Finite Element Methods for the Thermo-mechanical analysis of the Phase I Collimators

Workshop on Materials for Collimators and Beam Absorbers

3-5 Septermber, 2007

CERN Geneva

Alessandro Bertarelli[1]

Alessandro Dallocchio[1,2]

[1]TS department – Mechanical and Material Engineering Group
CERN, Geneva

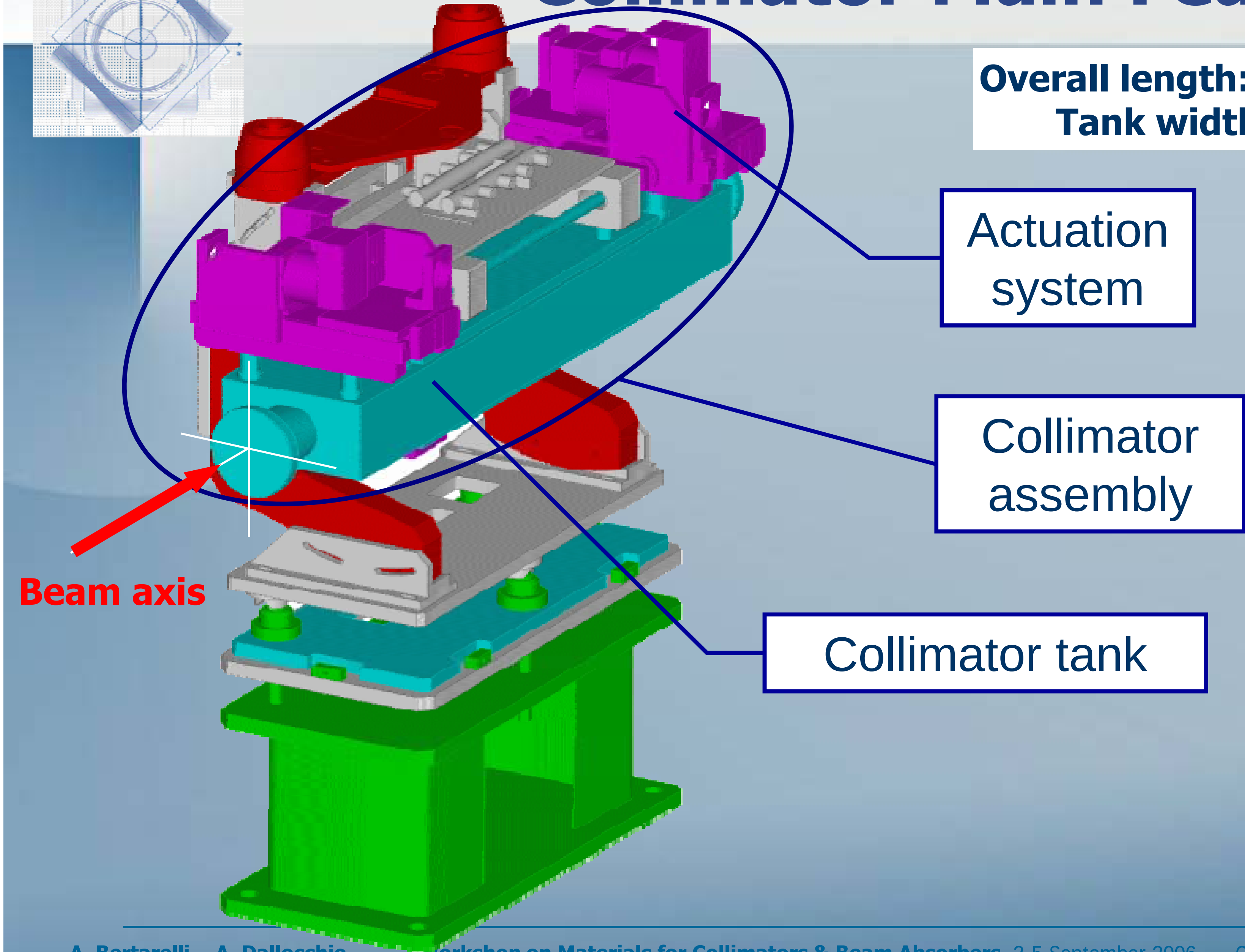
[2]Mechanical Engineering Department– Politecnico di Torino, Italy

Outline

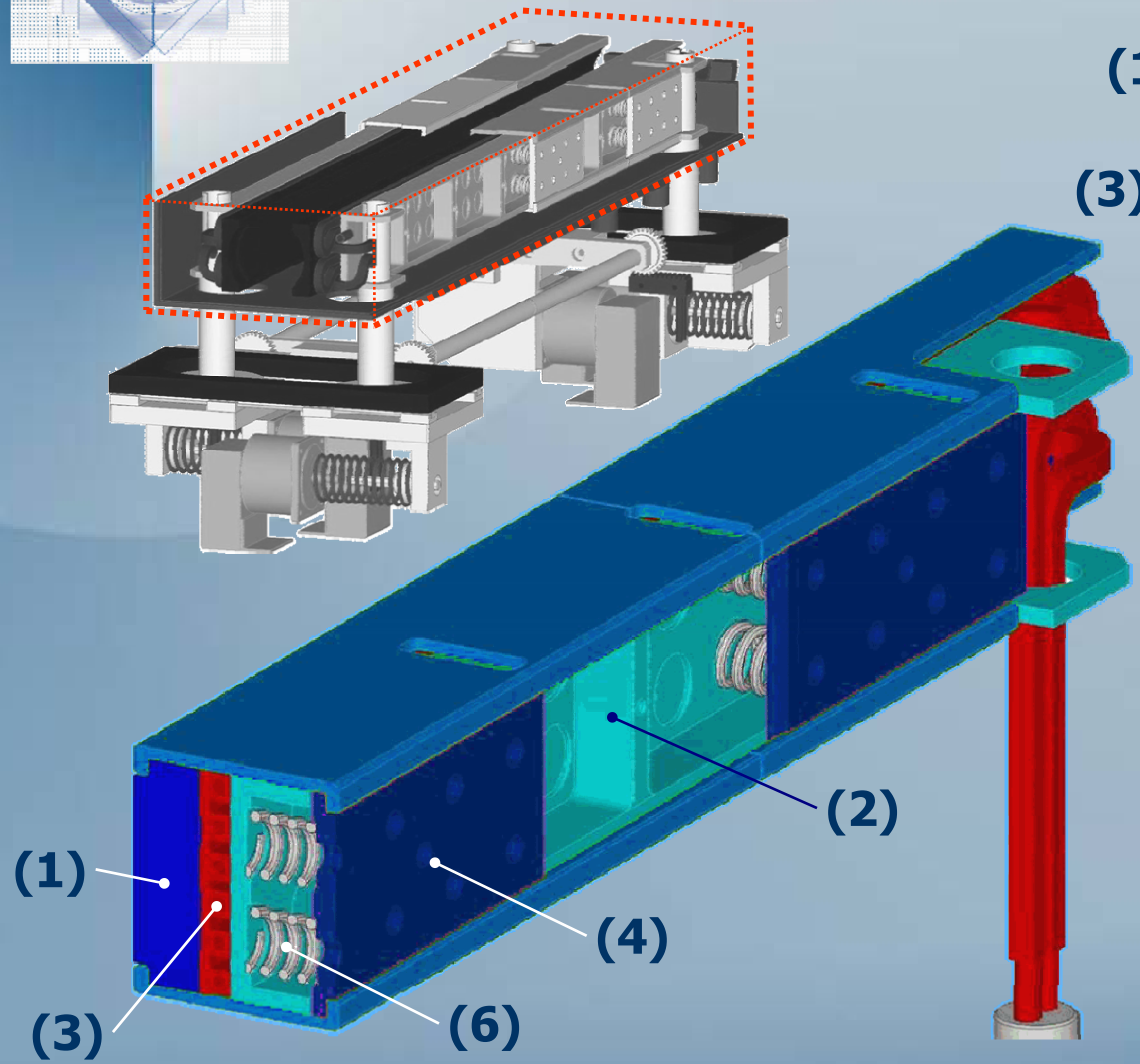
- Introduction
- Application of the Thermal load
- Coupled Thermo-structural analysis (Fully elastic)
- Sequential Fast-transient thermo-structural analysis (elastic-plastic)
- Conclusions

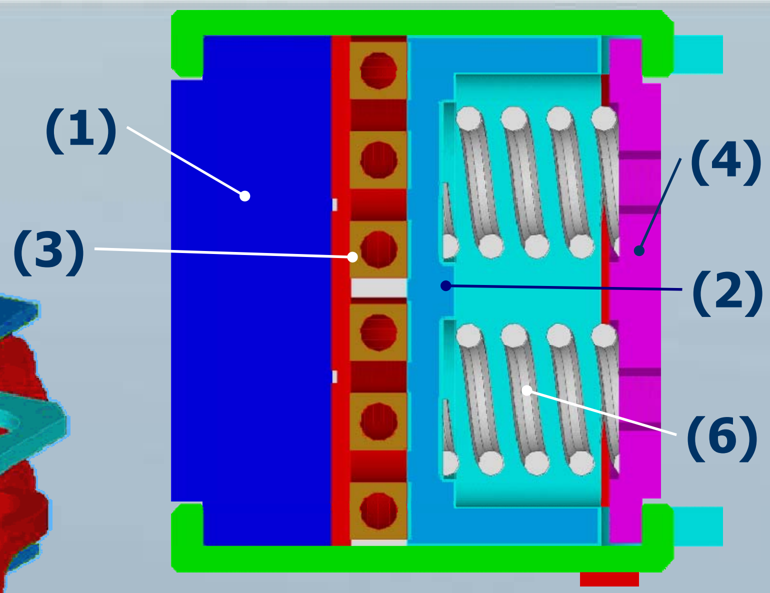
Collimator Main Features

Overall length: 1480mm
Tank width: 260mm



Collimator Main Features





Jaw Bloc Assembly

(1) Collimating Jaw (C/C composite)

(2) Main support beam (Glidcop)

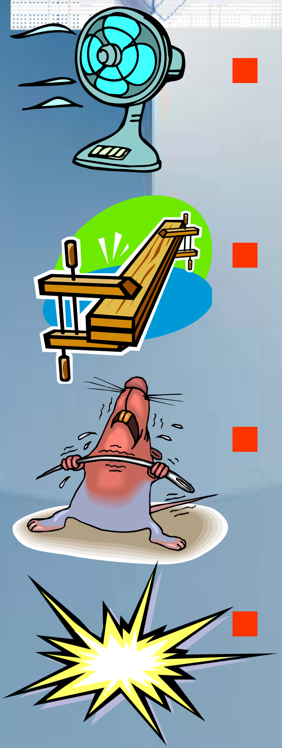
(3) Cooling-circuit (Cu-Ni pipes)

(4) Counter-plates (Stainless steel)

(5) Clamping plates (Glidcop)

(6) Preloaded springs (Stainless steel)

Objectives of the Thermo-structural analyses



- **Verify heat loads are effectively evacuated without attaining excessive temperatures**
- **Verify that good thermal contact between C/C jaw and metal support is ensured**
- **Verify that thermal deformations are kept to a minimum in steady-state and transient regimes**
- **Verify the jaw and metal support survive to thermal shocks (⇨ See also A. Dallocchio's talk)**

To meet all these requirements, given the complexity of the design and physics (coupled analysis, contacts, preloads, convection, thermal loads etc.), a general use, comprehensive F.E. code was required ⇨

Ansys Multiphysics

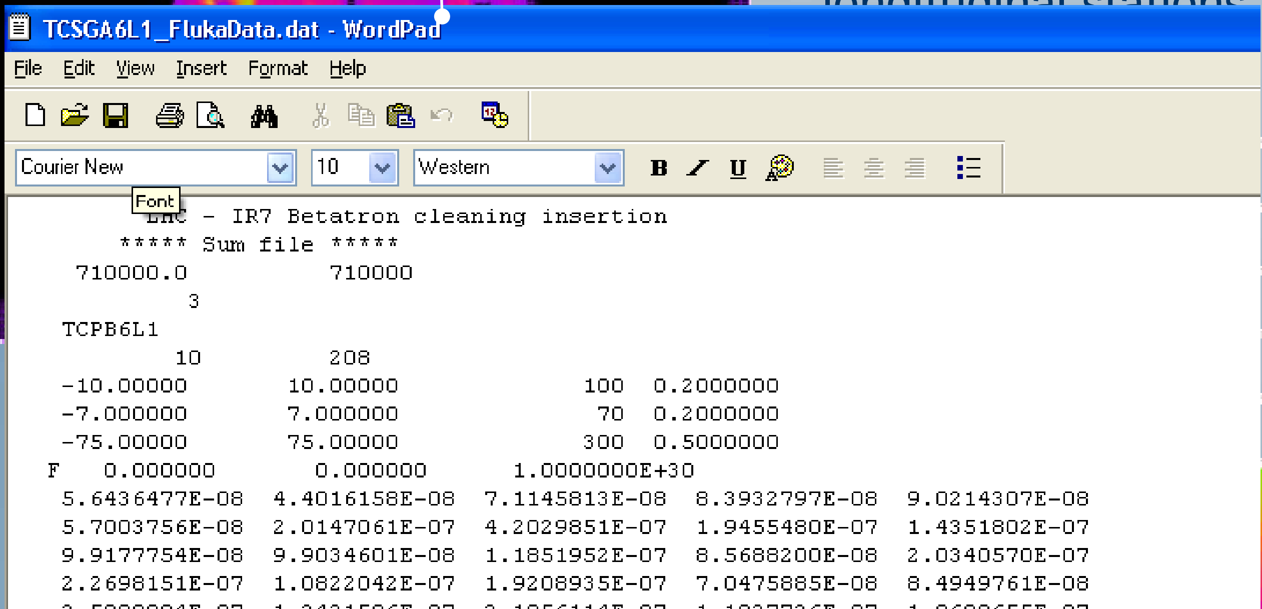
Outline

- Introduction
- Application of the Thermal loads
- Coupled Thermo-structural analysis (Fully elastic)
- Sequential Fast-transient thermo-structural analysis (elastic-plastic)
- Conclusions

Application of Thermal Loads

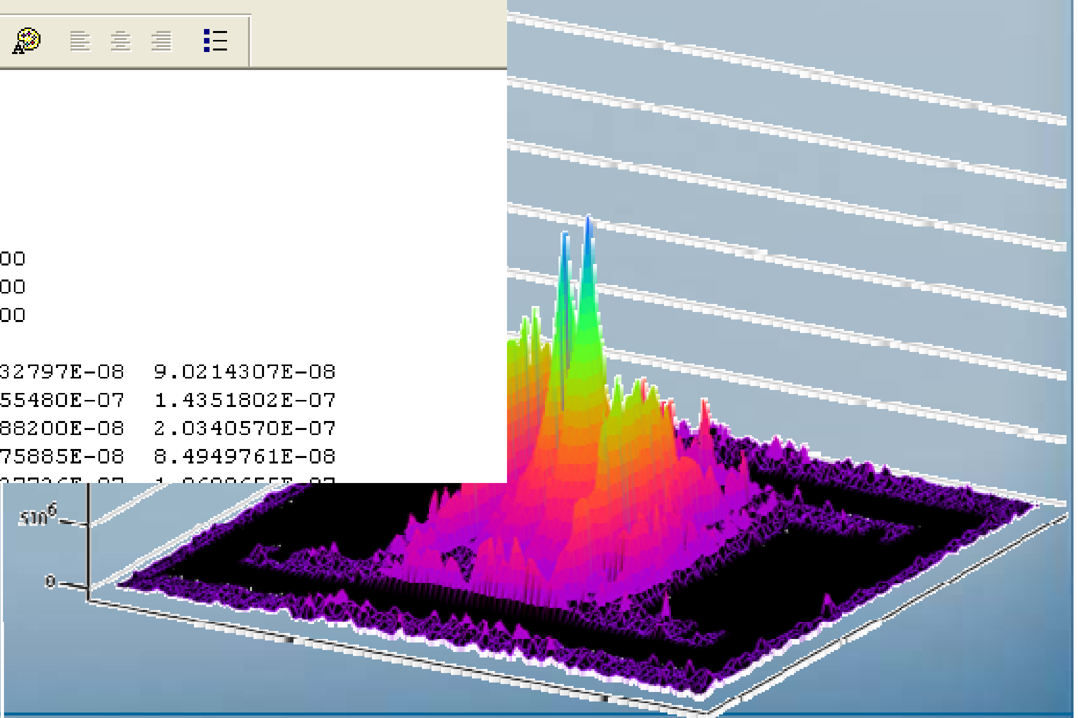
- **Step1** Thermal load from Fluka files ($GeV/cm^3/p$) are imported in Mathcad, rearranged and organized in a 3D matrix (XYZ) via a dedicated algorithm
- **Step2** 2D matrices (XY) at relevant longitudinal stations are extracted

5-columns, sequentially swept, Fluka input file



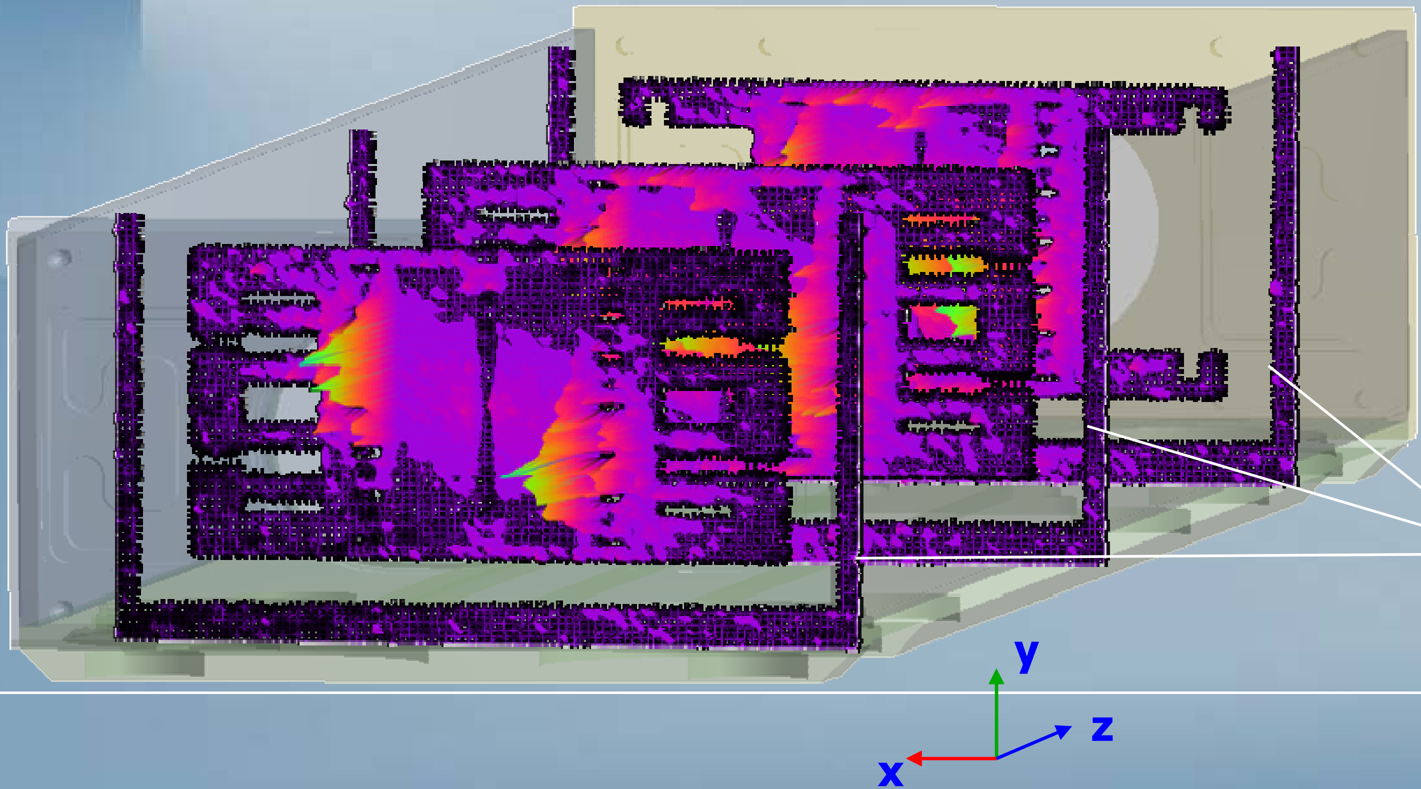

```
        - IR7 Betatron cleaning insertion
     ***** Sum file *****
  710000.0          710000
         3
 TCPB6L1
        10          208
 -10.00000       10.00000           100   0.2000000
 -7.000000       7.000000            70   0.2000000
 -75.00000       75.00000           300   0.5000000
F   0.000000       0.000000      1.0000000E+30
 5.6436477E-08  4.4016158E-08  7.1145813E-08  8.3932797E-08  9.0214307E-08
 5.7003756E-08  2.0147061E-07  4.2029851E-07  1.9455480E-07  1.4351802E-07
 9.9177754E-08  9.9034601E-08  1.1851952E-07  8.5688200E-08  2.0340570E-07
 2.2698151E-07  1.0822042E-07  1.9208935E-07  7.0475885E-08  8.4949761E-08
```

from Fluka) at a given longitudinal coordinate (z=1000 mm)



Application of Thermal Loads

Step 3 These 2D matrices (for each station) are read into ANSYS as a 3D table and applied to the FEM model as Internal Heat Generation. Ansys then linearly interpolates values between each station.



ANSYS linearly interpolates heat loads between matrices in longitudinal direction

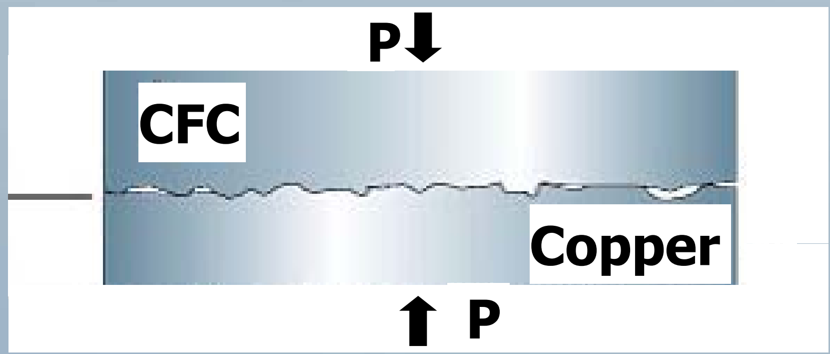
Outline

- Introduction
- Application of Heat Generation
- Coupled Thermo-structural analysis (Fully elastic)
- Sequential Fast-transient thermo-structural analysis (elastic-plastic)
- Conclusions

Thermo-mechanical analysis

A fully coupled thermal/structural analysis (i.e. the simultaneous resolution of temperature and displacement fields) is required since pressed-contact thermal conductance depends on contact pressure, which in turn varies with deformations .

$$h_c(P) = 1.49 \frac{k_S \Delta_a}{R_q} \left(\frac{2.3P}{E_G \Delta_a} \right)^{0.935}$$



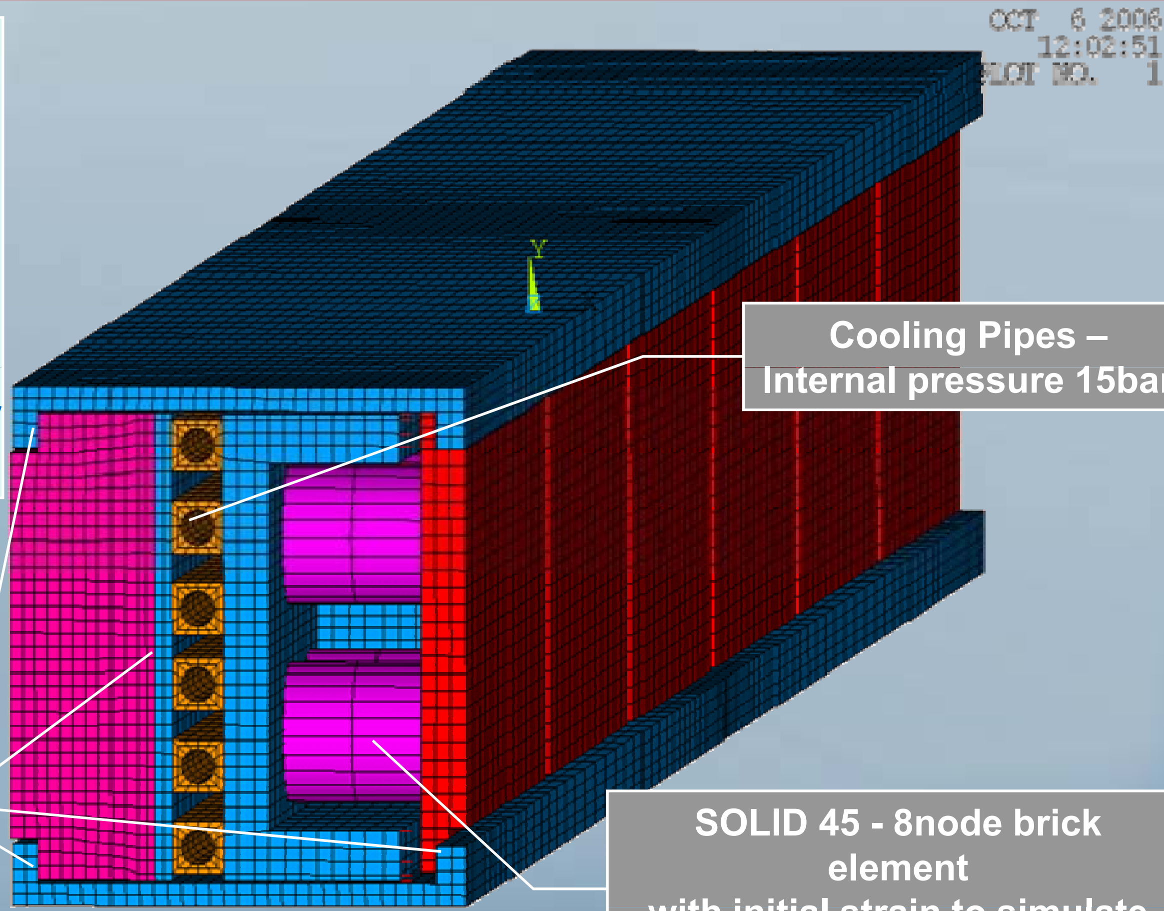
Equation relating thermal conductance *h* to Pressure *P*

This relation, implemented in Ansys FE model, has been experimentally validated

Thermo-mechanical analysis

Finite Element Model for 3-D Coupled Thermal-Structural Analysis
Steady-state (L.C. 1) and Slow-transient (L.C. 2)

- **D.O.F..: ux, uy, uz, T**
- **SOLID5 - 8node Coupled Elements ~70000nodes**
- **Linear elastic materials with temperature-dependent properties**
- **3-D linear orthotropic model for C-C jaw**



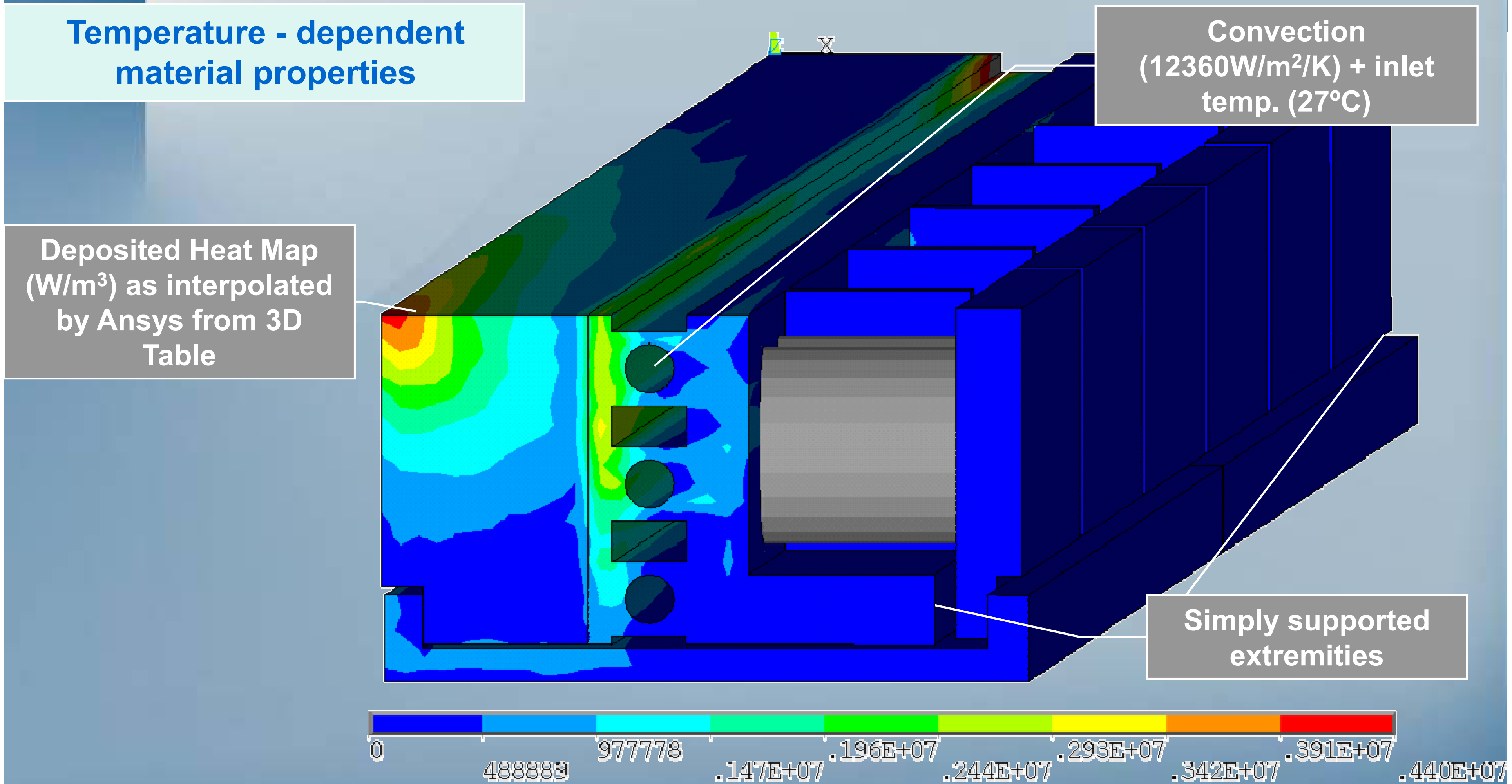
Cooling Pipes – Internal pressure 15bar

- **3D Contact elements: CONTA173 - TARGET170 d.o.f.: ux, uy, uz, T**
- **Lagrange and penalty method (for precision and convergence)**
- **Contact Friction taken into account**
- **Thermal contact conductance *h* function of contact pressure *P***

SOLID 45 - 8node brick element with initial strain to simulate spring preload (3bar)

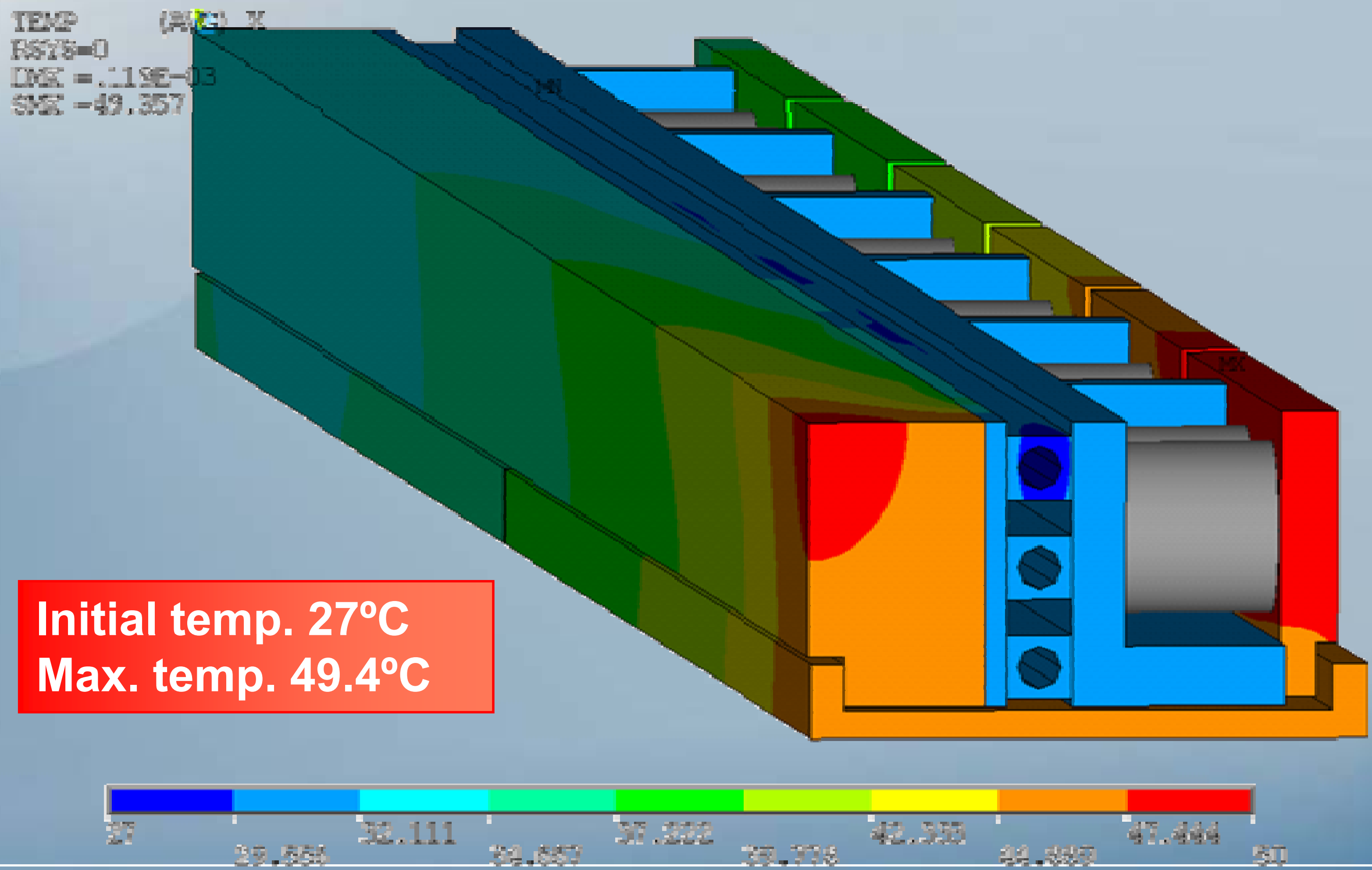
Thermo-mechanical analysis

Finite Element Model for 3-D Coupled Thermal-Structural Analysis



Thermo-mechanical analysis

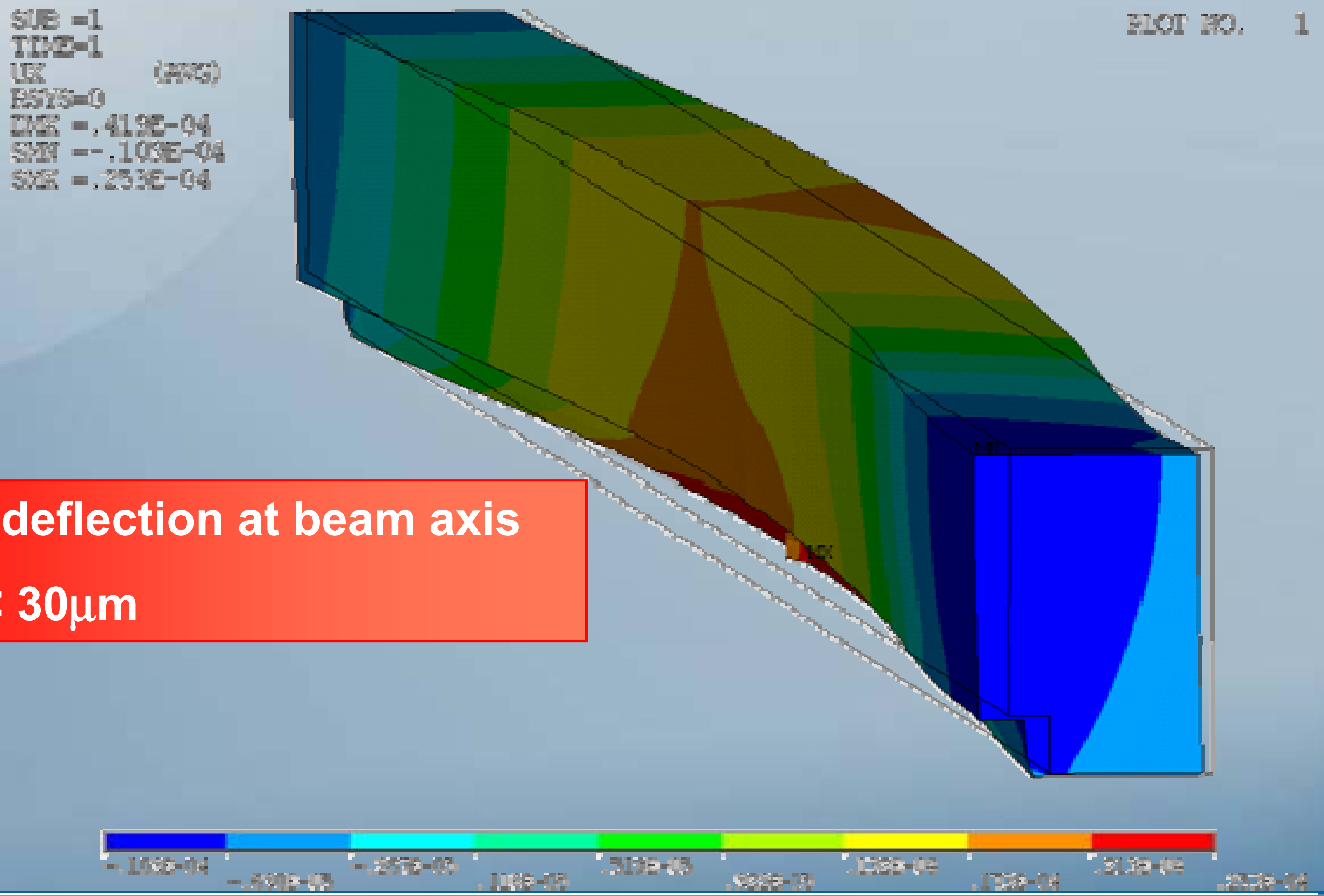
Thermal analysis – Load Case 1 (Nominal conditions) –
Steady state – Loss rate 8×10^{10} p/s – 4.5kW (Beam Lifetime 1h)



Initial temp. 27ºC
Max. temp. 49.4ºC

Thermo-mechanical analysis

Displacement analysis – Load Case 1 (Nominal conditions) – Steady state – Loss rate 8x10^{10} p/s – 4.5kW (Beam Lifetime 1h)



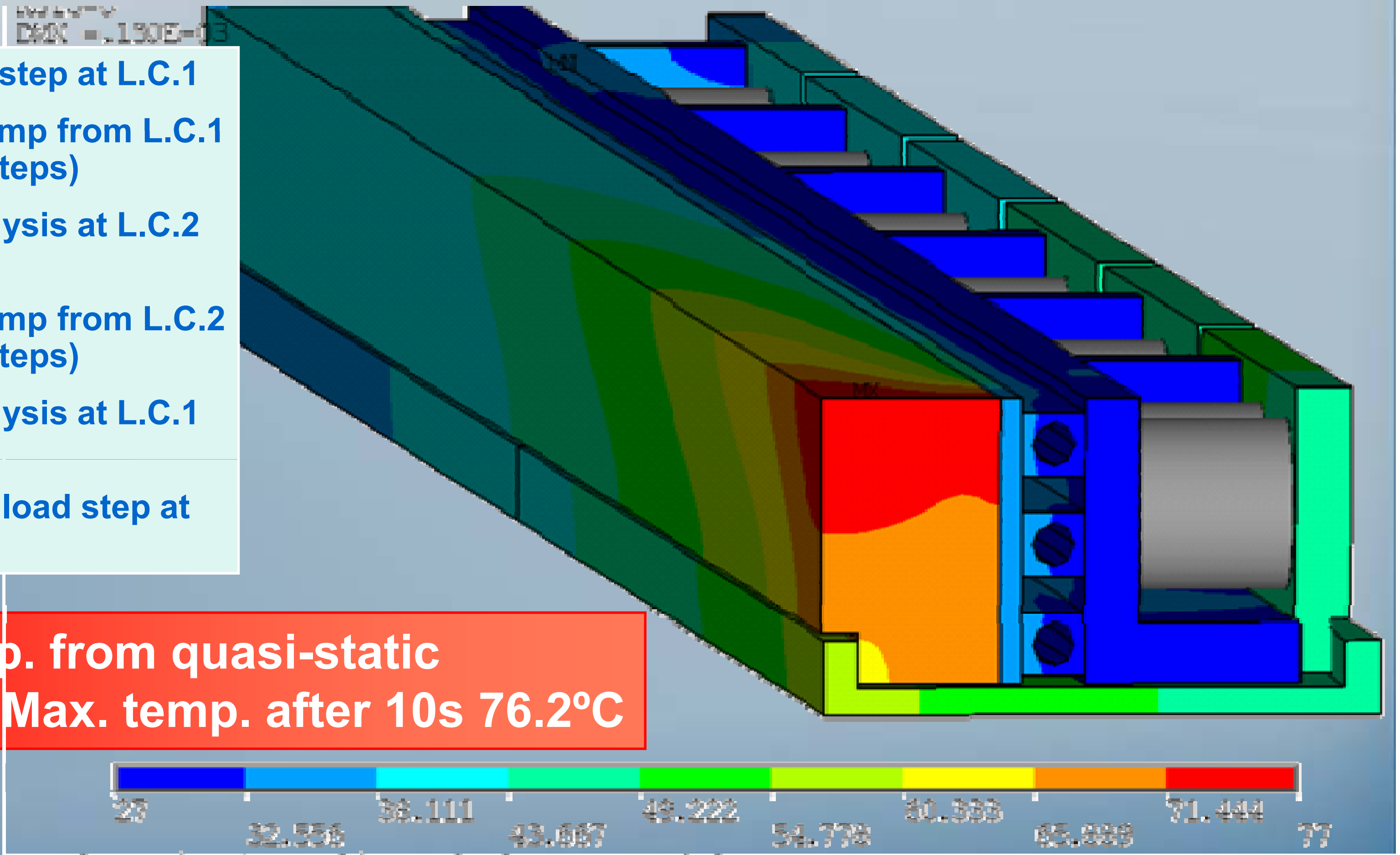
Total deflection at beam axis

δ_{Max} < 30μm

Thermo-mechanical analysis

Thermal analysis – Load Case 2 (Nominal conditions) –
10s Transient – Loss rate 4x10^{11} p/s - 22kW (Beam Lifetime 12min)

1. Quasi-static load step at L.C.1
2. 10ms transient ramp from L.C.1 to L.C.2 (20 sub-steps)
3. 10s transient analysis at L.C.2 (20 sub-steps)
4. 10ms transient ramp from L.C.2 to L.C.1 (20 sub-steps)
5. 20s transient analysis at L.C.1 (20 sub-steps)
6. Final quasi-static load step at L.C1



Initial temp. from quasi-static analysis - Max. temp. after 10s 76.2ºC

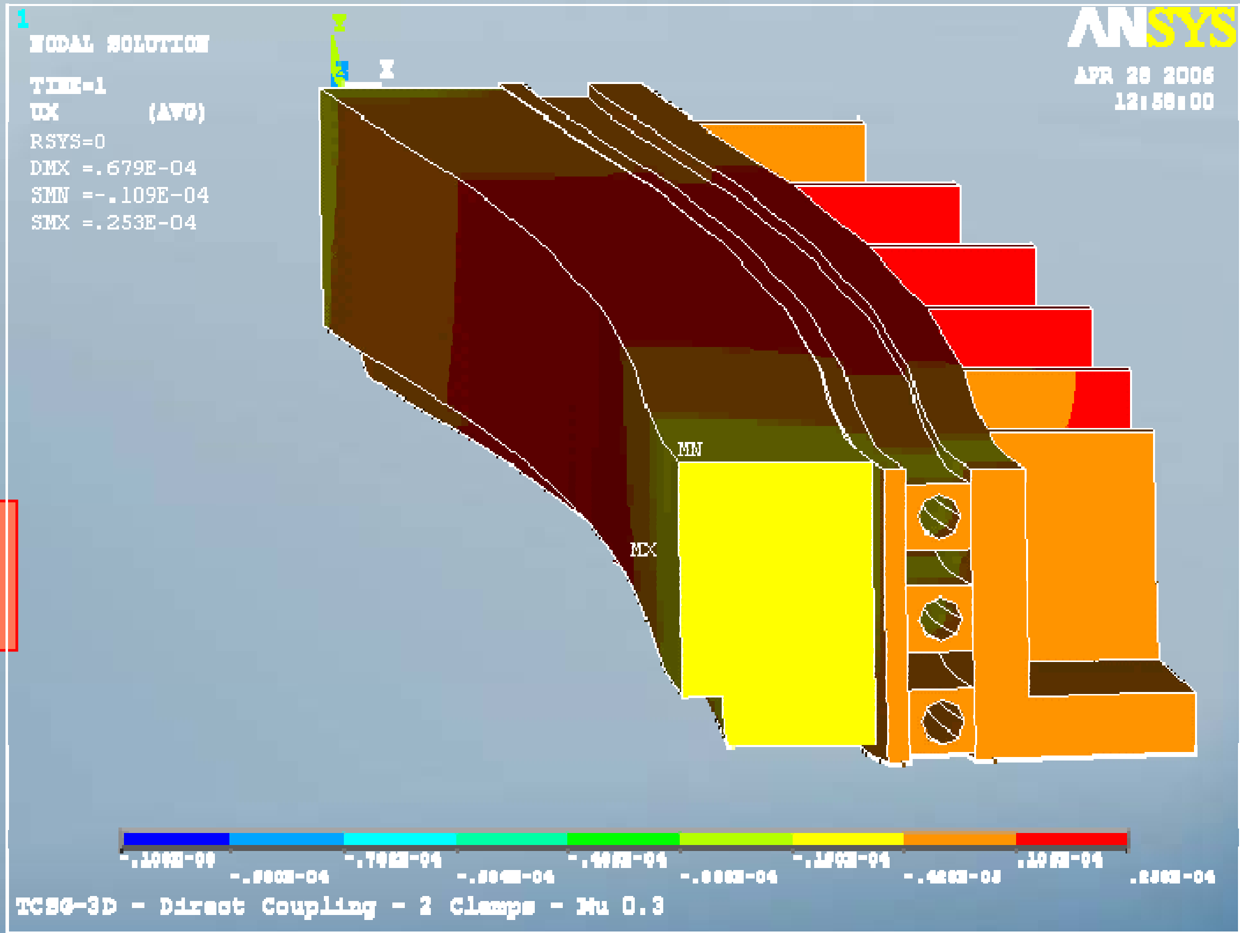
Thermo-mechanical analysis

Displacement analysis – Nominal conditions – Load Case 2
10s Transient – Loss rate 4×10^{11} p/s – 30kW (Beam Lifetime 12min)

Initial loss 8e10p/s
Max. deflect. ~20μm

Transient loss
4e11p/s during 10s

Max deflect. -108μm
Back to 8e10p/s



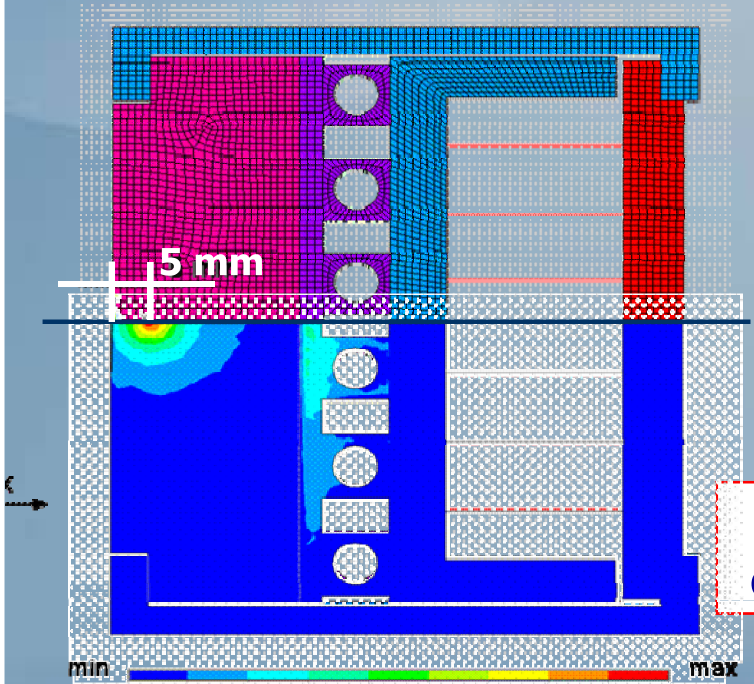
Outline

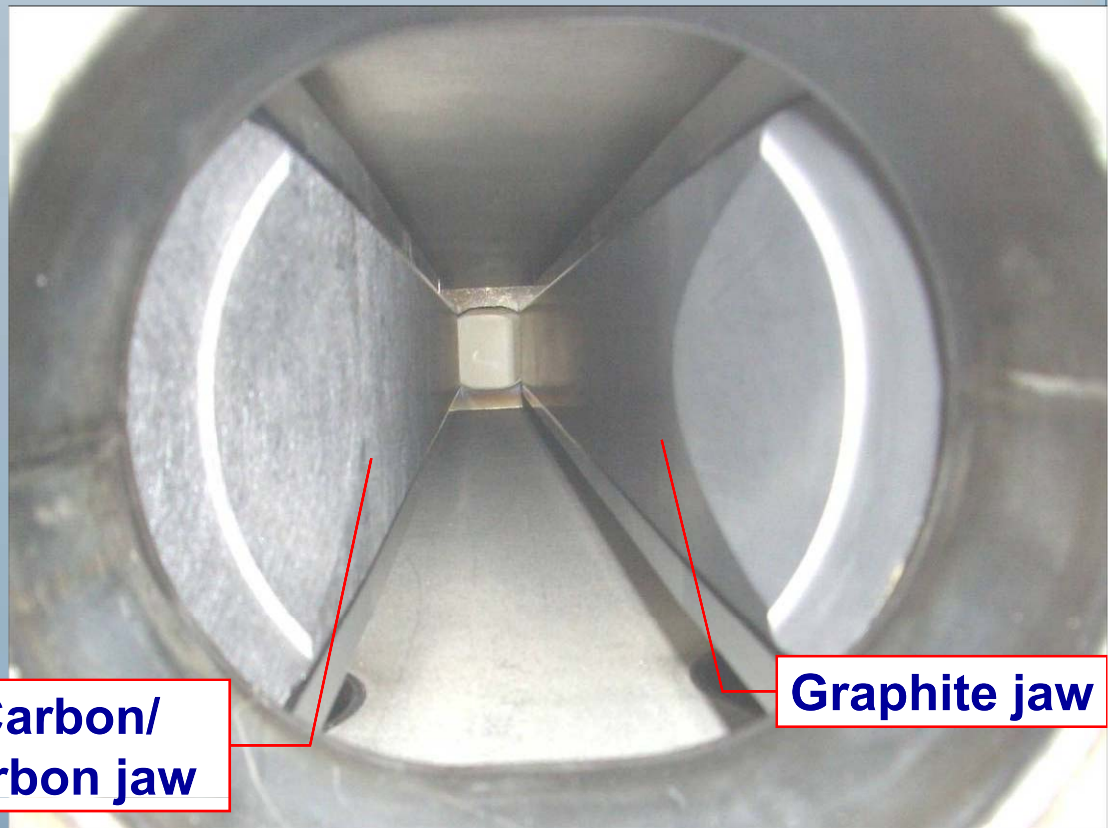
- Introduction
- Application of Heat Generation
- Coupled Thermo-structural analysis (Fully elastic)
- Sequential Fast-transient thermo-structural analysis (elastic-plastic)
- Conclusions

Simulation of Beam impact accidents

A fast-transient analyses are necessary to simulate accident scenarios: in particular 450 GeV Injection error test …

5 full intensity shots ranging from 1 to 5 mm, 3.2×10^{13} p, 7.2μs …
Each impact energy equivalent to more than ½ kg of TNT





Sequential analysis

Can we perform these analysis with Ansys Multiphysics, an implicit-integration scheme Finite Element code?

- **Ansys fully coupled thermal/structural analyses can only be performed in the linear elastic domain**
- **Numerical analyses in the elastic/plastic domain are necessary to simulate possible permanent deformation of metallic parts in case of accidents (as those found after 2004 Injection error test)**

But …

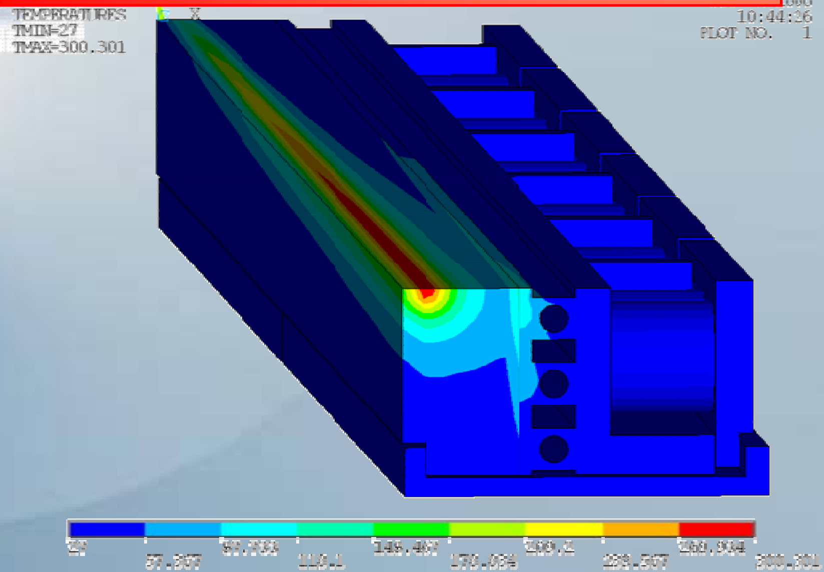
- **Accident cases entail fast energy depositions (a few *μs* or less)**
- **Given the typical thermal diffusion times (~*ms* or more – $\tau_{diff}=L^2/\kappa$), heat conduction plays a minor role in case of very short time-scale analyses.**
- **Hence, thermal and structural analyses can be decoupled and sequentially solved with Ansys Multiphysics**

The price to pay …

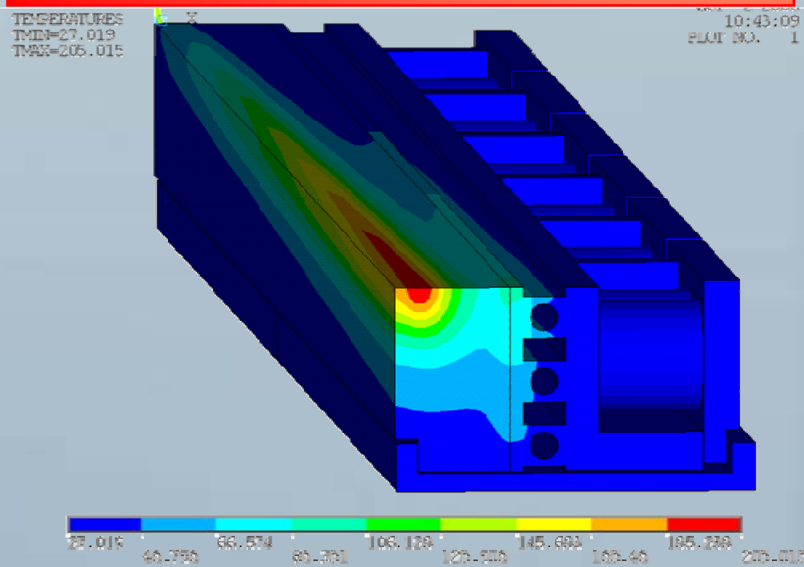
- **Be patient … very short time-steps ($\Delta T<0.9L_{mesh}/c<1\mu s$), at each step inversion of the stiffness matrix ⇨ very long CPU times**

Thermal analysis

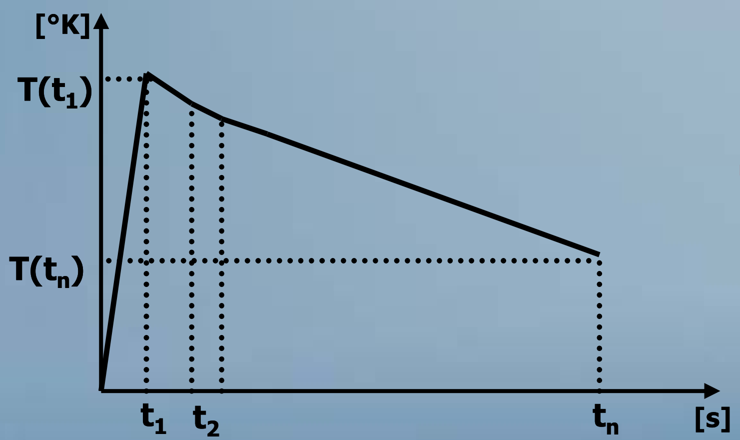
Temperature distribution at time t = 7.2μs



Temperature distribution at time t = 60ms



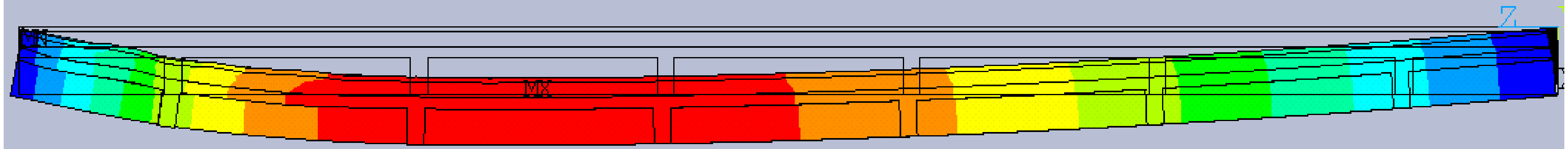
Temperature distribution as a function of time



- Temperature distributions from thermal analysis were applied at different sub-steps (time)
- Ansys linearly interpolates between sub-steps loads so quite closely following actual temperature evolution

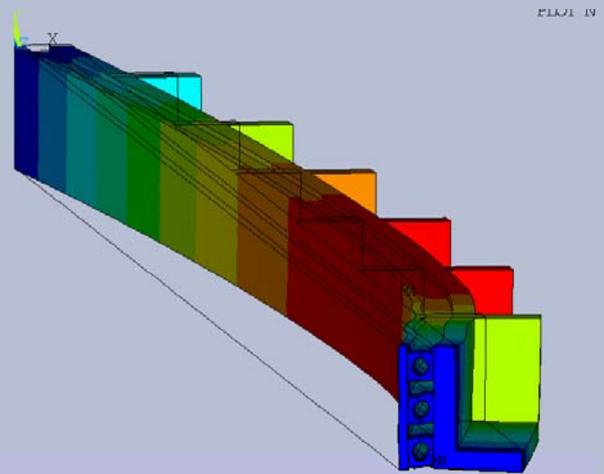
Fast-transient structural analysis

Collimator jaw oscillates several ms after the beam impact - *Thermally induced vibrations* have been fully studied with a 3D FE model in the elastic-plastic domain (see A. Dallocchio's talk for more details)



A multilinear-kinematic hardening plasticity model was used for metals

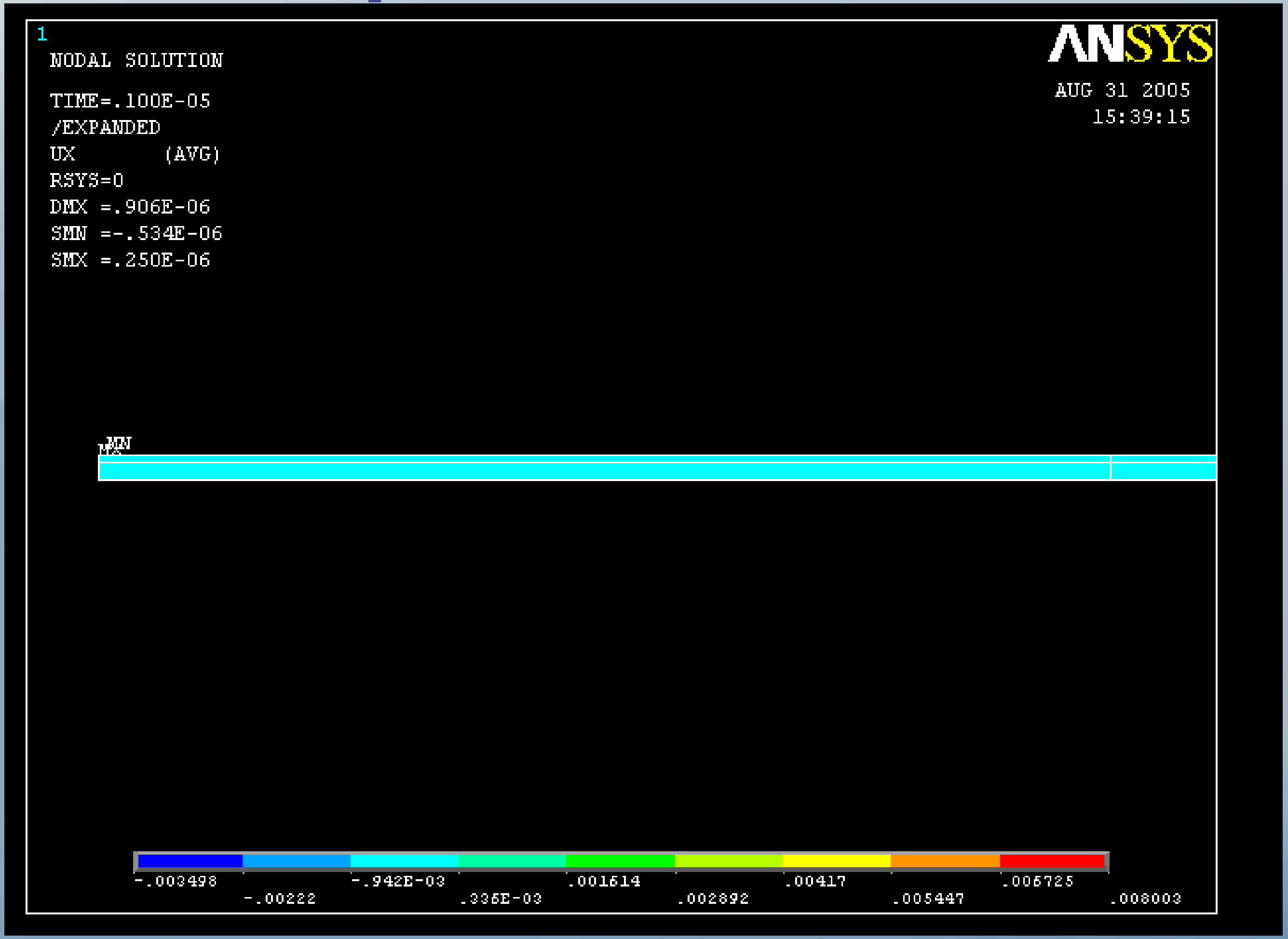
- **Over a long time scale the dynamics effects (inertia) were turned off.**
- **A last static load step was necessary to calculate equilibrium between plastic residual stresses and elastic forces.**
- **Numerical results are in line with analytical prediction**



Predicted permanent deformation is not zero (because of limited plasticity on the CuNi pipes) but well within limits (16 µm)

Thermo-mechanical Analysis

Results for the simplified model



Outline

- Introduction
- Application of Heat Generation
- Coupled Thermo-structural analysis (Fully elastic)
- Sequential Fast-transient thermo-structural analysis (elastic-plastic)
- Conclusions

Conclusions I

- In-depth thermo-structural simulations were performed on Phase I collimator models from the early stages allowing design optimization; variable thermal loads from FLUKA were applied making use of 3-D tables.
- Thermal/structural coupled elements used. On relatively long time-scales, it is not possible to separate the two domains (thermal conductance depends on contact pressure), making the problem intrinsically coupled.
- For contact elements (which include friction) the *Lagrange and Penalty* method was used allowing control of contact penetration.
- Studies of solutions accuracy, influence of mesh dimensions and time-steps were done.
- Comparisons with analytical calculations and experimental results were performed to validate FE models.

Conclusions II

- Sequential fast-transient thermo-structural analyses were performed in the elastic-plastic domain to predict collimator behavior in case of beam impacts.
- Solutions allowed to study thermally-induced vibrations on the short time scale and predict residual deformations on the long time-scale.
- We believe Ansys Multiphysics can be successfully applied for the reliable solution of complex fast-transient thermo-mechanical problems, provided changes of phase are not occurring or stress level is well below material Young's modulus (⇨ hydrodynamic theory)
- The main implicit method drawback is long solution times imposed by very short time-steps (this can be offset by powerful hardware)