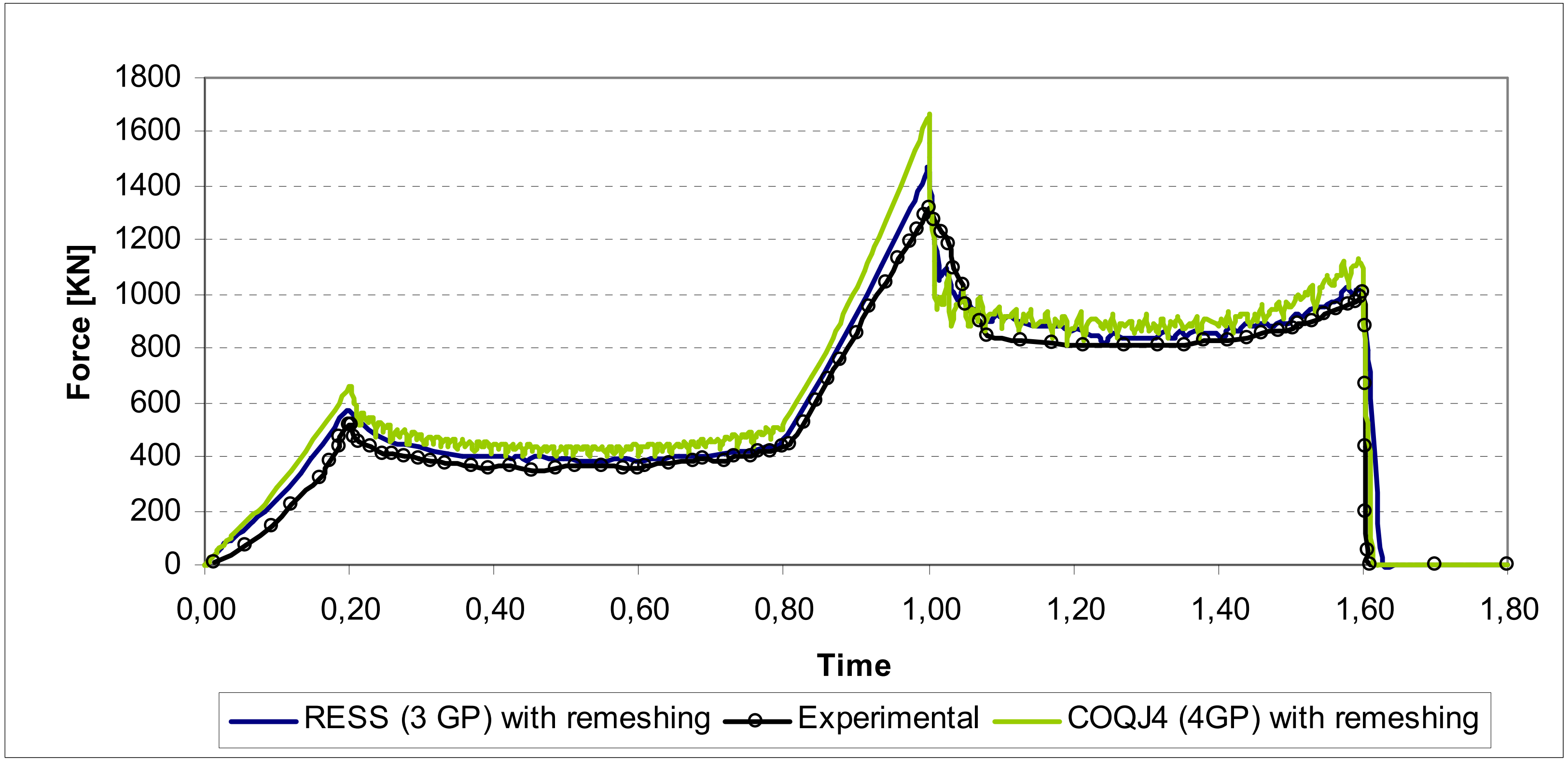


Shape in a cross-section



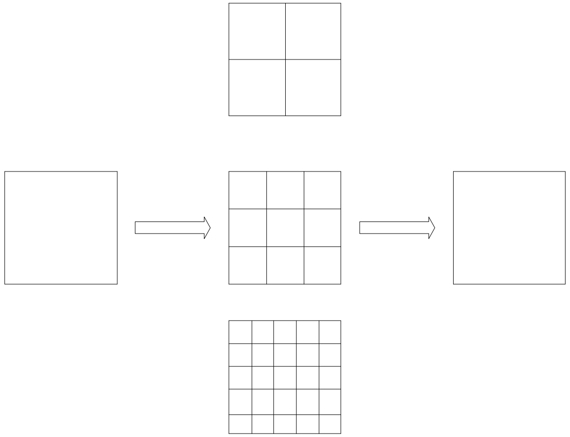
Numerical validation

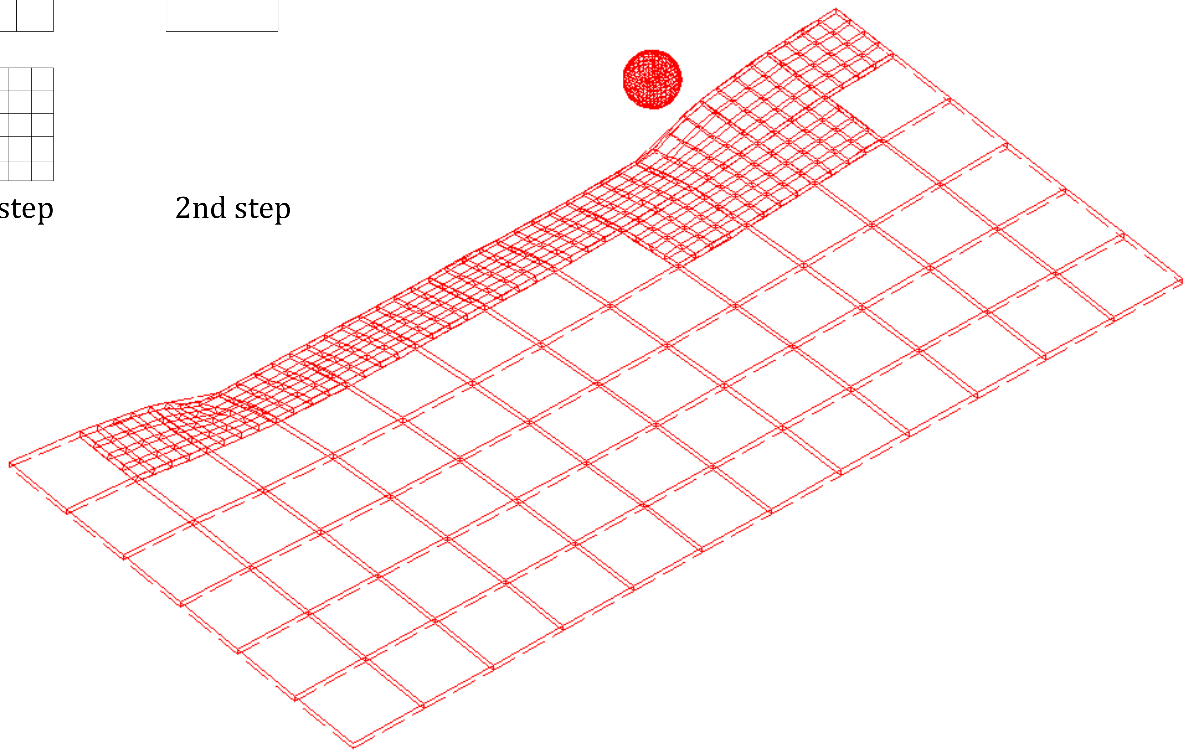
– Comparison between experimental and numerical results:



– Divisions per edge (n): 3 nodes (one element is divided by 16 new elements)

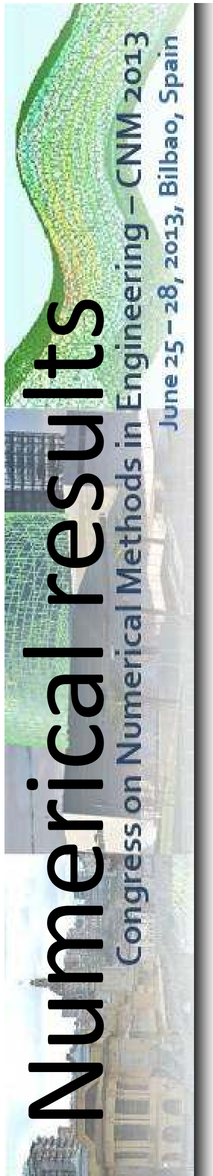
Final mesh using adaptive remeshing





Comparisons of time performance

Reference	COQJ4 4GP	RESS 3GP	Classic 8GP
CPU time	5m 36s	18m 54s	17m 9s
Nº of Iterations	2859	1194	1107
Nº of steps	579	167	203
N=1			
CPU time	1m 18s	2m 9s	1m 31s
Nº of Iterations	4164	907	725
Nº of steps	1041	153	158
N=2			
CPU time	1m 13s	5m 4s	4m 7s
Nº of Iterations	2792	1059	680
Nº of steps	698	197	133
N=3			
CPU time	1m 23s	6m 39s	5m 29s
Nº of Iterations	2076	1095	946
Nº of steps	519	163	170



GRIDS

Statistics results

• Total CPU time spent at the elements level:

	COQJ4	RESS
Element scheme	100%	89%
Contact elements	0%	11%
Constitutive law	24%	44%

• The CPU time spent with constitutive law corresponds to a 24% of the total time spent in the shell elements

• Using RESS the CPU time spent with constitutive law corresponds to a 44% of the total CPU time spent in the elements scheme

Final Considerations

- The results presented were expected due to the less number of elements and consequently less integration points using shell finite element.

- The major interest of the present work is the 3D analysis of single point incremental forming process.

- The current work in progress is the application of adaptive remeshing using RESS finite element in an efficient and accurate simulation framework of SPIF processes for general 3D analysis.

- The near future work will be the use of unstructured meshes using the adaptive remeshing technique.

GRIDS

Thank you for your attention

GRIDS