Double way interface between HFSS and ANSYS

HFSS ⇄ ANSYS

Autor: Grzegorz Zelek (Cracow University of Technology, technical student at TS-MME)

Supervised by: Adam Wróblewski (Cracow University of Technology),

Erk Jensen (AB-RF), Tadeusz Kurtyka (TS-MME)

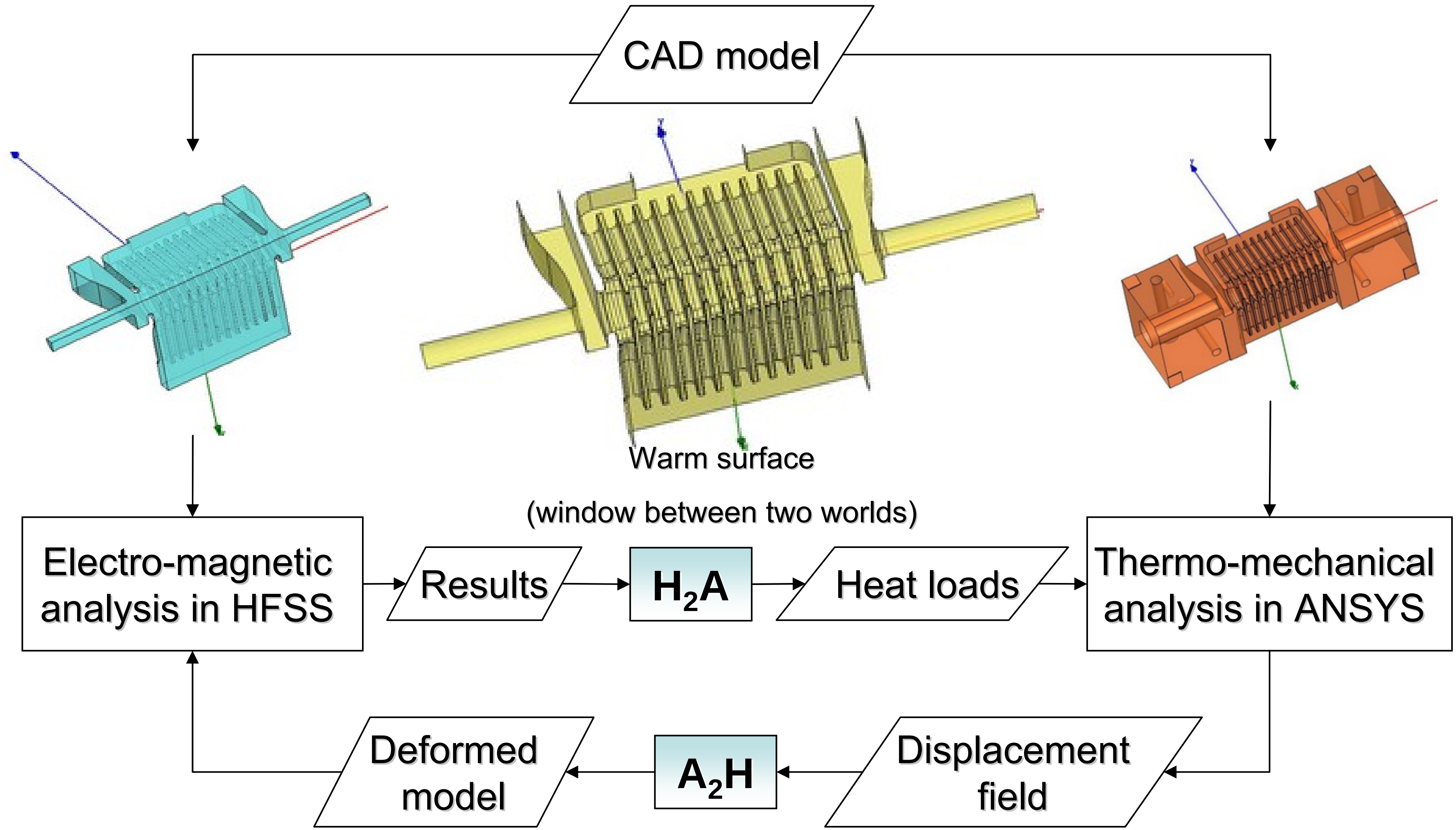
Why do we need that tool?

1. We need a bridge between HFSS and ANSYS to analyse what are the thermal consequences associated with electromagnetic field delivered to the structure
2. What is the thermal deformation and what influence it has for electromagnetic parameters of the cavity

ANSYS offers also an electromagnetic solver, and ANSOFT (HFSS) offers ePhysics for structural analysis. Nevertheless we want to have a link between them:

- we want each program to do what it does best,
- to have another comparison of obtained results,
- it is nice to have an alternative.

General conception of the interface



Outline of the presentation:

Presentation of HDX11 – a real accelerating RF structure

HFSS to ANSYS

- initial assumptions
- problem of „hot spots”

ANSYS calculation of the real RF cavity

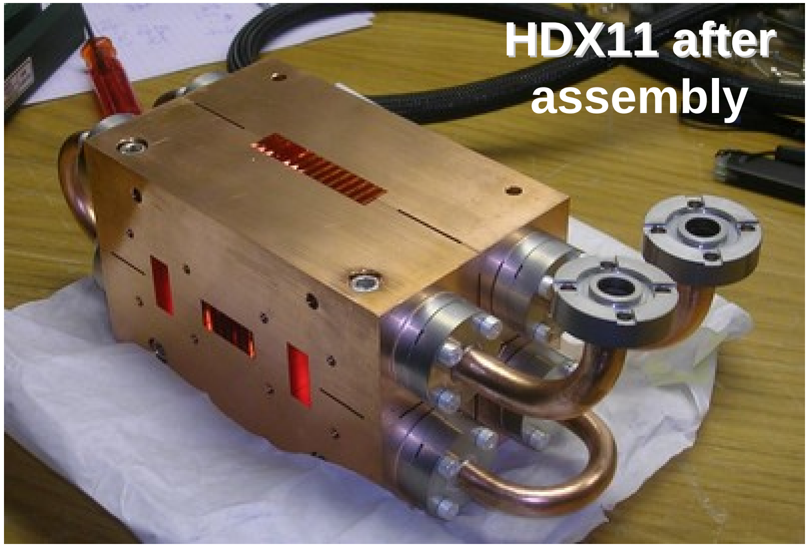
- result of steady-state thermo-mechanical analysis of HDX11 structure
- some aspects of transient analyses of RF cavities

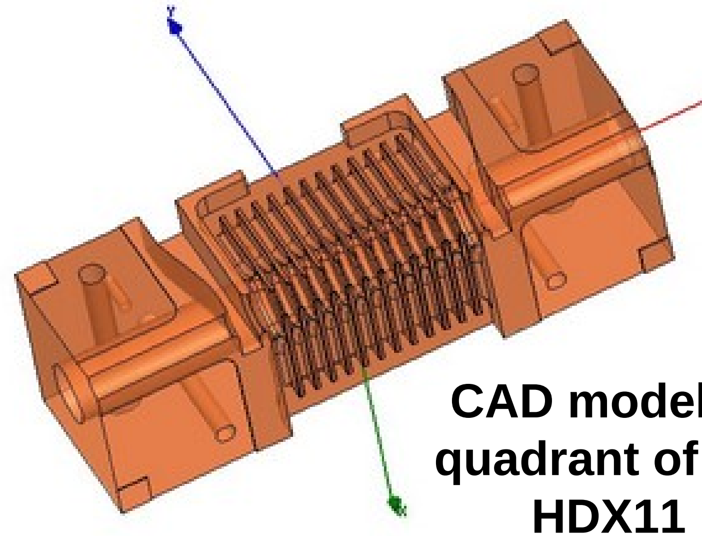
ANSYS to HFSS

- short description of the approach
- modification of interpolation's scheme
- updated mesh
- comparison of HFSS results obtained on genuine and deformed meshes

Things to develop

HDX11 – a real RF structure





Working parameters:

Frequency:	11.424GHz
Average RF power:	500W/quadrant
Pulse RF power:	100MW/quadrant
Duration of the pulse:	70ns
Repetition rate:	60Hz

General parameters:

Material:	copper,
Dimensions:	45x45x150 mm
Weight:	2.18kg

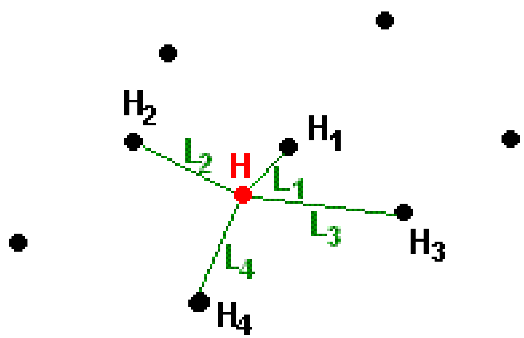
HFSS to ANSYS – initial assumptions

1. Heat is generated in the „skin" layer of the internal walls of the cavity hence the heat input to the ANSYS FE model is assumed to be heat flux.

$$q = \frac{1}{2} R \cdot H^2 \qquad R = \frac{1}{\delta \cdot \sigma} \qquad \delta = \sqrt{\frac{1}{\pi \cdot f \cdot \mu \cdot \sigma}}$$

* For HDX11 in room temperature skin depth=0.62μm

2. HFSS and ANSYS meshes are independent – a special interpolating formula is needed.



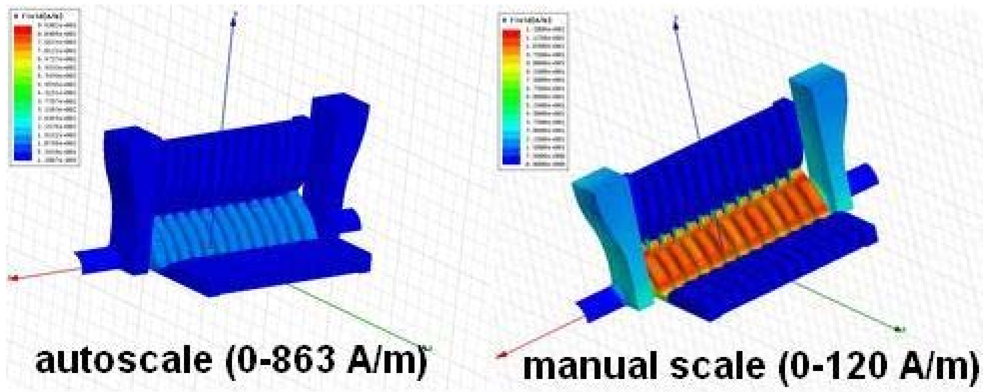
$$H^{ANSYS} = \sum_{n=1}^{k} w_n \cdot H_n^{HFSS}$$

$$\sum_{n=1}^{k} w_n = 1$$

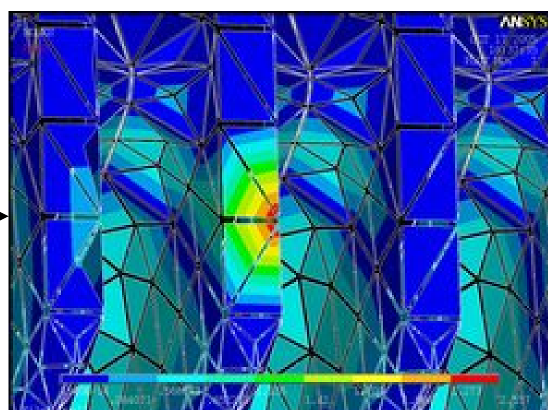
$$w_n = \frac{\frac{1}{l_n}}{\frac{1}{l_1} + \frac{1}{l_2} + \ldots \frac{1}{l_k}}$$

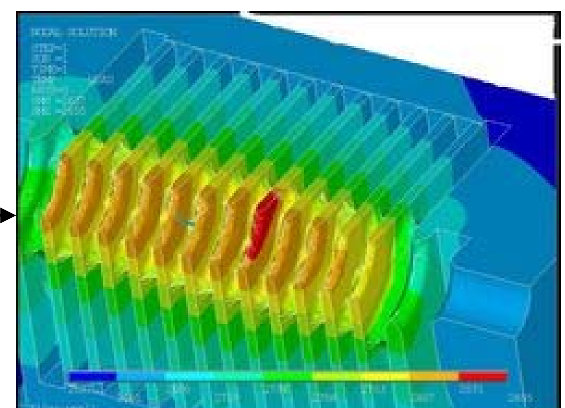
$$l_1 < l_2 < \ldots < l_k$$

„Hot spots" – what is that?



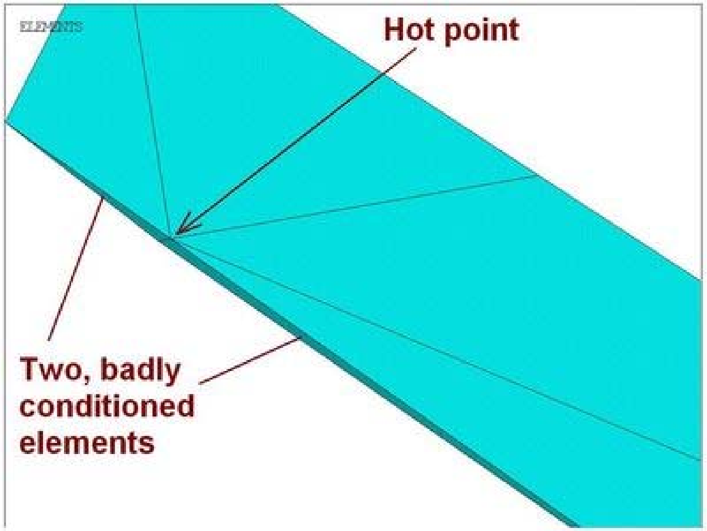
Display file → Interpolation in H_2A



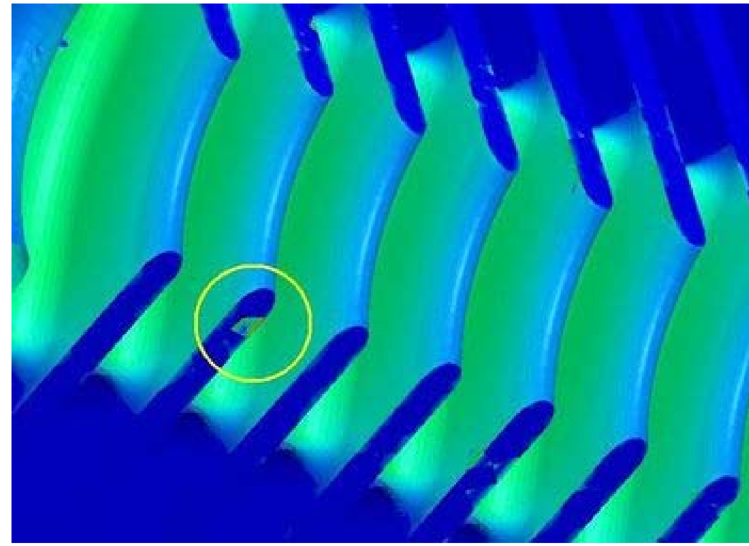


Origin of the „Hot spots”

Hot spots may occur in neighbourhood of poor elements



or at sharp edges of the model.



-We need a filtering method.

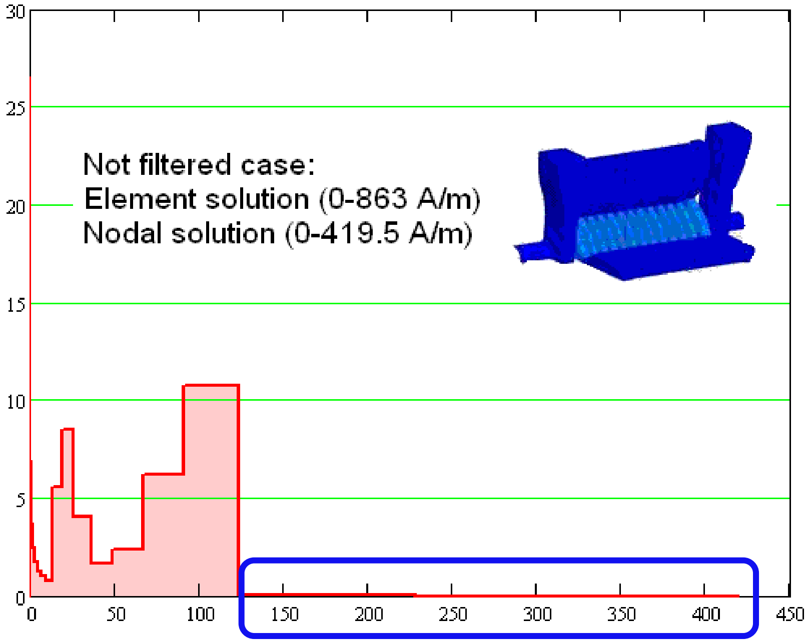
-Because occurrence of the „hot spots” it is a numerical artefact, the filtering method can be independent from the electromagnetic theory.

-The proposed filtering method should be very general, meaning it should work properly not only with one task.

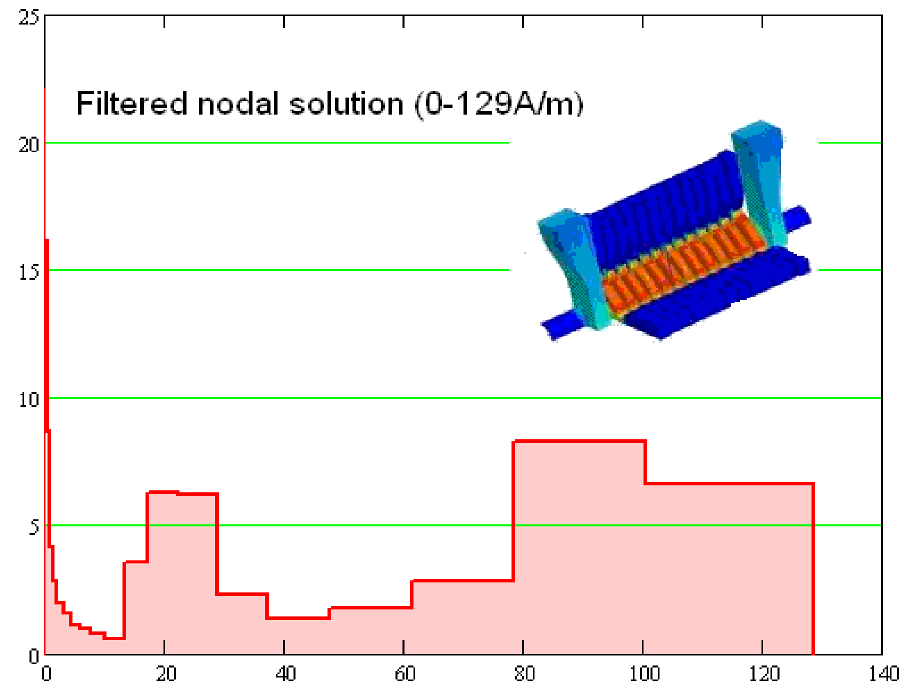
Filtering of the „Hot spots”

Histogram: Number of surface elements

with field in certain value



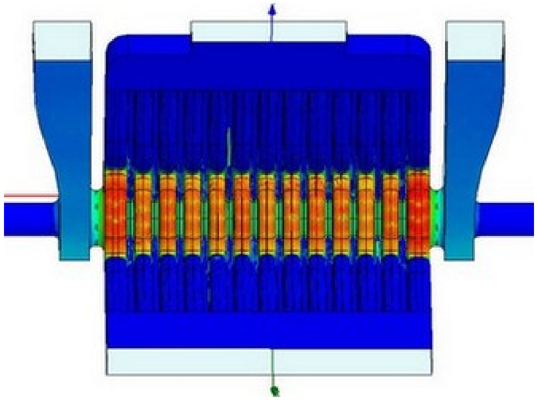
The filtering method base on observation that number of points with abnormally high value is very small.



What we want to do is to „correct” value of the „hottest” points.

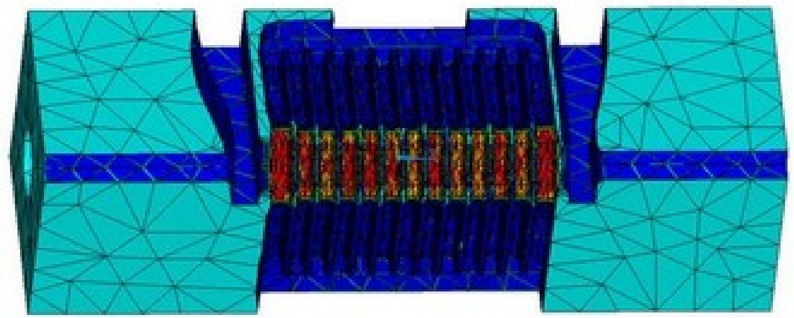
Steady-state thermal analysis (HDX11)

H field (output from HFSS)



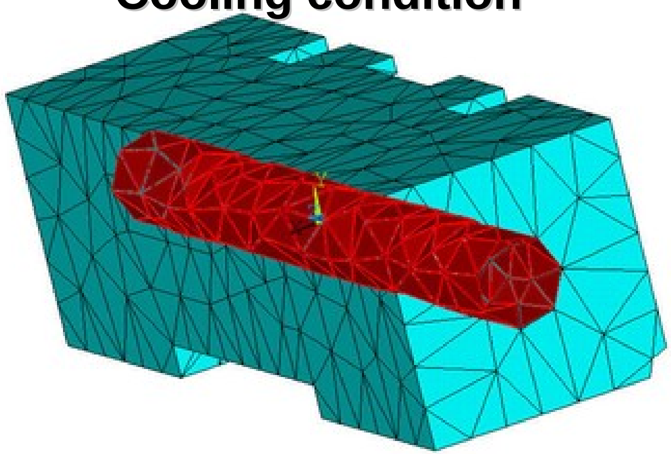
H_{max} = 53.85A/m (RF power =1W)

Heat flux (input for ANSYS)



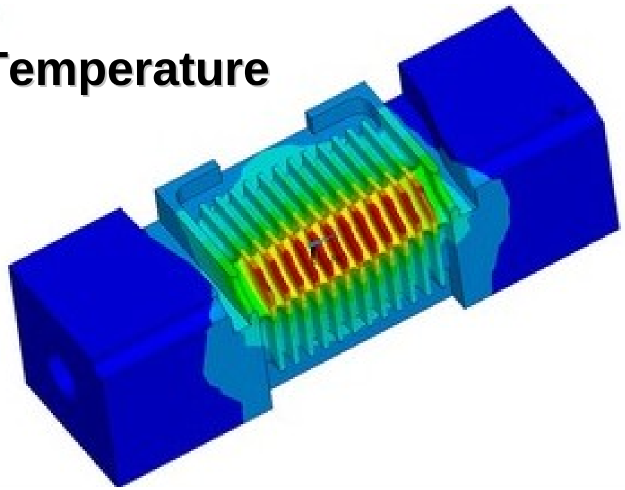
q_{max} = 22’836W/m^2 (RF power =500W)
Q_{tot} = 41.4W/quadrant

Cooling condition



T=44°C, α=3’500W/(m^2·K)

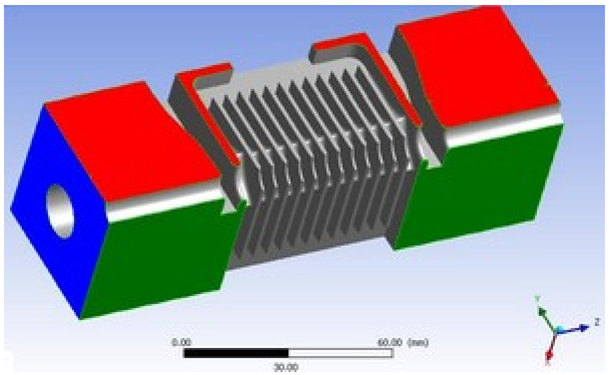
Temperature



(45.241°C – 48.188°C)

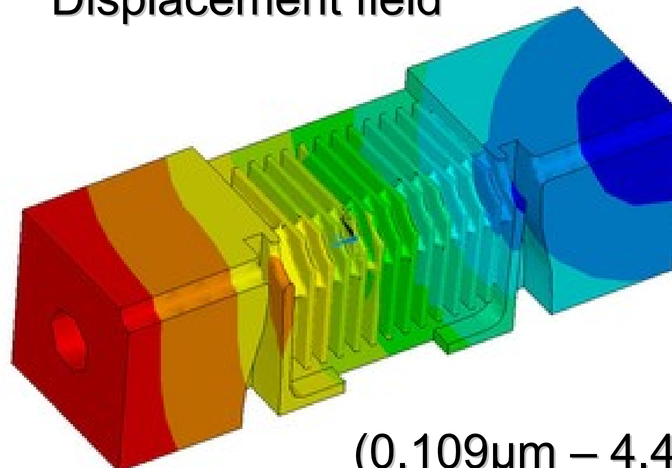
Steady-state mechanical analysis

Boundary conditions



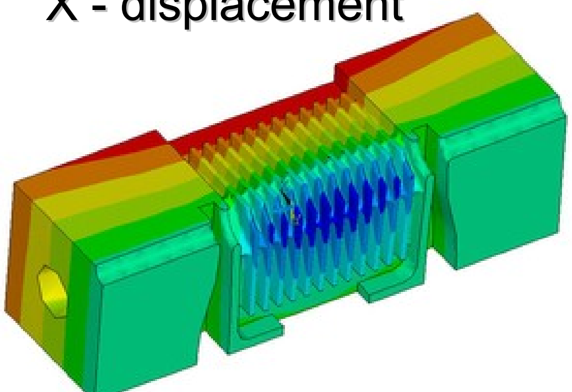
YZ symmetry plane – X blocked
ZX symmetry plane – Y blocked
Z – blocked

Displacement field



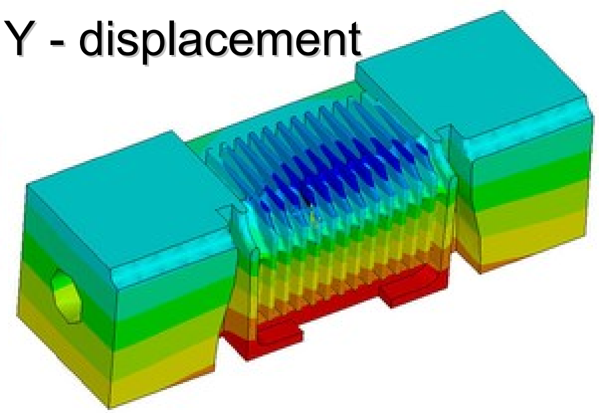
(0.109µm – 4.41µm)

X - displacement



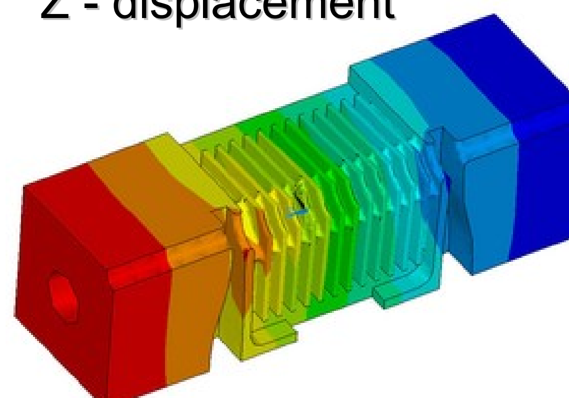
(-0.776µm – 1.3µm)

Y - displacement



(-0.563µm – 1.53µm)

Z - displacement



(0 – 4.17µm)

Transient thermal analyses

Because a full cycle consist of „nanosecond" heating, we need to use time steps of nanosecond order.

To catch all transient effects in thermal analysis the following condition must be satisfied $L \leq \sqrt{\alpha \cdot \tau}$ where $\alpha = \frac{k}{\rho \cdot c_p}$ is a material property.

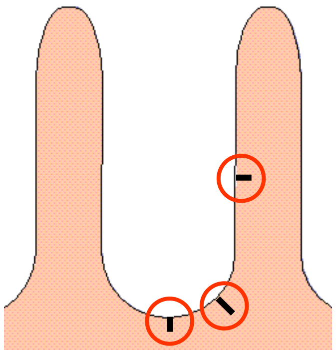
$\alpha_{Cu} = 1.118 \cdot 10^{-4} \frac{m^2}{s}$	
Typical time - t	**Typical length - L**
20ms	1.496mm
70ns	2.798µm
10ns	1.058µm

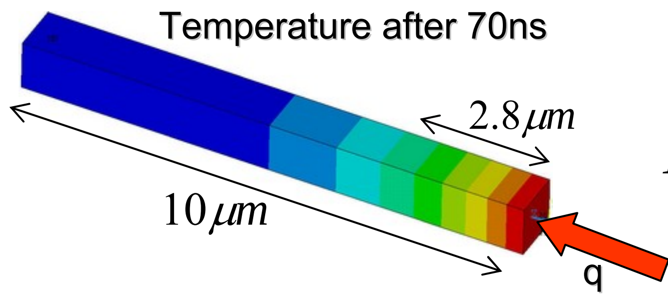
To mesh a cubic 1x1x1 with tetrahedral elements of 0.1 size it is necessary to use ~7.8e3 elements

The volume of metal used for quadrant of HDX11 is ~244cm^3. To mesh that volume with tetrahedral elements (size 1µm), desired number of elements would be about 1.9e15!

Transient thermal analyses (2)

As we cannot afford for the transient thermal analysis of full model let concentrate on small enough part of the model.





$$L = \sqrt{\alpha \cdot t} \quad L_{Cu,70ns} = 2.798 \mu m$$

Due to the low penetration of heat up the material during the pulse and despite the complex shape of the real cavities, the problem locally might be considered as a 1D heat conduction problem (a semi-infinite solid with heat flux load), for which analytical exist.

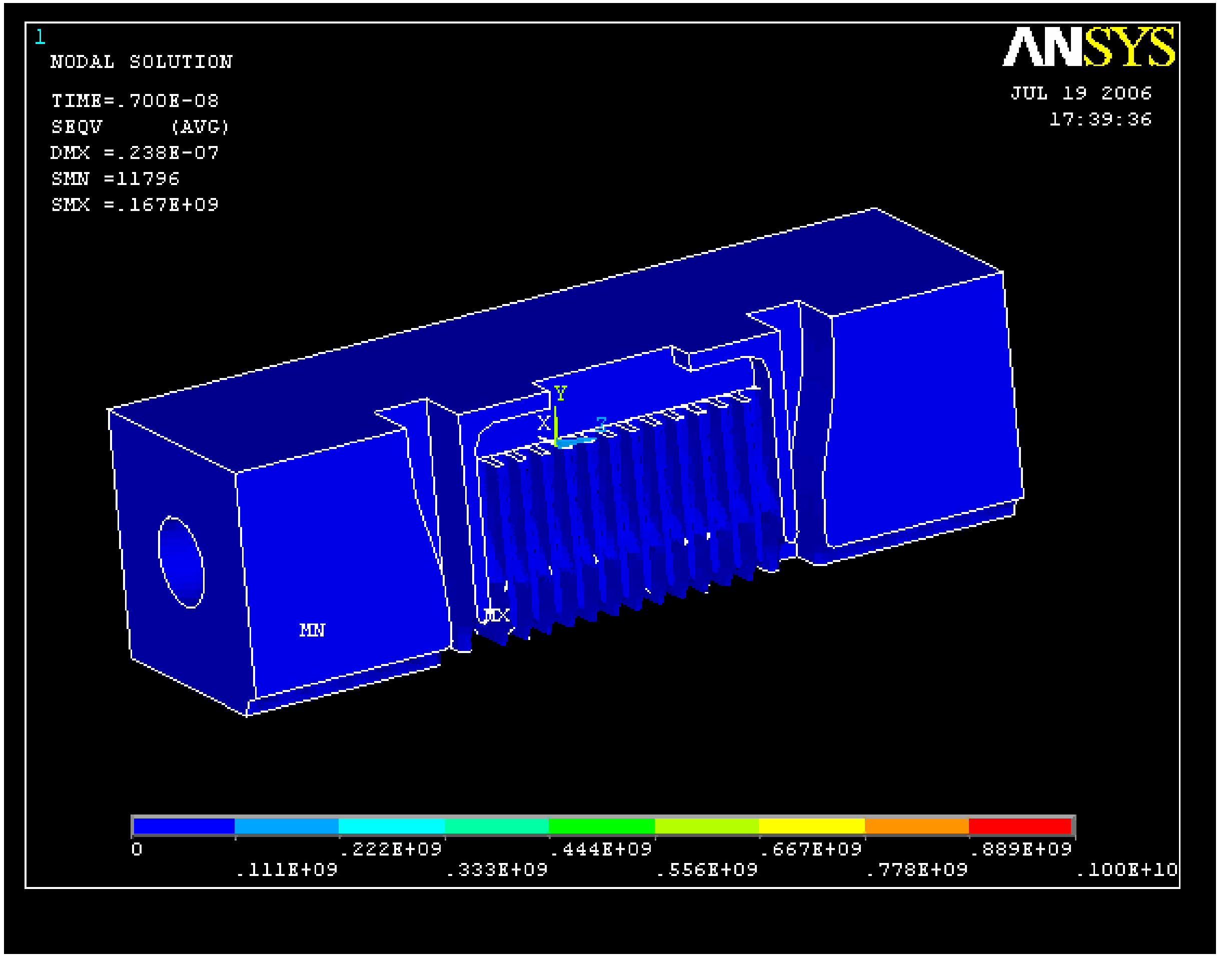
Total temperature

Temperature at any point from WS	Steady-state component (FE analysis)	Peak component (analytical formula)

For the HDX11 structure the max. temp increase during 70ns pulse is ~36K and max. absolute temperature ~84°C.

Transient mechanical analysis

As long as we cannot do reliable transient thermal analysis of the full model we neither can rely on results of mechanical analysis.



ANSYS to HFSS – two different approaches

Update of the model

- CAD geometry — (- change of the topology)
- FE geometry — (+ no problems with reassignment of BC)

YourProject.hfss
Ansoft HFSS File
2 120 KB

YourProject.hfssresults

\ HFSSDesign*.results\ DV*.cmesh

current.fac
FAC File
26 344 KB

current.lin
LIN File
4 078 KB

current.hyd
HYD File
49 075 KB — Definition of HFSS elements

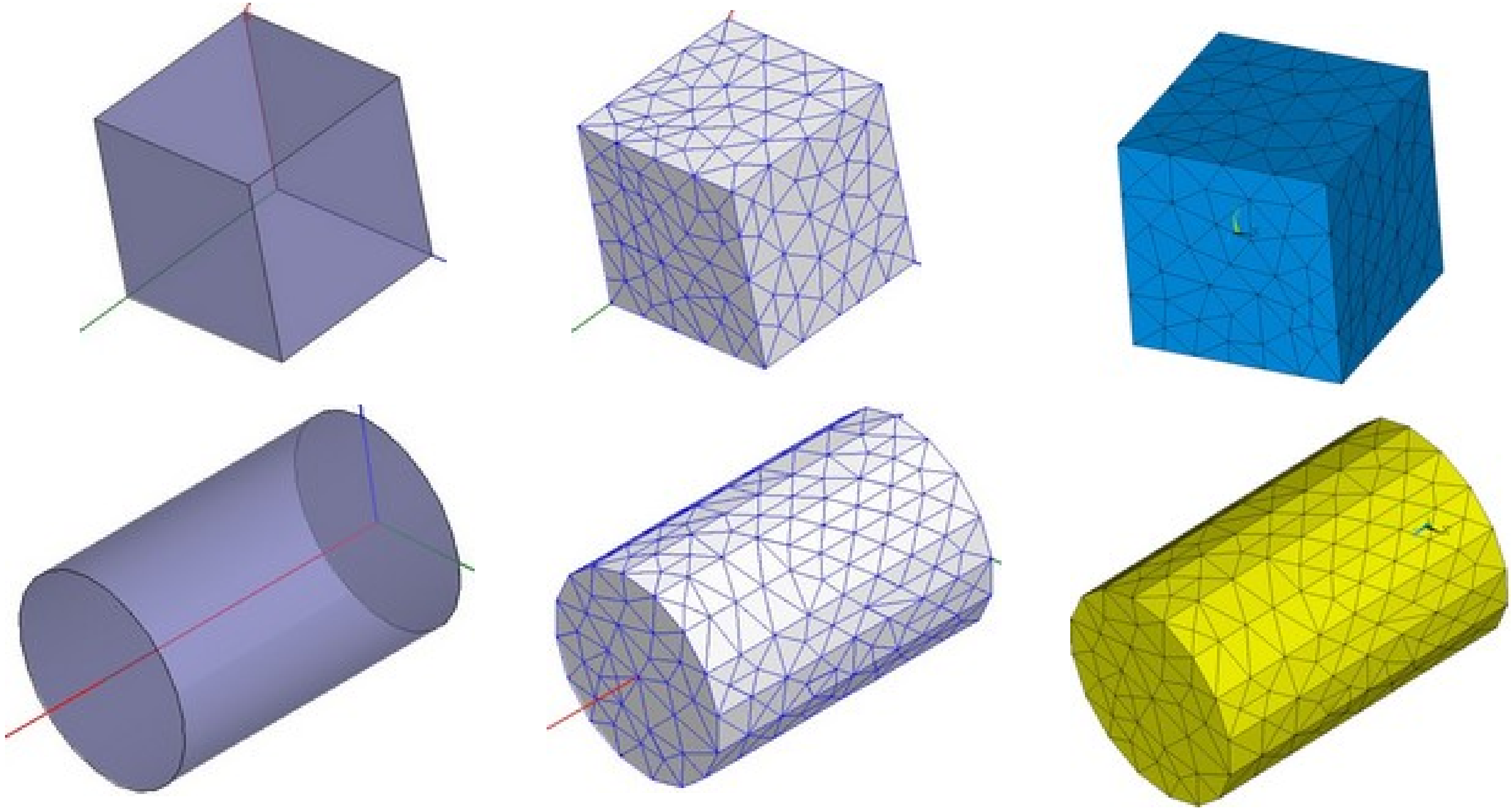
current.plk
PLK File
1 KB

current.sld
SLD File
1 KB

current.pnt
PNT File
4 240 KB — Coordinates of HFSS vertices

Verification of data stored in „current” files

CAD model | **FE model (HFSS)** | **FE „proper” mesh (external code)**

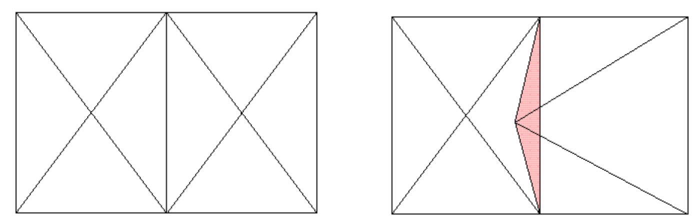


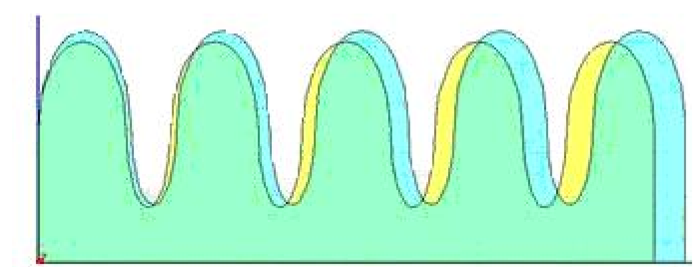
General assumptions of the update process

As in the „HFSS to ANSYS" part, nodes from the ANSYS and HFSS models have different localisation. Hence again the interpolation formula is needed

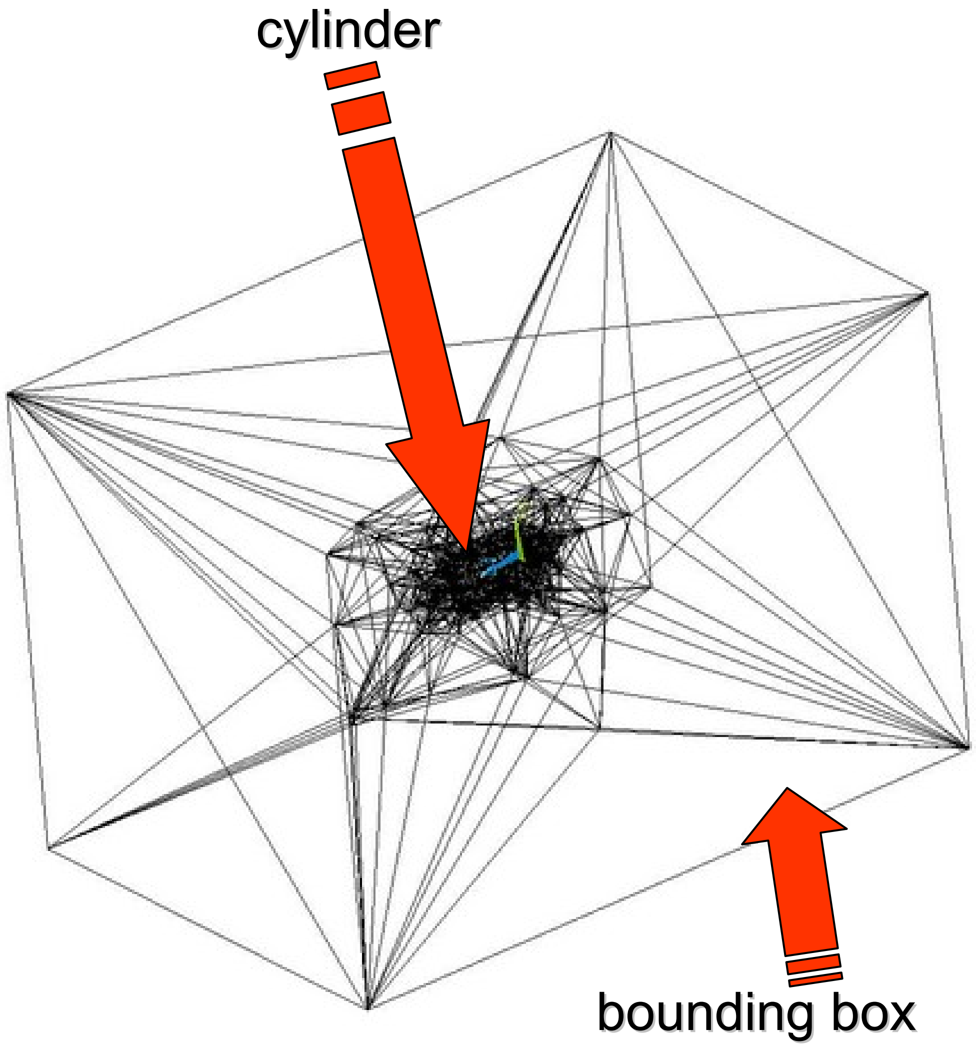
Unfortunately the situation is more complicated than in the previous case:

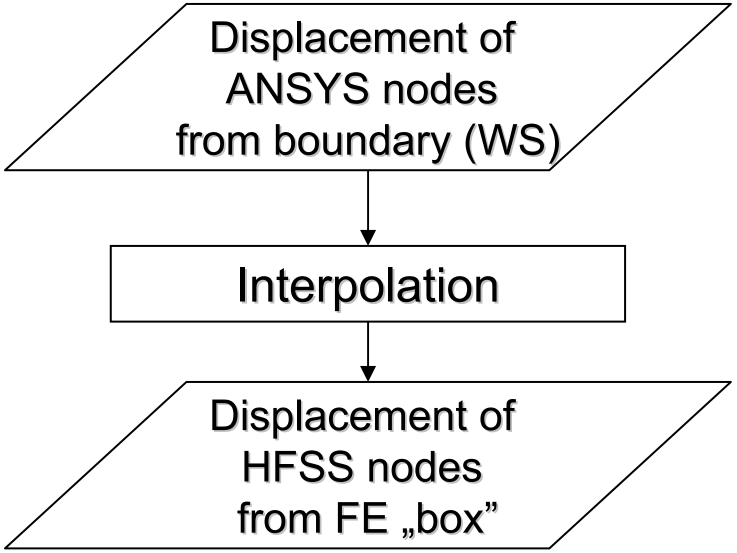
1. The interpolated field is a displacement field which changes position of HFSS nodes. It may lead to degeneration of the mesh.
2. The interpolation process cannot be restricted only to the common boundary (warm surface).





Background mesh – additional complication

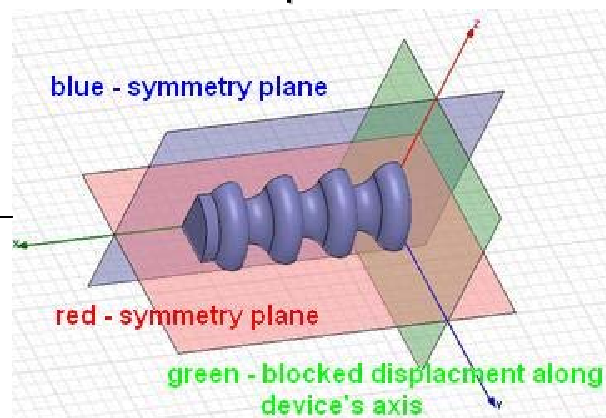




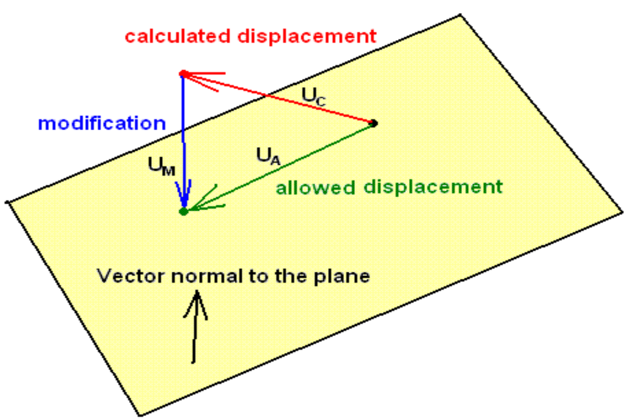
The objective is to stretch the FE mesh without moving nodes which are distant from the common boundary (warm surface).

„Constraining planes”

As a consequence of interpolation scheme which has power to change localisation of all HFSS nodes, a special procedures must be introduced to protect position of some special points. These are for instance points from symmetry planes. On the other hand it does not mean that those points should be definitely locked



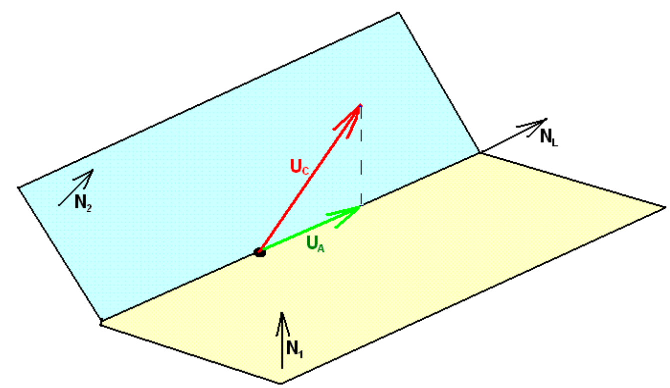
Point belongs to 1 constraining plane



Point belongs to min. 3 constraining planes

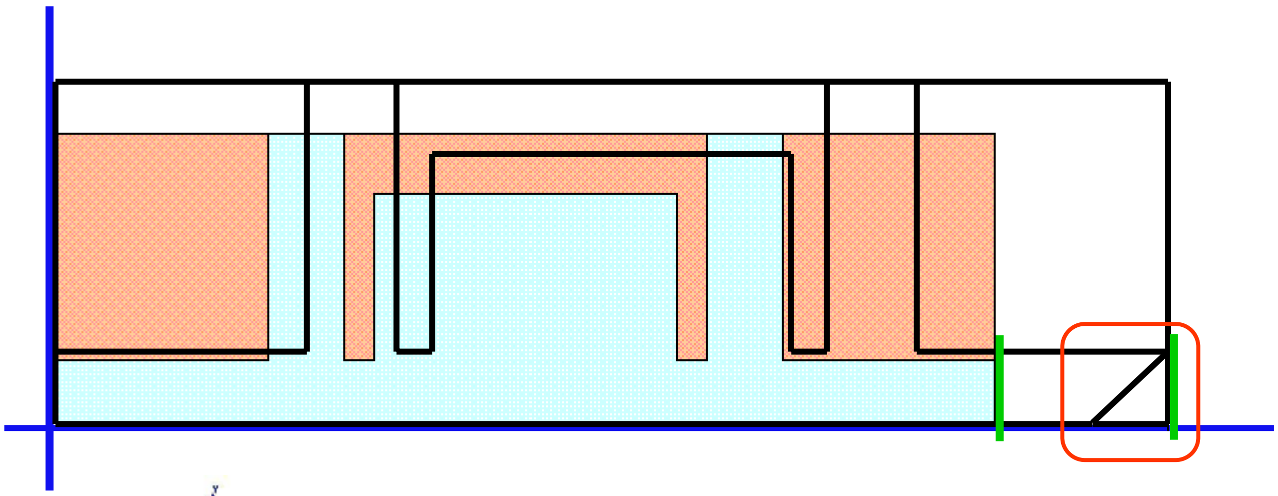
$$\vec{U}_A = \vec{0}$$

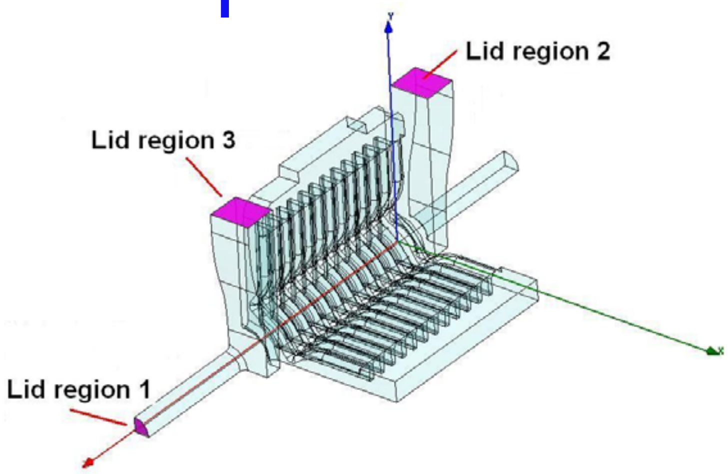
Point belongs to 2 constraining planes



„Lid regions”

Another family of „special” nodes contains those which after the update process are supposed to belong to common plane as they belonged before, while the planes „before” and „after” are not the same.

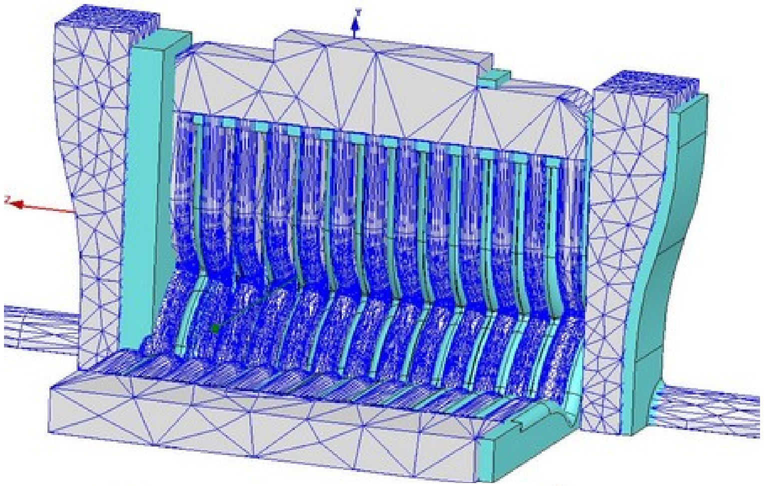


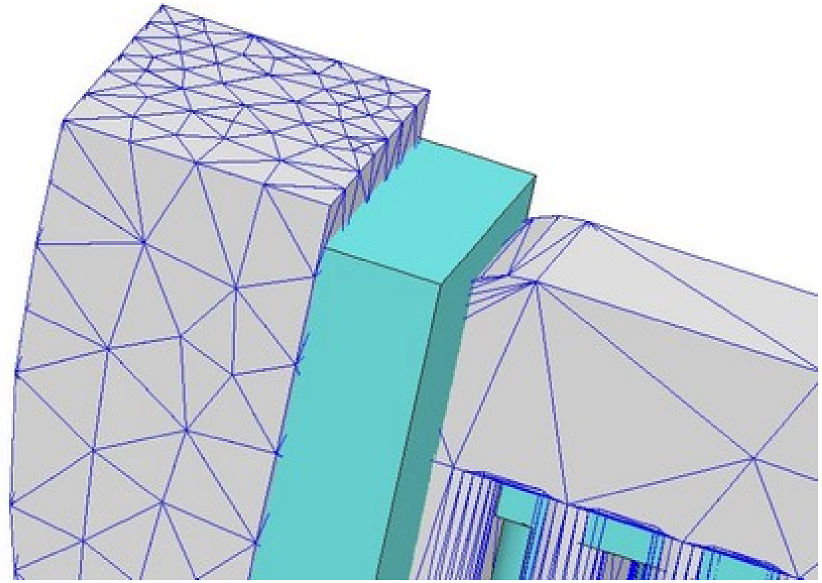


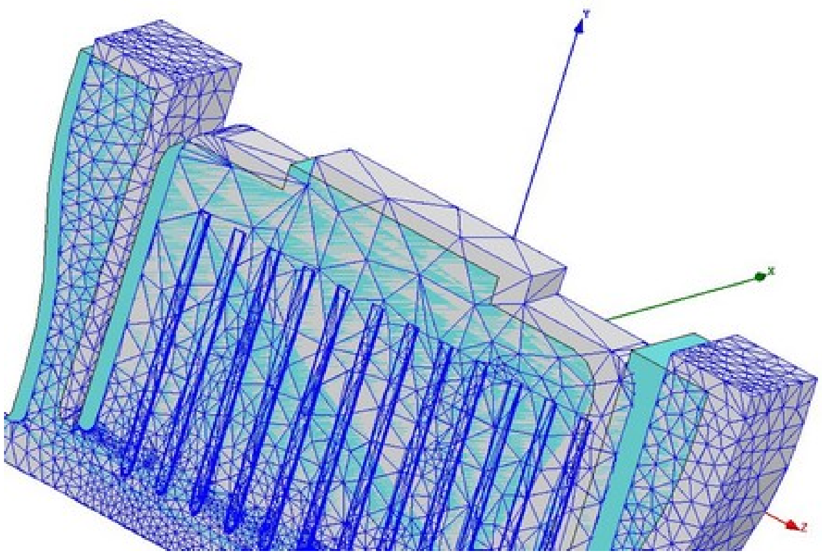
Lid region 1 – outlet of the beam

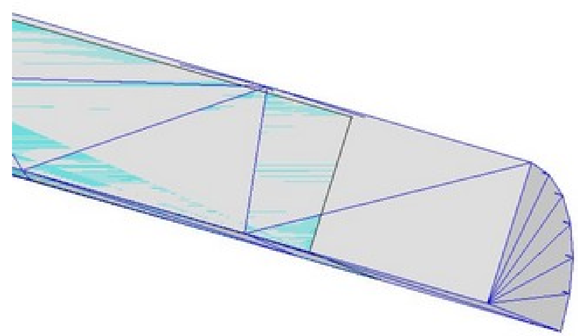
Lid regions 2 & 3 – wave ports 1 & 2

HFSS mesh after update



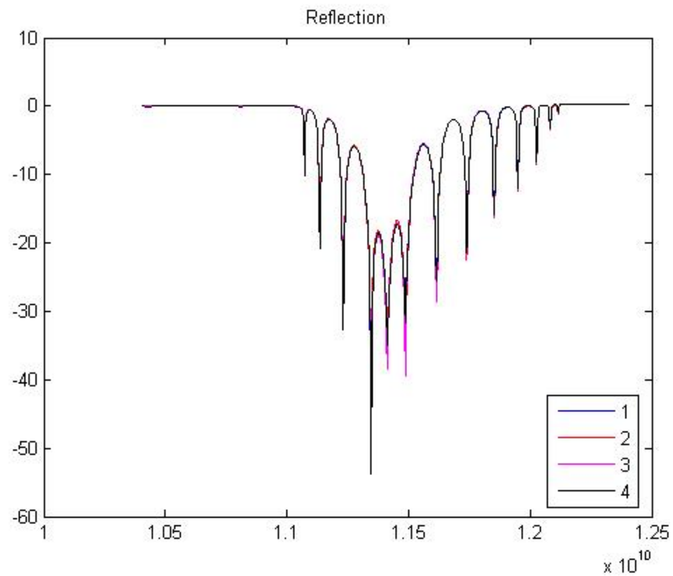


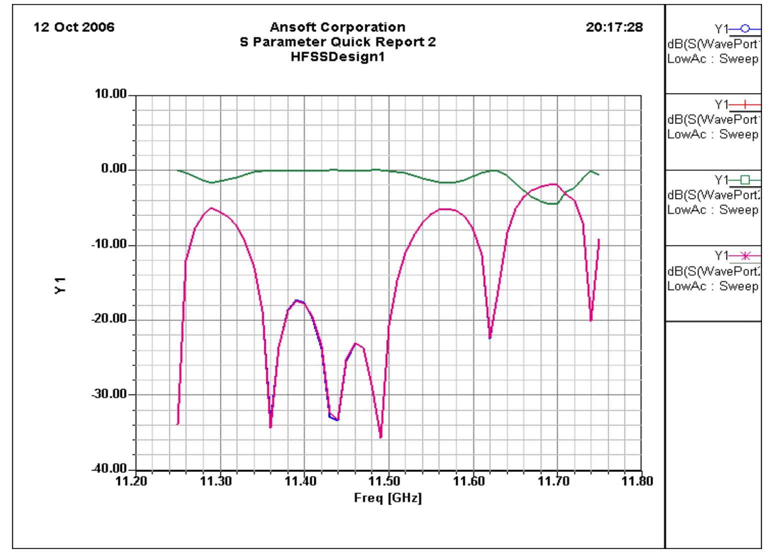


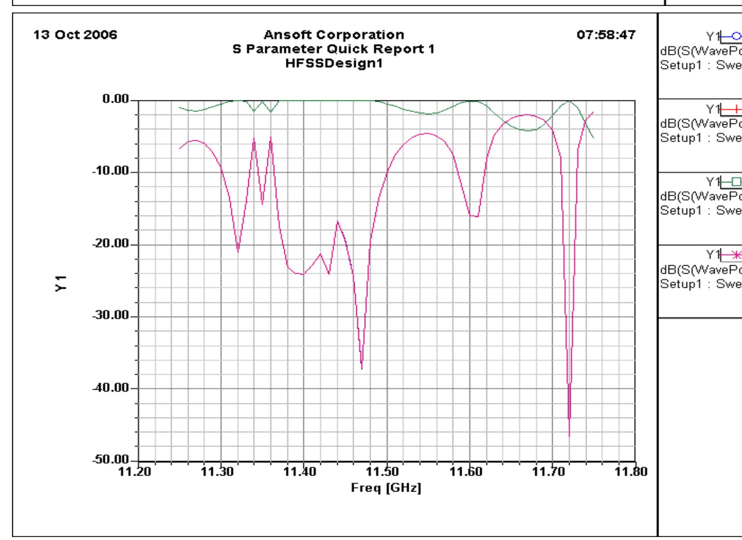


HFSS results before and after deformation

Complex Magnitude of S11 in dB







$$df = -f\,\frac{dL_z}{L_z} =$$

$$= -11.424GHz \cdot \frac{0.3mm}{150mm} = -22.8MHz$$

Things to develop

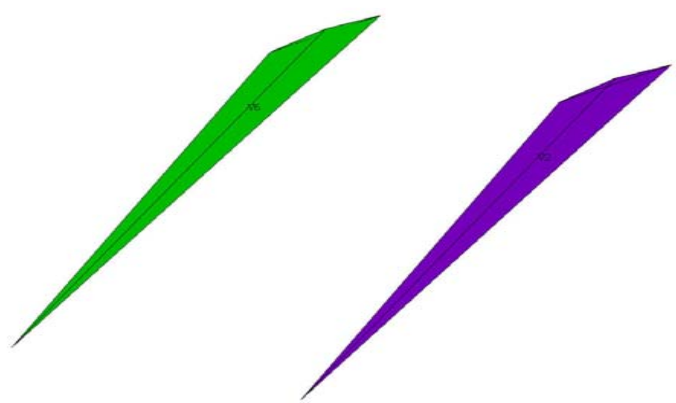
- Transient thermo-mechanical analysis of full model

  (different FE formulations, different codes, etc.)

- Procedures for A2H code with the prevention of mesh degeneration.

Two mechanism which lead to violation of the FE topology

Badly conditioned tetrahedrons in HFSS mesh



Specific design of the RF cavities