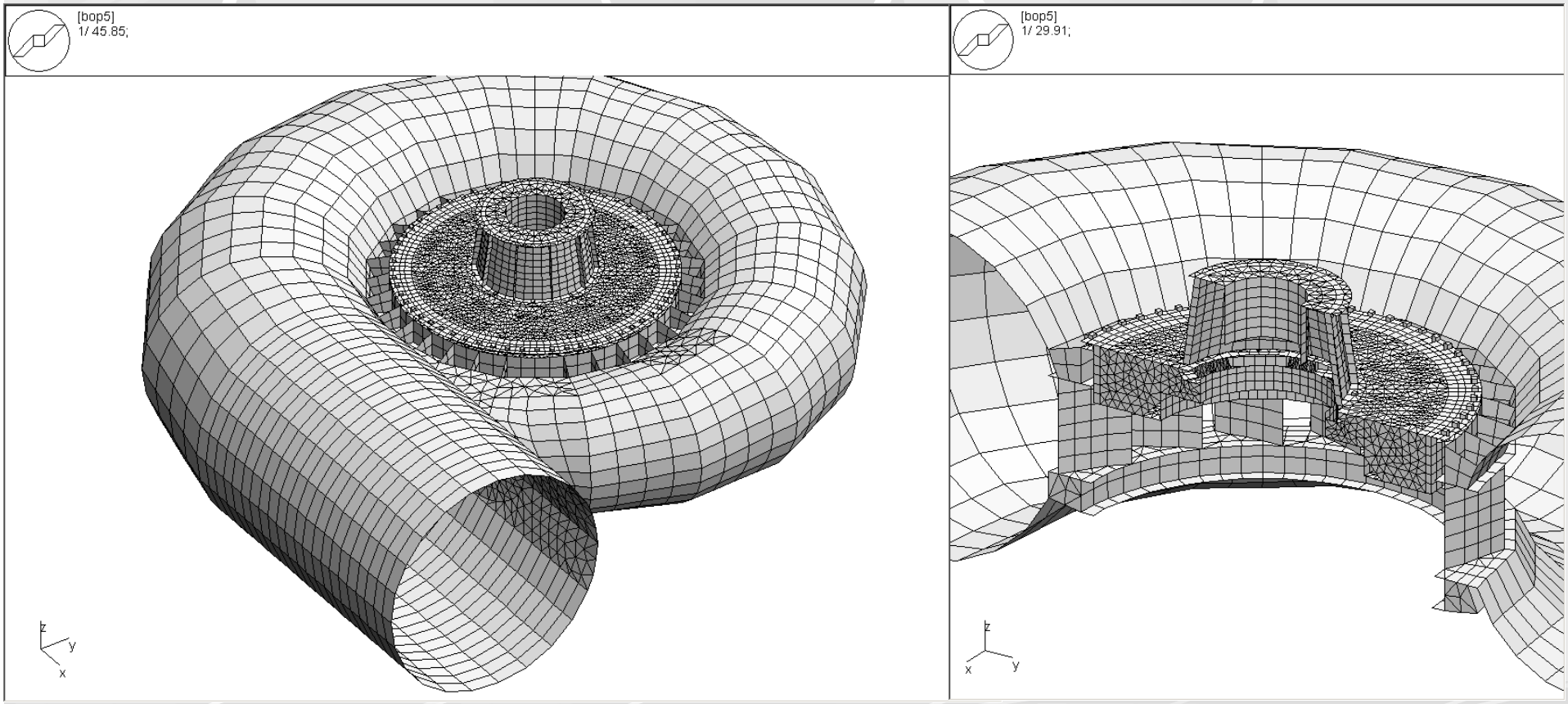
# ICAB
# FEA applied to hydraulic structures

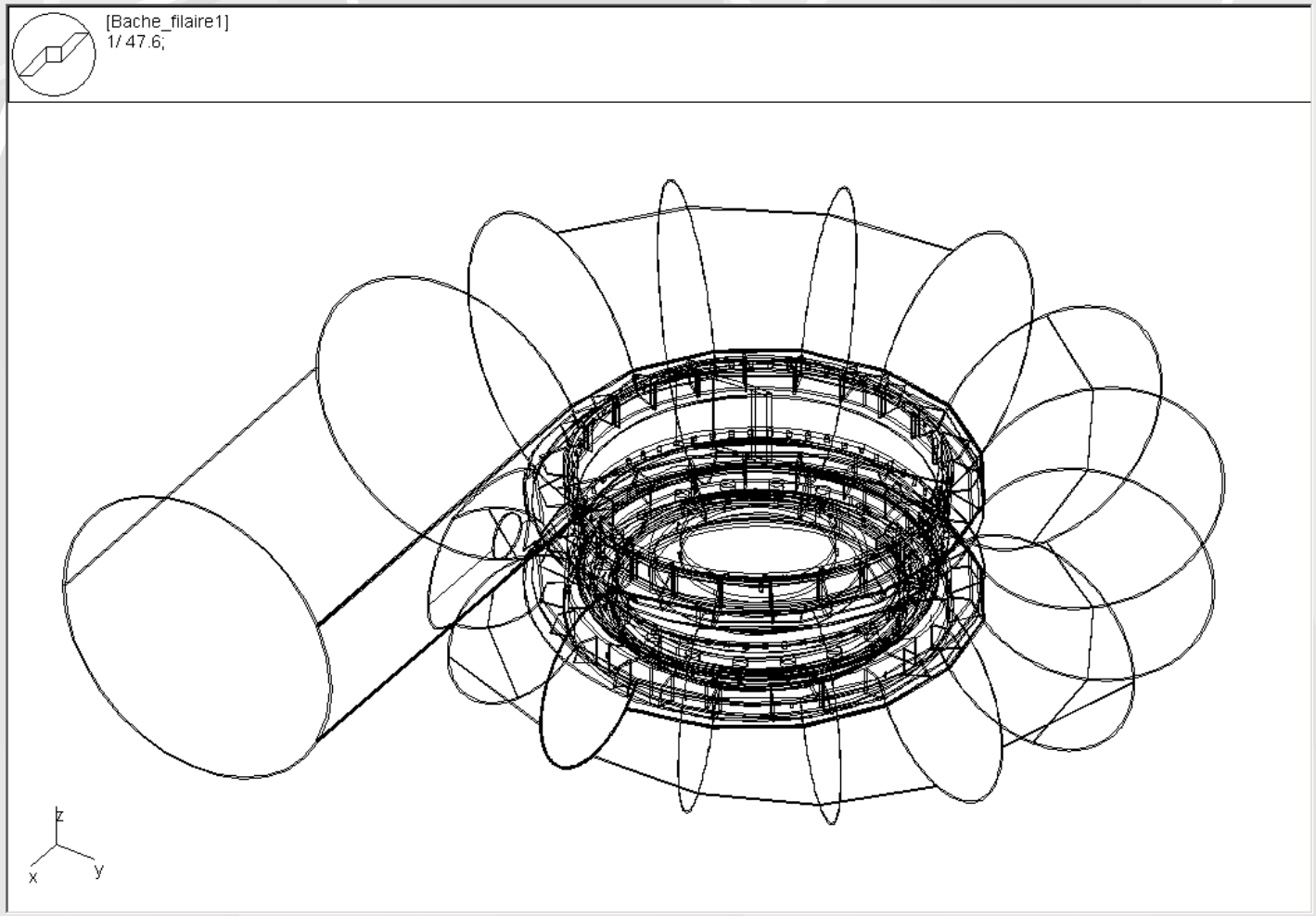
# Design and FE Analysis

- Graphical data capture interface :
  - General view:



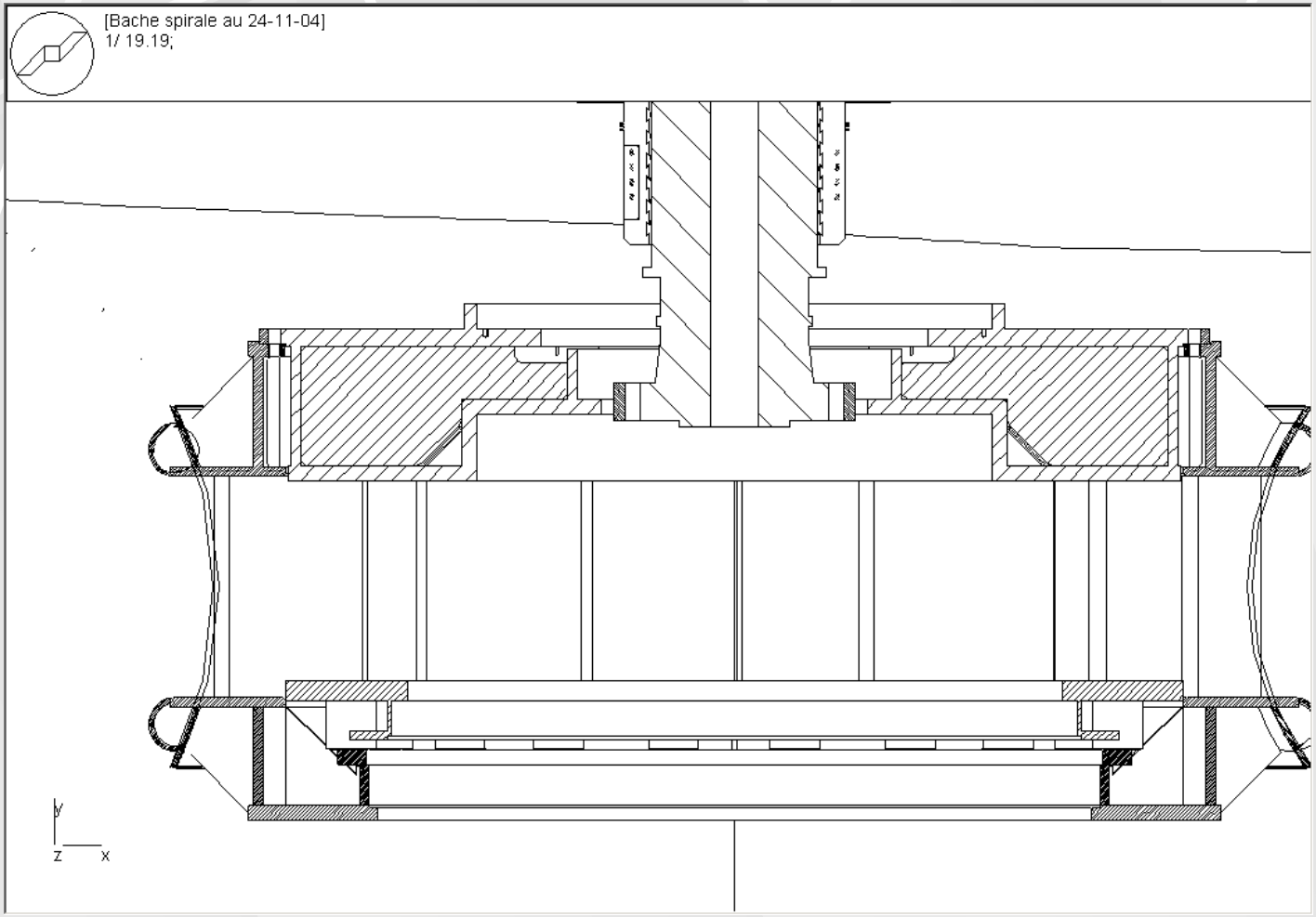
# CAD data input

- Import IGES (or DXF AutoCAD) file



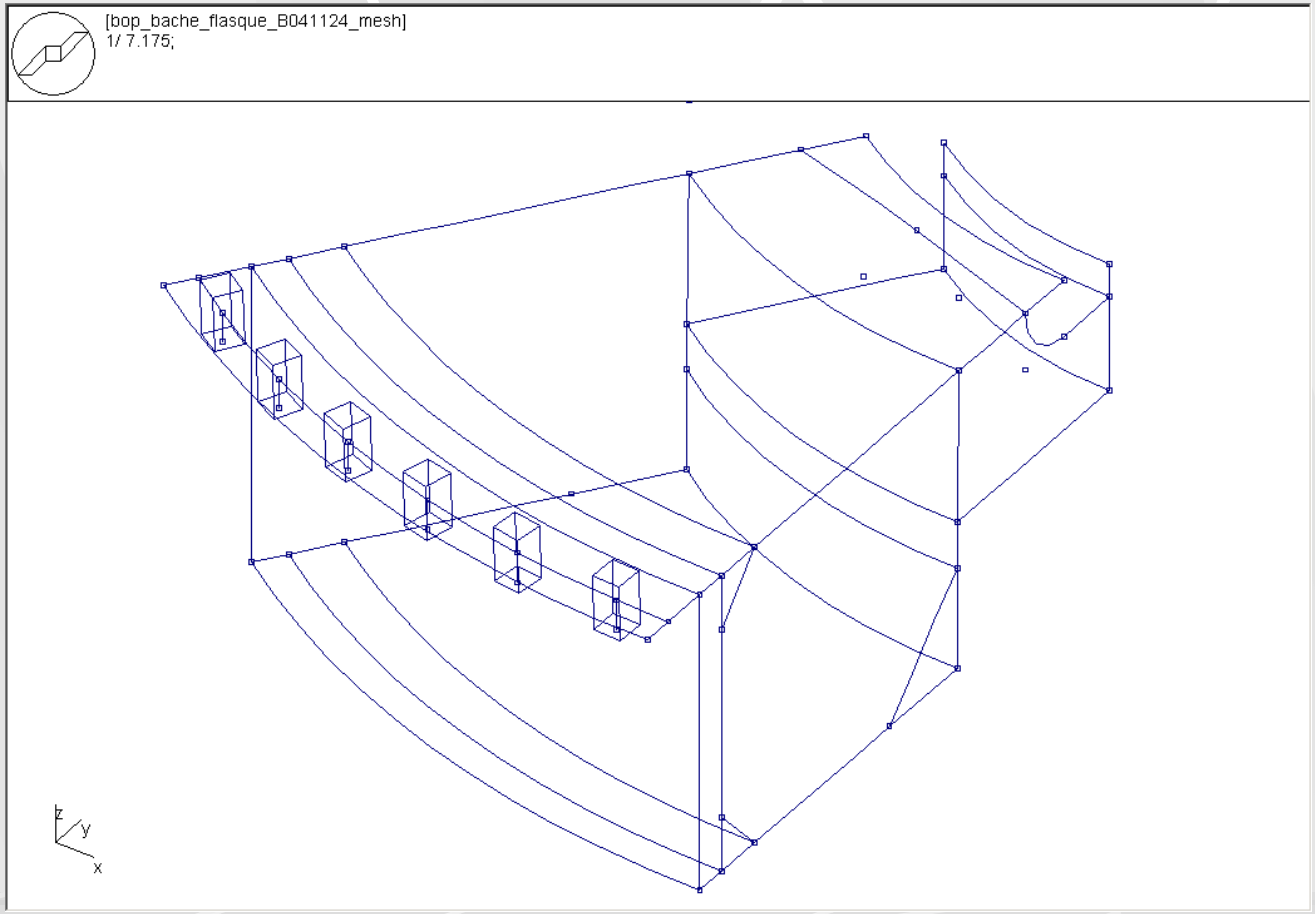
# CAD detail

- Detail of the import



# Meshing

- Meshing :
  - Preliminary mesh



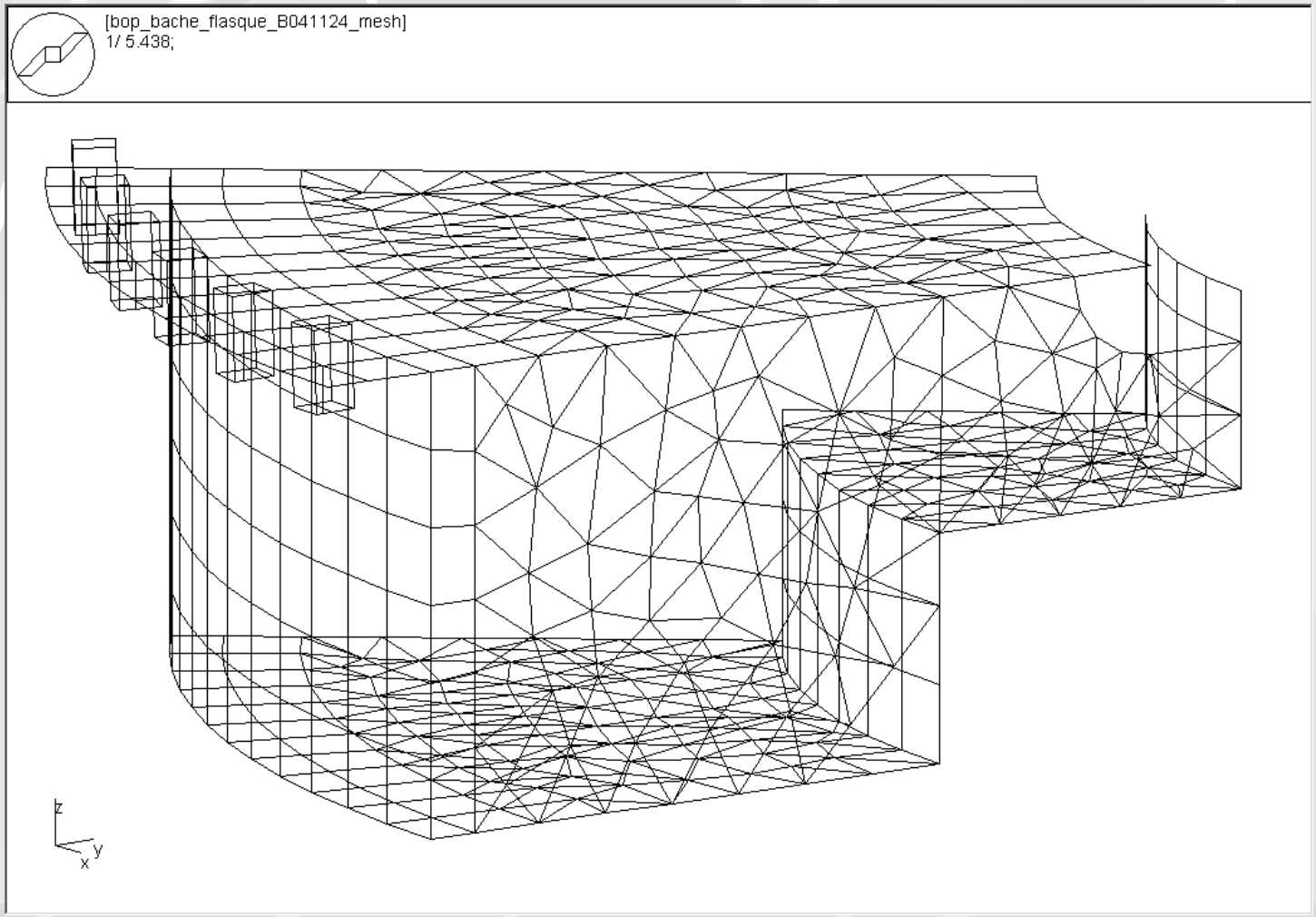


<< home

next >

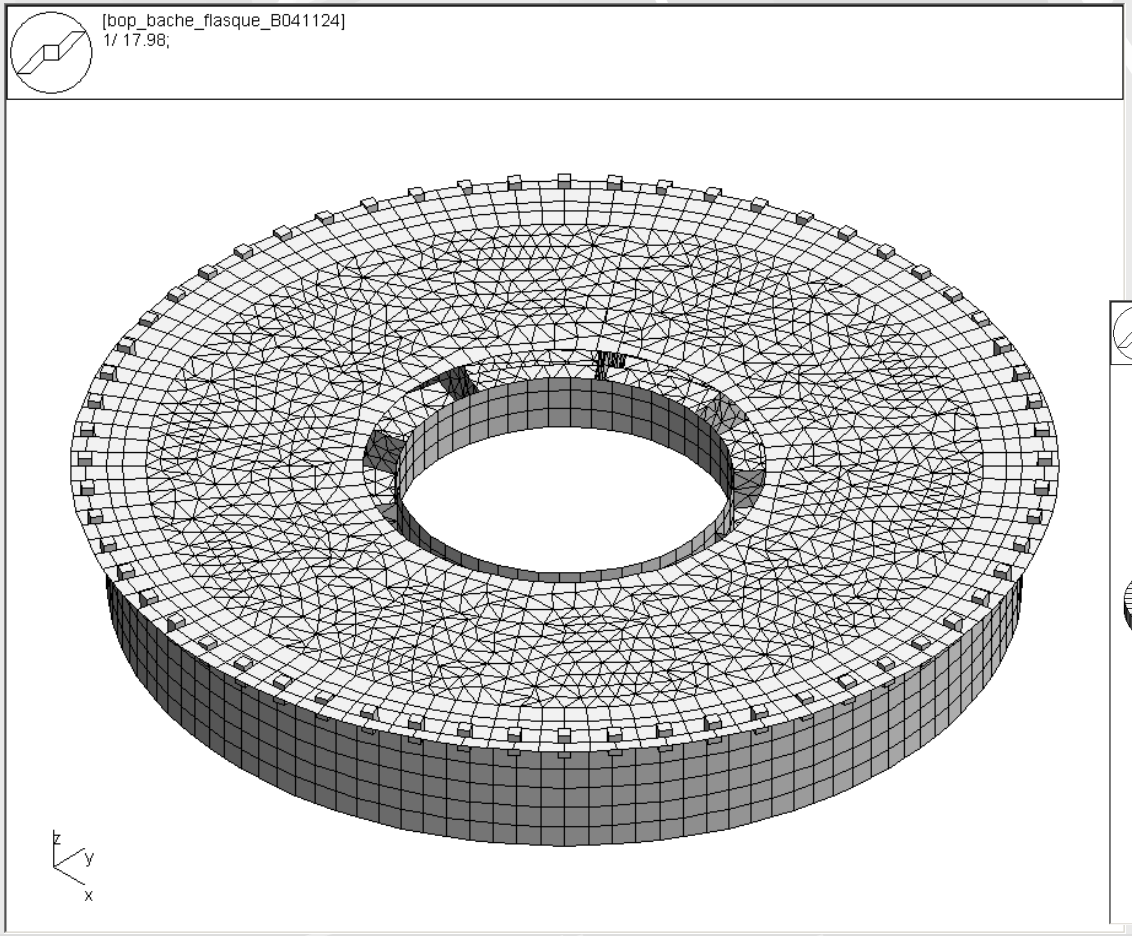
# Finite element Meshing

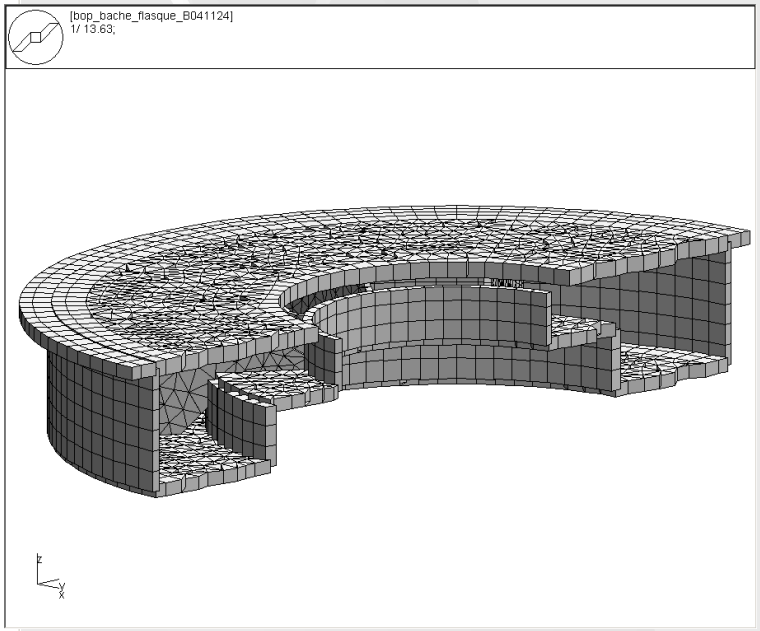
- Creation of the shell elements



# Model completion

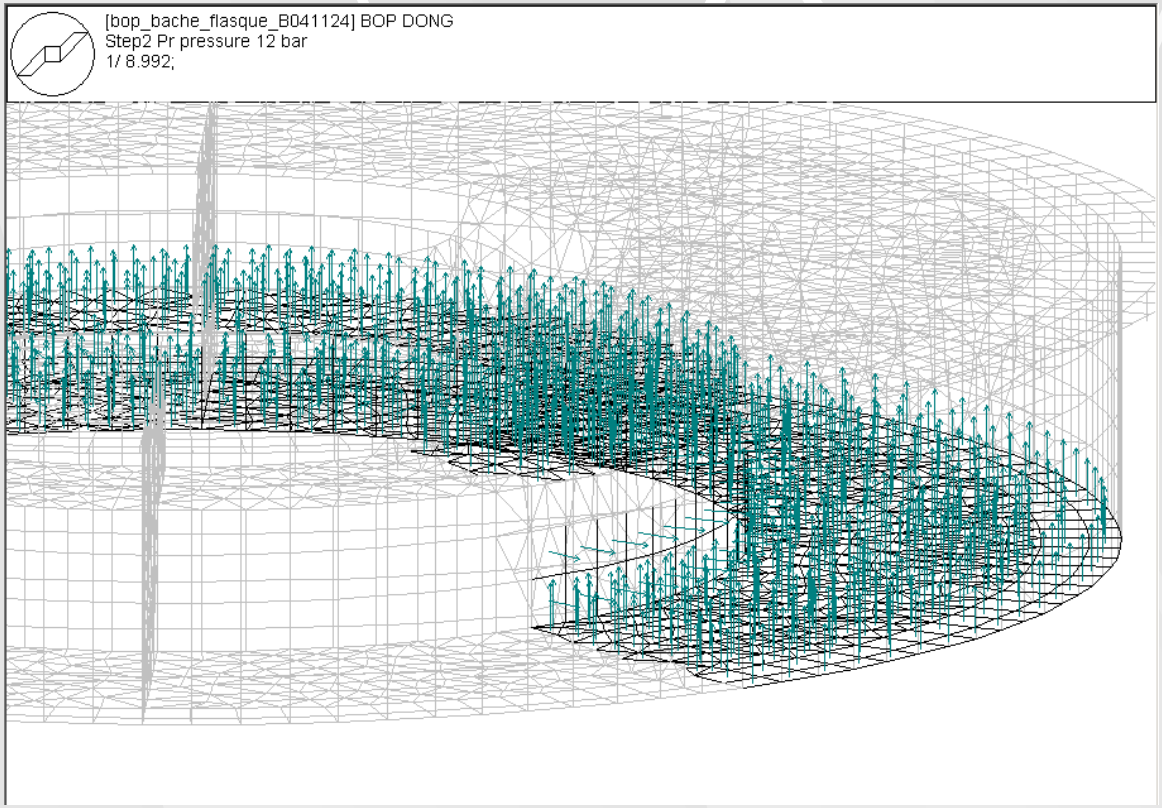
- Completion of the meshing by rotation



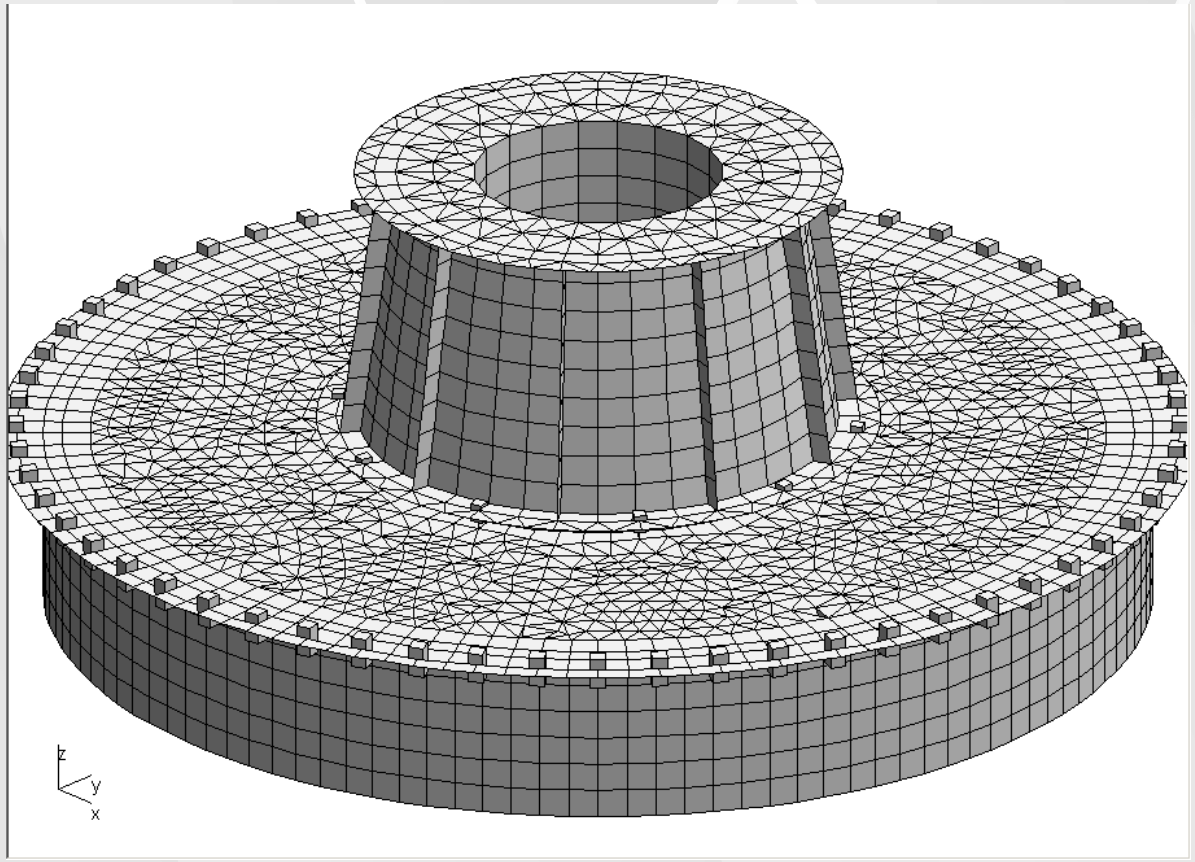
# Pressure load

- Applying of boundary conditions :
  - Pressure = 12 bars :



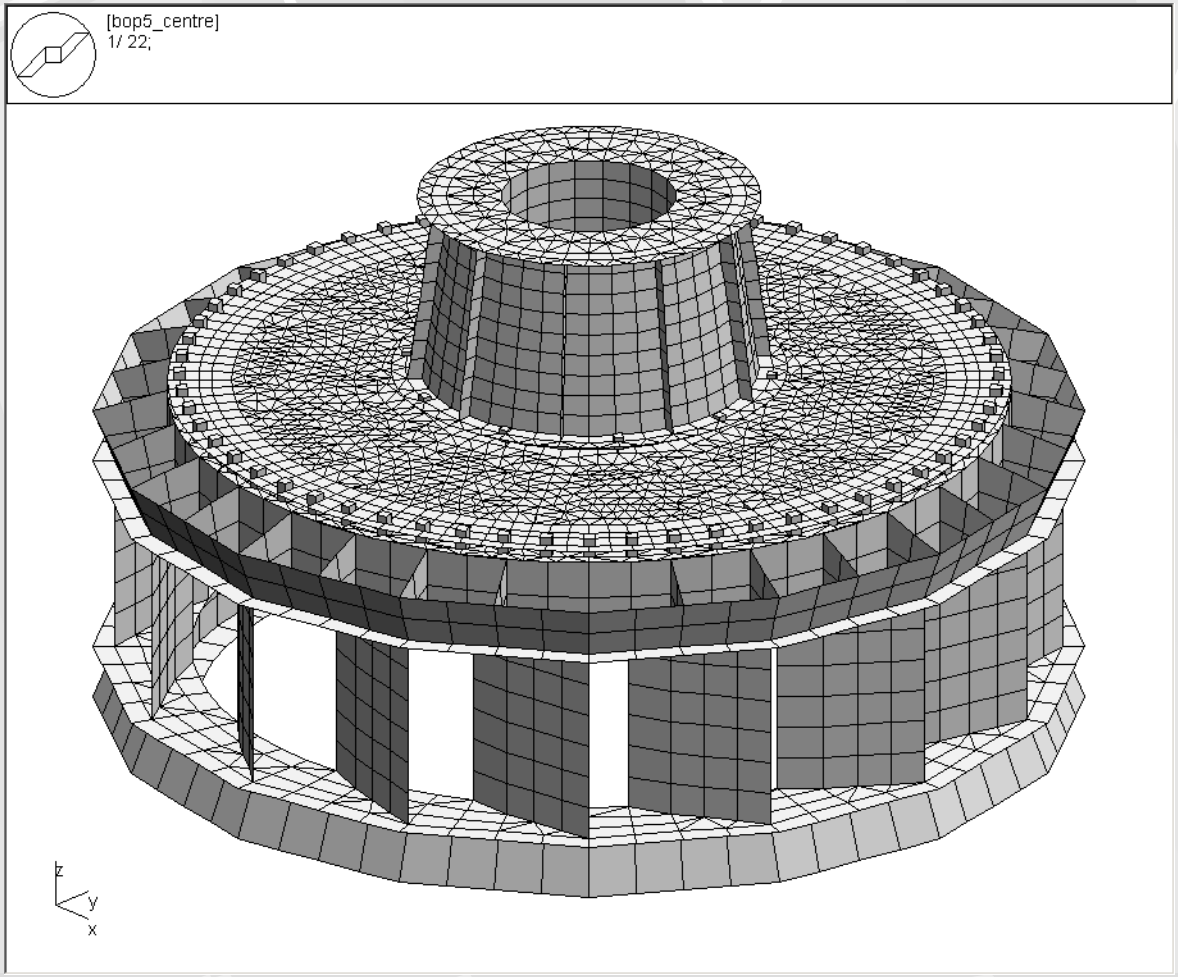
# Assemble parts

- Assembly of the other parts :
  - + cover with bearing



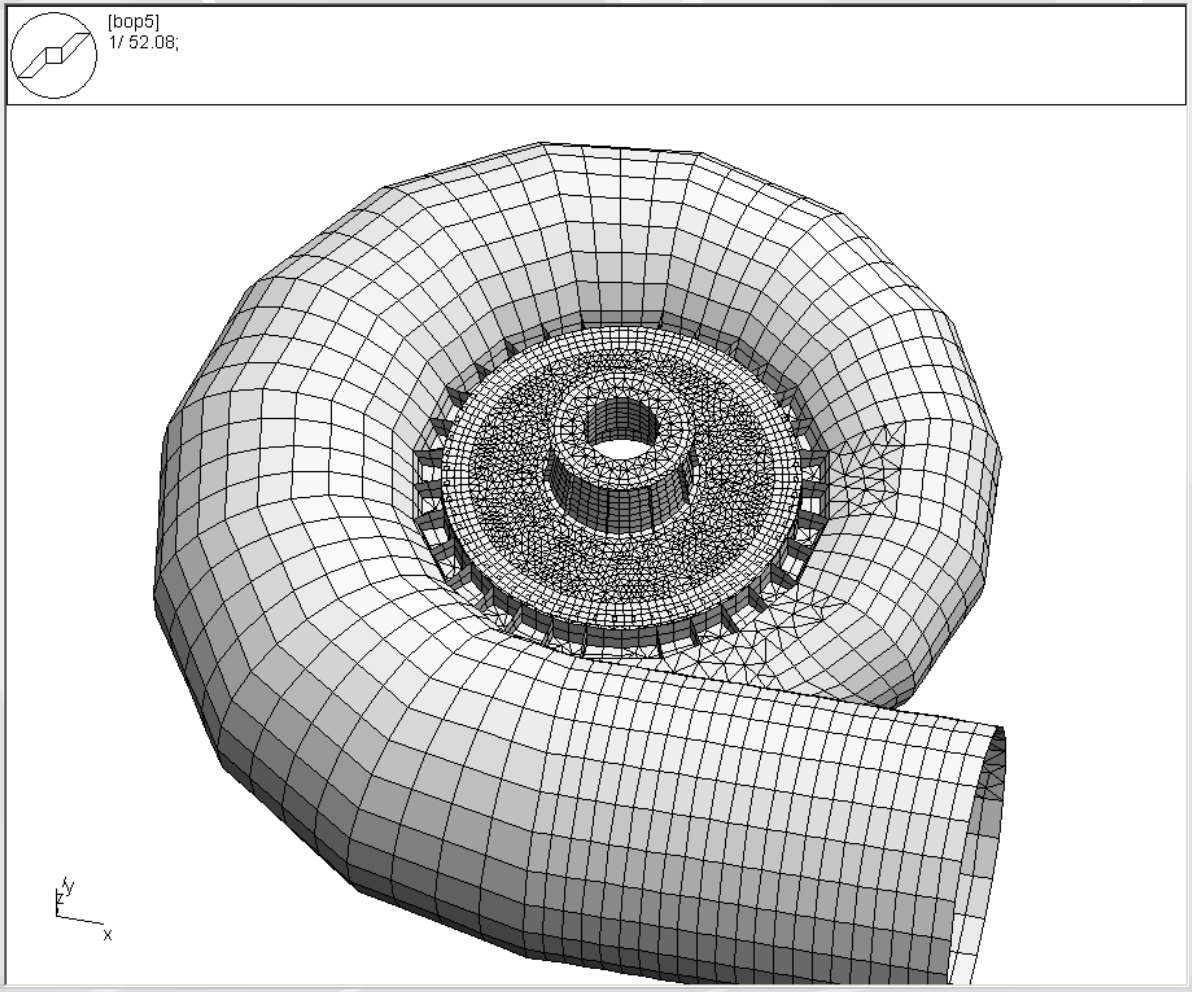
# Assemble parts (2)

- + support of the case



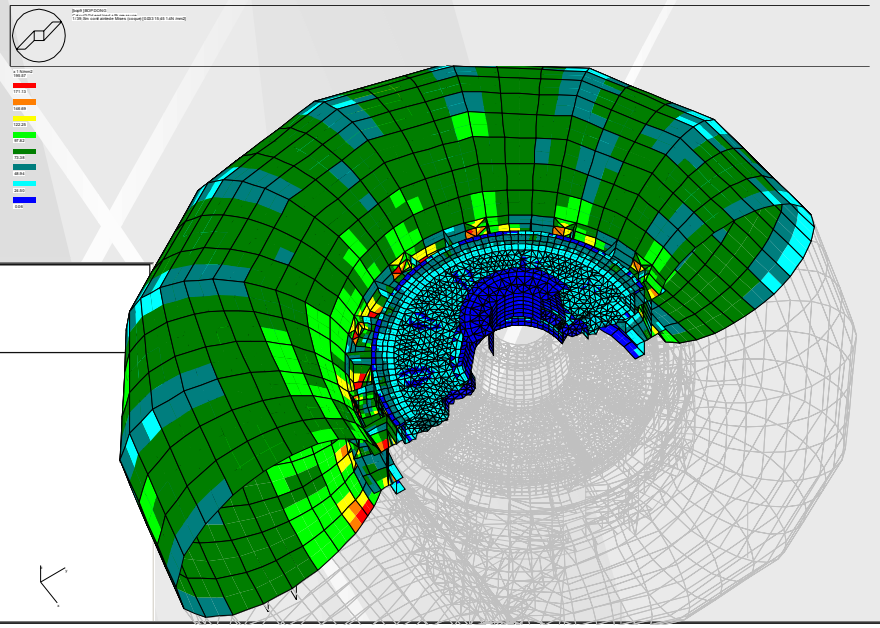
# Assemble parts (3)

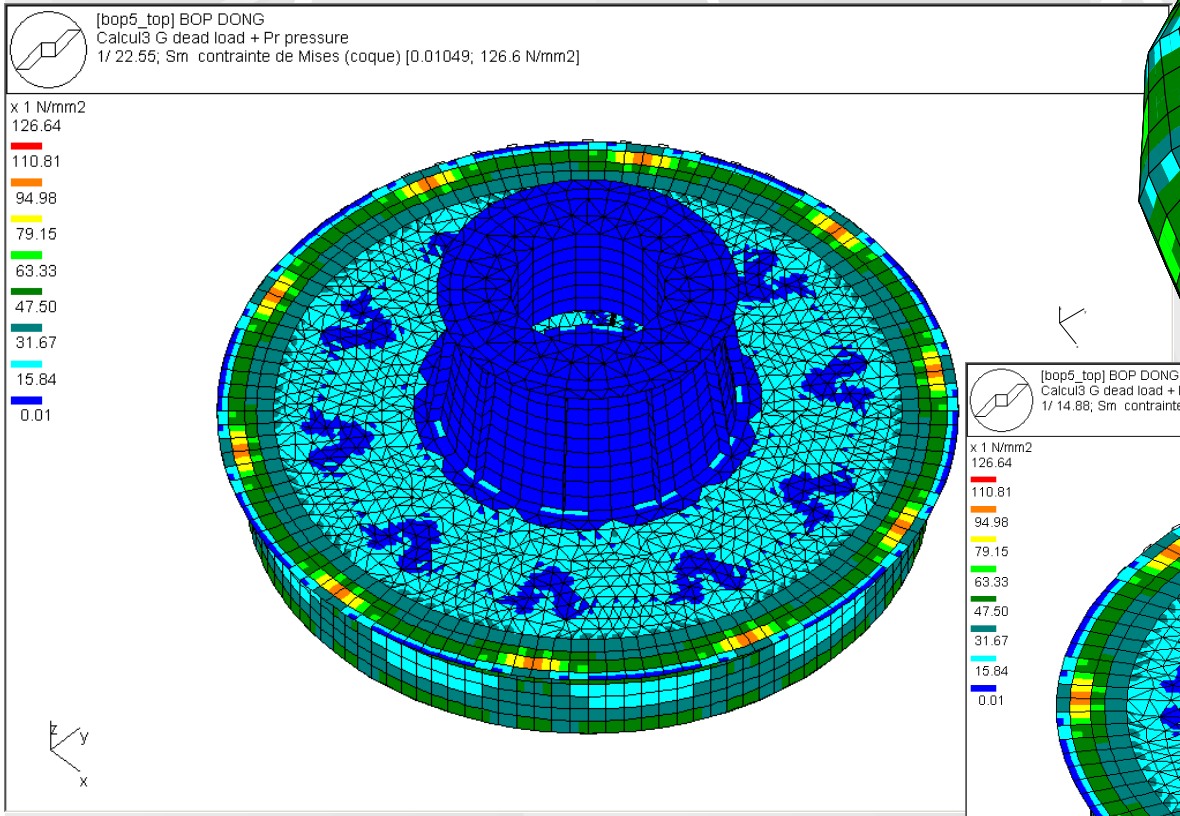
- The full structure

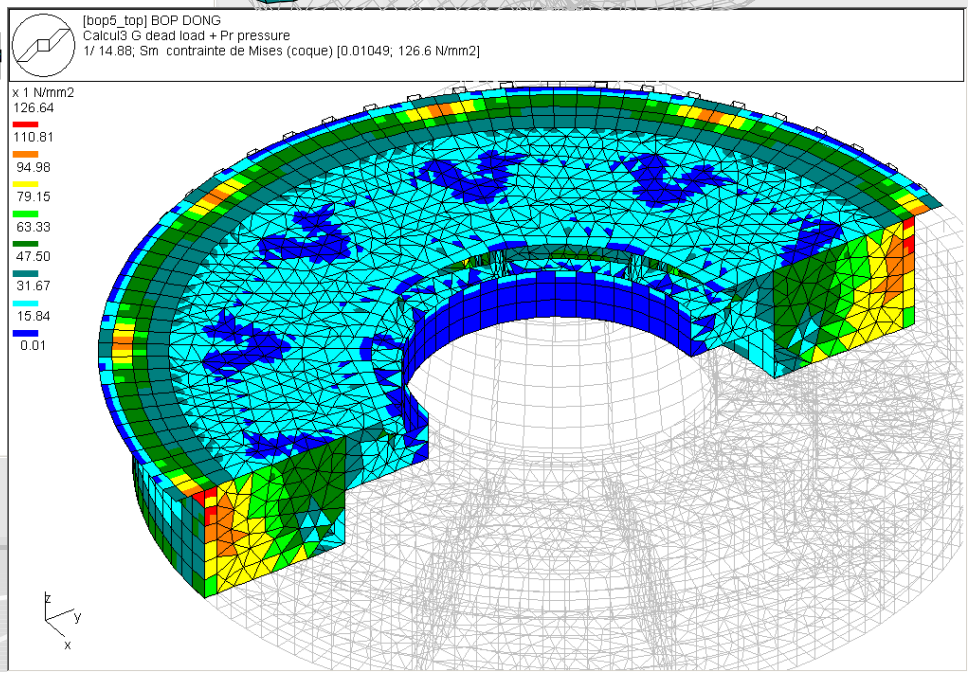


# Results - stresses

- Visual output :
  - Stresses





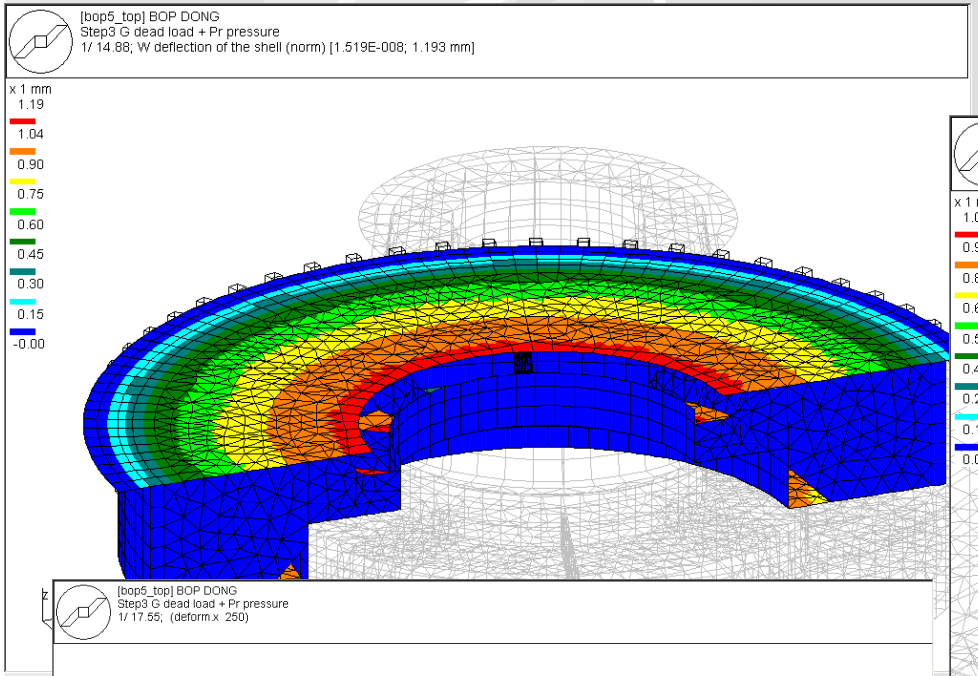


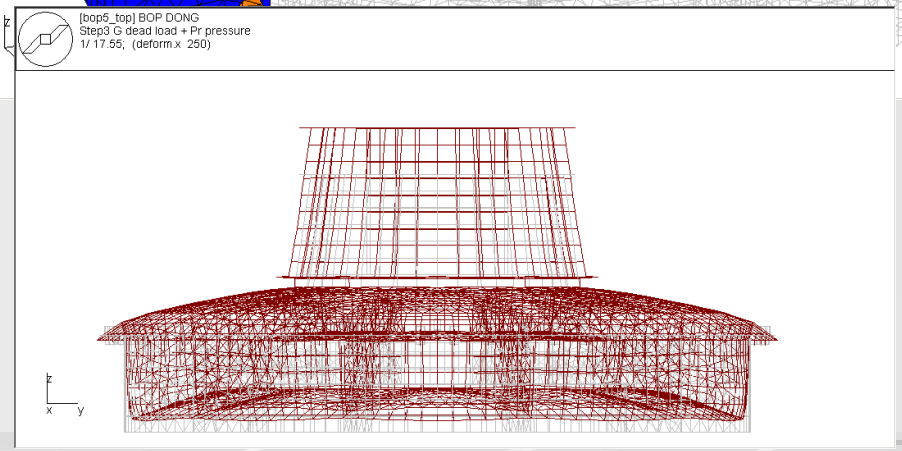
<< home

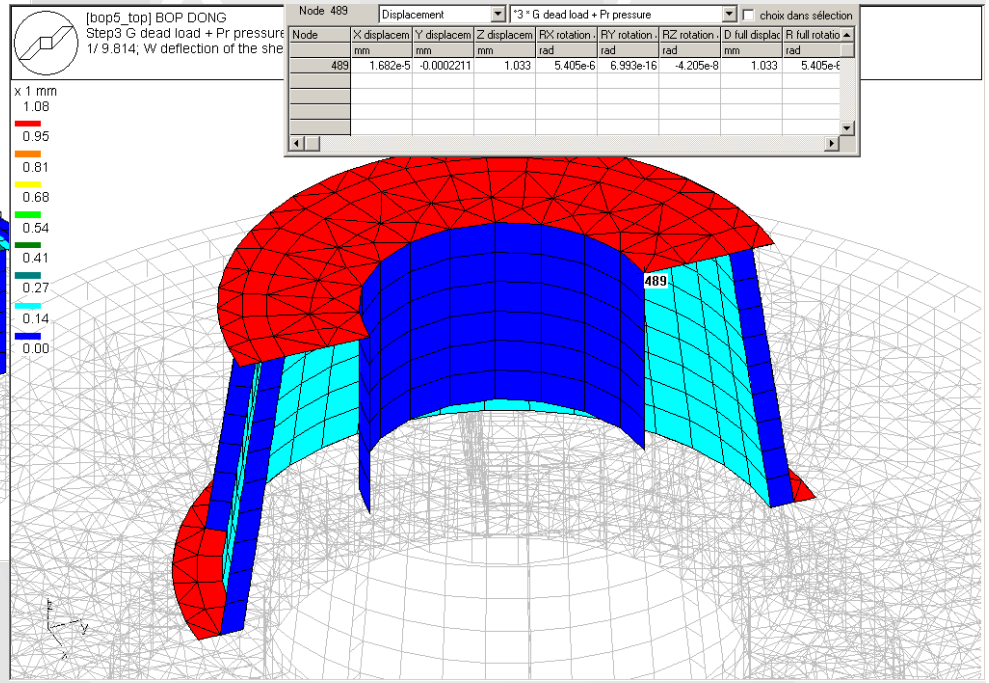
next >

# Results -displacement

- Deflection





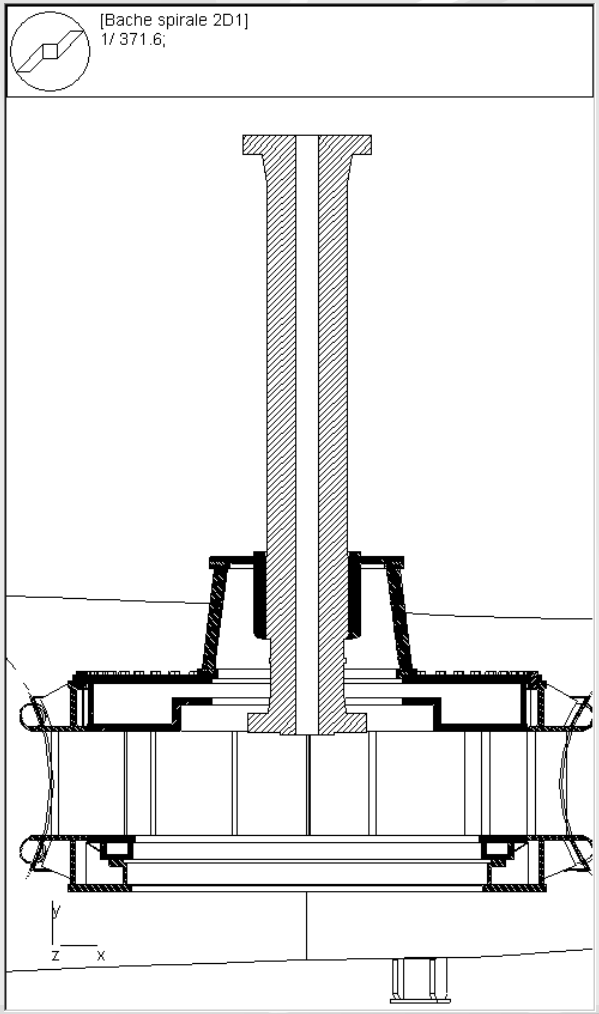


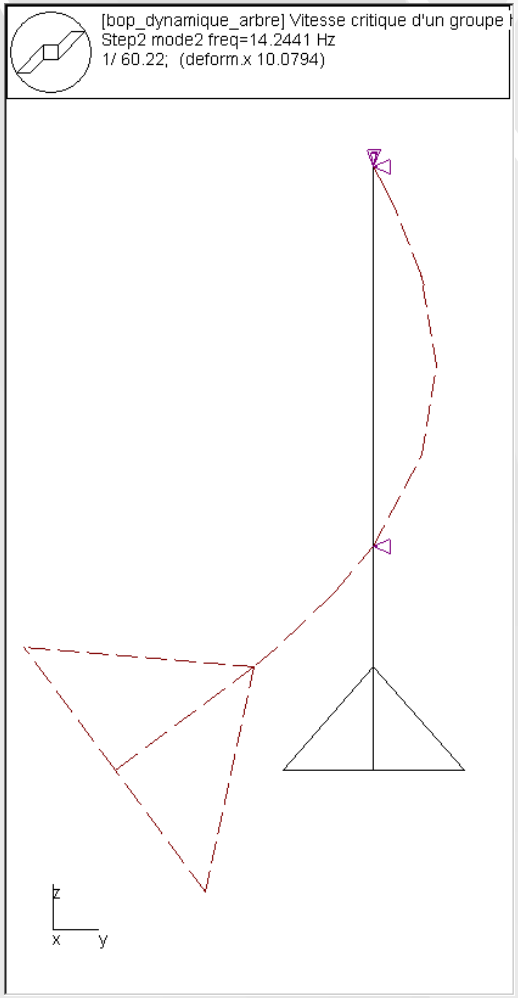

<< home

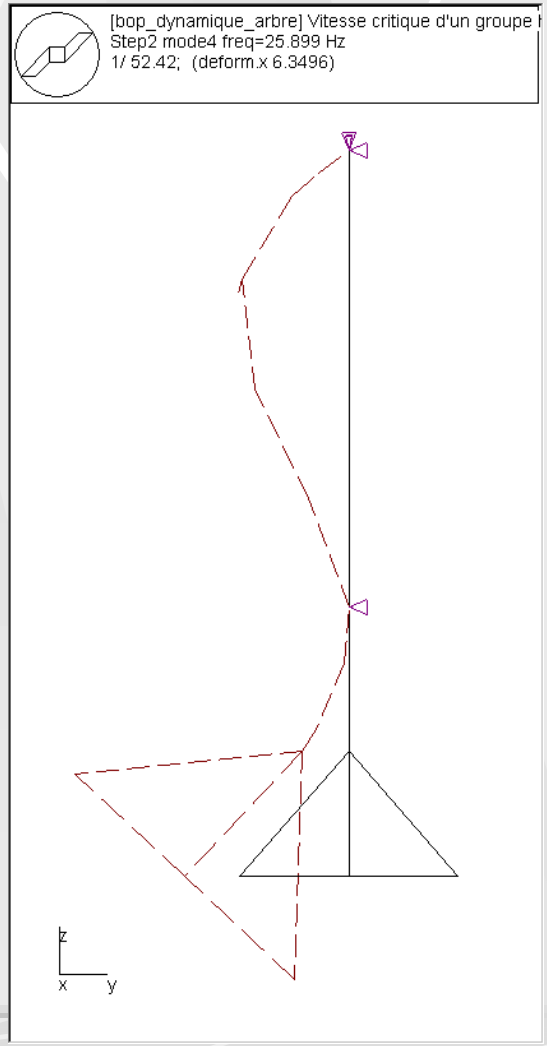
next >

# Dynamic analysis

- Critical rotating speed of the shaft

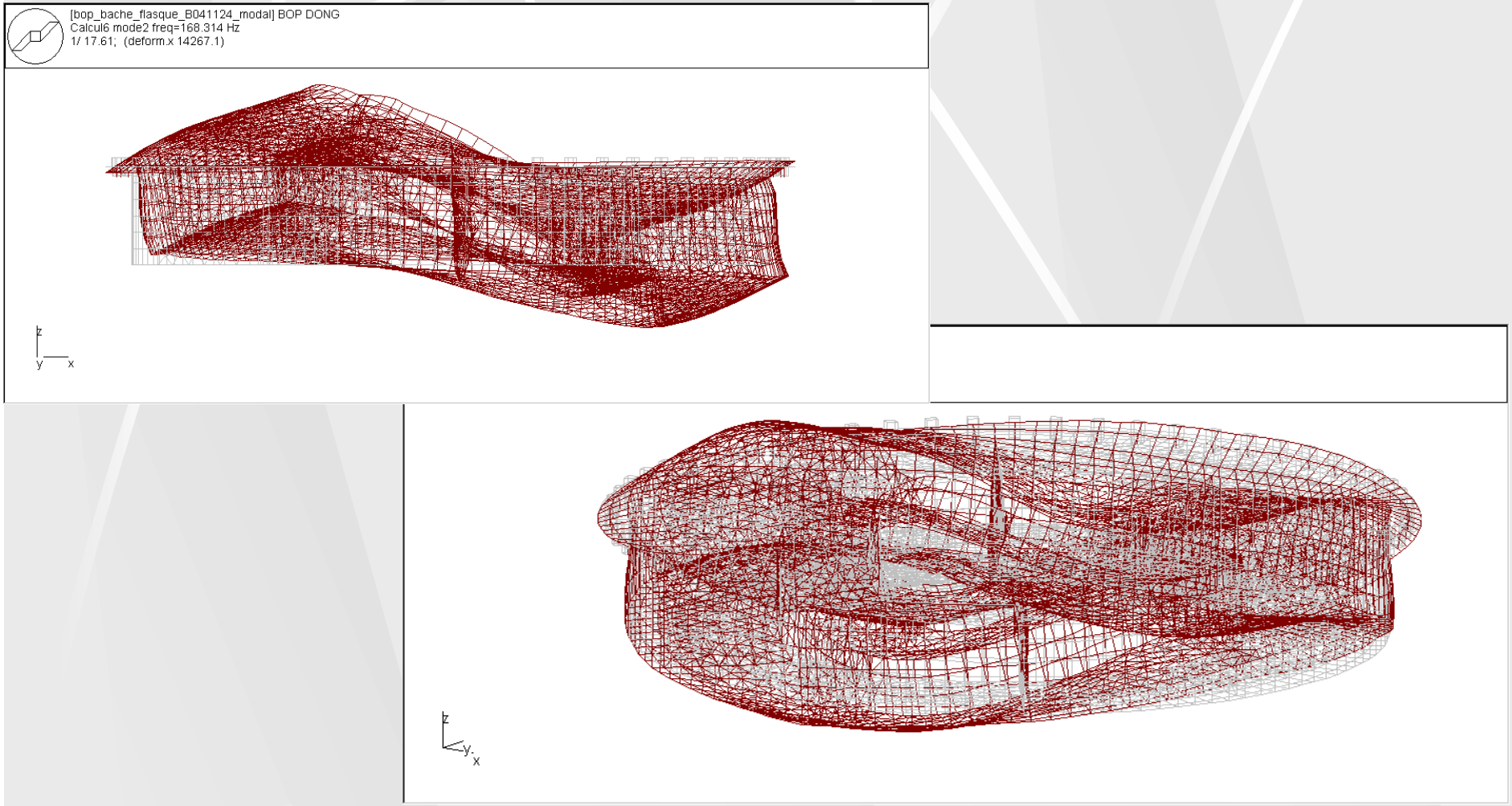


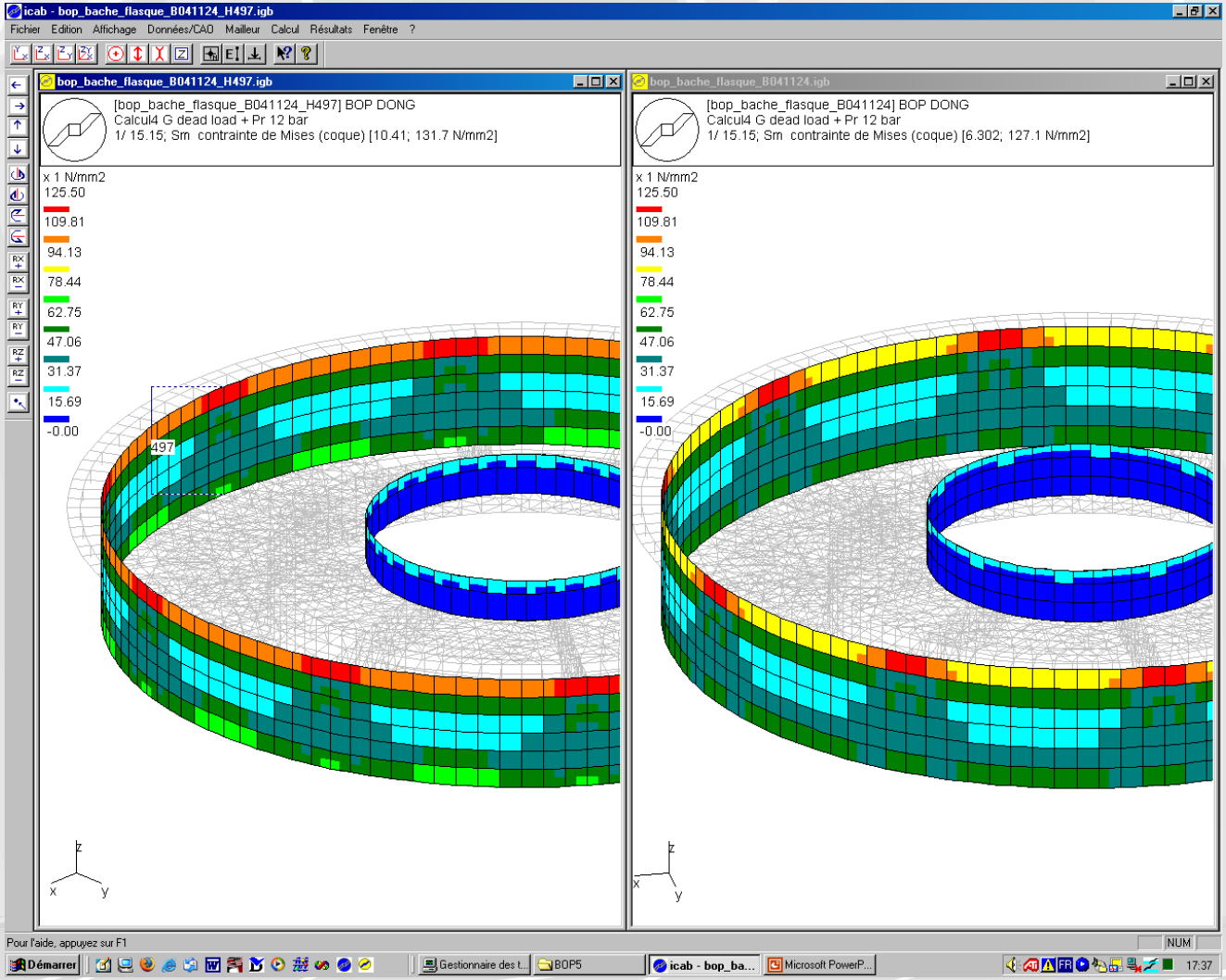
# Vibration analysis

- modal analysis to extract eigen modes:



# Parametric study

- +10% increase of stiffener height reduces stresses (-6% - 20%)



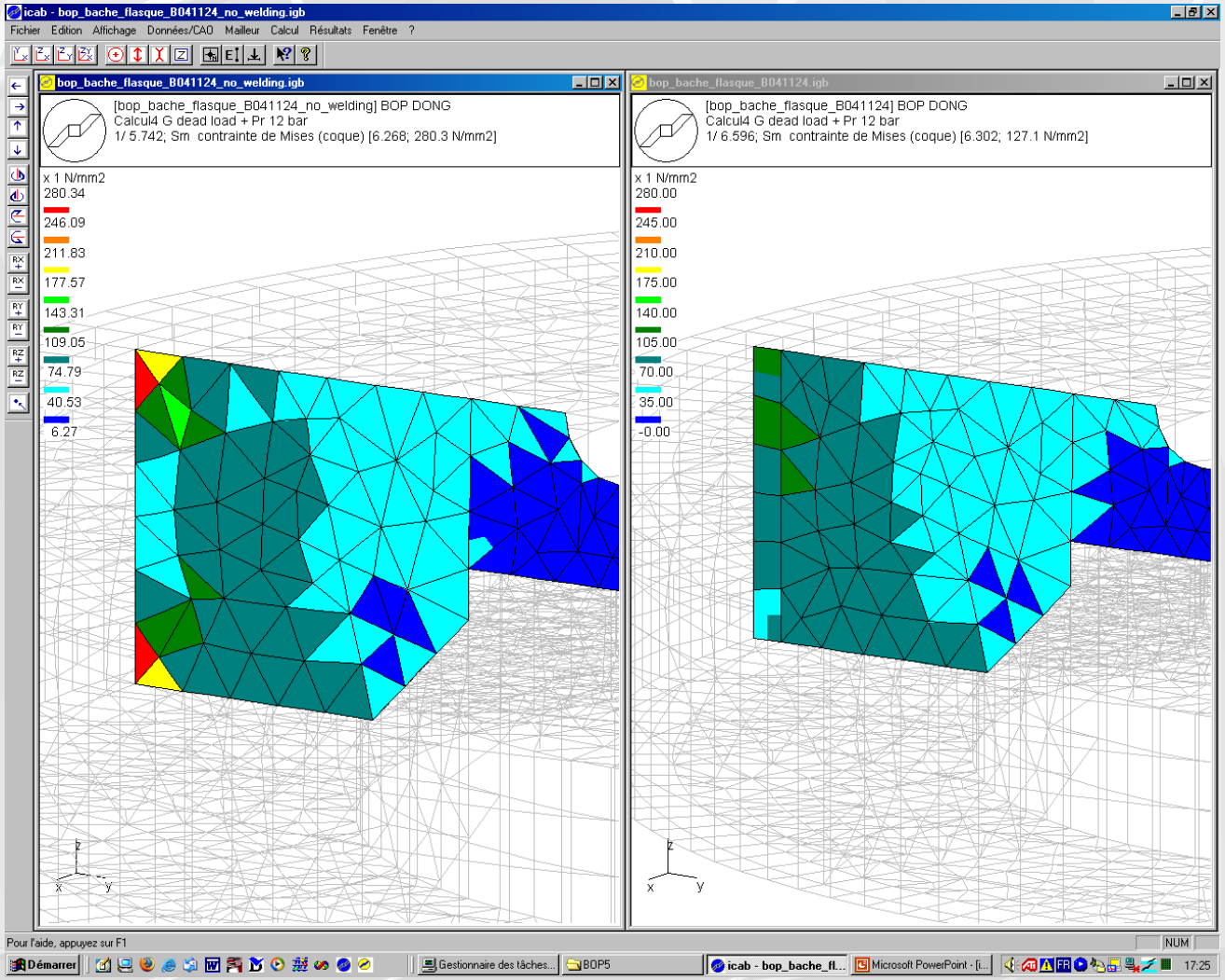


<< home

next >

# Welding effect

- Stress concentration in missing welding area





<< home

next >

# ICAB
# FEA software used to predict

- deflection
- Stresses
- Dynamic response

**to improve design and reliability**