FINITE ELEMENT ANALYSIS IN DESIGN OF SYNCHROTRON INSTRUMENTATION – ISSUES, GOOD PRACTICES AND NEW HORIZONS

BRANISLAV BRAJUSKOVIC
Principal Mechanical Engineer
Mechanical Engineering and Design Group
Advanced Photon Source
Argonne National Laboratory

September 12th, 2016
Barcelona

OUTLINE

- A bit of history
- FEA is just a tool, you are the problem solver! (Introduction, kind of)
- Our equipment sometimes fails, do we understand why? (The physics behind the component failures)
- How can we limit/eliminate the failures? (Examples of thermal and structural optimization via FEA)
- NEW Horizons! (Less mainstream applications of FEA)

A BIT OF HISTORY…

…about me

- I am a mechanical engineer,
- I am **NOT** an analyst!
- I have 35+ years of experience in heat transfer engineering and mechanical design,
- I work in the Mechanical Engineering and design Group at APS, ANL since September 2000,
- Currently, I am a principal mechanical engineer working full time on the APS-U project.

Same generation

(deep intra-generational understanding?)

…about FEA

- According to Wikipedia, the origins of FEA can be tracked to China in the **later 1950s and early 1960s** where, based on computations of dam constructions, K. Feng proposed a systematic numerical method for solving partial differential equations

LET THE TALKS BEGIN!

INTRODUCTION

FEA IS JUST A TOOL, YOU ARE THE PROBLEM SOLVER!!

BRAN BRAJUSKOVIC
Principal Mechanical Engineer
Mechanical Engineering and Design Group
Advanced Photon Source
Argonne National Laboratory

September 12th, 2016
Barcelona

WHAT IS FEA?

Wikipedia definition:

- **The finite element method (FEM)** is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It is also referred to as **finite element analysis (FEA)**. **FEM** subdivides a large problem into smaller, simpler, parts, called finite elements.

Autodesk definition:

- **Finite element analysis** (**FEA**) is a computerized method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow, and other physical effects. **Finite element analysis** shows whether a product will break, wear out, or work the way it was designed.

Bran's definition:

- **The finite element method (FEM)** is a **tool,** widely used in the design of stuff, that can cause you, depending on how **you** use it, a lot of **good** or a lot of **harm**!

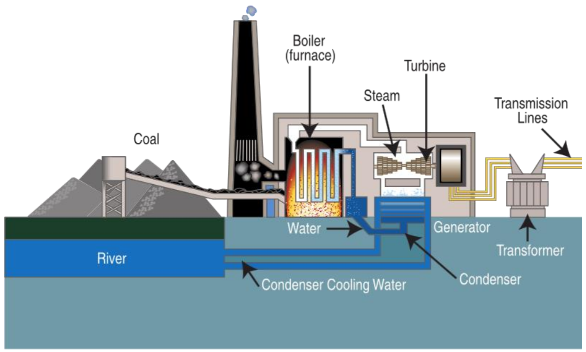
FIVE COMMANDMENTS OF FEA

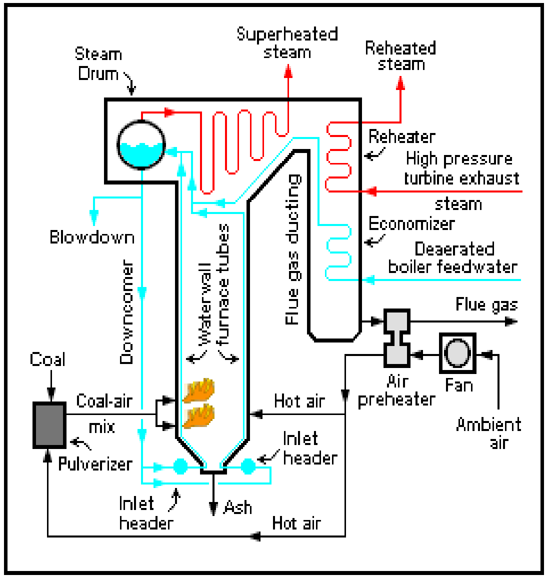
P-H-B-M-R!

- Use your tool properly!
  - Know your **P**hysics!
  - Do your **H**omework!
  - Know the **B**oundaries!
  - Know your **M**aterials!
  - Understand your **R**esults!

FEA COMMANDMENT #1: KNOW YOUR PHYSICS!

Here's an example of FEA analysis implemented to analyze temperatures and stresses in the tubes of the furnaces of large power plant boilers

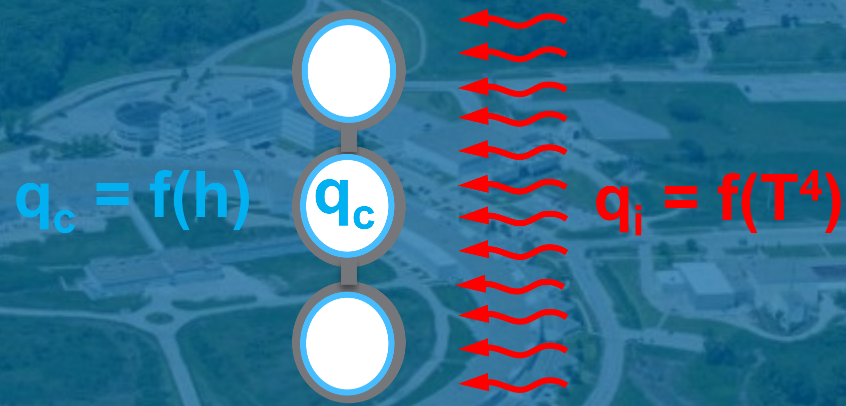




Schematics of power plant and power plant furnace

- **The Physics!!**
  - Power plants produce electricity converting the potential energy of steam into kinetic energy in the turbines and then into the electricity in the generators
  - Steam is generated by evaporating water in the furnaces of the boilers. The generated Steam is then superheated in the superheaters
  - The heat needed for evaporation and superheating is generated by combustion of fossil fuels
  - The generated heat is transferred from the products of combustion to the boiler tubes by radiation and convection
    - Radiation is the primary mode of heat transfer in the furnaces
    - Convection is the primary mode of heat transfer in the rest of the boiler
  - Water and steam are convectively heated while flowing through the tubes

KNOW YOUR PHYSICS—DEFINE THE PROBLEM!



FEA COMMANDMENT #2: DO YOUR HOMEWORK!

Here's an example of FEA analysis implemented to analyze temperatures and stresses in the tubes of the furnaces of large power plant boilers

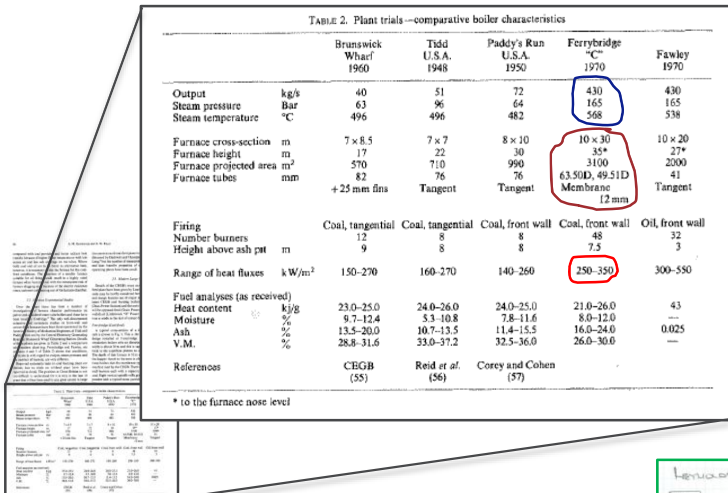
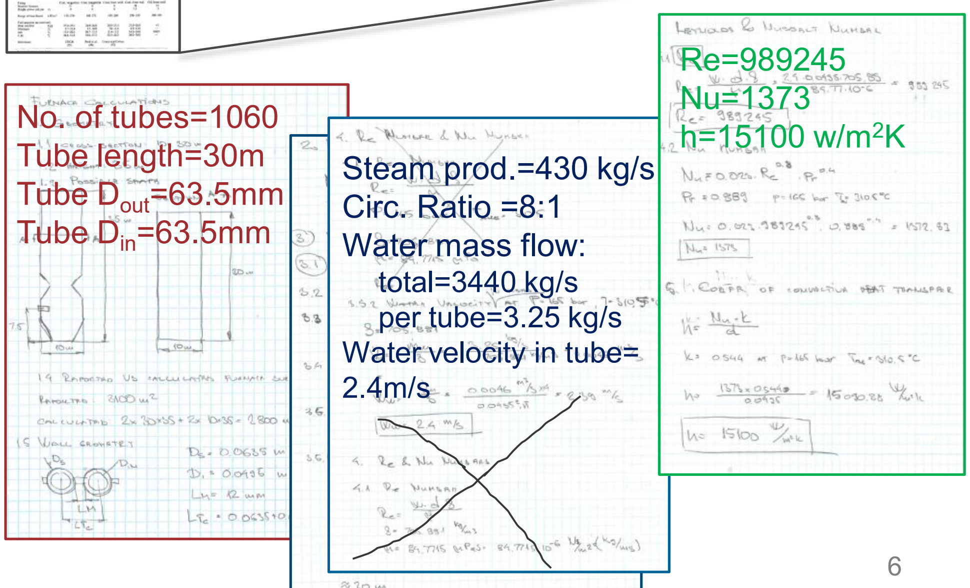


Table 2. Plant trials—comparative boiler characteristics

		Brunswick Wharf 1960	Tidd U.S.A. 1948	Paddy's Run U.S.A. 1950	Ferrybridge "C" 1970	Fawley 1970
Output	kg/s	40	51	72	430	430
Steam pressure	Bar	63	96	64	165	165
Steam temperature	°C	496	496	482	568	538
Furnace cross-section	m	7 × 8.5	7 × 7	8 × 10	10 × 30	10 × 20
Furnace height	m	17	22	30	35*	27*
Furnace projected area	m^2	570	710	990	3100	2000
Furnace tubes	mm	82 + 25 mm fins	76 Tangent	76 Tangent	63.5OD, 49.5ID Membrane 12 mm	41 Tangent
Firing		Coal, tangential	Coal, tangential	Coal, front wall	Coal, front wall	Oil, front wall
Number burners		12	8	8	48	32
Height above ash pit	m	9	8	8	7.5	3
Range of heat fluxes	kW/m^2	150–270	160–270	140–260	250–350	300–550
Fuel analyses (as received)						
Heat content	kj/g	23.0–25.0	24.0–26.0	24.0–25.0	21.0–26.0	43
Moisture	%	9.7–12.4	5.3–10.8	7.8–11.6	8.0–12.0	—
Ash	%	13.5–20.0	10.7–13.5	11.4–15.5	16.0–24.0	0.025
V.M.	%	28.8–31.6	33.0–37.2	32.5–36.0	26.0–30.0	—
References		CEGB (55)	Reid et al. (56)	Corey and Cohen (57)		

* to the furnace nose level



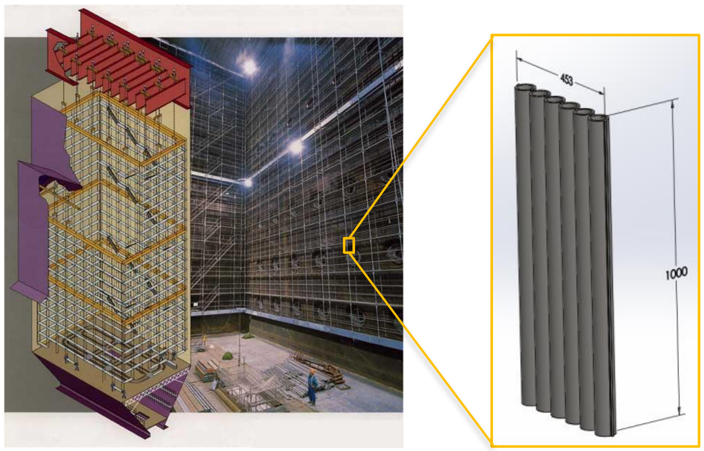
- **The Homework!!**
  - Heat is transferred from the products of combustion to the outer surfaces of the furnace tube walls primarily by radiation:
    - Exact computations possible but very time consuming and CFD module is needed,
    - Luckily, there is plenty of experimental data on radiative heat fluxes in furnaces
  - Heat is transferred from the inner surfaces of the furnace tube walls to the water running thru them by convection,
    - Exact computations possible and less time consuming than in the previous step, CFD module still required,
    - Number of semi-empirical equations are available for calculation of convective heat transfer coefficient from tube walls to the water

DO YOUR HOMEWORK—QUANTIFY THE PROBLEM!

FEA COMMANDMENT #3: KNOW THE BOUNDARIES!

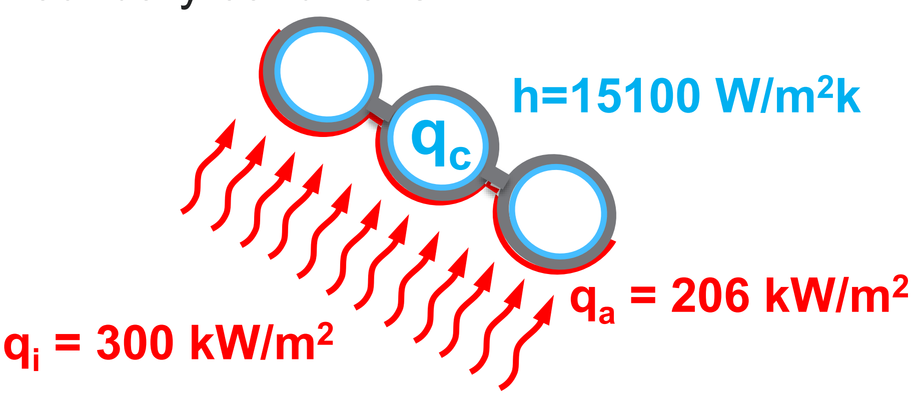
In order to start FEA analysis we need the geometry (2D or 3D model) and the boundary conditions (loads and constrains)

Geometry



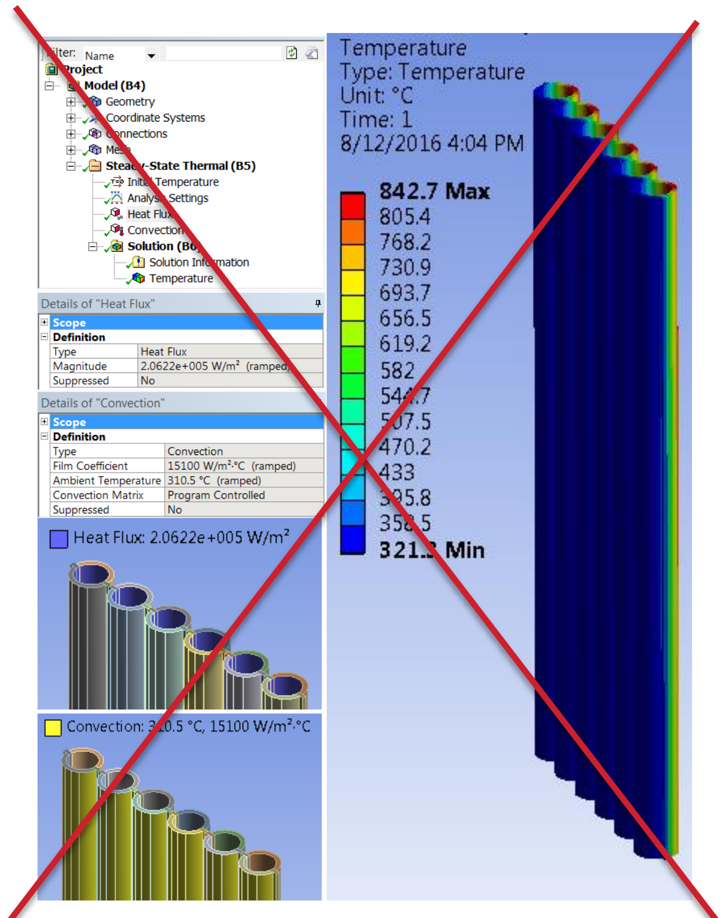
- The geometry can be simplified
  - Although the boiler furnaces are very large objects with a relatively complex geometry, the model used in analysis can be very simple
- The boundary conditions for thermal analysis ARE simple
  - As the heat is transferred from the hot products of combustion to the tube walls, almost exclusively by radiation, the heat absorbed by the tube walls equals the incident heat
  - The cooling is fully defined by the coefficient of the convective heat transfer and the water temperature

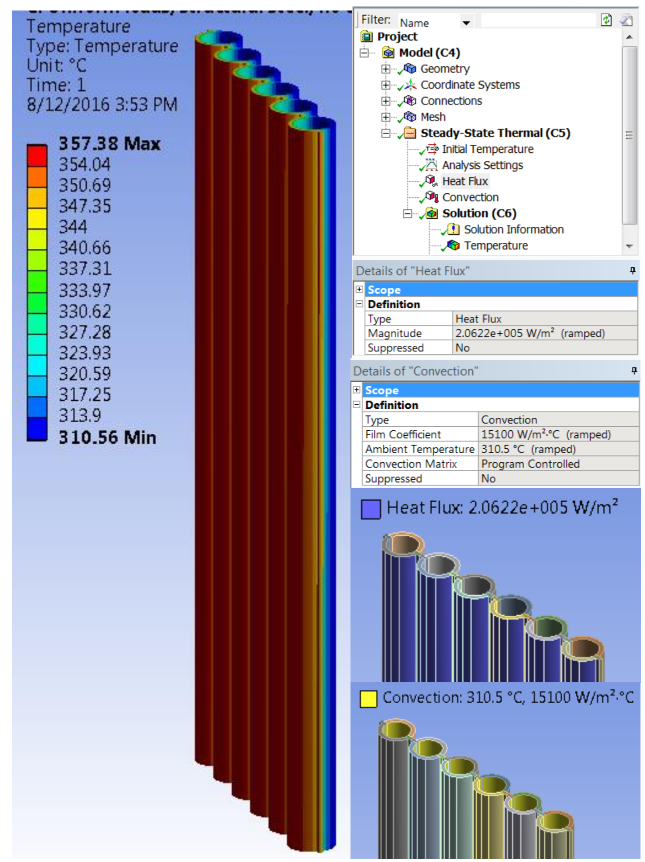
Boundary conditions



FEA COMMANDMENT #3: KNOW THE BOUNDARIES!

Be careful when applying boundary conditions!

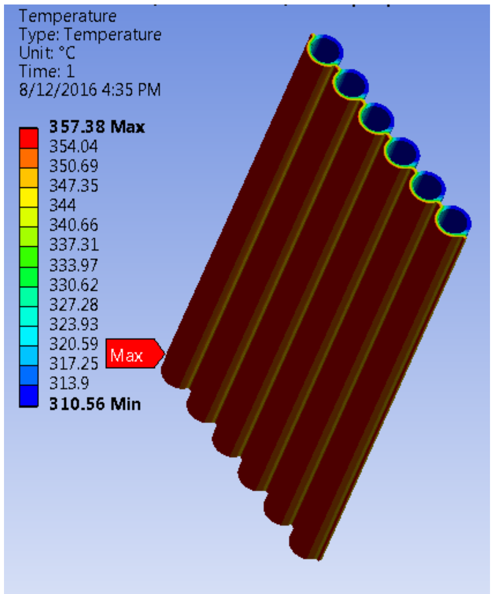


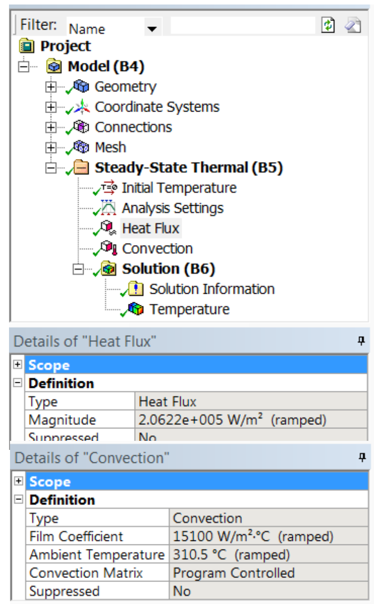


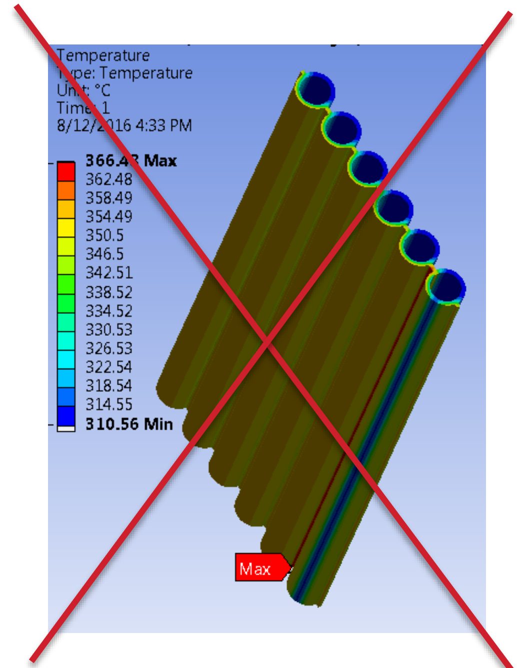
We can apply boundaries with identical values, and get very different results if we apply them to different areas of the model!

FEA COMMANDMENT #3: KNOW THE BOUNDARIES!

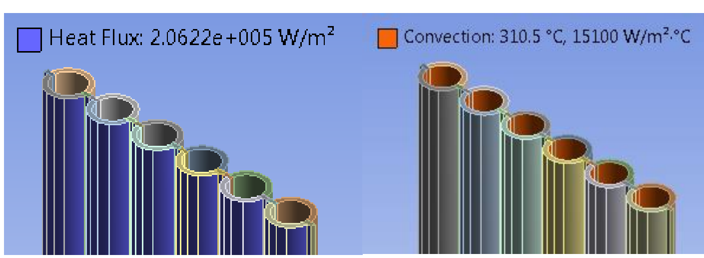
Very, very careful!

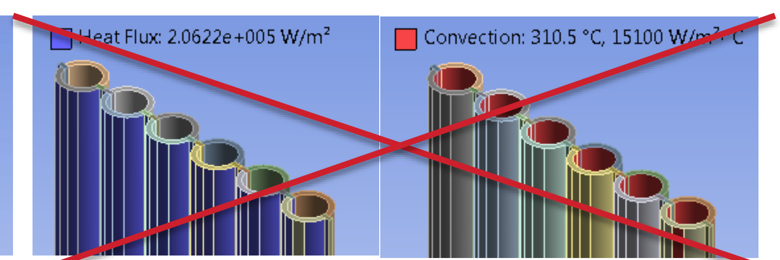






- Sometimes errors come from convenience features of the software!
- Even one misplaced load out of fifty can make a significant difference

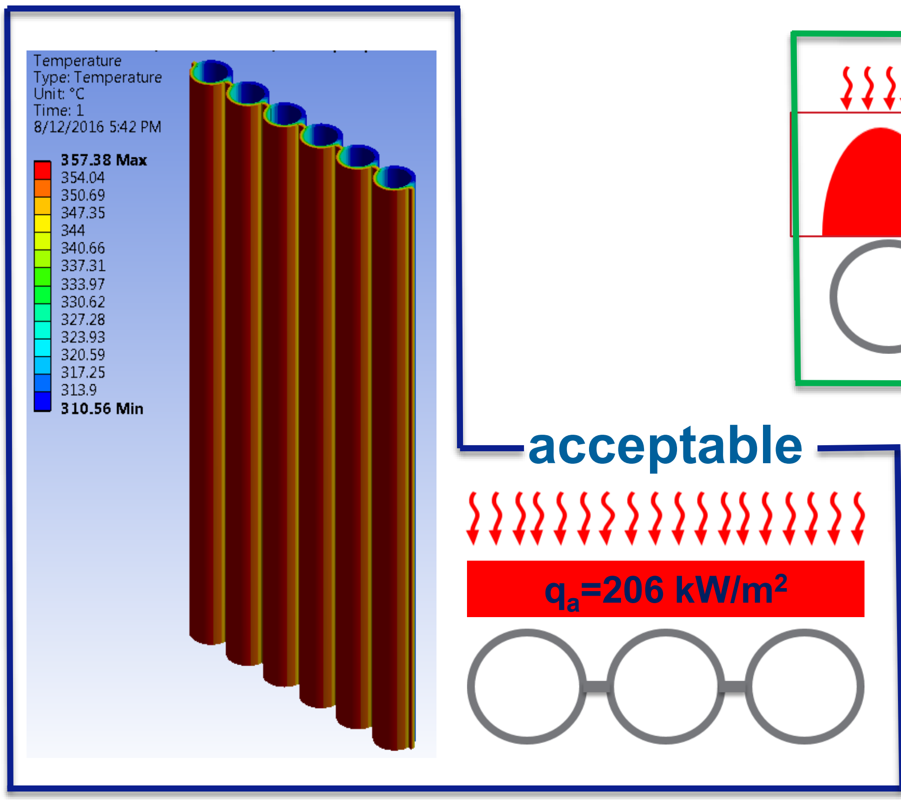


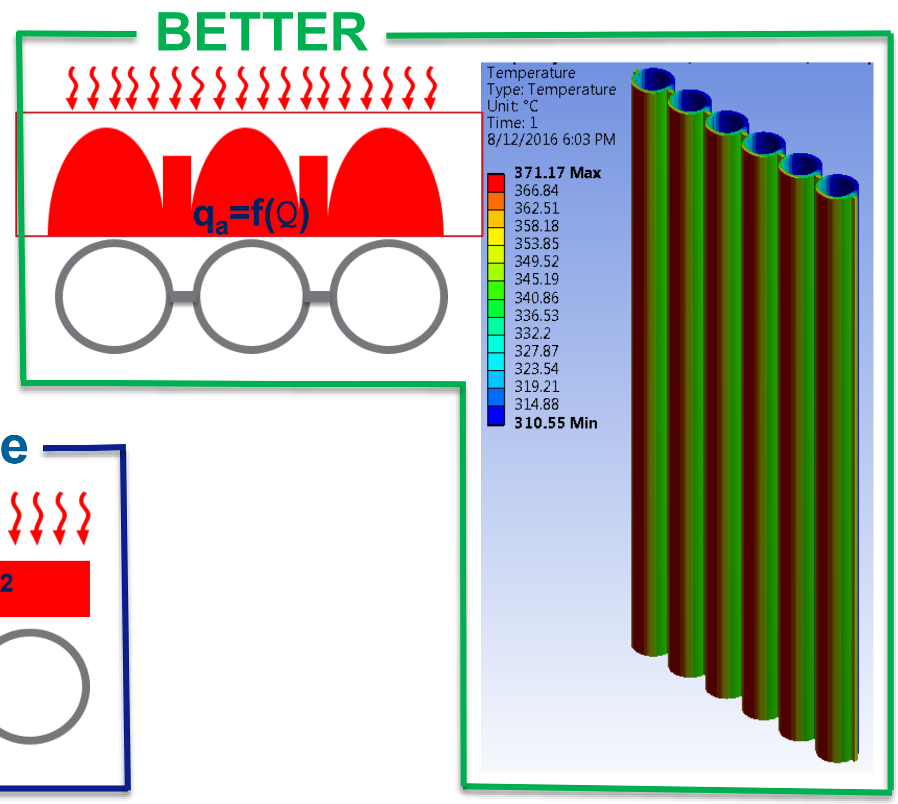


Argonne NATIONAL LABORATORY

FEA COMMANDMENT #3: KNOW THE BOUNDARIES!

Back to Physics, know your boundaries REALLY well!

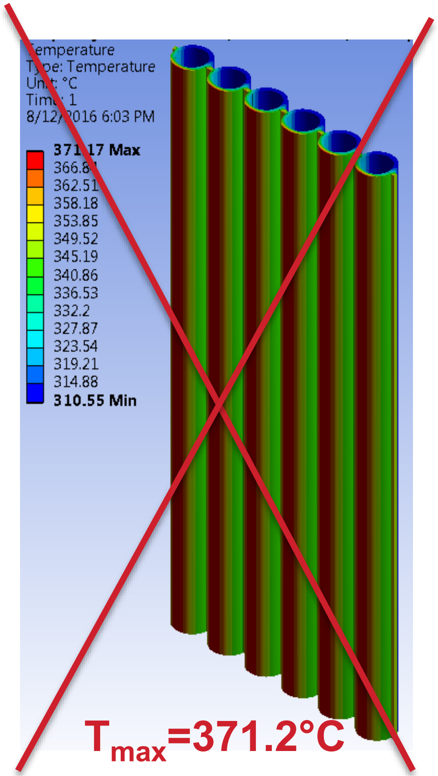




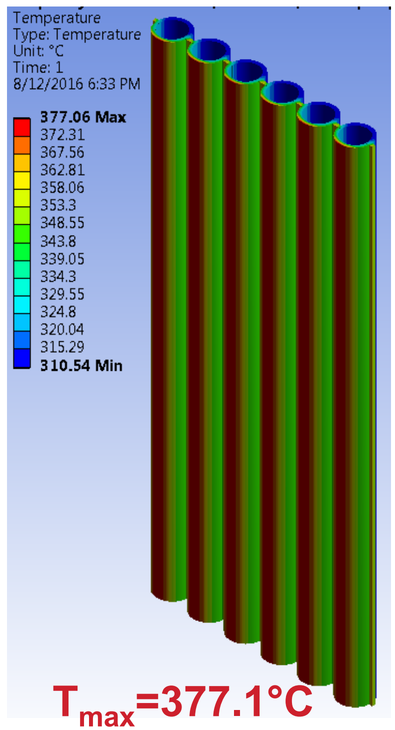
The more accurately our boundaries represent real loads the more accurate results we will get!

FEA COMMANDMENT #4: KNOW YOUR MATERIALS!

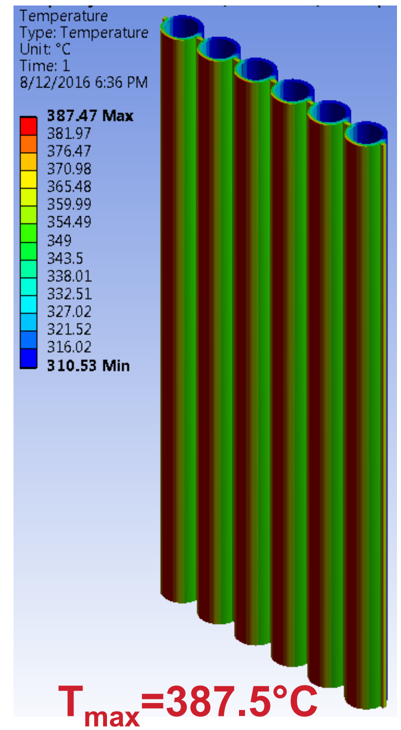
Yes, MATERIALS!!



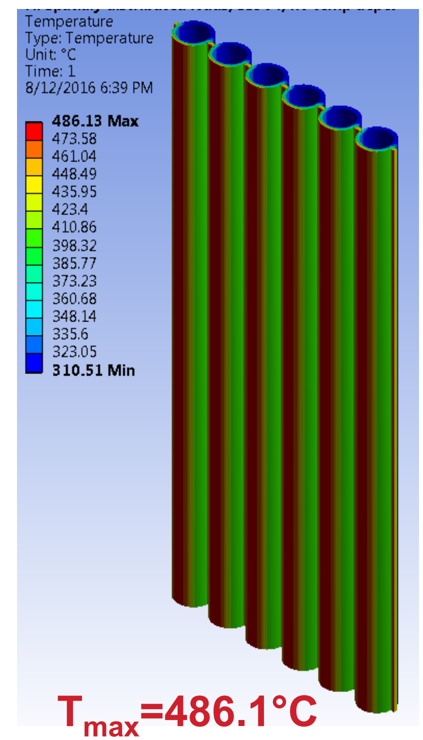
Construction Steel



A 106 Gr. B



1Cr 0.5Mo

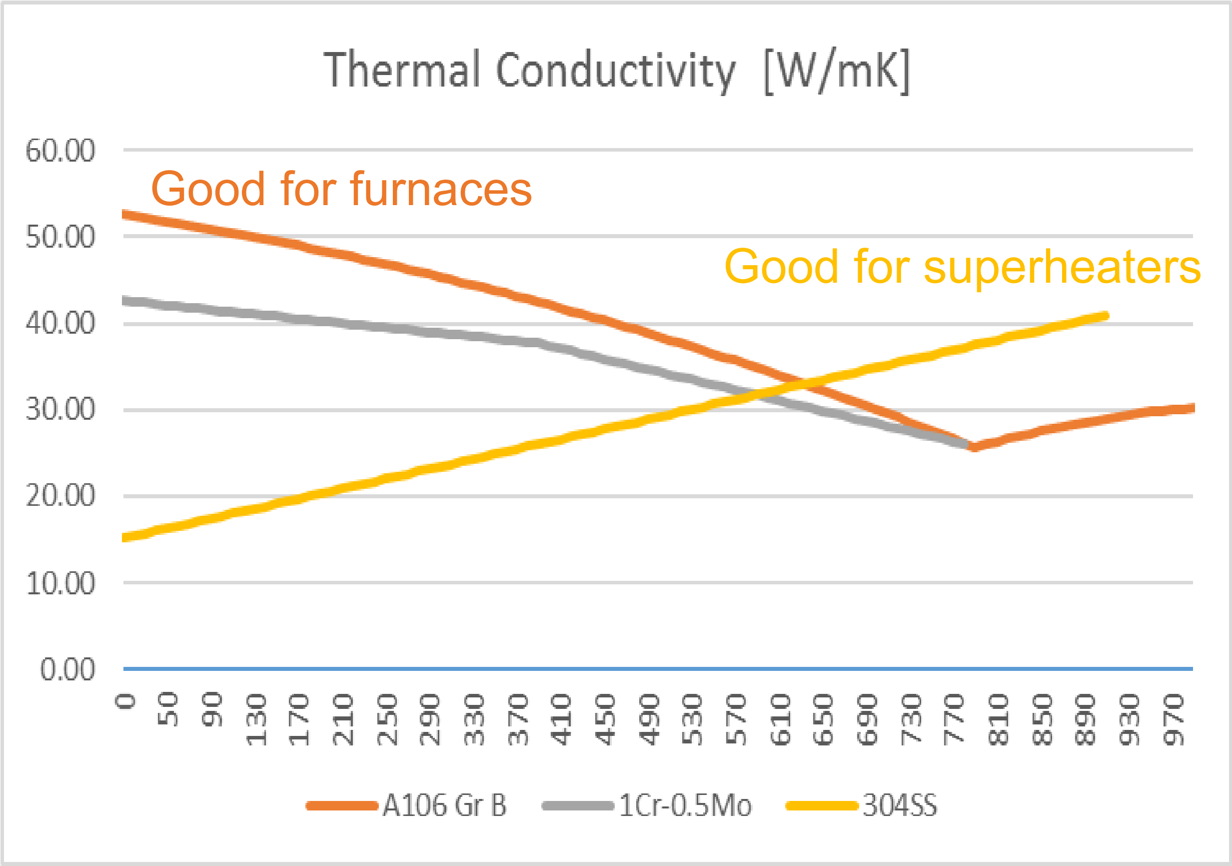


SS 304H

Your best bet

- **relatively cheap carbon steel**
- **higher thermal conductivity**

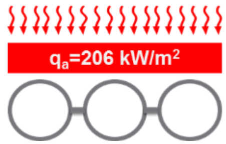
FEA COMMANDMENT #4: KNOW YOUR MATERIALS!

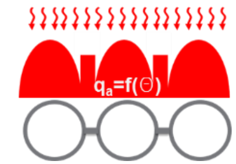


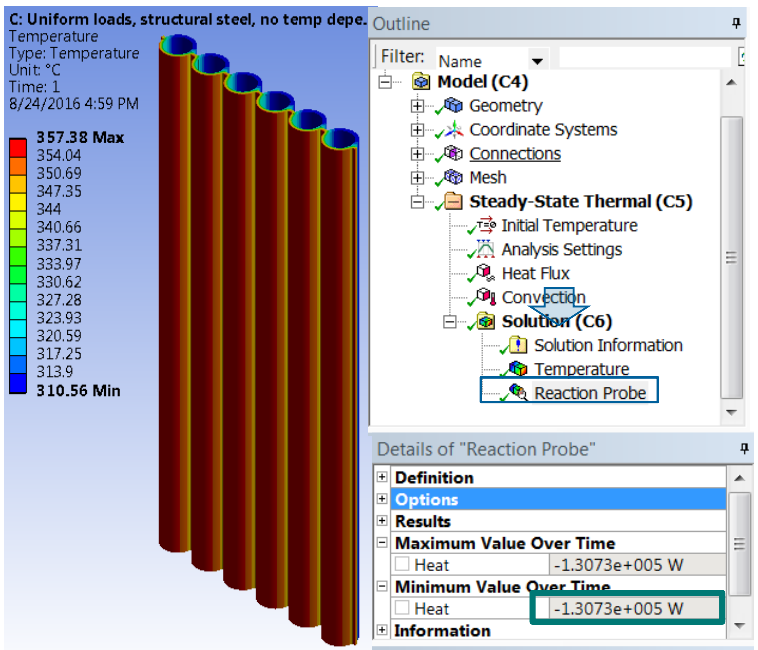
A material's properties can define its range of use

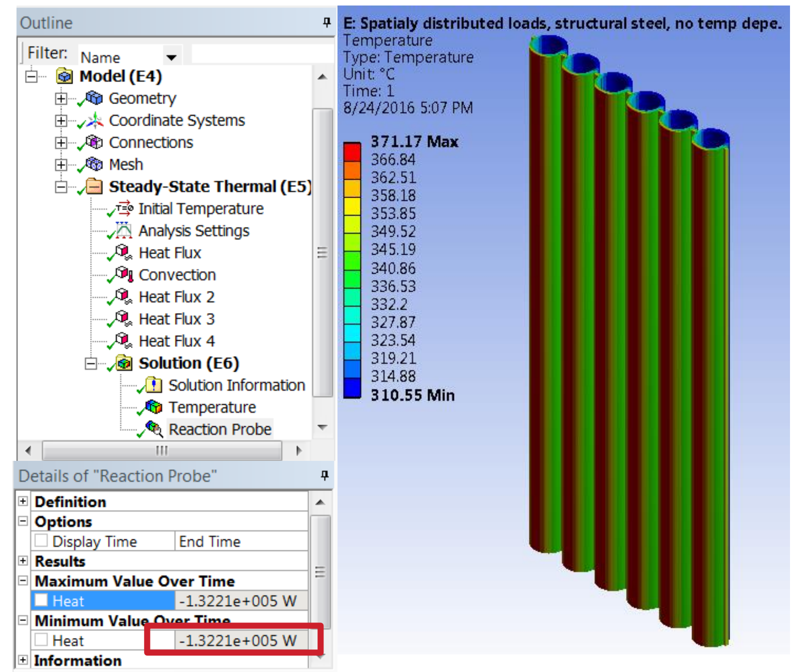
FEA COMMANDMENT #5: UNDERSTAND YOUR RESULTS!

Do the cross-checking!









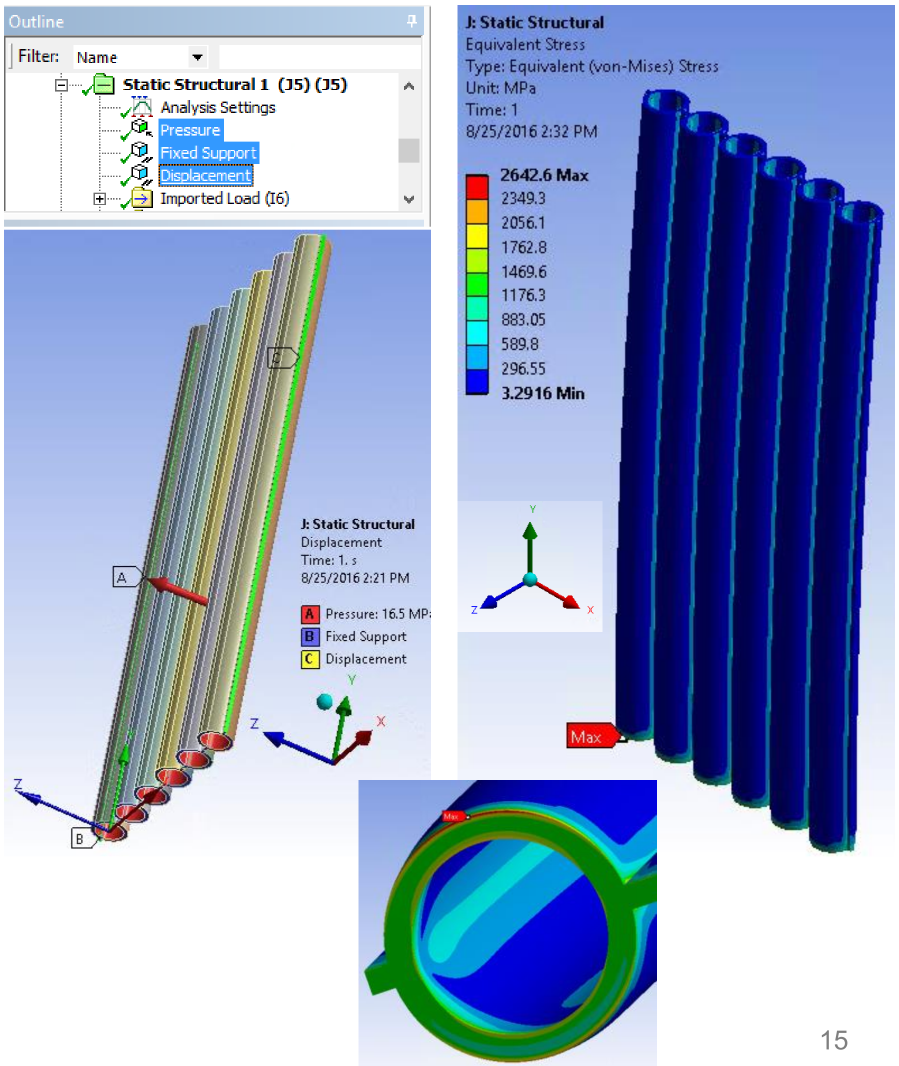
- From the Homework and the boundaries:
  - q_{inc}= 300kW/m² ➡ q_a= 206 kW/m²
  - S_a= 0.633 m²

} Q_a= 130.4 kW/m² ≈ **130.7 kW/m²** ≈ **132.2 kW/m²**

Argonne NATIONAL LABORATORY

FEA COMMANDMENT #5: UNDERSTAND YOUR RESULTS!

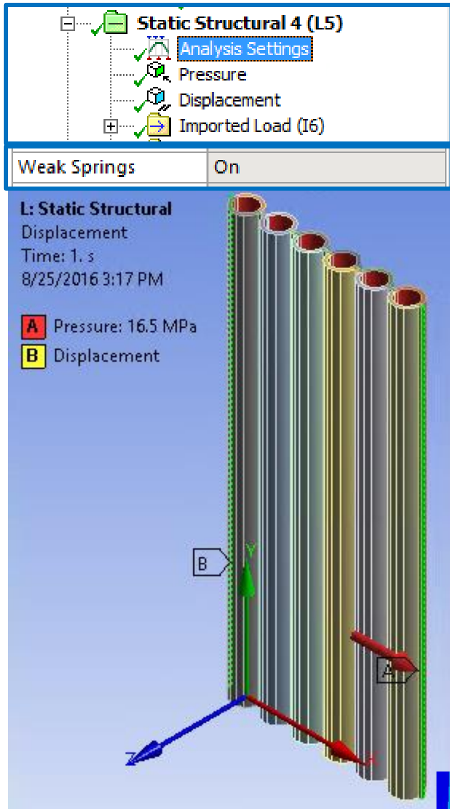
Understand the effect of the boundaries!

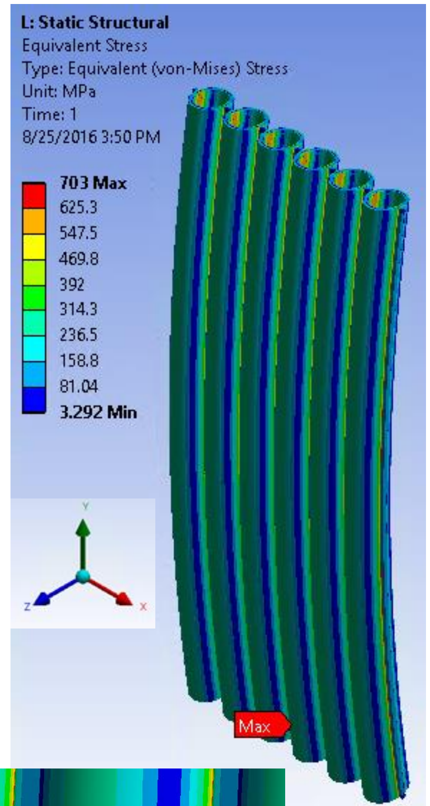


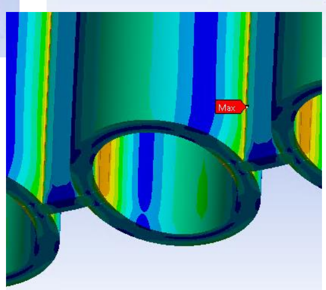
- Loads and boundary conditions:
  - Uniform pressure on the inner tube walls (representing the hydrostatic pressure of the water flowing through the tubes)
  - Previously computed temperature distribution
  - Fixed support applied to bottom surfaces (representing the supportive action of the lower portion of the furnace wall)
  - Displacement Dx=0 applied to two end surfaces at each side of the model (representing the neighboring parts of the furnace wall)
- Computed stresses:
  - s_{max}= 2642.6 Mpa,
  - **10x** the s_y for the material
  - The maximum stress levels are on the bottom surfaces
  - The bottom surfaces are unrealistically constrained as fixed surfaces can't thermally expand!

FEA COMMANDMENT #5: UNDERSTAND YOUR RESULTS!

Understand the effect of the boundaries! - continued



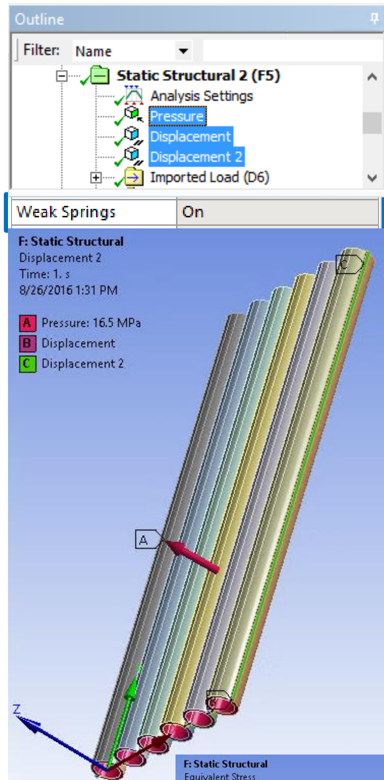


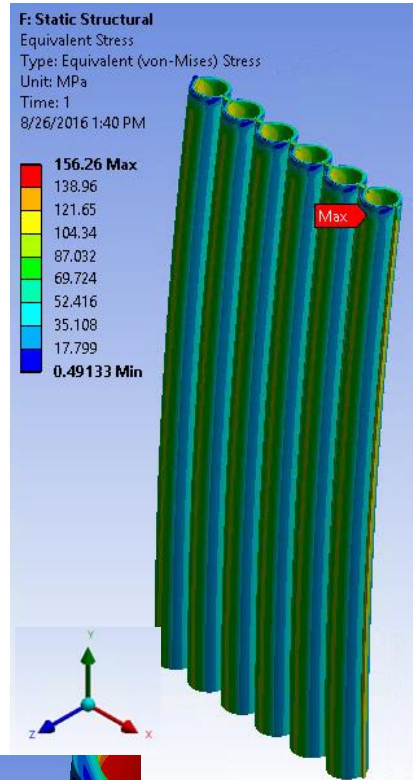


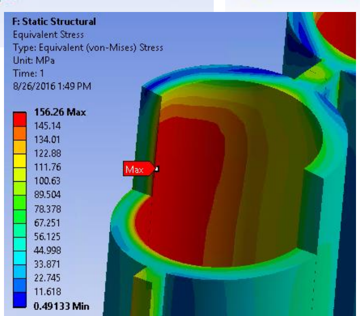
- Loads and boundary conditions:
  - Uniform pressure on the inner tube walls
  - Previously computed temperature distribution
  - Displacement Dx=0 applied to two end surfaces at each side of the model
  - A weak spring option in the analysis is turned on to prevent rigid body motion
- Computed stresses:
  - s_{max}= 703 Mpa
  - Max. computed stresses are 3x the s_y for the material
  - Max. computed stresses are in the areas where the membrane fins are in contact with the tubes
  - The elimination of the ability for walls to expand in x direction (Dx=0) is too restrictive

FEA COMMANDMENT #5: UNDERSTAND YOUR RESULTS!

Understand the effect of the boundaries! – (even more) continued



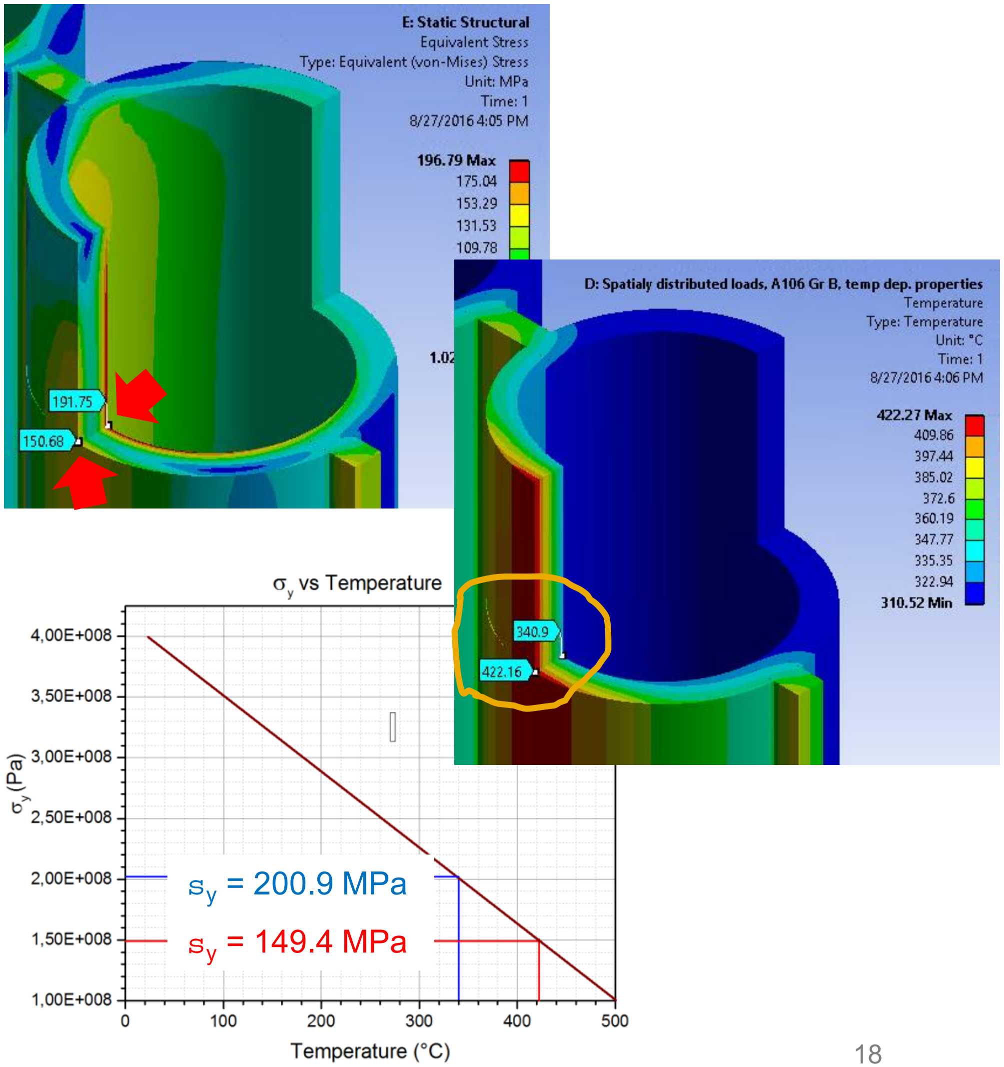




- Loads and boundary conditions
  - Uniform pressure on the inner tube walls
  - Previously computed temperature distribution
  - Displacement Dx=0 applied to only one surface (the structure can expand in that direction)
  - Displacement Dy=0 applied to the bottom of the model (the structure can expand in that direction too)
  - Weak springs turned on
- Computed stresses
  - s_{max}= 156 Mpa,
  - maximum computed stresses are 0.6x the s_y for the material
  - Maximum values computed on the inner surface of the tubes on the heated side
  - Eliminating the expansion restrictions created more realistic boundaries and results

FEA COMMANDMENT #5: UNDERSTAND YOUR RESULTS!

Analyze results in the full context of your engineering problem!



- Always analyze your results having in mind all aspects of the problem,
- This is particularly important in the coupled analyses,
- In the shown example of coupled thermo-mechanical analysis
  - Stress analysis indicates that the inner surface of the tube can locally have stresses up to 30% higher than the outer surface,
  - However, inner surface is at 80 ◦C lower temperature in the same area, and
  - Temperature dependent yield strength of the tube material indicates that outer surface could be plastically deformed!

FEA DOES NOT STAND FOR DEUS EX MACHINA

- YOU ARE THE ONE WHO SOLVES THE PROBLEM!
- EVERYBODY MAKES MISTAKES!
- FEA MIGHT MAKE YOUR MISTAKES LOOK SEXY BUT IT WON'T FIX THEM!
- BE PREPARED! THE MORE YOU ARE PREPARED, LESS MISTAKES YOU WILL MAKE!
- SCRUTINIZE YOUR RESULTS! THE MORE YOU STUDY THEM, EASIER WILL BE TO CATCH THE MISTAKES!

QUESTIONS?

OUR EQUIPMENT SOMETIMES FAILS, DO WE UNDERSTAND WHY?

BRAN BRAJUSKOVIC
Principal Mechanical Engineer
Mechanical Engineering and Design Group
Advanced Photon Source
Argonne National Laboratory

September 12th, 2016
Barcelona

CAUSES BEHIND OUR COMPONENT FAILURES

Outline

- What makes the design of synchrotron components distinct?
  - Dealing with particle and photon beams,
  - Handling high power densities and spatial heat flux distribution,
  - Understanding the failure mechanisms caused by thermal stresses,
  - Limitations of the design for ultrahigh vacuum components.
- What are the consequences?
  - Coupled thermo-mechanical analysis in almost all cases,
  - Means for accurate computing thermal loads needed,
  - Model meshes capable of accurately capturing thermal loads necessary.

WHAT MAKES THE DESIGN OF SYNCHROTRON COMPONENTS DISTINCT?

Why are our designs ‘special’?

- The majority of the components that we design have to contain or condition synchrotron beams
  - Ring and straight section components deal with particle and/or photon beams
  - Front end and beamline components deal exclusively with photon beams
- Both particle and photon beams are characterized by high power and small dimensions
  - High power and small dimensions result in very high power densities
  - The power distribution is spatially non-uniform
  - The contact between the high power density beams and the components that contain or condition those beams results in very localized heating of the components
- In order to minimize scattering, the synchrotron beams travel in vacuum
  - Particle beams travel exclusively in ultra high vacuum (UHV)
  - Photon beams travel either in high or ultrahigh vacuum (and sometimes in air but that is an entirely different story).

WHAT MAKES THE DESIGN OF THE SYNCHROTRON COMPONENTS DISTINCT?

What is the cost of being special?

- Localized heating of our components, caused by contact with high power density synchrotron beams, results in:
  - Our components generally operating at elevated temperatures,
    - Thus, our components need cooling.
  - The formation of localized areas with very high temperatures and very high thermal gradients in our components
    - Although high capacity cooling is rarely required, we frequently require high efficiency cooling
    - Thermal stresses are the major contributor to stress levels in our components
- All of the components that contain or condition synchrotron beams are either HV or UHV components, and:
  - They are exposed to the vacuum force (this is easy to forget),
  - The choice of the materials available for our designs is limited in most cases to:
    - Stainless steel
    - Aluminum
    - Copper

THERMAL STRESSES ARE THE MAJOR CONTRIBUTOR TO THE STRESS LEVELS IN OUR COMPONENTS!

What are thermal stresses?

- Thermal stresses are a consequence of the tendency of the materials to either expand or contract with the change of temperature
- Thermal stresses occur if restrictions on thermal expansion or contraction are imposed by:
  - Continuity of the body or
  - The conditions at the boundaries
- In the absence of constraints, thermal stresses are self-equilibrating

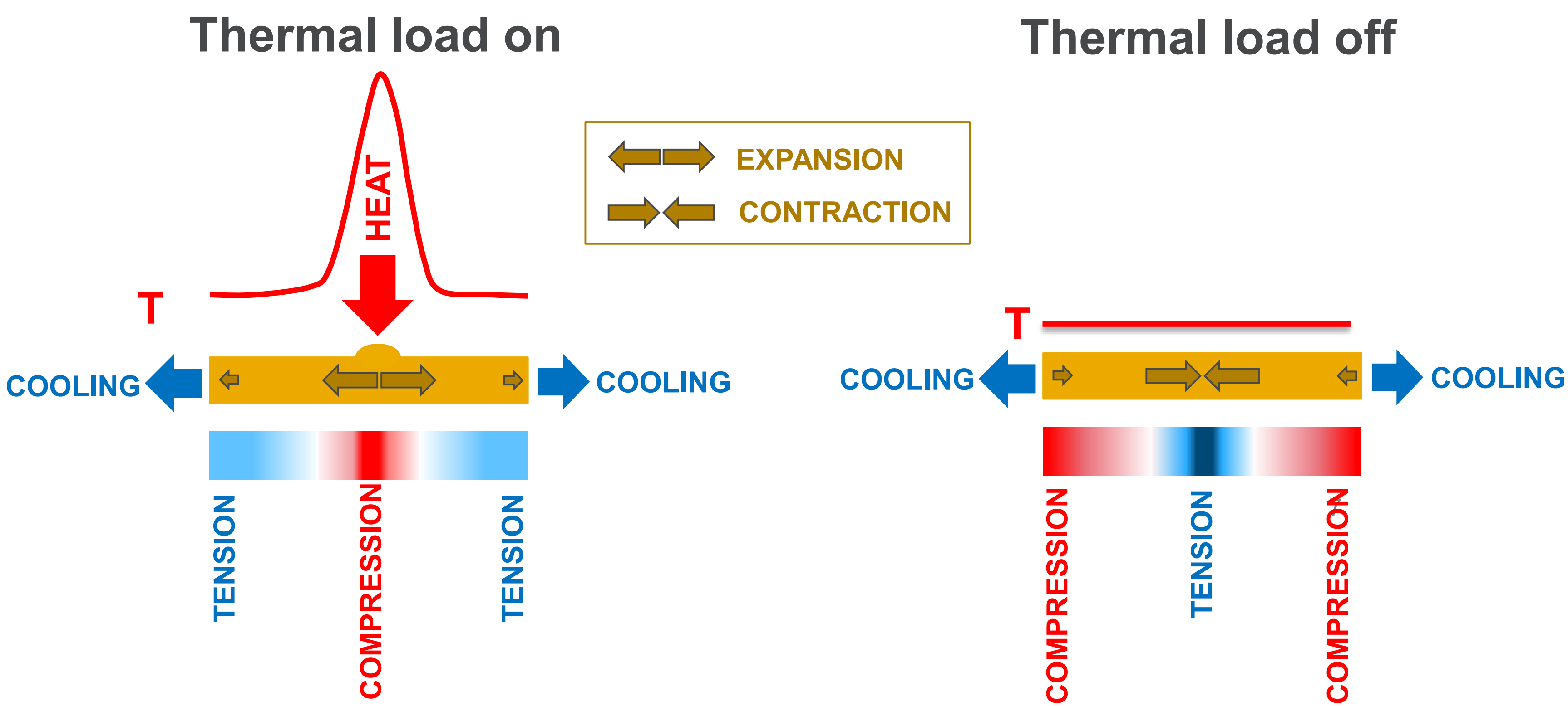
THERMAL STRESSES ARE THE MAJOR CONTRIBUTOR TO THE STRESS LEVELS IN OUR COMPONENTS!

Under what conditions do thermal stresses occur?

- In bodies made of a single material, thermal stresses occur if there is a thermal gradient throughout the material
  - Material in colder areas of the body will expand less or contract more and constrain the expansion/contraction of the material in the hotter areas
  - As a consequence of the constraining action, colder areas will be stressed in tension while hotter areas will be stressed in compression
- In bodies made of different materials bonded together, thermal stresses occur even at a uniform temperature (different than the one at equilibrium) due to different coefficients of thermal expansion
  - A material with a lower coefficient of thermal expansion will expand/contract less than a material with a higher coefficient of thermal expansion
  - As a consequence, the material with a lower coefficient of thermal expansion will be in tension if the temperature is rising and in compression if the temperature is falling

THERMAL STRESSES ARE MAJOR CONTRIBUTOR TO THE STRESS LEVELS IN OUR COMPONENTS!

A seemingly paradoxical nature of material failure due to the thermal stresses – Part 1

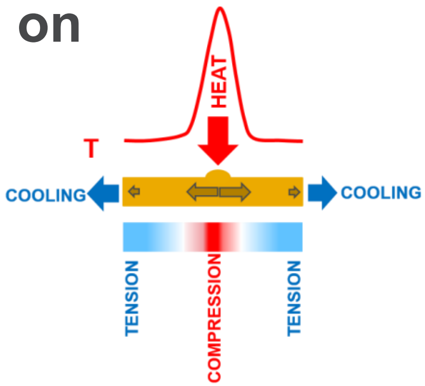


The behavior of a monolithic body during a thermal cycle

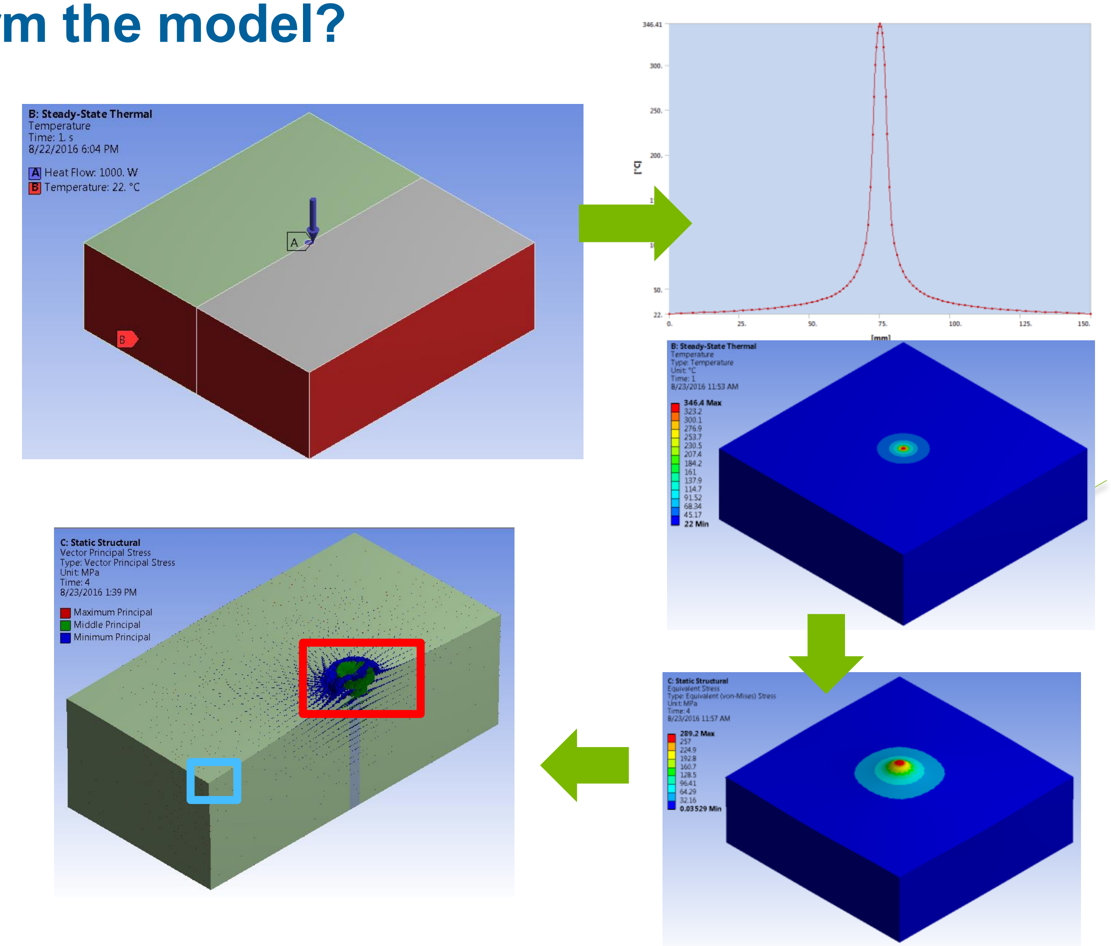
A SEEMINGLY PARADOXICAL NATURE OF MATERIAL FAILURE DUE TO THE THERMAL STRESSES

Part 1 – Will FEA confirm the model?

Thermal load on



These two areas need to be blown up

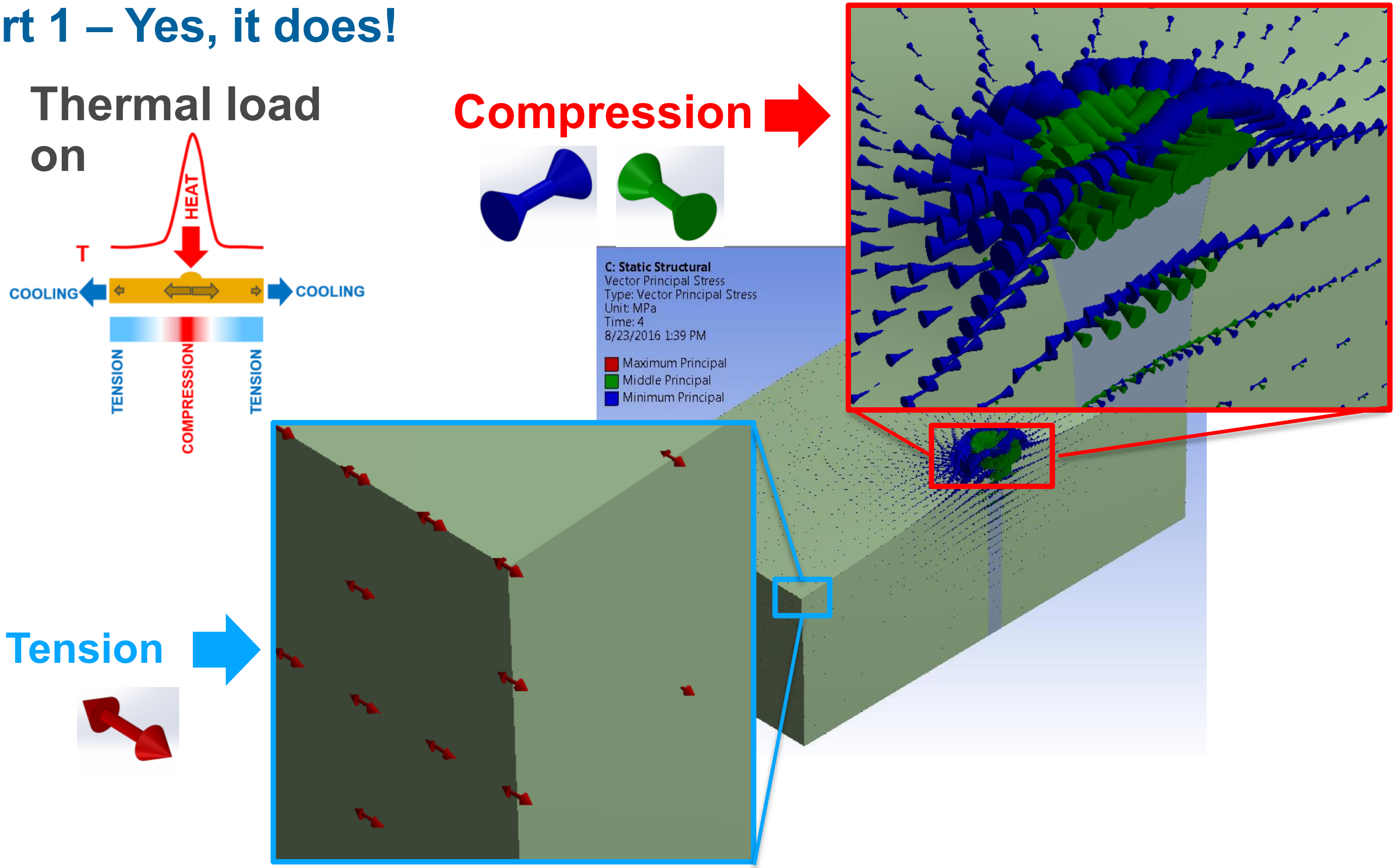


FEA analysis of a monolithic body exposed to a thermal cycle

Argonne NATIONAL LABORATORY

A SEEMINGLY PARADOXICAL NATURE OF MATERIAL FAILURE DUE TO THE THERMAL STRESSES

Part 1 – Yes, it does!



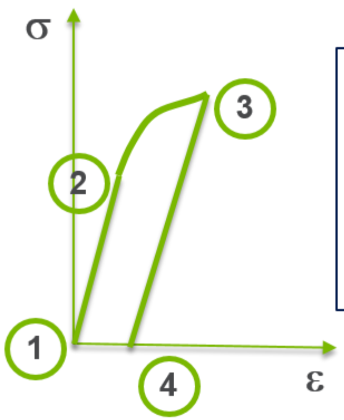
A detail of the previous slide

Argonne NATIONAL LABORATORY

THE SEEMINGLY PARADOXICAL NATURE OF MATERIAL FAILURE DUE TO THERMAL STRESSES

Failure model

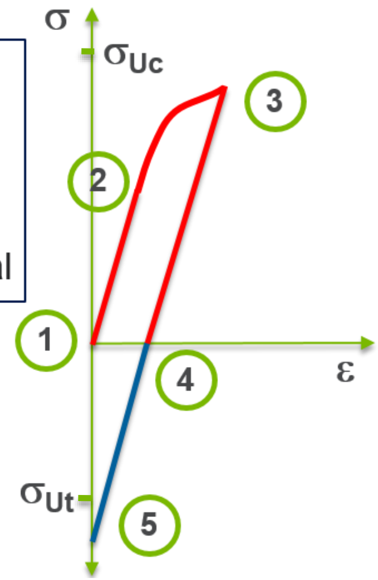
Mechanical stress



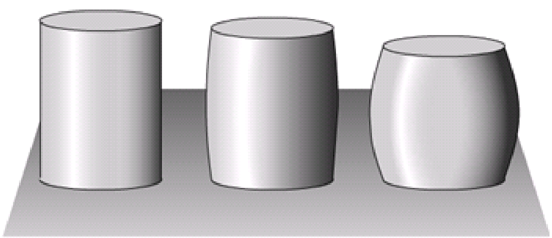
1-3: load on
- 1-2: elastic
- 2-3: plastic

3-4/5: load off
- 4-5: stress reversal

Thermal stress



Brittle Failure

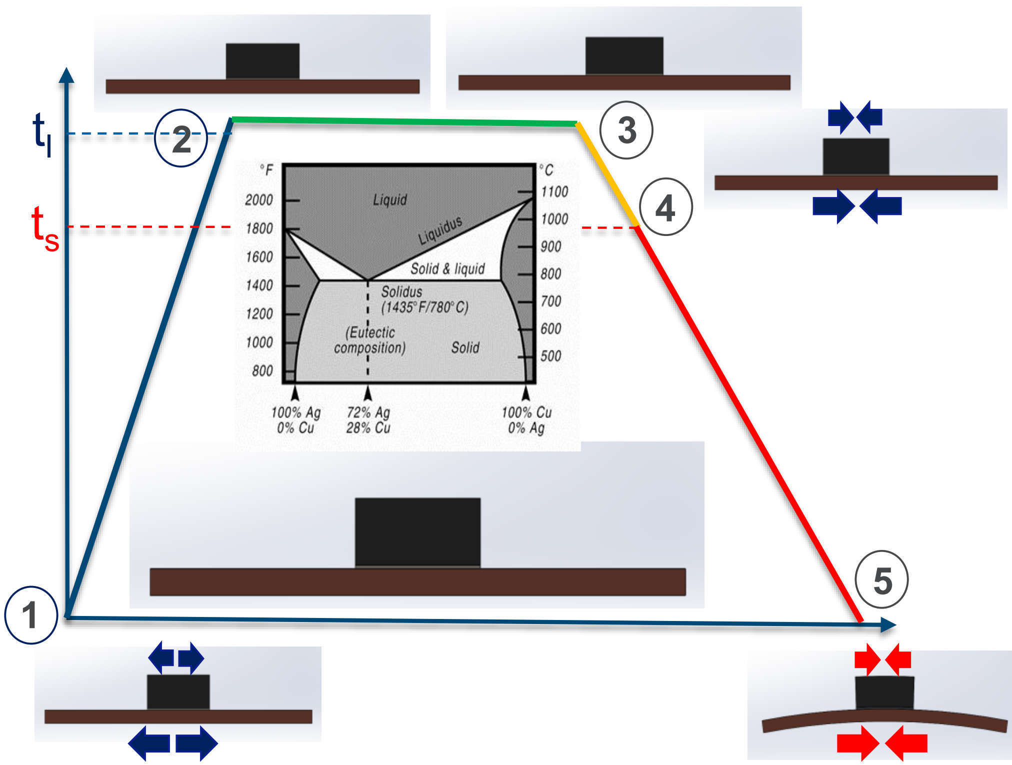


Ductile Failure

- If the compression under load is high enough to induce localized plastic deformation then, after cooling, areas previously in plastic compression can end up in plastic tension
- Ductile materials can survive substantial compressive deformation before developing cracks
- **Thus our designs can survive when the loads are on but fail when the loads are off!**

THE SEEMINGLY PARADOXICAL NATURE OF MATERIAL FAILURE DUE TO THERMAL STRESSES

Part 2 – A brazed joint



Brazing cycle

1-2: Temperature rises above liquidus temperature of the brazing alloy

- All components of the (future) joint expand freely. Cu expands much more than SiC
- No stresses developed

2-3: Temperature stays constant

- Melted brazing alloy wets the surfaces of SiC and Cu

3-4: Temperature drops to solidus temperature of the alloy:

- All components of the (future) joint contract freely. Cu contracts much more than SiC

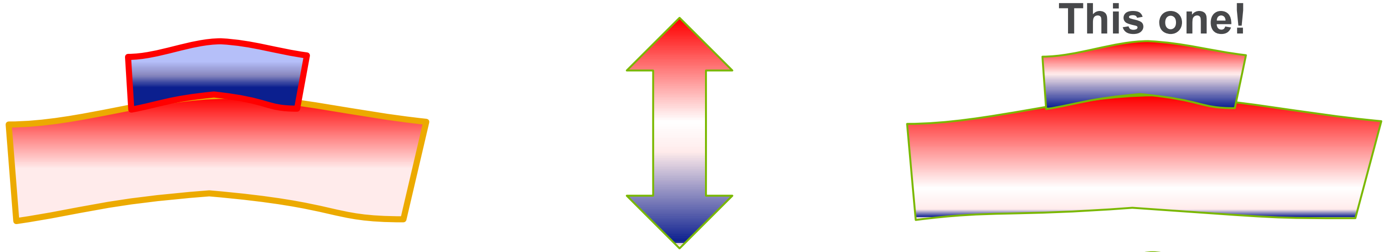
4-5: Temperature goes down to the room temperature

- Solid joint is formed, Components cannot contract freely. Cu wants to contract much more than SiC. Stress levels rise!

THE SEEMINGLY PARADOXICAL NATURE OF MATERIAL FAILURE DUE TO THERMAL STRESSES

Part 2 – How will the brazed Cu/SiC joint fail?

This one!

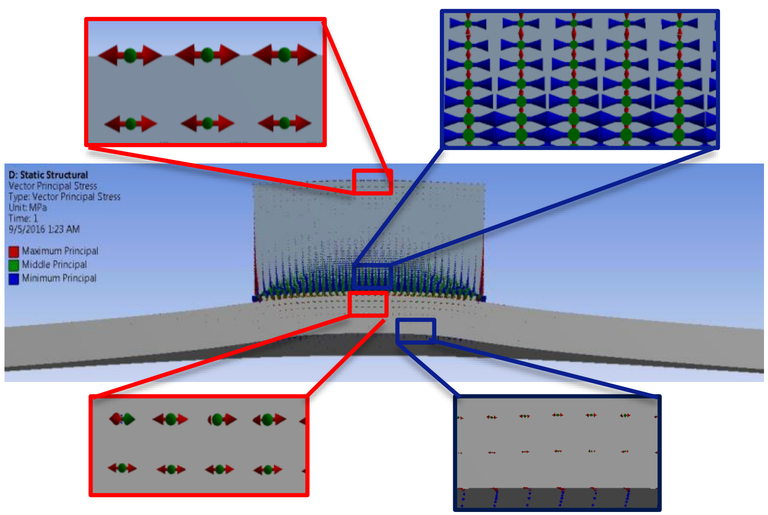


1

Which mode is correct?

2

- Once the brazed assembly starts cooling both components start to shrink
- Further away from the joint area both materials are less constrained in shrinking
- Cu shrinks more while SiC shrinks less
- The whole structure bends and in both components areas both in compression and in tension develop
- As a result, cracks develop in the top portion of the very brittle SiC and propagate towards the copper



Principal stress vectors indicate Mode 2 is correct

HOW DO THE SPECIFICS OF THE DESIGN OF SYNCHROTRON COMPONENTS AFFECT FEA SIMULATIONS

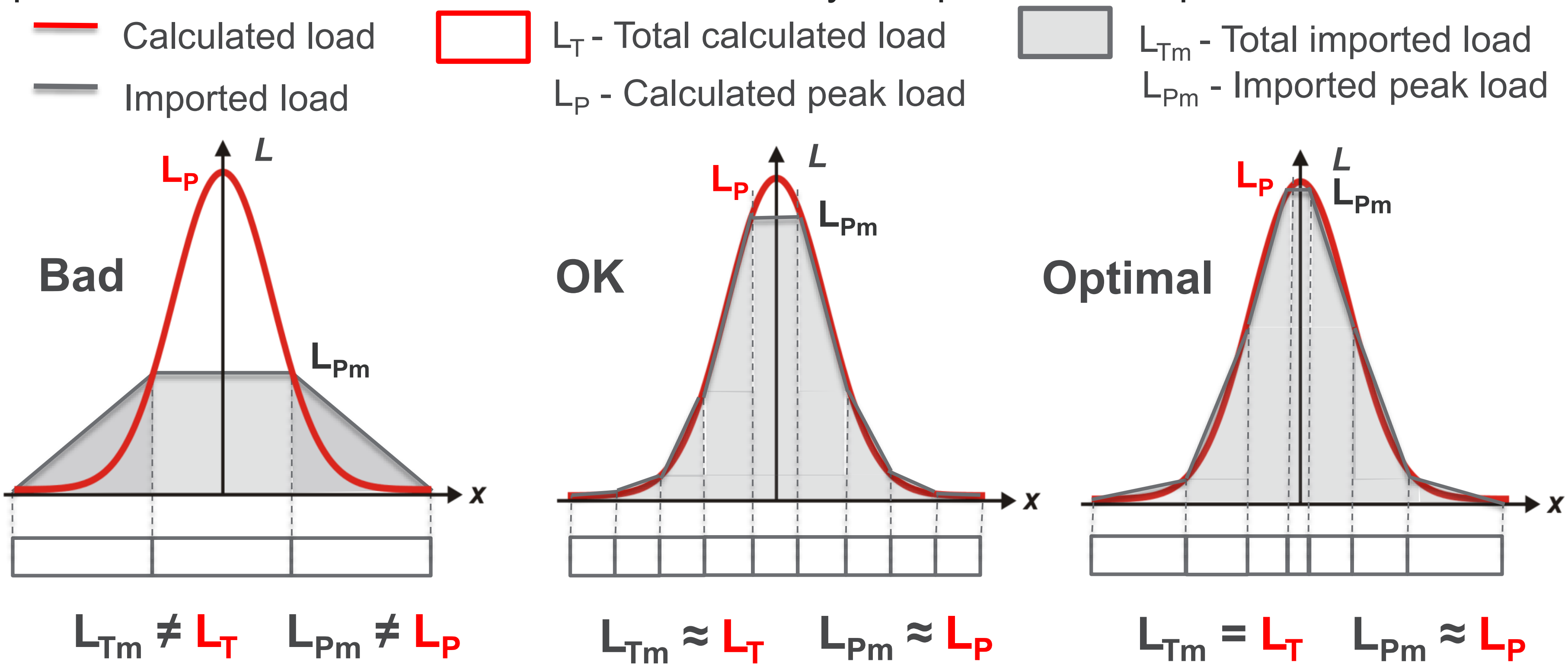
- Most of our components need cooling
  - Thermal analysis is frequently a first step
- Components are exposed to high power densities that are spatially non-uniform
  - Accurate methods of computing of power densities as well as accurate methods of importing the results into FEA are a must
    - Meshing has to allow for accurate capturing of spatial distribution
- High efficiency cooling is required
  - A good understanding of turbulent convective heat transfer is a must
    - Either calculations of convective heat transfer coefficient using semi-empirical equations or
    - CFD analysis of cooling are needed.
- Thermal stresses are the major contributor to stress levels in our components
  - Structural analysis is coupled analysis
  - A good understanding of particularities of component failures due to the thermal stresses is a must!
- Our components are vacuum components
  - The pressure difference across the component walls has to be accounted for,
  - A good understanding of material (Cu, Al) properties is needed

ACCURATE CALCULATION OF THERMAL LOADS

- There are two types of interactions of the beams with the components
  - The surface interaction of synchrotron radiation with synchrotron components
    - Due to the **relatively** shallow penetration depth of the photon beams through the walls of our components, it is safe/conservative to assume that all beam energy is deposited on the surface
  - The volumetric interaction of the particle beam with synchrotron components
    - The ring particle beam penetrates component walls much more deeply and thus the volumetric generation of heat must be computed, as the assumption of surface energy deposition would be too conservative
- And three ways of computing thermal loads
  - Analytical computations of the total power and power distribution of the photon beams using theoretical formulas
  - Using software packages for the calculation of total power and power distribution of photon beams,
    - SYNRAD+ for the bending magnet photon beams
    - SRUF for undulated photon beams.
  - Using software packages to compute the volumetric heat generation that occurs when ring particle beam and component walls interact:
    - MARS
    - GEANT4
    - Fluka
    - Etc.

ADEQUATE MESHING

- Thermal gradients in the components are directly related to the spatial distribution of loads
- Meshing of the component models has to be dense enough to capture the spatial profile of the thermal load in order to correctly compute the temperature distribution

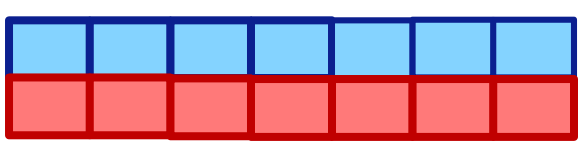


The effect of mesh density on the import of load data

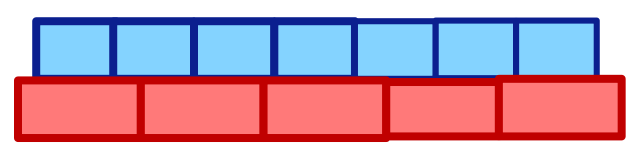
CONTACTS AND WHY AND HOW TO AVOID THEM WHEN FEASIBLE (ANSYS SPECIFIC)

- Definition of contact by ANSYS
  - When two separate surfaces touch each other such that they become mutually tangent, they are said to be in *contact*
  - In the common physical sense, surfaces that are in contact have these characteristics:
    - They do not interpenetrate
    - They can transmit compressive normal forces and tangential friction forces
    - They often do not transmit tensile normal forces. They are therefore often free to separate and move away from each other
- Contact elements were introduced into FEA in order to enable the analysis of the assemblies with different types of contacts between components
- This achieved two main things:
  - Assemblies with contacts between components other than rigid-bond contacts could be analyzed
  - There was no longer a requirement that the nodes of meshes of two different components have to coincide at the surface of their contact

No more prerequisite



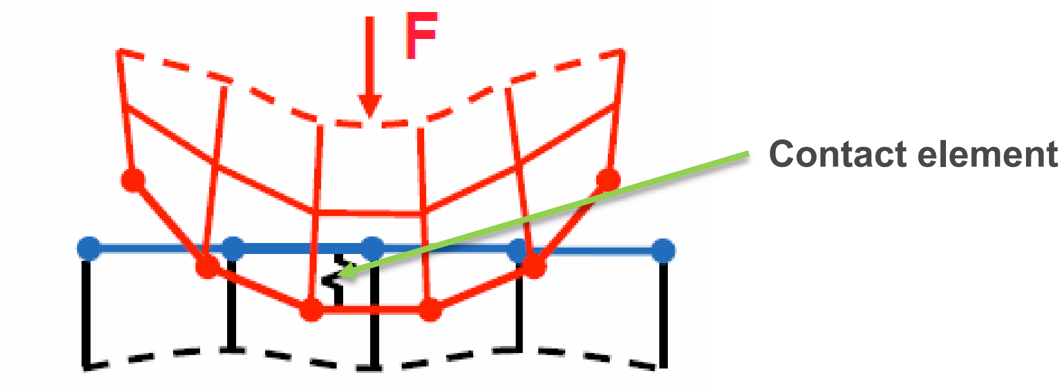
OK



WHY TO AVOID CONTACT ELEMENTS WHEN FEASIBLE

My (very unofficial) definition of contact elements

- Contact elements are the interpreters that enable communication between nodes of different component meshes that do not coincide



- Communication through the interpreter is not direct and errors are possible,
- Contact elements should be avoided when:
  - dealing with contact between components that have to transmit both compressive and tensile normal forces that cannot be separated without damaging at least one of them
  - you can live with the loss of flexibility and automatism in meshing
- If the above is not true or feasible, use bonded contacts.

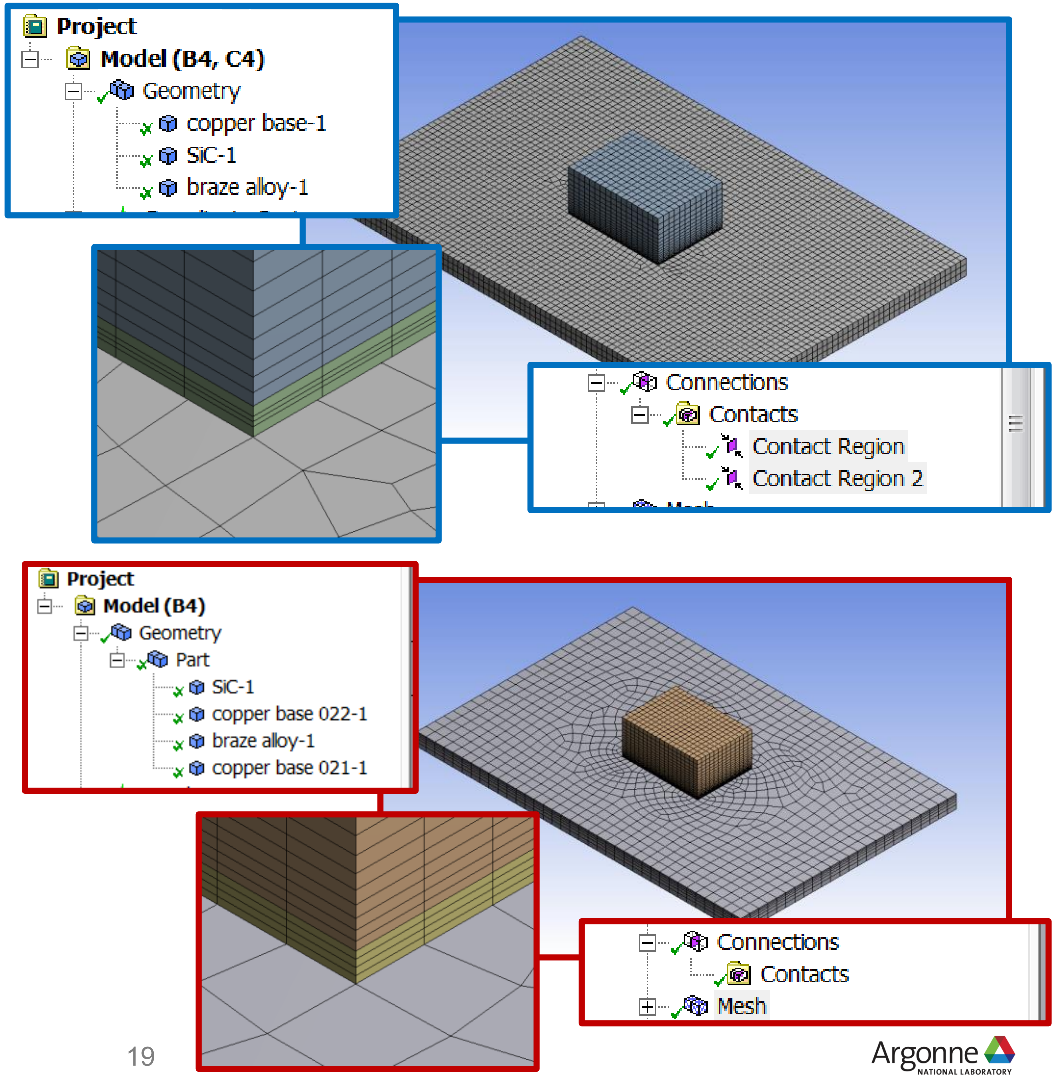
HOW TO AVOID CONTACT ELEMENTS WHEN FEASIBLE

- You can avoid the creation of contacts in ANSYS using the concept of multibody parts
  - Multi-body part concept enables combining multiple components into a single continuous part while still treating each component as a distinct body,
  - Each body can have its own material (with all material properties) assigned to it.
    - One multi-body part can consists of N bodies with up to N materials assigned to them,
  - Each body of a multi-body part can be individually meshed
    - Mesh nodes of two individual bodies will match at the contacting surfaces,
    - No contact elements will be created at the contacting surfaces
- The model of a multi-body part will differ from an identical model structured as a multi-part assembly as follows:
  - No contacts will be created at the contact surfaces between the bodies,
  - Meshes of the individual bodies will share nodes at the surface of contact,
- There is an increased possibility that the mesher will fail in creating meshes for the multi-body part

HOW TO AVOID CONTACT ELEMENTS WHEN FEASIBLE

Multi-part assembly VS Multi-body Part – an example

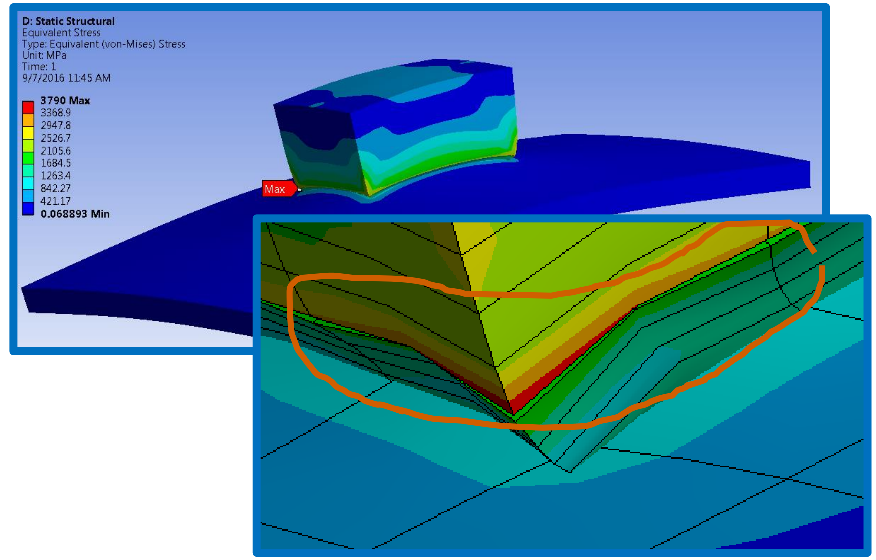
- Multi-part assembly:
  - 3 individual parts,
  - Mesh nodes in the surface of contact do not necessarily match,
  - Two contact pairs created,
- Multi-body part:
  - 1 one part with 4 bodies
    - Copper base body was split in two for purpose of promoting sweep method of meshing,
  - Mesh nodes in the surface of contact DO match,
  - NO contacts created.

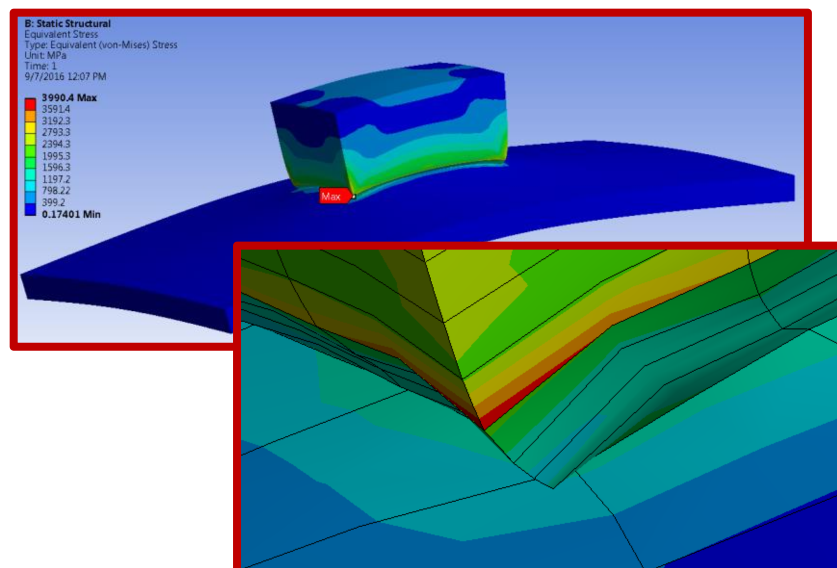


MULTI-PART ASSEMBLY VS MULTI-BODY PART

So what's the big deal?

- Multi-part assembly:
  - Simulation produced somewhat lower maximum equivalent stress values
  - Separation between parts in contact was observed
    - This should not happen as the bonded (no separation) contact was used!
- Multi-body part
  - Higher maximum values of equivalent stress produced
  - **NO** separation between the bodies observed.





Argonne NATIONAL LABORATORY

CORRELATIONS FOR CALCULATION OF CONVECTIVE HEAT TRANSFER COEFFICIENT

- The coefficient of convective heat transfer, h, for a turbulent flow is a function of Nusselt number, *Nu:*

$$h = \frac{Nu \cdot k}{D_H}$$

- The Nusselt number is a function of the Reynolds number, Re:
  - Reynolds number:

$$Re = \frac{\overline{U} D_H}{\nu}$$

  - Nusselt number:
    - Dittus-Boelter correlation

$$Nu = 0.023\ Re^{0.8}\ Pr^{n}$$

    - Gnielinski correlation (better for large water temperature rises, rough tubes, generally slightly more conservative)

$$Nu = \frac{\left(f/8\right)(Re - 1000)Pr}{1 + 12.7\left(f/8\right)^{1/2}\left(Pr^{2/3} - 1\right)}$$

$$f = \left(\frac{1}{-1.8log\left[\left(\frac{\epsilon/D_H}{3.7}\right)^{1.11} + \frac{6.9}{Re}\right]}\right)^2$$

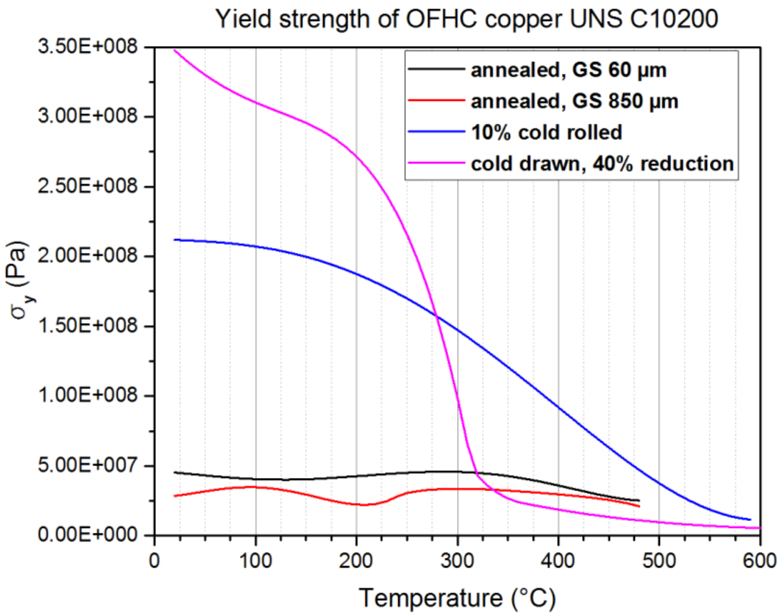
Darcy friction factor, can be obtained from Moody Chart

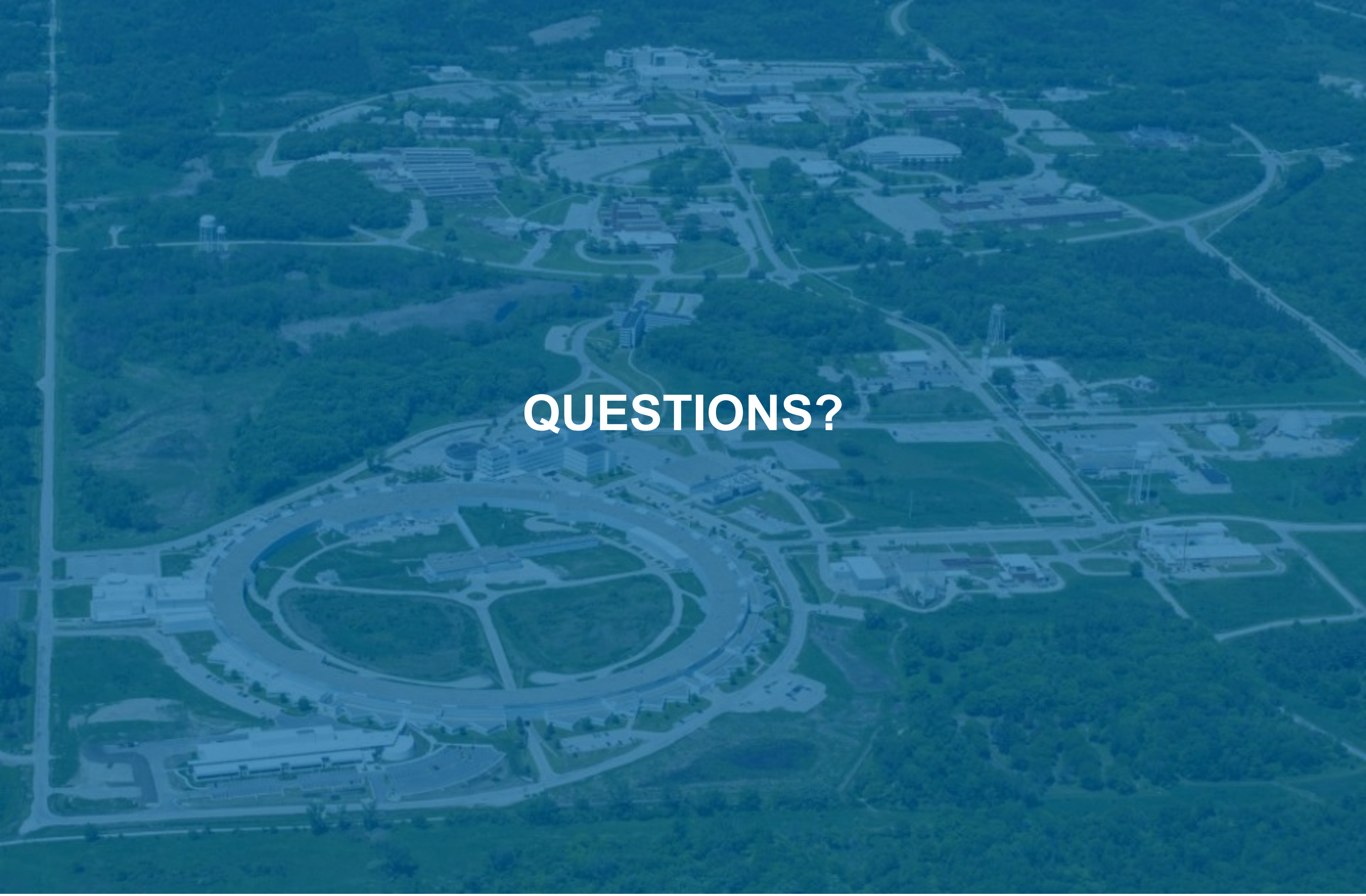
FEA AND UNDERSTANDING OF THE SPECIFICS OF COMPONENT FAILURES DUE TO THE THERMAL STRESSES

- In monolithic components, where thermal stresses develop due to thermal gradients, the compressive stresses develop under the thermal load
- However it makes sense to establish ultimate and yield **tensile** strength as the design criteria for ductile materials (copper, aluminum) as stress reversal occurs
  - once the load is removed **and**
  - if the part was in plastic compression under the load
- In multi-material components that consist of bonded materials with a different coefficient of thermal expansion, the analysis makes sense only if the residual stresses originating from the process of bonding are accounted for
  - If these stresses are not accounted for in the analysis of the components that heat up during the operation, then the results of the analysis can be too conservative!
  - If the component is cooled down during the operation and the residual stresses of bonding are not accounted for in the analysis, then the analysis results tend to be overly optimistic!

LIMITED CHOICE OF MATERIALS AS A FACTOR IN FEA ANALYSIS

- Copper and aluminum are frequently used in the design of synchrotron components
- Both copper and aluminum are materials with thermally unstable mechanical properties that are dependent on temperature history
  - The use of temperature dependent material properties in the analysis is very important,
  - It is even more important to know the ‘temper’ of these materials expected in component application, e.g.
    - If brazing will be used in the fabrication of the component made of copper, then the yield strength of fully annealed copper should be used in the evaluation of the structural analysis results,
    - If 10% cold rolled copper will only be explosively bonded in the process of fabrication, then the yield strength of 10% cold rolled copper can be used.




QUESTIONS?

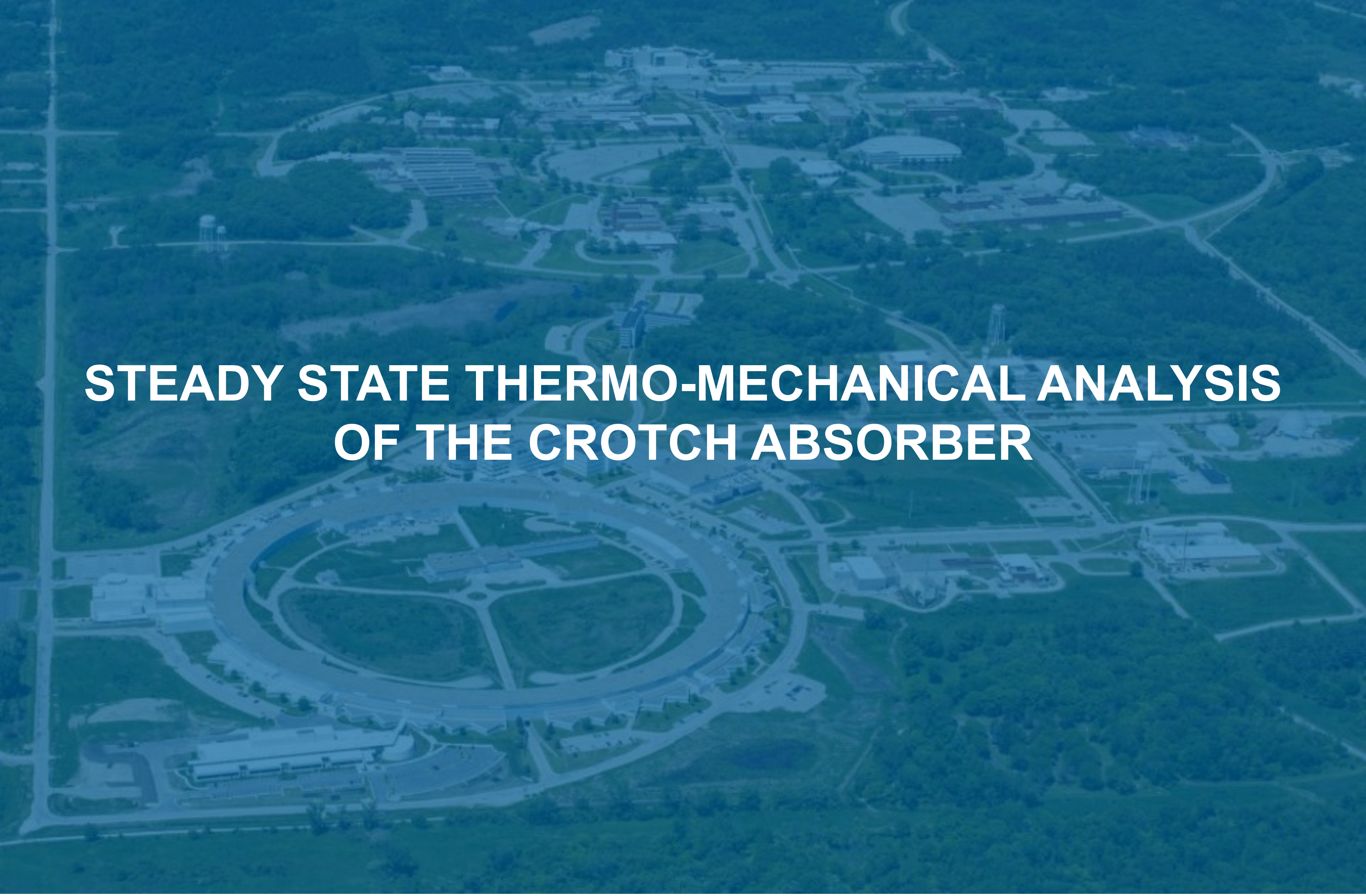
EXAMPLES OF COUPLED THERMO-MECHANICAL SIMULATIONS

BRAN BRAJUSKOVIC
Principal Mechanical Engineer
Mechanical Engineering and Design Group
Advanced Photon Source
Argonne National Laboratory

September 12th, 2016
Barcelona

OUTLINE

- Steady state thermo-mechanical analysis of the crotch absorber
  - Photon beam surface thermal loads defined via analytical formula
- Transient thermal analysis of scraper
  - Particle beam volumetric thermal loads defined using MARS code
- Steady state thermo-mechanical analysis of FODO section inline absorbers
  - Surface thermal loads defined by SYNRAD+
    - Data import issues
  - Meshing and elimination of contacts
    - Multibody parts
- Transient thermo-mechanical analysis of thermal fatigue in Glidcop AL15
  - Experimentally obtained multi-linear stress-correlation
  - Experimental validation of the analysis results



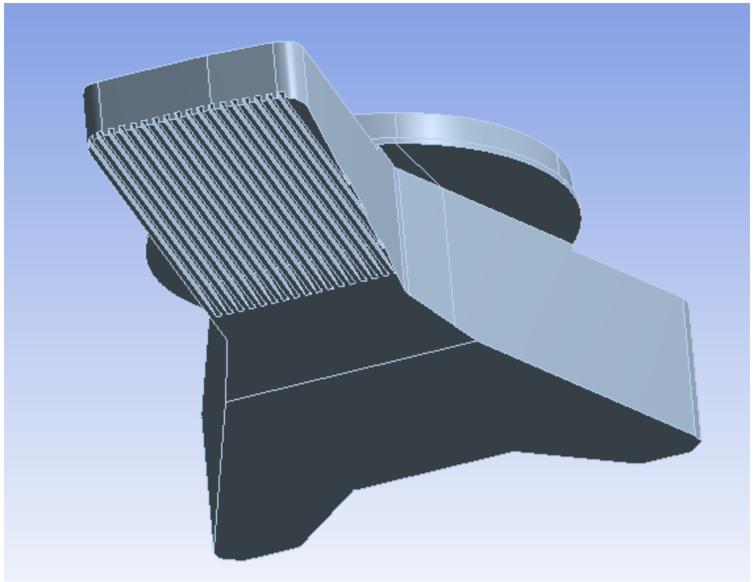
STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE CROTCH ABSORBER

Argonne NATIONAL LABORATORY

STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE CROTCH ABSORBER

Courtesy of Jie Liu

- Objective
  - Modify the design of the crotch absorber used for the existing 0.6 T dipole at 100 mA to be able to handle synchrotron radiation coming from the new 1.2T dipole at 150mA,
  - The design should not include significant changes to the storage ring
    - Same location of the absorber
    - Similar dimensions
  - The maximum computed temperature and stress should be below the Glidcop criteria
- Strategy
  - Reduce the beam incidence angle (‘spread the beam to lower incident heat flux values),
  - Increase water flow rate to improve the coefficient of convective cooling and thus increase the efficiency of cooling
  - Use FEA analysis for design optimization



3D model of the crotch absorber

Argonne NATIONAL LABORATORY

STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE CROTCH ABSORBER

Physics

- On-axis (peak) power density:

$$P_{d0}(W/mrad^2) = 5.421 * E_e^4(GeV) * I(A) * B(T) .$$

- Power distribution integrated over all frequencies:

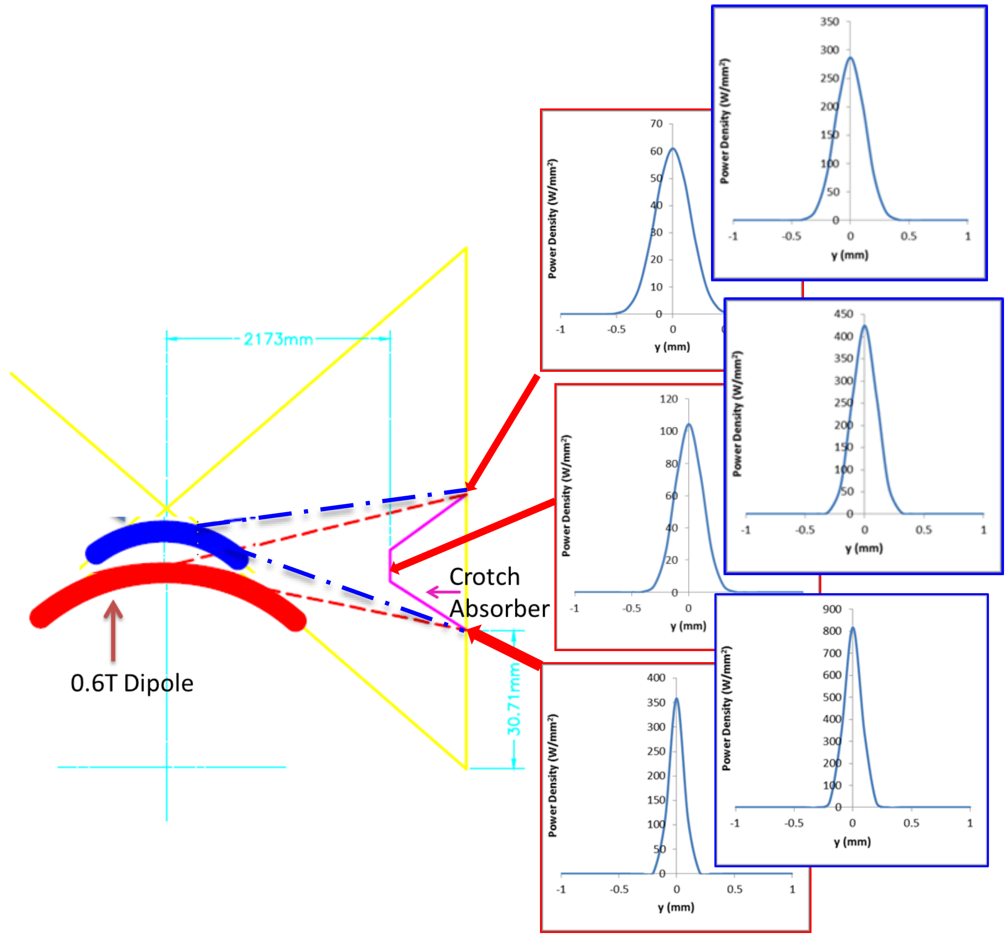
$$P_d\,(W/mrad^2) = P_{d0}\frac{1}{(1+X^2)^{5/2}}\left[1+\frac{5}{7}\frac{X^2}{1+X^2}\right];\; X=\gamma\psi\ ,$$

- Integrated power distribution projected on the intercepting surface

$$pP_d(W/mm^2)=P_d*\sin\theta/l^2$$

- Total power per unit horizontal angle:

$$P_{tot}\,(W/mradh) = 4.221 * E_e^3(GeV) * I(A) * B(T) .$$



Schematics of the bending magnet synchrotron radiation hitting the crotch absorber

STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE CROTCH ABSORBER

Homework

- Magnet parameters

0.6T (Current APS dipole)		Unit
E	7	GeV
B	0.5990	T
L	3.0600	m
γ	13699	
R	38.9611	m
Straight length	3.0577	
Total radiation arc	4.5°	
	78.5000	mrad
total power generated by BM	6811.8510	W

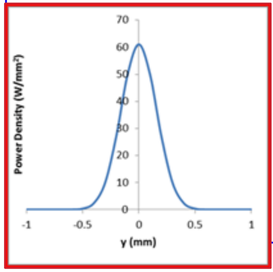
1.2 T (Superbend)		Unit
E	7	GeV
B	1.2	T
Beam Current	150	mA
L	1.5264	m
γ	13699	
R	19.4444	m
Straight length	1.5260	
Total radiation arc	4.5000	degree
	78.5000	mrad
total power generated by BM	20473.4990	W

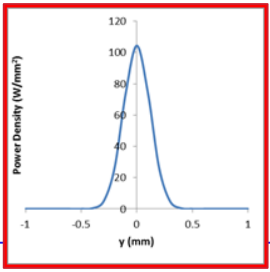
  - Calculated incident heat loads
    - Total heat load:

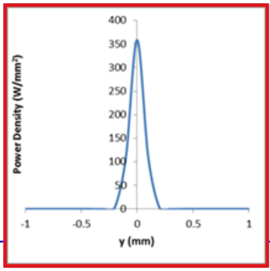
$$P = 0.96 \times 10^{-15} \frac{\gamma^4}{\rho[m]} \times I\ [mA] \times 49.74[mrad]$$

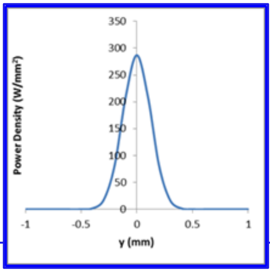
$BM_{(0.6T)}$ – 4317.6 W; $BM_{(1.2T)}$ – 14299.8 W

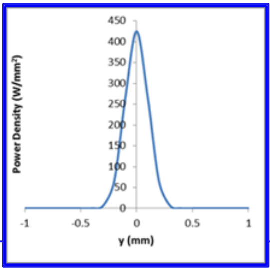
    - Incident heat load distribution

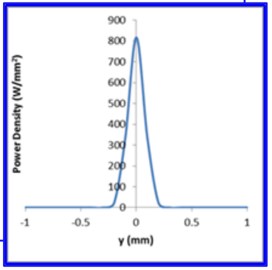








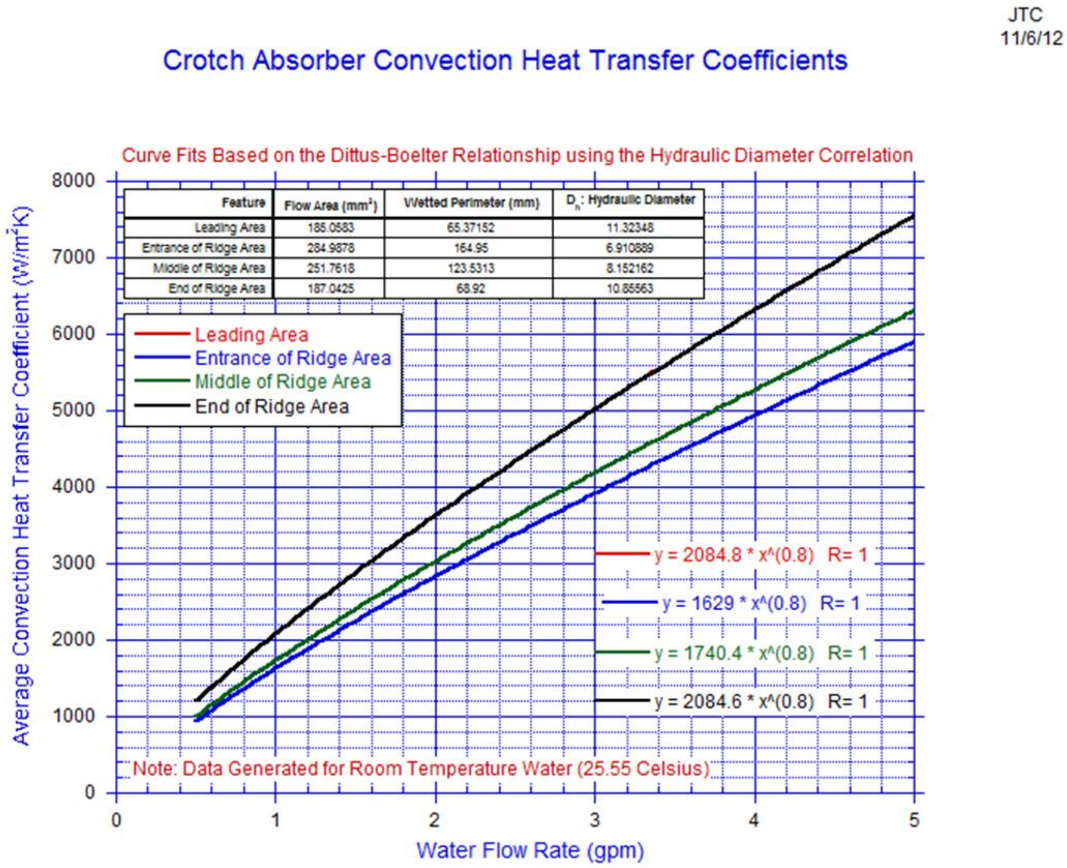




STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE CROTCH ABSORBER

Homework (continued)

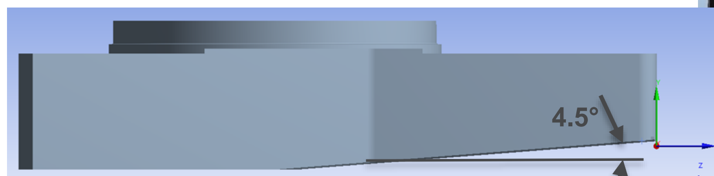
- Cooling parameters

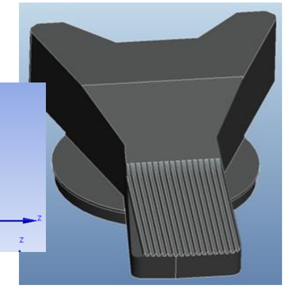


Feature	Flow Area (mm^2)	Wetted Perimeter (mm)	D_h: Hydraulic Diameter
Leading Area	185.0583	65.37152	11.32348
Entrance of Ridge Area	284.9878	164.95	6.910889
Middle of Ridge Area	251.7618	123.5313	8.152162
End of Ridge Area	187.0425	68.92	10.85563

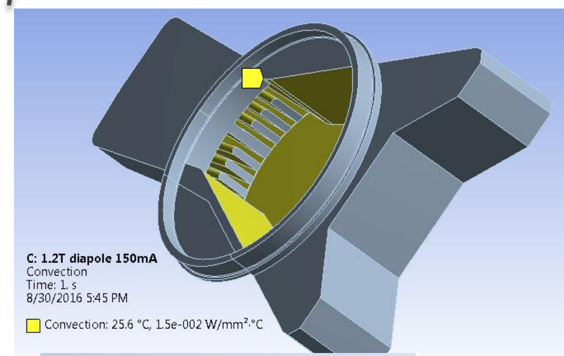
STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE CROTCH ABSORBER

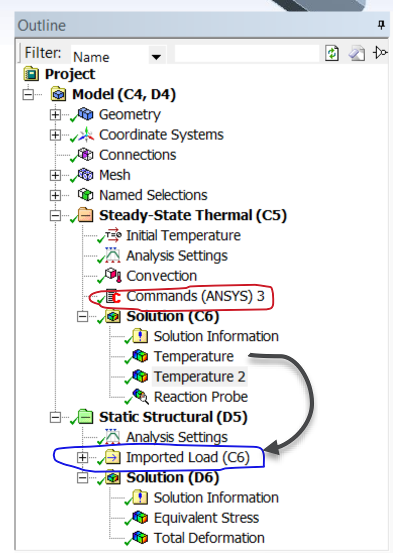
Boundary conditions





- Geometry
  - Incidence angle changed from 11° to 4.5°
  - Added chamfers and rounds
- Thermal analysis
  - Cooling
    - Convective heat transfer coefficient increased to 15 kW/m^2K
  - Thermal load
    - An analytical function was used to define thermal loads on crotch absorbers intercepting surfaces
    - A command snippet was developed to calculate power distribution in normal plane and project the values onto the intercepting surfaces
- Static structural Analysis
  - A previously computed temperature distribution was used as the only load
  - A weak spring was used to prevent rigid body motion

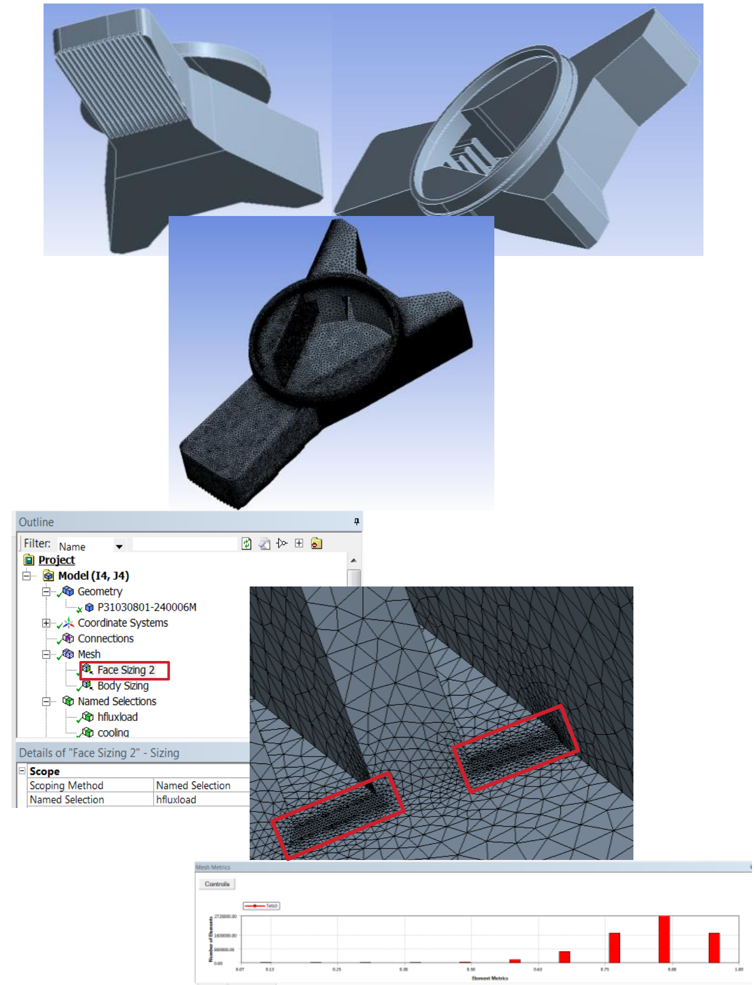




STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE CROTCH ABSORBER

Meshing

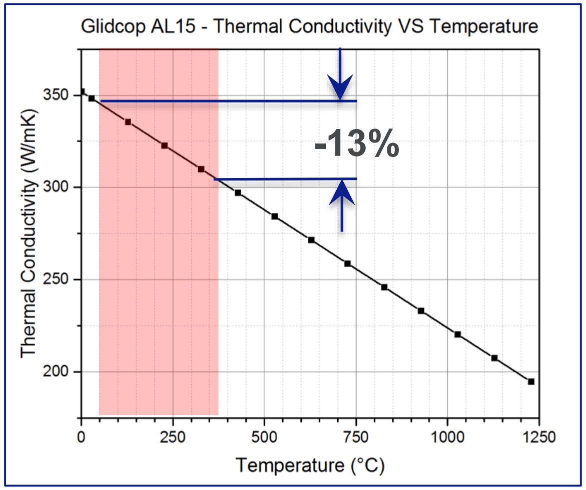
- When the geometry is complex the default meshing method is your best bet!
- Regardless of the meshing method used
  - A denser mesh (more elements) produces more accurate results
  - Tet elements give you the most of the flexibility in meshing
    - But they are less accurate and the mesh will be more ‘node-heavy’
  - The size of meshes can be controlled by using mesh controls
    - Mesh controls let you apply a denser mesh selectively, in the areas of particular interest
    - Named selections are a very effective way to do that (at least in ANSYS)
  - Different metrics can tell you a lot about the mesh quality
    - However, a good-quality mesh does not guarantee that the mesh is optimal for your problem!

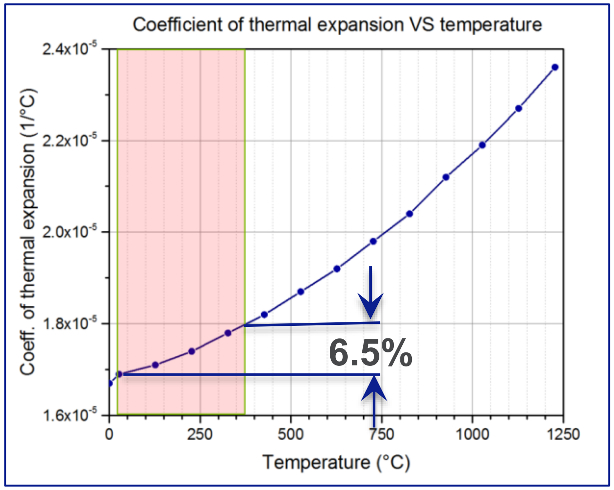


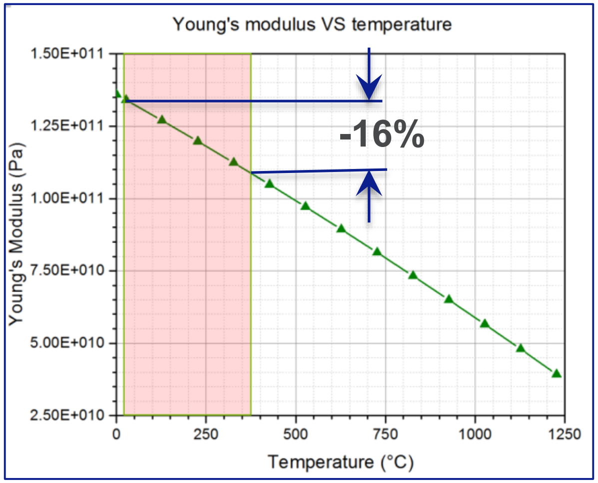
STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE CROTCH ABSORBER

Material

- Glidcop Al15 was the material used in the analysis
  - For the analysis, only thermal conductivity, thermal expansion, and Young's modulus are needed (Poisson's ratio is assumed to be 0.33)
  - The property data used in the analysis were at room temperature
  - The calculated temperature range was 26.6-373.6 °C
    - Thermal conductivity changes -13% across the range
    - The coefficient of thermal expansion changes +6.5%
    - Young's modulus changes -16%
  - The temperatures and expansion are underestimated, and the stiffness is overestimated



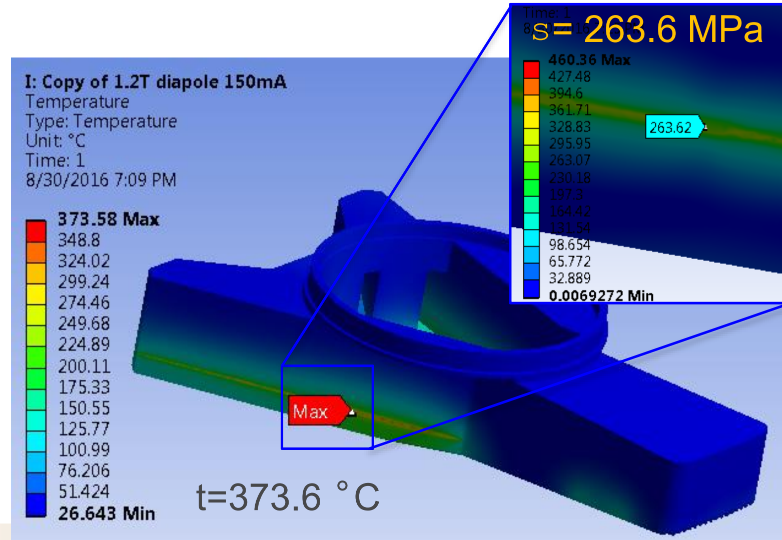


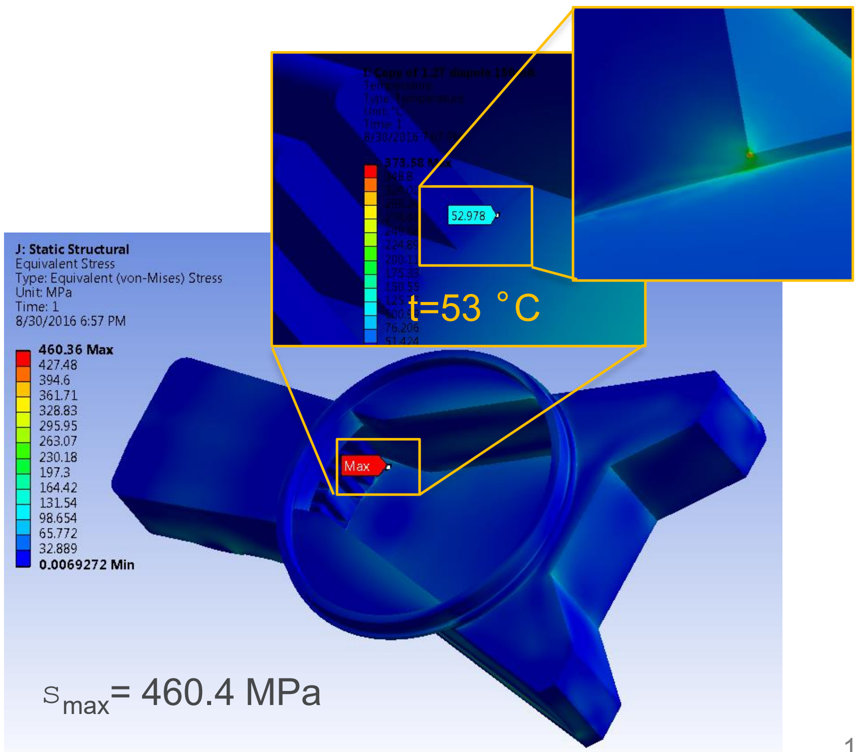


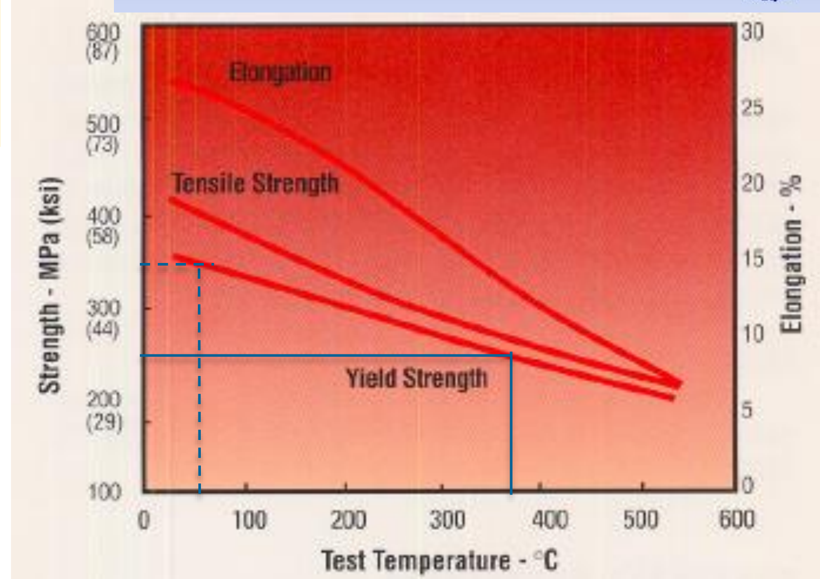
STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE CROTCH ABSORBER

Results

- Max. comp. temperature – 373.6 °C
  - Max. comp. stress at the location – 264 MPa
  - s_y z 270 MPa at 373.6 °C
- Max. comp. stress – 460.4 MPa (?)
  - Max. comp. temperature at the location – 53 °C
  - s_y z 350 MPa at 53 °C







STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE CROTCH ABSORBER

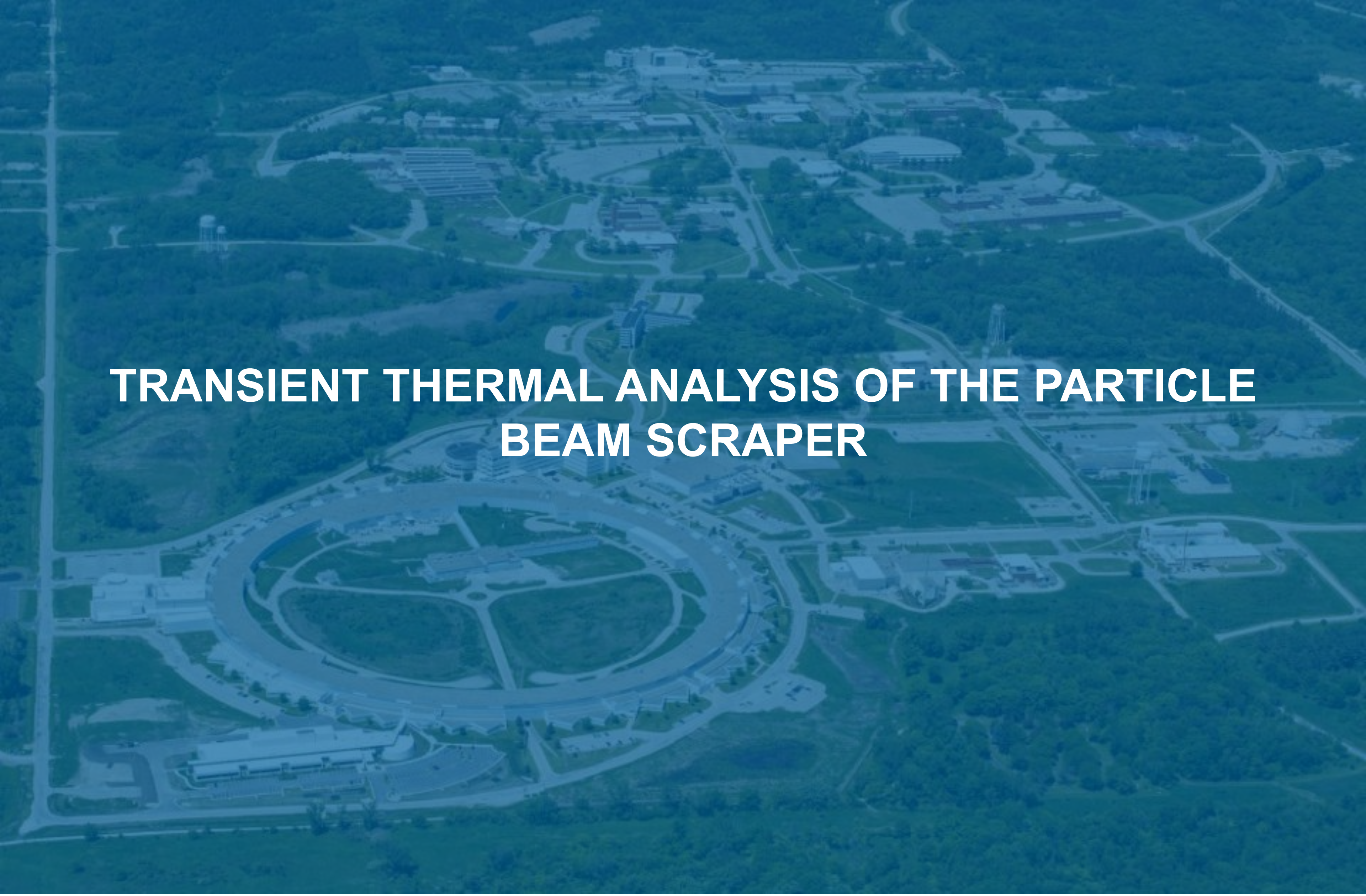
The Main Challenge

Crotch-absorber snipet - Notepad

File Edit Format View Help

```
!    Commands inserted into this file will be executed just prior to the Ansys SOLVE command.
!    These commands may supersede command settings set by Workbench.

!    Active UNIT system in Workbench when this object was created:  Metric (mm, kg, N, s, mV, mA) with temperature units of C


!    Commands inserted into this file will be executed just prior to the Ansys SOLVE command.
!    These commands may supersede command settings set by Workbench.

!    Active UNIT system in Workbench when this object was created:  Metric (mm, kg, N, s, mV, mA)

/prep7
allsel,all
et,99,surf152
type,99
!keyopt,99,4,1
keyopt,99,8,1

finish

/sol
csys,12
cmsel,s,hfluxload
type,99
esurf
allsel,all

esel,s,type,,99
selelms=elmiqr(0,13)

!numelms=elmiqr(0,14)
!*dim,loadvect,array,numelms
currelm=0
*do,i,1,selelms
  currelm=elnext(currelm)
  !xcoord=nx(currnode)
  !ycoord=ny(currnode)
  *get,xcoord,elem,currelm,cent,x
  *get,ycoord,elem,currelm,cent,y
  *get,zcoord,elem,currelm,cent,z

   q1=(2173-zcoord)**2
   q2=(19454.33-xcoord)**2
   q3=19444**2
   d=(sqrt(q1+q2-q3))/1000

   ppeak=2342/d/d
   q4=(ycoord/d)**2

  *get,a,elem,currelm,area
  *get,ap,elem,currelm,aproj,z
  pp = ap/a
  p=pp*1000*ppeak*exp(-258.3*q4)


  *if,ycoord**2,gt,1,then
  p=0
  *endif


   sfe,currelm,1,hflux,,p

*enddo
allsel,all
```

- Developing a command snippet for the input of a thermal load
  - ANSYS commands do not have intuitive syntax,
  - Each command has multiple options,
  - Absolute accuracy in the syntax is required
  - Units in the analysis have to be same as in the snippet
    - This is particularly important when you are using or modifying existing snippets

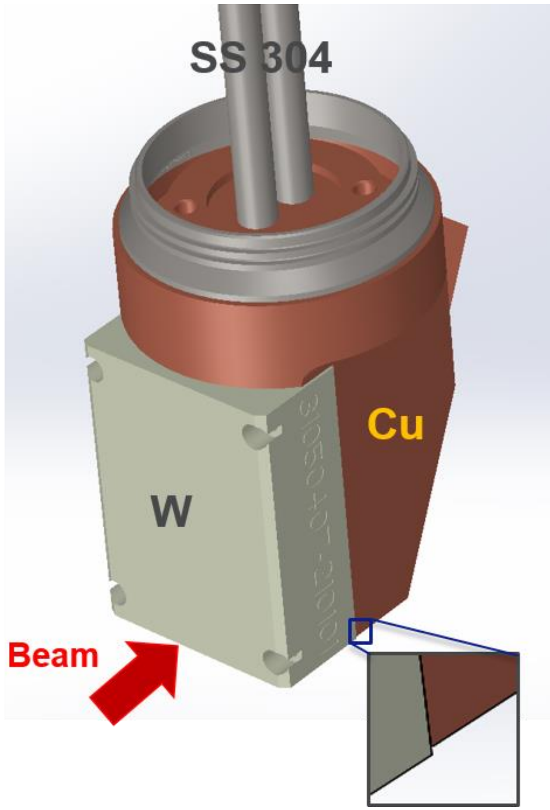


TRANSIENT THERMAL ANALYSIS OF THE PARTICLE BEAM SCRAPER

Argonne
NATIONAL LABORATORY

TRANSIENT THERMAL ANALYSIS OF THE PARTICLE BEAM SCRAPER

- Objective
  - Determine the cause of the material damage observed on the beam scraper installed at Sector 37 of the APS
    - Pit-like damage was observed on the bottom surface of the tungsten part
    - Gray powder was found on the bottom of the chamber with the scraper right underneath it
    - Only the tungsten ‘sees’ the beam, the copper part is protected by the tungsten
    - The intercepted beam is the particle beam
- Analysis strategy
  - Use FEA analysis to find out if excessive thermal stresses are the cause of the observed damage



Beam scraper

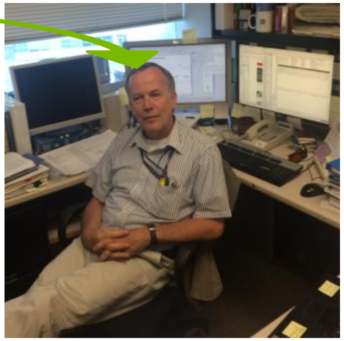
TRANSIENT THERMAL ANALYSIS OF THE BEAM SCRAPER

Physics

- The beam is a particle beam, not a photon one!
- The accelerator particle beam has a considerably larger matter penetration depth than the photon beams
  - The heat that results from the interaction of the beam particles with the intercepting matter is generated within a certain volume
    - The assumption that the transfer of energy and the resulting heat generation occurs only on the intercepting surface would be overly conservative
  - Dedicated software codes have to be used to compute the interactions between the particles and the intercepting matter
    - The computation of the heat generated in the interactions is definitely not a primary output of these packages
    - A careful interpretation of the computed results is necessary to accurately predict the generated heat and its volumetric distribution
    - The ‘translated’ heat generation results are in the form of comma-delimited or tab-delimited text files that have to be imported into the FEA software in a meaningful way.

TRANSIENT THERMAL ANALYSIS OF THE BEAM SCRAPER

Homework



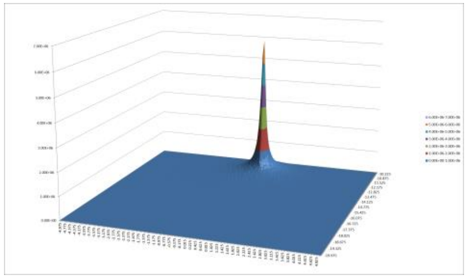
- We called Jeff Dooling, our resident radiation physicist, to the rescue
- He used the MARS code, developed at FermiLab, as a computational tool

MARS: Intro/Manual

What is MARS (Code birthday: October 1974)

MARS is a Monte Carlo code for inclusive and exclusive simulation of three-dimensional hadronic and electromagnetic cascades, muon, heavy-ion and low-energy neutron transport in accelerator, detector, spacecraft and shielding components in the energy range from a fraction of an electronvolt up to 100 TeV.

- He helped us to translate MARS data outputs into tab delimited files
  - We used EXCEL to create the structure of the data tables required for input in ANSYS



X-coor.

Z-coor.

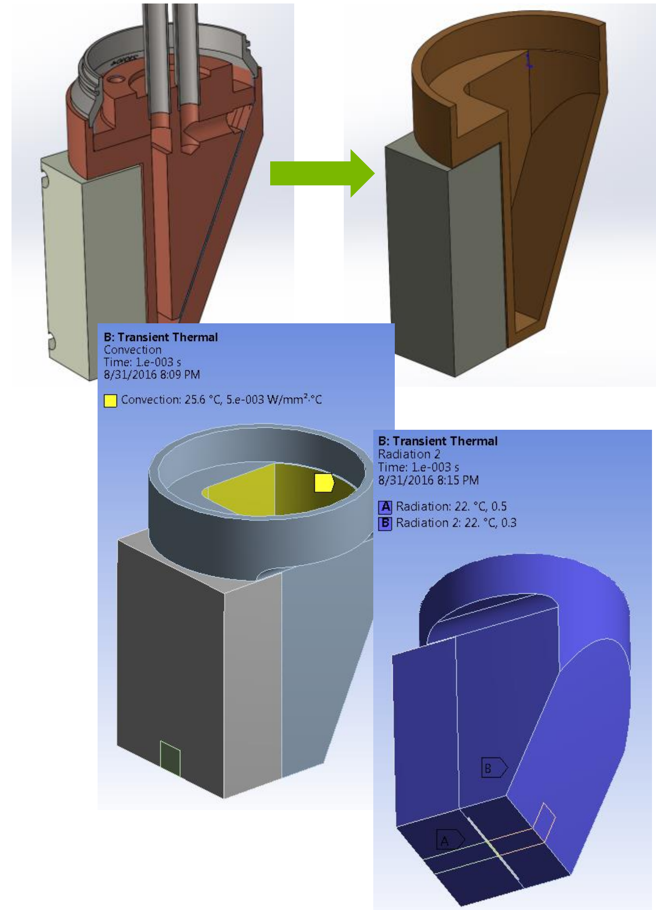
	A	B	C	D
1	2.03730E+01	-4.97500E+00	-4.92500E+00	-4.87500E+00
2	-1.99750E+01	1.87851E+05	1.45755E+05	1.93283E+04
3	-1.99250E+01	3.42801E+04	1.76024E+05	1.69340E+05
4	-1.98750E+01	3.68253E+04	2.23010E+04	1.65423E+05
5	-1.98250E+01	4.53617E+04	4.92466E+04	3.26166E+06
6	-1.97750E+01	1.74787E+05	9.17388E+04	8.57998E+04
7	-1.97250E+01	3.55525E+04	1.38988E+06	7.04175E+04

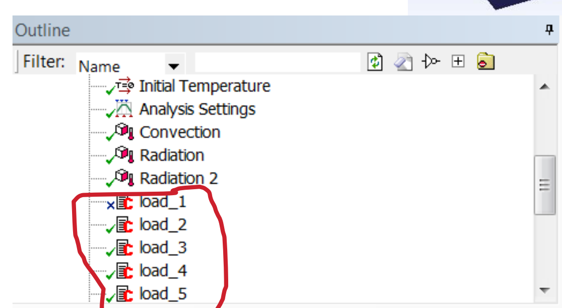
Y-coor.

TRANSIENT THERMAL ANALYSIS OF THE BEAM SCRAPER

Boundary conditions

- Geometry
  - The geometry was simplified
  - Only the relevant parts were analyzed
- Thermal analysis
  - Cooling
    - A convective heat transfer coefficient of 5 kW/m^{2}K was used
    - Radiative cooling to the environment was used with coefficients of emissivity of 0.3 for copper and 0.5 for tungsten
  - Thermal load
    - Command snippets were developed to import data computed with MARS
    - Due to the size of files, the data was imported in several segments computed for specific areas of the scraper

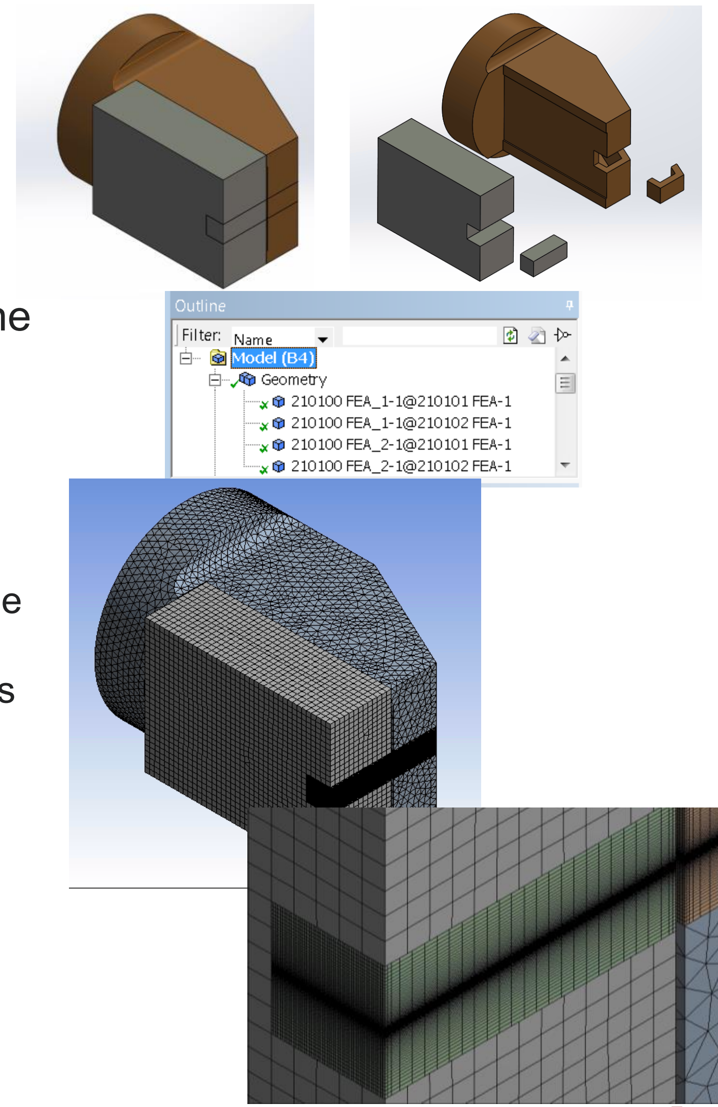




TRANSIENT THERMAL ANALYSIS OF THE BEAM SCRAPER

Meshing

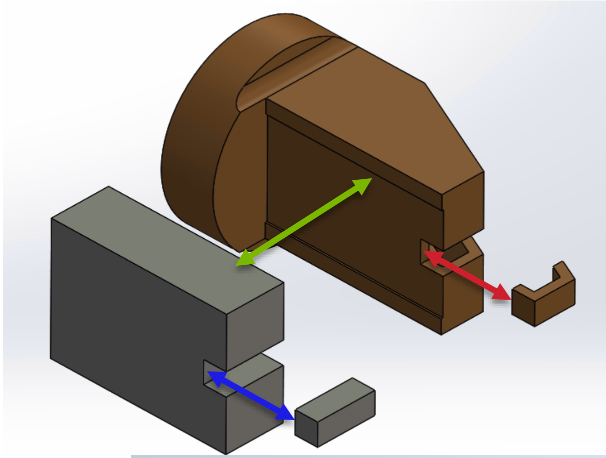
- To minimize areas with a high density mesh the components were broken into subcomponents
- The subcomponents that were exposed to the beam
  - Had a much denser mesh to accurately capture the imported loads
  - They were meshed using a sweep method
    - A sweep method produces identical/similar hex elements and defines the size of the elements in the direction of the sweep
    - The size of the elements in other two directions can be controlled using mesh size controls
    - Using a bias option (ANSYS), the size of the elements can be gradually increased/decreased
- Subcomponents that were far from the area of interaction with the beam had a much coarser mesh in order to minimize the number of elements
  - Very important as this was a transient analysis

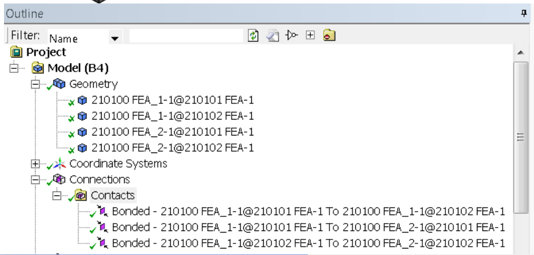


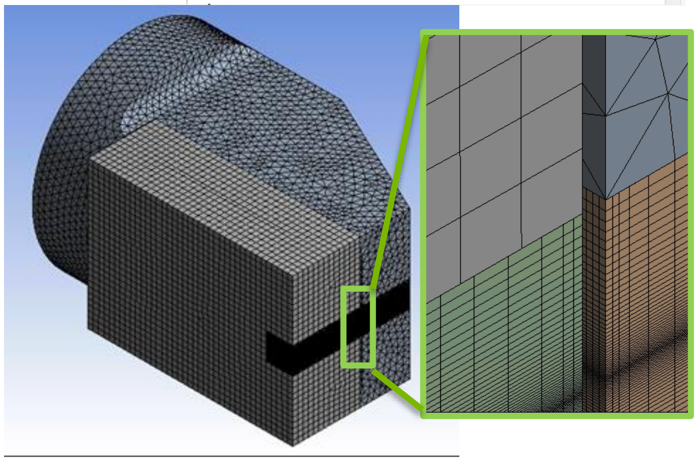
TRANSIENT THERMAL ANALYSIS OF THE BEAM SCRAPER

Contacts

- The model consisted of four parts
- Three contacts were created
  - ANSYS allows multiple surfaces to create one contact
  - When models have a very small gap, it is necessary to make sure that contact is not created between the surfaces that are separated by the gap
    - In ANSYS this is done using the contact tolerance option,
    - The contact tolerance has to be smaller than the smallest gap
- Interaction between the parts of the model is defined with contacts
  - In addition, contacts allow for non-conformal meshing where the nodes of the parts in the contact do not have to match
    - This brings a lot of flexibility into the meshing process



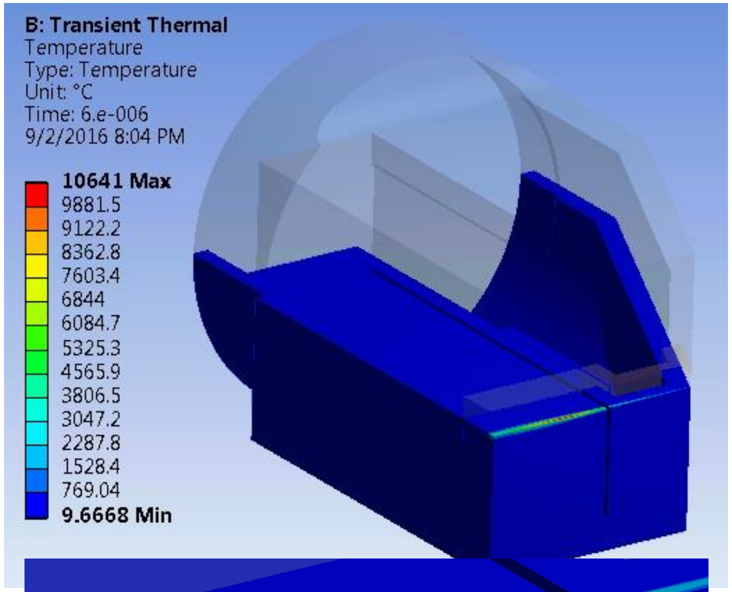


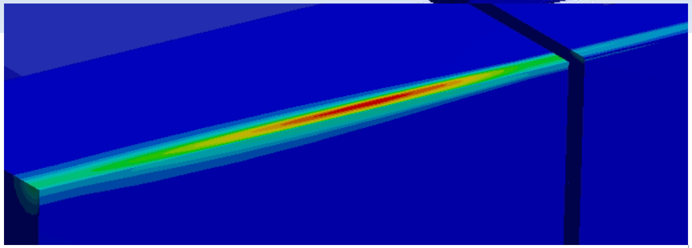


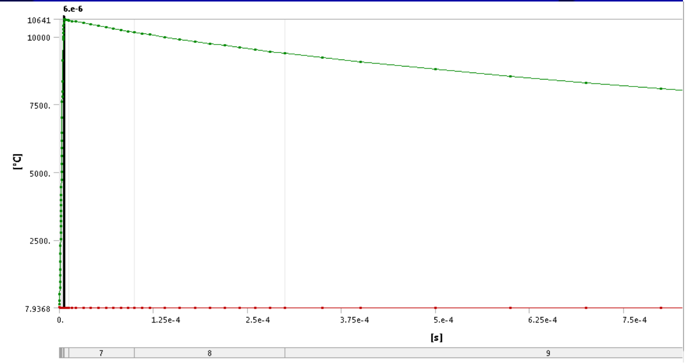
TRANSIENT THERMAL ANALYSIS OF THE BEAM SCRAPER

Results

- The highest computed temperature was >10000°C
  - Computations were accurate only until the computed temperature reached the melting point of tungsten
- The area with the highest temperatures was approximately 0.5 mm away from the surface of the material
  - This was in a good agreement with the volumetric distribution of the absorbed energy calculated by MARS
  - The data import in ANSYS was successful
- The temperature change over time indicated that the melting of tungsten started immediately after contact between the particle beam and the tungsten block
  - There was no need for structural analysis







TRANSIENT THERMAL ANALYSIS OF THE BEAM SCRAPER

The Main Challenge

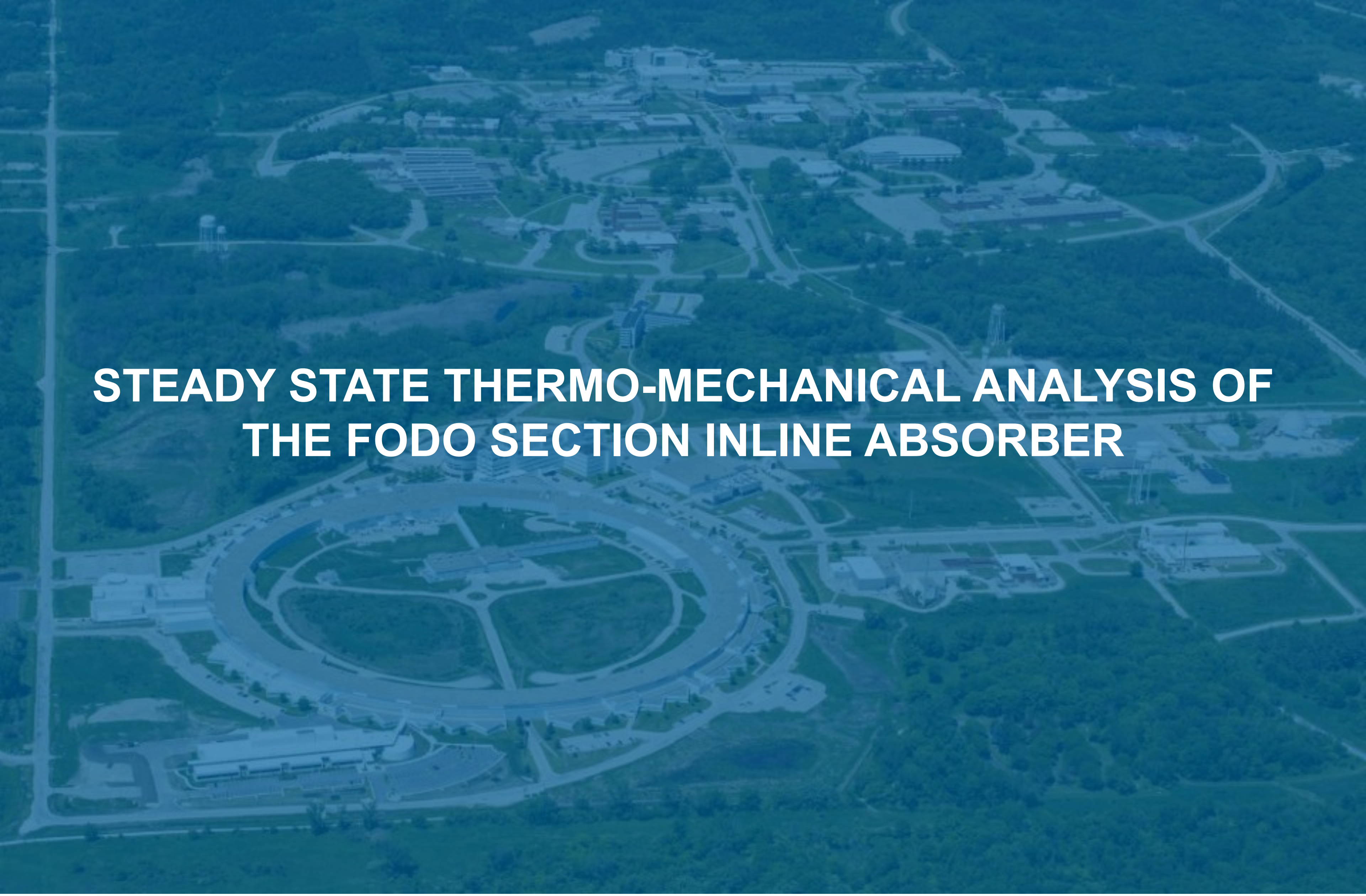
- The location of the model
  - ANSYS has to have an identical global coordinate system to the one in MARS,
  - The location of the model in ANSYS had to be in the exactly same location as in MARS.
- Accurate data input
  - Correct tabular distribution of input data was necessary
    - The excel table had to have a particular form for correct data input
  - The mesh had to be sensitive (fine) enough for correct interpolation of the input data
- Computations were very time consuming due to the extremely short initial time steps (10^{-7} sec)

X-coor.

Z-coor.

	A	B	C	D
1	2.03730E+01	-4.97500E+00	-4.92500E+00	-4.87500E+00
2	-1.99750E+01	1.87851E+05	1.45755E+05	1.93283E+04
3	-1.99250E+01	3.42801E+04	1.76024E+05	1.69340E+05
4	-1.98750E+01	3.68253E+04	2.23010E+04	1.65423E+05
5	-1.98250E+01	4.53617E+04	4.92466E+04	3.26166E+06
6	-1.97750E+01	1.74787E+05	9.17388E+04	8.57998E+04
7	-1.97250E+01	3.55525E+04	1.38988E+06	7.04175E+04

Y-coor.

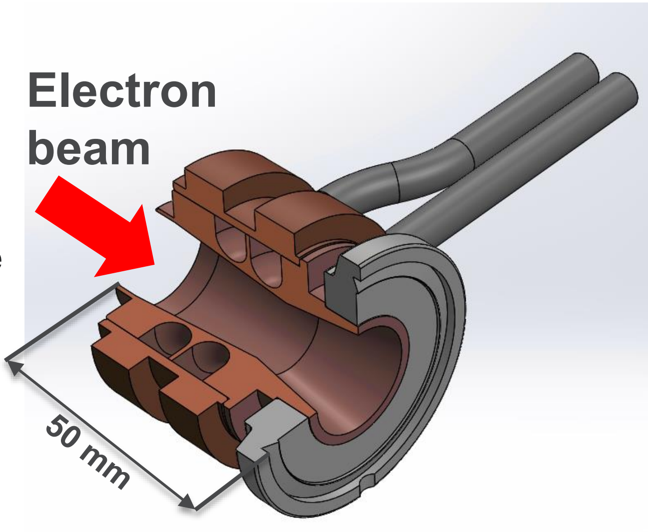


STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

Inline absorbers are integral parts of the vacuum chambers in the FODO section

- Objective
  - Optimize the cooling of the absorber
    - Inline absorbers create a shadow and protect the downstream components of the vacuum system from the synchrotron radiation
    - They are located just in front of the components they protect (BPMs, gate valves, vacuum crosses and flanges) thus they have to fit in a very limited space
    - As a consequence they have to have a dedicated, efficient, and compact cooling system



The inline absorber

- Analysis strategy
  - Use FEA analysis to optimize cooling
    - Keep the stresses within an acceptable range
    - Keep the cooling channel wall temperatures significantly below onset of boiling

STEADY STATE THERMO-MECHANICAL ANALYSIS OF FODO SECTION INLINE ABSORBER

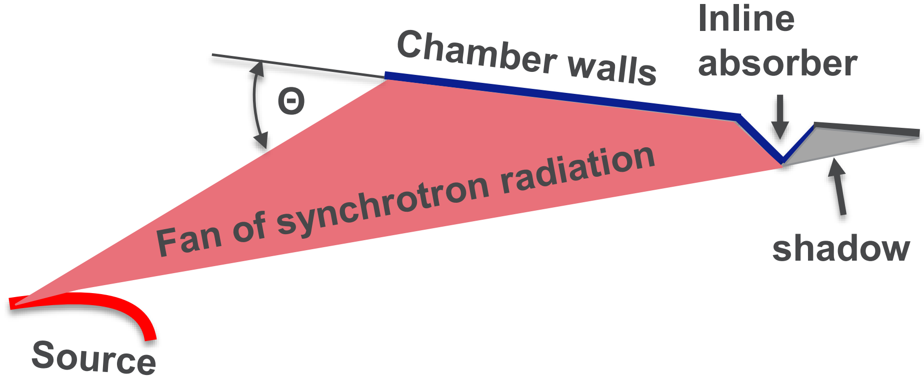
Physics

- Same as in the crotch absorber case

$$P_{d0}(W/mrad^2) = 5.421 * E_e^4(GeV) * I(A) * B(T).$$

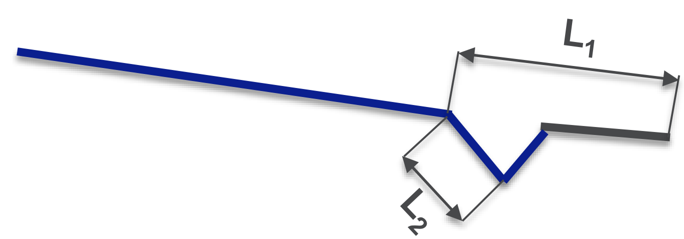
$$P_d(W/mrad^2) = P_{d0}\frac{1}{(1+X^2)^{5/2}}\left[1+\frac{5}{7}\frac{X^2}{1+X^2}\right];\ X = \gamma\psi,$$

$$pP_d(W/mm^2) = P_d * \sin\theta / l^2$$



RF synchrotron radiation hitting the inline absorber

- The power incident on the inline absorber:
  - Has a higher peak value due to the increase in incident angle, Θ
    - Thus the peak values of the heat flux absorbed at the surface of the inline absorber wall are higher
  - Has higher per length unit value as the absorber intercepts the fan that would otherwise hit downstream components
    - Total heat power absorbed by inline absorber is higher than total heat power absorbed by the chamber wall of the same length (by factor of L_1/L_2).



Length ratio of the inline absorber to the protected area

STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

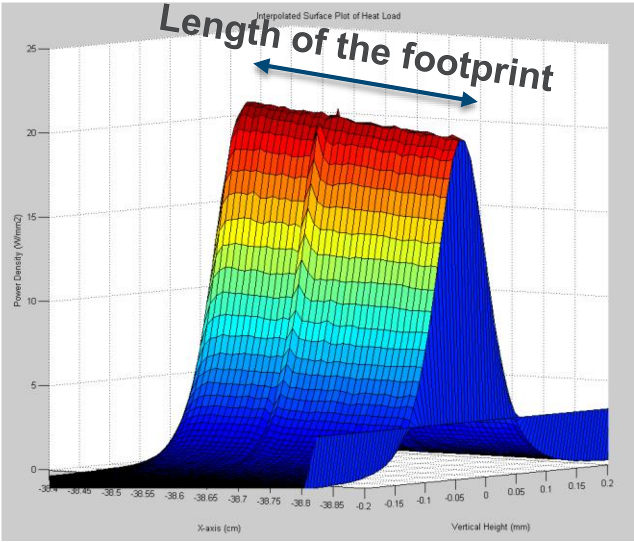
Homework

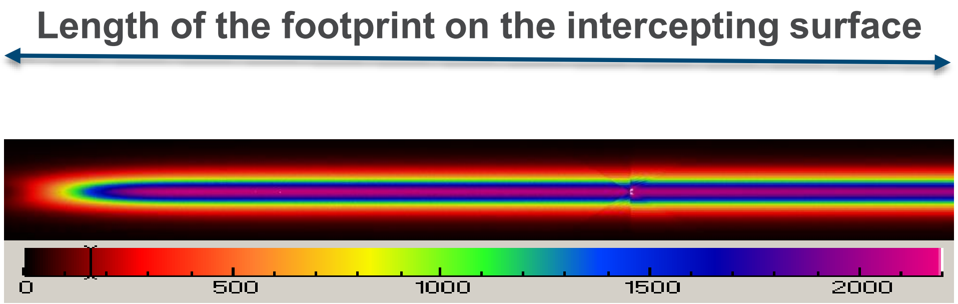
- Use the SynRad+ software package to calculate and apply heat loads
  - Synrad+ is a software package developed at CERN in late 2012
    - The developers of the package are Roberto Kersevan and Marton Adu
    - The package was released to public in 2013.
  - It is used for calculating the flux and power distribution of synchroton radiation in geometries of arbitrary complexity
  - SynRad+ traces photons to calculate the flux and power distribution
- Develop a method to accurately apply heat fluxes calculated with SynRad+ to the model used in ANSYS analysis
  - SynRad+ is a Monte Carlo based program and the ‘smoothness’ of the data depends on the number of ‘hits’
    - A higher number of hits will produce a ‘smoother’ power distribution data,
    - A higher number of hits will require longer computational time
  - SynRad uses 2D surface meshes, but ANSYS models are 3D bodies
    - A denser mesh with a larger number of small mesh elements will give ‘smoother’ results
    - Small mesh elements in a 2D mesh have a smaller impact on computation time than in a 3D mesh
    - For best results, during data import, the size of the elements in the critical direction has to be similar in both packages.

STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

The computational results in SynRad+, courtesy of Jason Carter

- SynRad+ calculates the vertical power distribution of synchrotron radiation and projects it along the intercepting surfaces

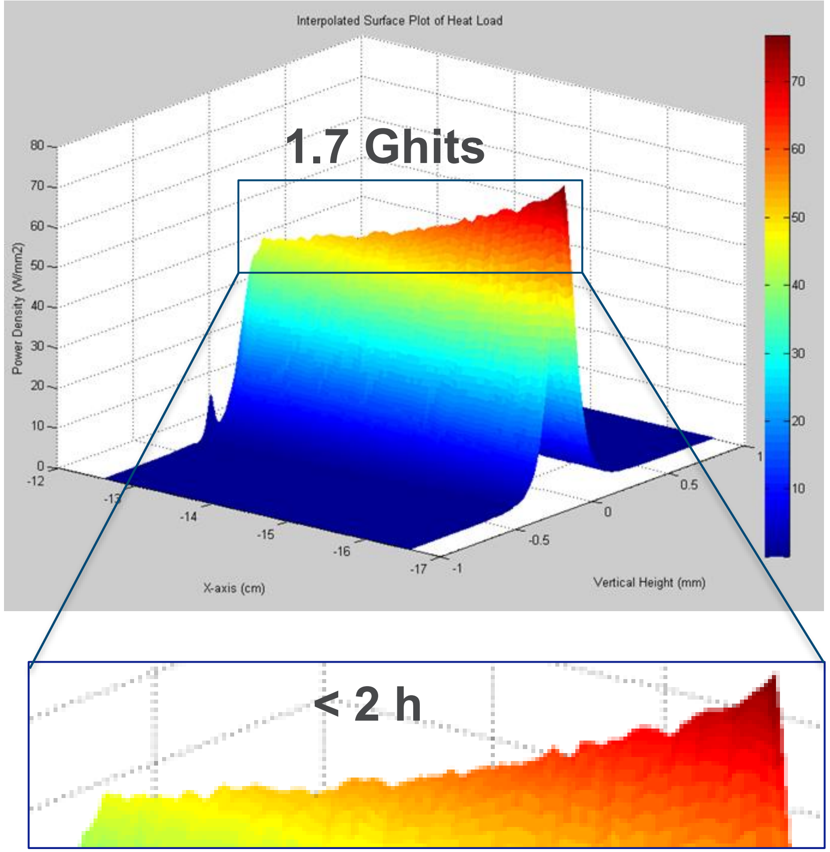




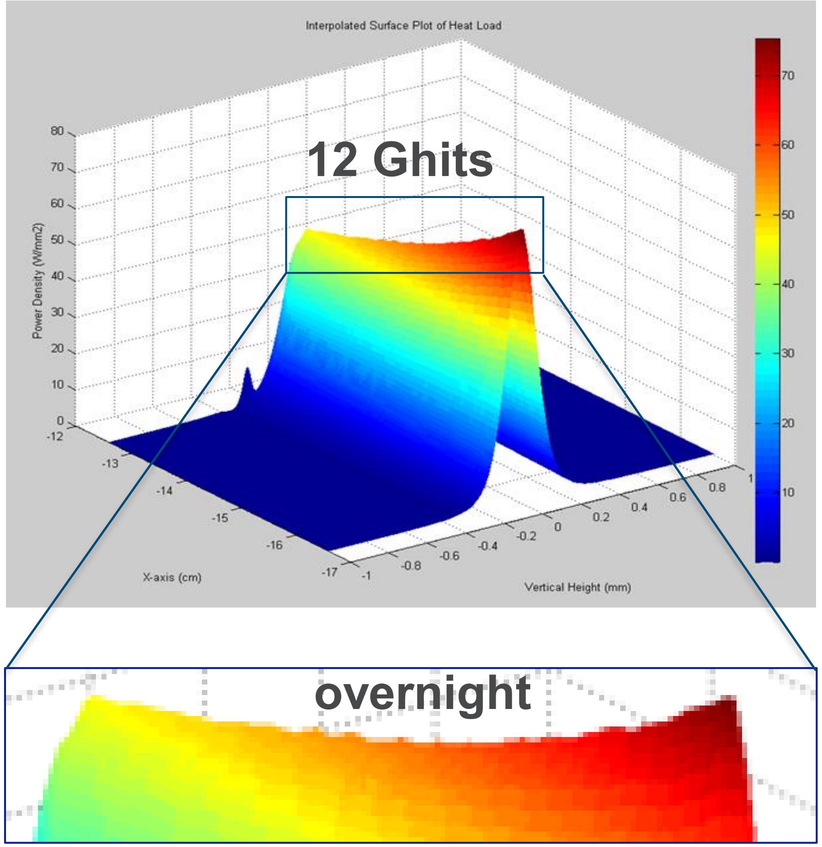
- If one surface sees radiation from multiple sources, or one source 'shines' on multiple surfaces, SynRad+ photon tracking accounts for this

STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

The effect of number of hits on the results of Synrad+ computations



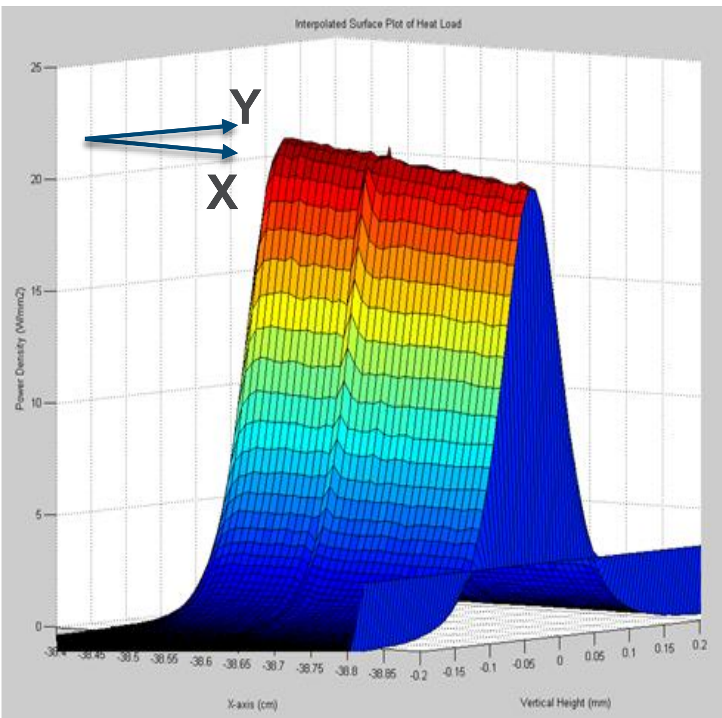
Computed power distribution at 1.7 Ghits



Computed power distribution at 12 Ghits

STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

SynRad+ VS ANSYS mesh element size



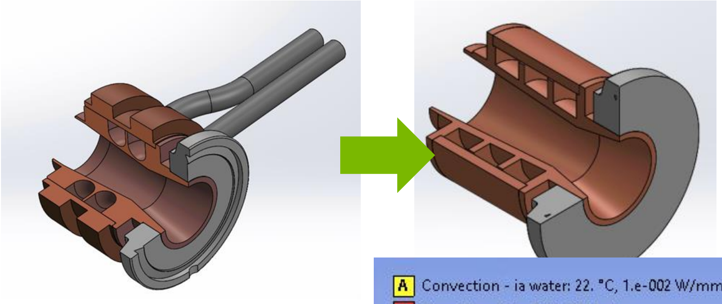
Power distribution showing the mesh used in computation

- SYNRAD+ has a 2D mesh with square elements
  - A square mesh facilitates data analysis
  - In our case, the element size was 20x20 microns
- The ANSYS mesh is a 3D mesh
  - Having 20 micron cubic mesh elements would be highly impractical
  - To capture the power distribution in a proper way, the vertical dimension of mesh elements is kept at 20 microns in the high mesh density area of the beam footprint
  - The other two dimensions of the mesh are determined from
    - The requirement that the walls exposed to bending have at least 3 layers of elements
    - The requirement that the mesh aspect ratio should not exceed 20:1

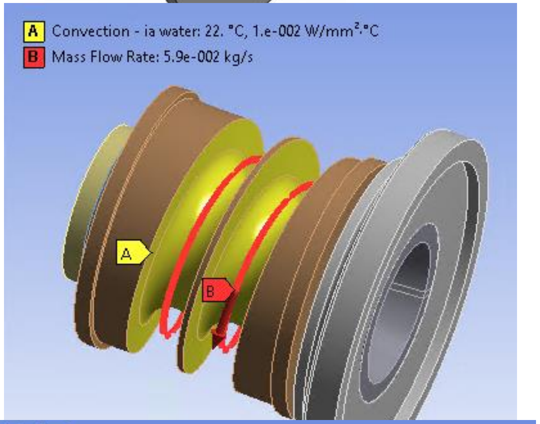
STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

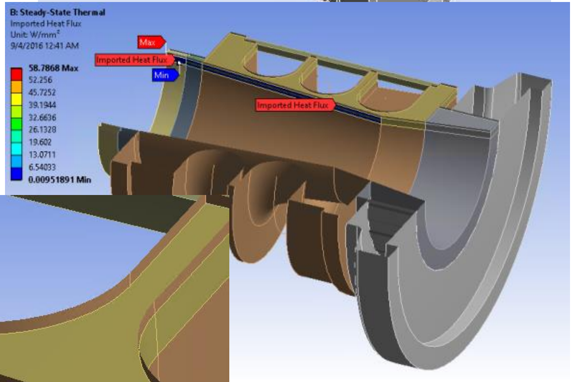
Boundary conditions

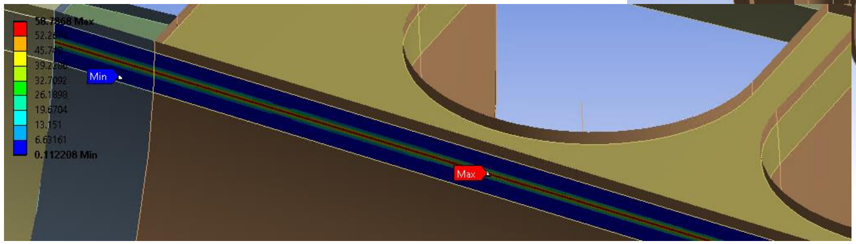
- Geometry
  - The geometry was simplified,
  - Only the relevant parts were analyzed



- Thermal analysis
  - Cooling
    - A convective heat transfer coefficient of 5 kW/m^{2}K was used
    - A thermal fluid element was introduced to simulate the water temperature change.
  - Thermal load
    - Heat flux values computed with SynRad+ were imported using External Data module



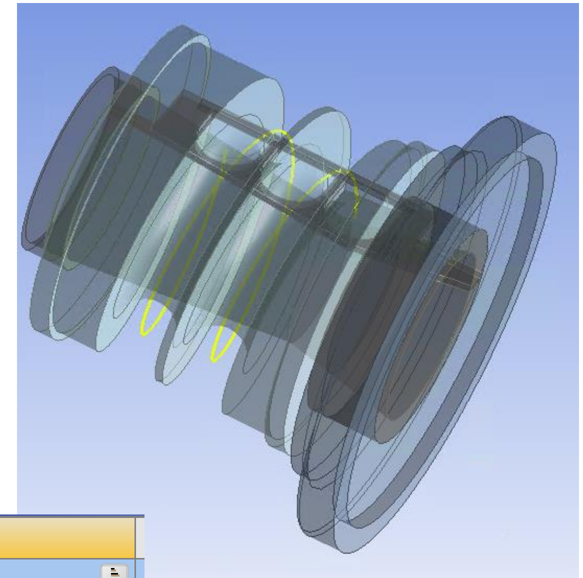


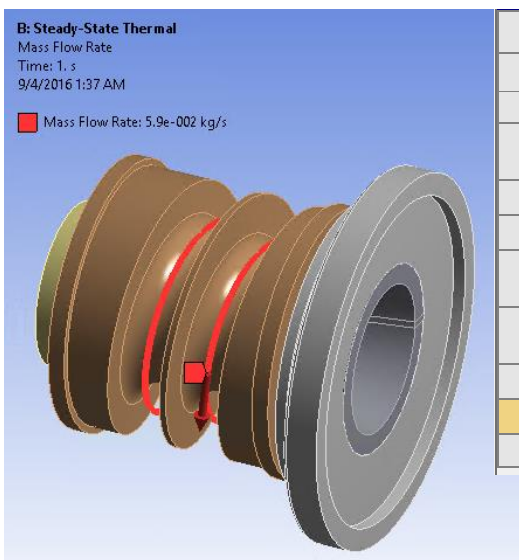


STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

Boundary conditions – Thermal fluid element (ANSYS specific)

- The geometry was created in Design Modeler
  - One-dimensional element
  - Follows the cooling channel trajectory
- Material properties defined in the engineering data module
  - The water liquid properties were taken from the ANSYS data library
- The mass flow rate was assigned as a load in the thermal analysis module



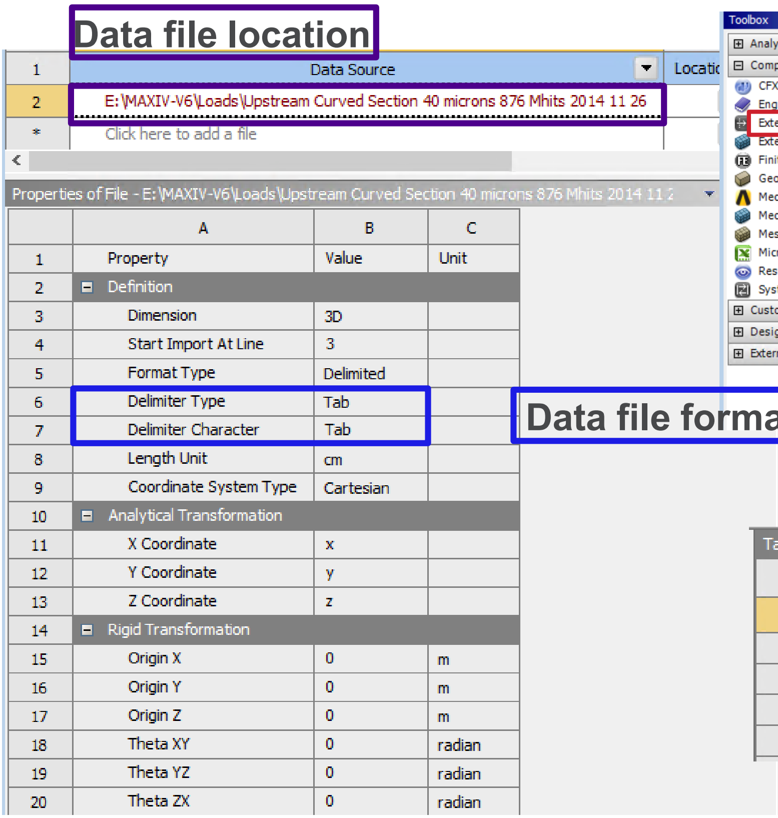


	A
1	Contents of Engineering Data
2	Material
3	6063 (UNS A96063)
4	Glidcop Al-15
5	OFE Cu
6	SS 316L
7	Structural Steel
8	UNS C10400 (UNS C10400)
9	Water Liquid
*	Click here to add a new material

STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

Boundary conditions – External data import (ANSYS specific)

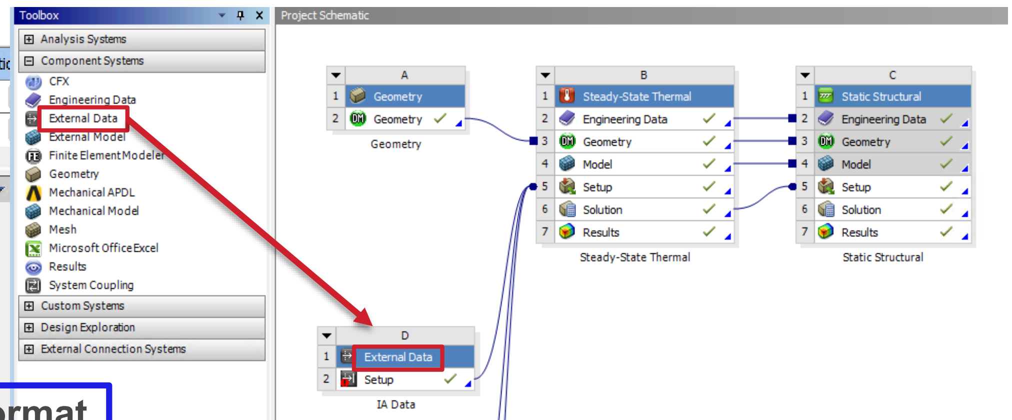
- External data module used for data import
- Setup simple, with three windows where data file location, data file format and data types and units are easily defined



	Data Source	Locatio
1	Data Source	Locatio
2	E:\MAXIV-V6\Loads\Upstream Curved Section 40 microns 876 Mhits 2014 11 26	
*	Click here to add a file	

Properties of File - E:\MAXIV-V6\Loads\Upstream Curved Section 40 microns 876 Mhits 2014 11 2

	A	B	C
1	Property	Value	Unit
2	⊟ Definition		
3	Dimension	3D	
4	Start Import At Line	3	
5	Format Type	Delimited	
6	Delimiter Type	Tab	
7	Delimiter Character	Tab	
8	Length Unit	cm	
9	Coordinate System Type	Cartesian	
10	⊟ Analytical Transformation		
11	X Coordinate	x	
12	Y Coordinate	y	
13	Z Coordinate	z	
14	⊟ Rigid Transformation		
15	Origin X	0	m
16	Origin Y	0	m
17	Origin Z	0	m
18	Theta XY	0	radian
19	Theta YZ	0	radian
20	Theta ZX	0	radian



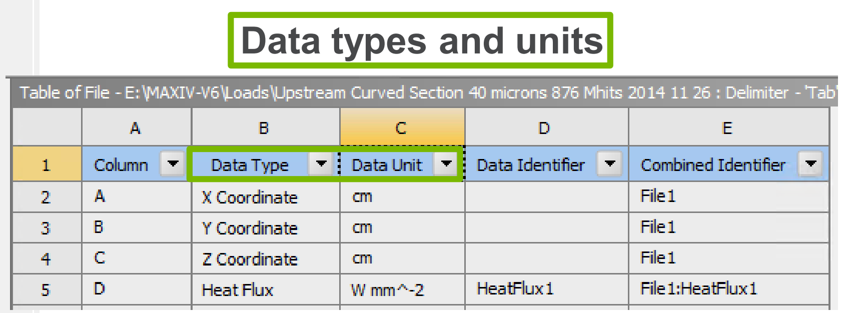


Table of File - E:\MAXIV-V6\Loads\Upstream Curved Section 40 microns 876 Mhits 2014 11 26 : Delimiter - 'Tab'

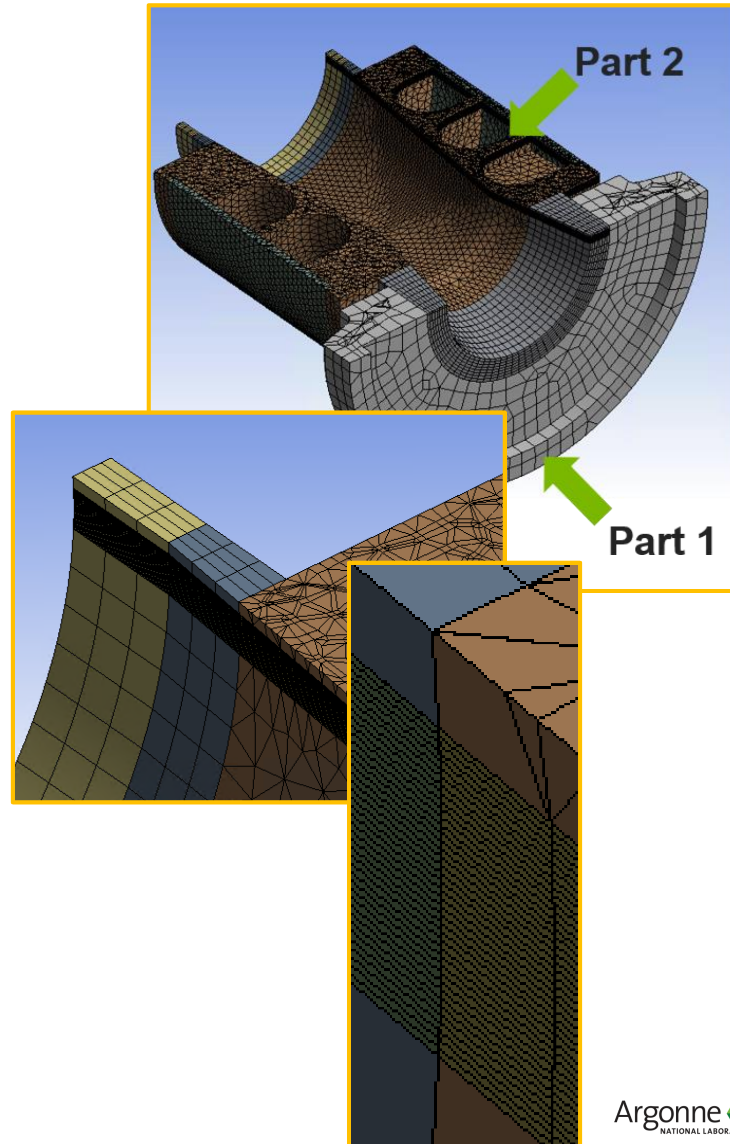
	A	B	C	D	E
1	Column	Data Type	Data Unit	Data Identifier	Combined Identifier
2	A	X Coordinate	cm		File1
3	B	Y Coordinate	cm		File1
4	C	Z Coordinate	cm		File1
5	D	Heat Flux	W mm^-2	HeatFlux1	File1:HeatFlux1

Argonne NATIONAL LABORATORY

STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

Meshing and contacts

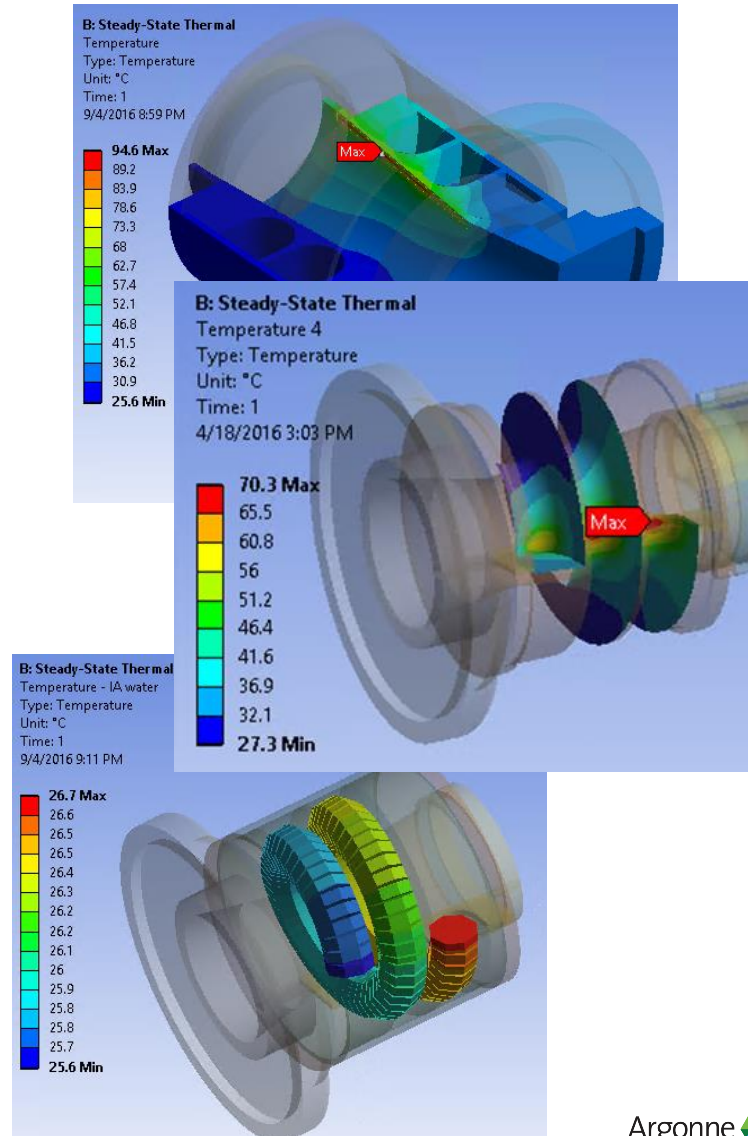
- Combination of standard part and multibody part meshing
  - The copper structure exposed to the synchrotron radiation is represented as multibody Part 2
    - The geometry of Part 2 is broken into five pairs of bodies to promote a Sweep method in meshing
    - All the pairs were broken into a narrow member representing the area of direct contact with the beam, which has a very dense mesh, and a member representing the bulk of the pair with a mesh that has a much lower density
  - As the bodies of the copper structure belong to a single part, the nodes on the contacting faces match
- As the model consisted of only two parts, a single contact was created



STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

Results

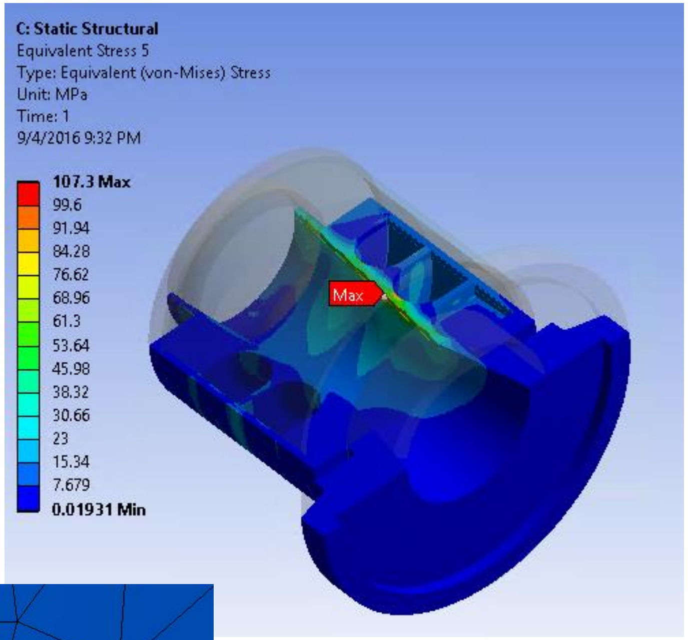
- Temperatures
  - The maximum computed temperature is 94.6 °C
    - It is located on the tilted surface of the absorber, right in the middle of the area of contact with the beam
  - The maximum cooling channel wall temperature is 70.3 °C
    - It is less than 50 °C higher than the temperature of cooling water
  - The temperature of cooling water rose from 25.6 °C at the inlet to 26.7 °C at the outlet

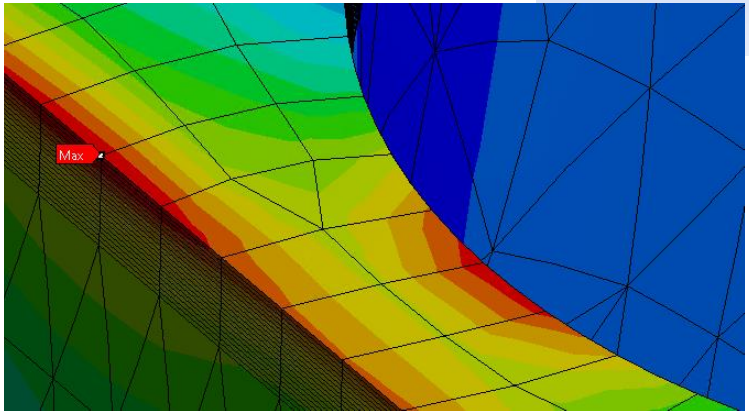


STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

Results

- Stresses
  - The maximum computed equivalent stress is 107.3 MPa
    - The temperature at the same location is 88.6°C and the yield strength of Glidcop AL15 at that temperature is ~325 MPa
  - The stress computed at the location of maximum computed temperature is 85.5 Mpa
    - The yield strength of Glidcop AL15 at the temperature of 94.6 °C is ~320 MPa



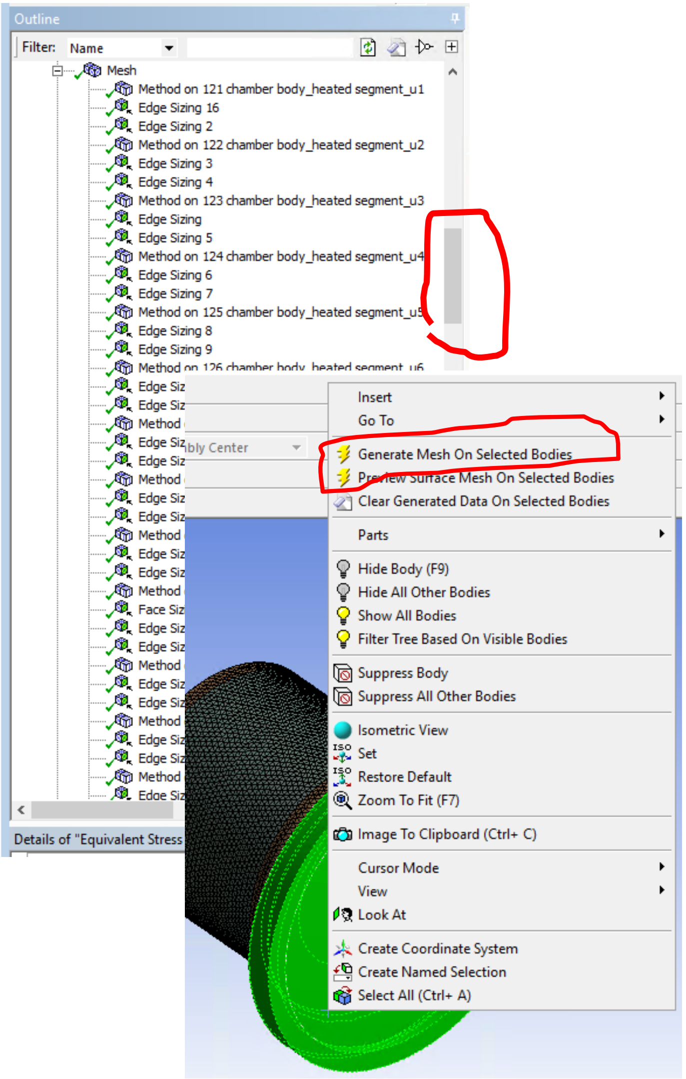


STEADY STATE THERMO-MECHANICAL ANALYSIS OF THE FODO SECTION INLINE ABSORBER

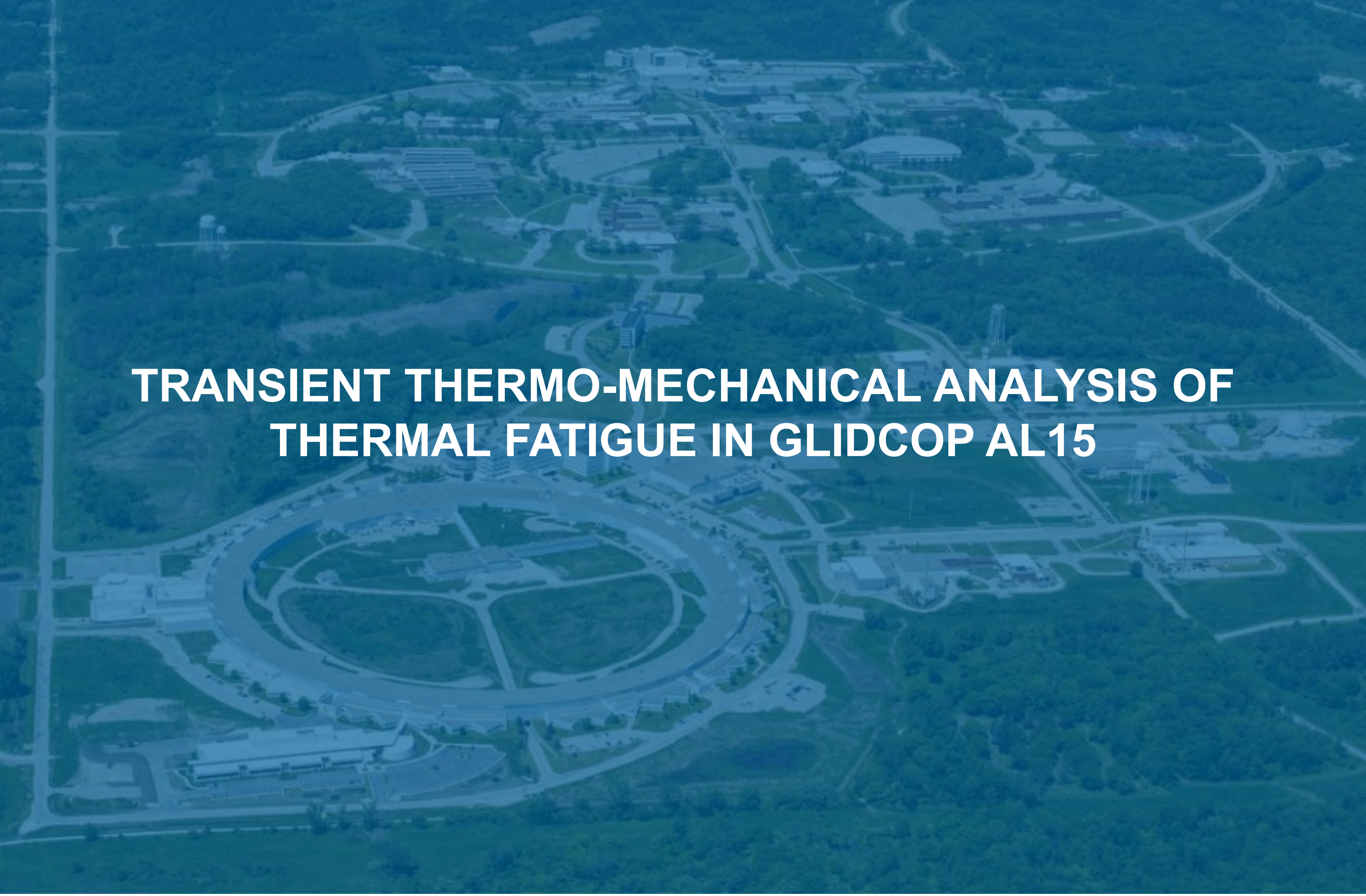
The Main Challenge

- Contrary to the previous two cases, and thanks to our talented young colleague, Jason Carter, and the simplified import procedure in ANSYS, the data import related issues were NOT the biggest challenge

- Meshing was the biggest challenge!
  - The inline absorber was only a small part of the large vacuum chamber that was analyzed, and number of standard and multibody parts was much higher
    - Thus the number of mesh control commands was significant
  - A step-by-step meshing of the individual parts was more efficient than automatic meshing



TRANSIENT THERMO-MECHANICAL ANALYSIS OF THERMAL FATIGUE IN GLIDCOP AL15

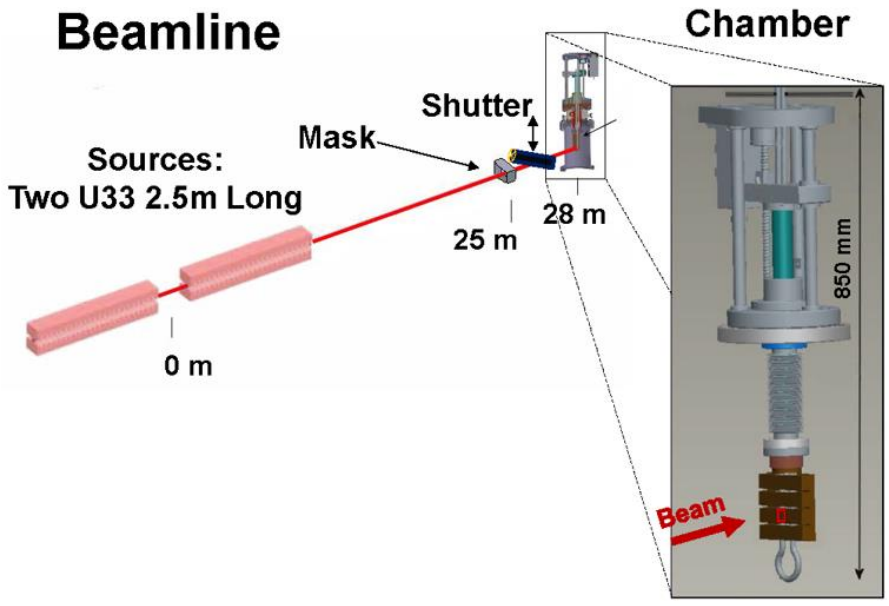


Argonne NATIONAL LABORATORY

TRANSIENT THERMO-MECHANICAL ANALYSIS OF THERMAL FATIGUE IN GLIDCOP AL15

Courtesy of Jeremy Nudell

- Objective
  - Establish the thermal fatigue limits of Glidcop Al-15
- Analysis strategy
  - Combine a beamline experiment with a nonlinear FEA model of the experiment to correlate the stress state of the material to the observed failure



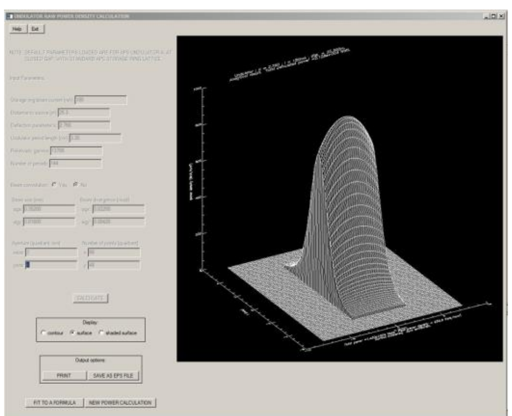
Schematics of the experiment

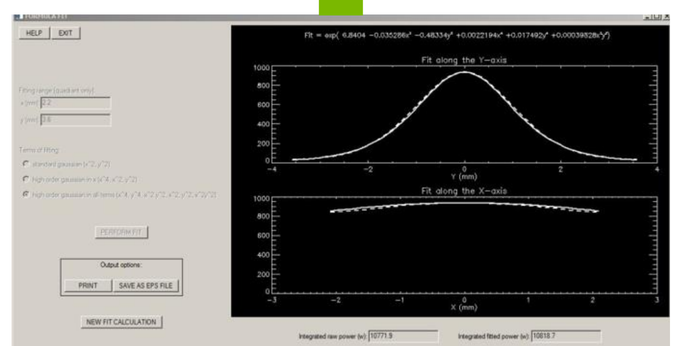
TRANSIENT THERMO-MECHANICAL ANALYSIS OF THERMAL FATIGUE IN GLIDCOP AL15

Physics

- The beam is a photon beam generated with two 2.5 m long inline U33 undulators!
- The total power and power density distribution was calculated using the SRUFF code developed at APS
  - SRUFF gives, as one of the outputs, the polynomial approximation of the beam power distribution
    - The computed power distribution is imported in ANSYS as a heat flux load using a command snippet
    - Normal incidence of the undulator beam was simulated

$$\text{Fit} = \exp(6.8404 - 0.035286x^2 - 0.48334y^2 + 0.0022194x^4 + 0.017492y^4 + 0.00039828x^2y^2)$$



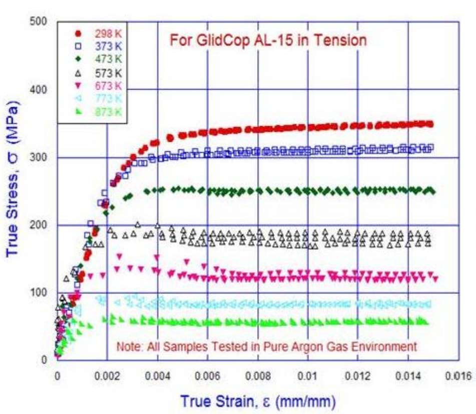


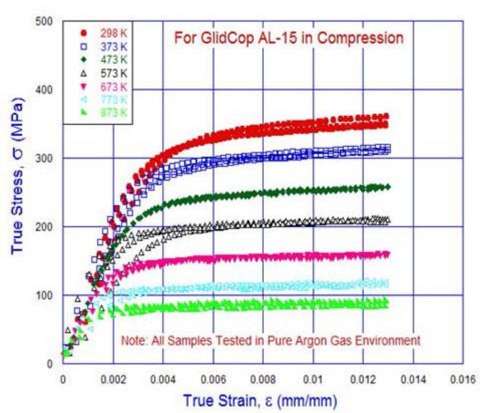
SRUFF results window

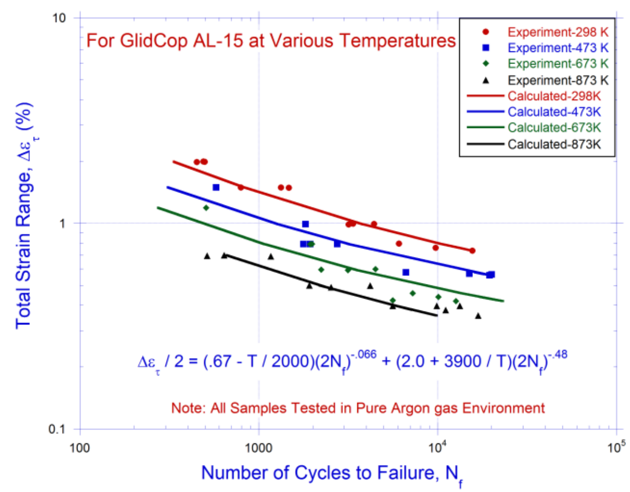
TRANSIENT THERMO-MECHANICAL ANALYSIS OF THERMAL FATIGUE IN GLIDCOP AL15

Homework

- The temperature dependent material properties of Glidcop AL15 were extensively researched
  - Physical properties relevant for the analysis were obtained from the literature
  - The mechanical properties of Glidcop were experimentally determined
    - True-stress-versus-true-strain tests were conducted in the 100-600 °C range with 100 °C increments
    - Based on test results, poly-linear stress-strain curves were developed to be used in simulations
  - Uniaxial mechanical fatigue tests were conducted at 20 °C, 200 °C, 400 °C, and 600 °C and the uniaxial model, described with the empirical equation for temperature-dependent total strain as a function of number of cycles to failure, was developed
  - The temperature-dependent uniaxial mechanical model was transformed into thermal fatigue model



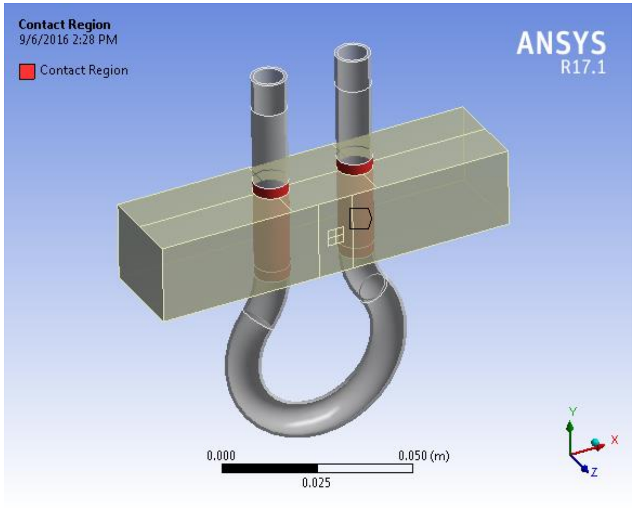




TRANSIENT THERMO-MECHANICAL ANALYSIS OF THERMAL FATIGUE IN GLIDCOP AL15

Boundary conditions

- The actual geometry did not require further simplification
- Thermal analysis
  - A convective heat transfer coefficient of 5 kW/m^2K was applied to the inner surfaces of the cooling tube
  - The power distribution calculated with the snippet containing the polynomial developed with SRUFF was applied to the limited area of the beam footprint
- Structural analysis
  - The temperature distribution computed in thermal analysis was used as a load,
  - The Weak Link option was turned on in order to stabilize the model
  - The Large Deflection control was turned on in order to capture the plastic deformation

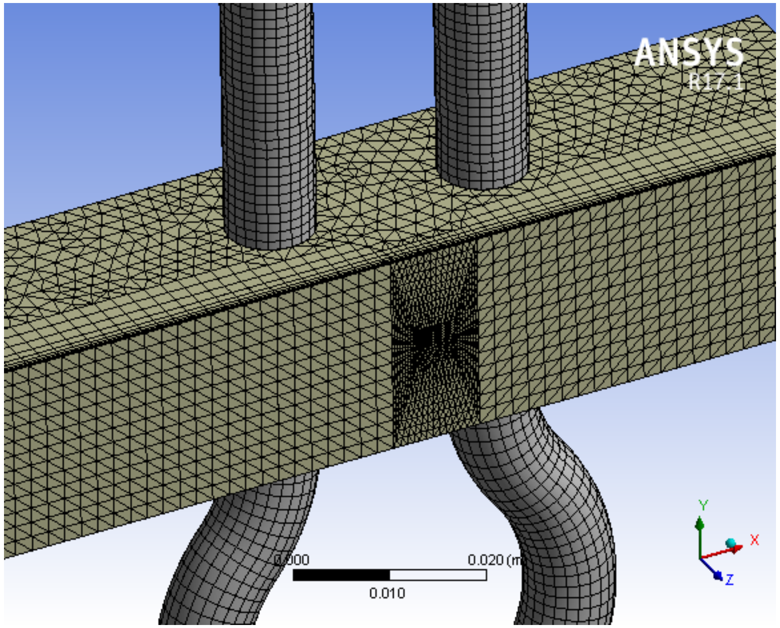


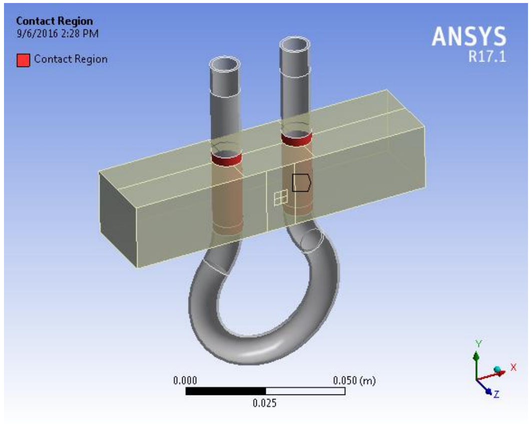
Argonne NATIONAL LABORATORY

TRANSIENT THERMO-MECHANICAL ANALYSIS OF THERMAL FATIGUE IN GLIDCOP AL15

Meshing and contacts

- A simple tetrahedral mesh was used to mesh the target exposed to the photon beam
  - Due to the effect of the mesh density on the already long computational times, a lot of effort was invested in the mesh density optimization
    - The most dense mesh was implemented in the area of beam footprint in order to capture the beam power distribution
- Contacts
  - Bonded contacts between the target and cooling tube were applied
    - The contacts were far enough from the region with the highest computed stresses such that there were no issues

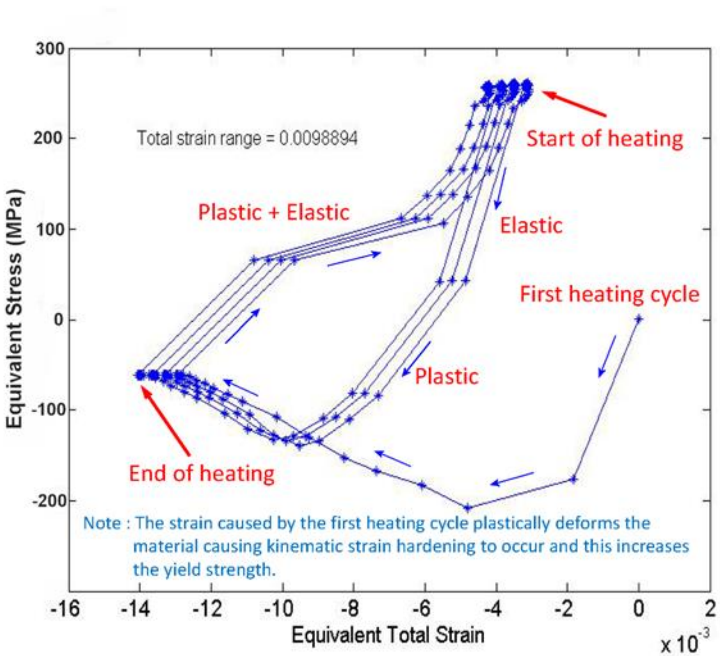




TRANSIENT THERMO-MECHANICAL ANALYSIS OF THERMAL FATIGUE IN GLIDCOP AL15

Results

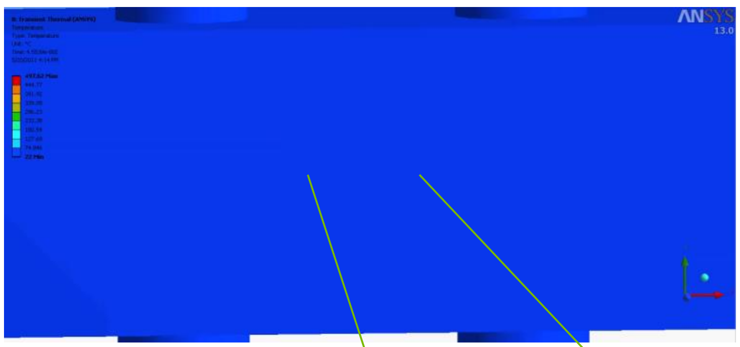
Hysteresis plots showing the strain range for several heating/cooling cycles were created for all sample conditions

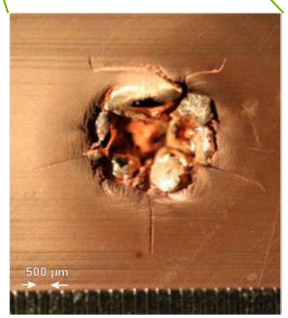


Stress-strain hysteresis loop

Reference: J. Collins, "The Establishment of New Design Criteria for GlidCop® X-ray Absorbers"

Temperature change during one thermal cycle





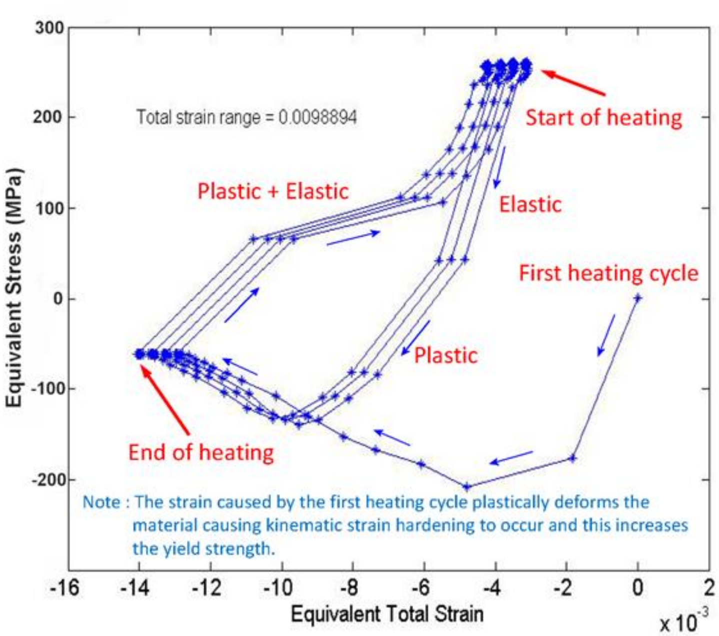
Damage from thermal cycling

Argonne NATIONAL LABORATORY

TRANSIENT THERMO-MECHANICAL ANALYSIS OF THERMAL FATIGUE IN GLIDCOP AL15

Validation flow diagram

Temperature dependent true stress VS true strain mechanical tests → Poly-linear stress-strain curve, f(t). → Cyclical transent thermo-structural analysis (plastic)



Beamline experiment

Uniaxial Mechanical fatigue tests → Uniaxial mechanical fatigue equation, f(t) → Thermal fatigue equation, f(t)

TRANSIENT THERMO-MECHANICAL ANALYSIS OF THERMAL FATIGUE IN GLIDCOP AL15

The challenges

- **There were many challenges!**
  - Accurate modeling of the experiment
    - Inaccuracies were mitigated with comprehensive testing of nonlinear material properties
  - Problems with convergence
    - Experimentation with mesh size and number of substeps
  - Analyzing output from ANSYS in order to produce a useful hysteresis plot
    - Created MATLAB program to automate data analysis
  - Extremely long run times
    - Utilize symmetry, start with simpler problem first

QUESTIONS?

NON-MAINSTREAM USES TO FEA

Argonne NATIONAL LABORATORY

NEW HORIZONS!

BRAN BRAJUSKOVIC
Principal Mechanical Engineer
Mechanical Engineering and Design Group
Advanced Photon Source
Argonne National Laboratory

September 12th, 2016
Barcelona

MEDSI 2016 BARCELONA

OUTLINE

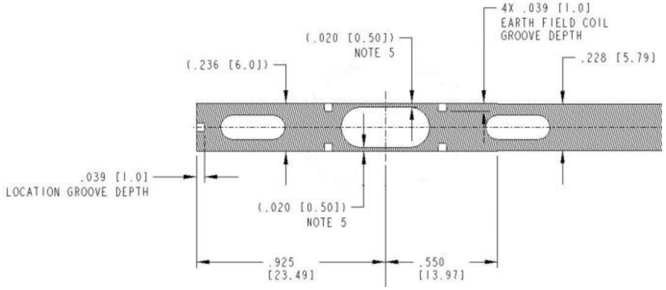
- CFD in the design of synchrotron instrumentation
- Multiphysics analysis in the design of RF cavities
- Tolerance Analyses of Quadrupole Magnets for the Advanced Photon Source Upgrade
- Topology optimization for the APS-U magnet support structure
- Structural dynamics simulation
- Analysis of acoustic levitation supports

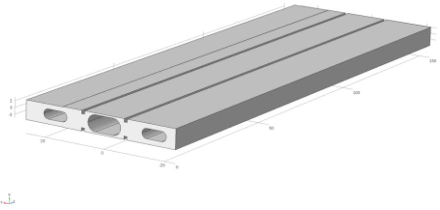
USE OF CFD IN THE DESIGN OF SYNCHROTRON INSTRUMENTATION

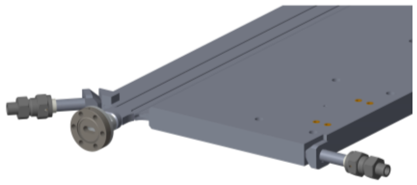
CFD IN THE DESIGN OF SYNCHROTRON INSTRUMENTATION

An example of the right and wrong use of CFD (in the same analysis)—Courtesy of Jason Lurch

- Objective
  - To design and manufacture Undulator vacuum chambers for the Hard X-Ray and Soft X-Ray line for SLAC's LCLS-II Project
- Strategy
  - Use coupled CFD and FEA in the design process due to the specific design constraints
    - The temperature of the vacuum chamber should remain in 20±0.1°C range
    - The total heat load to be removed is 3.3 W
    - The water velocity should not exceed 3 m/s
    - The material of the chamber is Aluminum 6063-T5
    - The geometry of the chamber is given
  - The CFD module determines the convective heat transfer parameters and FEA computes temperature distribution in the chamber walls







CFD IN THE DESIGN OF SYNCHROTRON INSTRUMENTATION

The Dilemma! How do we determine the convective heat transfer parameters?

- Using an analytical approach to compute convective heat transfer parameters
  - Use equations for Re and Nu number to calculate coefficient of convective heat transfer
  - Use energy balance to determine temperature of the cooling fluid (*ambient temperature* in ANSYS, *external temperature* in COMSOL) where temperature of the cooling fluid can be:
    - Constant with the value equal to the average of inlet and the outlet temperature
    - Linearly approximated temperature varying with the distance
- Using CFD to compute convective heat transfer parameters

CFD IN THE DESIGN OF SYNCHROTRON INSTRUMENTATION

The Dilemma!

$$Q = \dot{m} c_p \Delta T \rightarrow \Delta T = \frac{Q}{\dot{m} c_p} \rightarrow T_{ext} - T_i = \frac{\frac{Q}{L} * L}{c_{p_{water}} * \rho_{water} * \bar{U} * A}$$

$$\therefore T_{ext} = \frac{\frac{Q}{L} * L}{c_{p_{water}} * \rho_{water} * \bar{U} * A} + T_i$$

Where:

- $Q/_L$ is the energy per length
- T_{ext} is the external fluid temperature away from the boundary wall
- T_i is the initial temperature
- $\bar{U}$ is the average velocity of the fluid
- ρ_{water} is the density of water
- $c_{p_{water}}$ is the specific heat of water
- L is the length of the system
- A is the cross sectional area of the cooling channel

$$\therefore T_{ext} = \frac{3.3\frac{W}{m} * 3.8m}{4182\frac{J}{kg \cdot K}\left(2\left(998\frac{kg}{m^3} * 2.2\frac{m}{s} * 2.21E-5m^2\right)\right)} + 20°C \cong 20.031°C$$

This?

$$Re = \frac{\bar{U} D_H}{\nu} = \frac{(2.2[\frac{m}{s}])(\frac{4(2.21E-5[m^2])}{0.01942[m]})}{1.004E-6[\frac{m^2}{s}]} \cong 9{,}975 \quad ; \quad D_H = \frac{4A}{P}$$

$$Pr = \frac{c_p \mu}{k} = \frac{4182\left[\frac{J}{kg \cdot K}\right] * 1.002E-3[Pa \cdot s]}{0.6[\frac{W}{m \cdot K}]} = 6.98$$

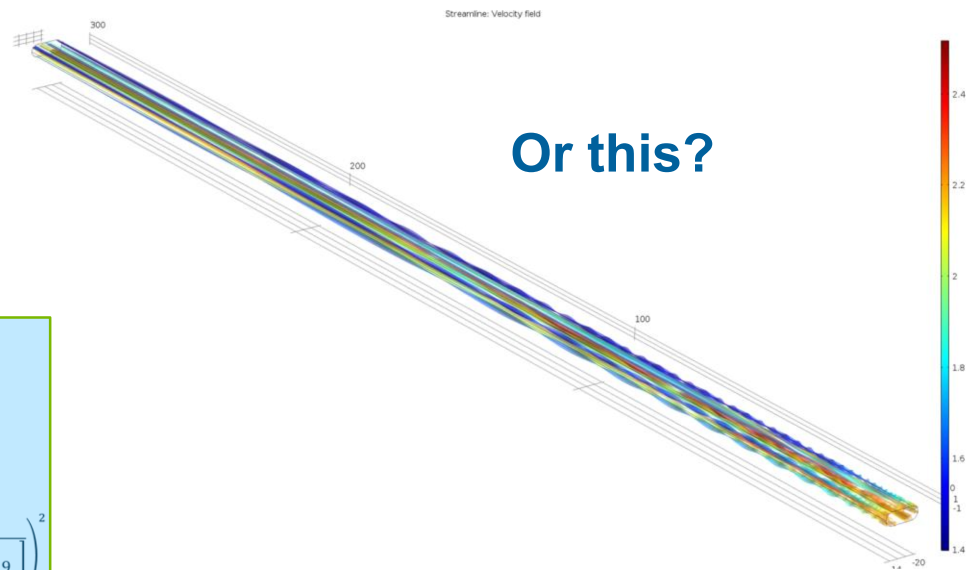
$$f = \left(\frac{1}{-1.8log\left[\left(\frac{\epsilon/D_H}{3.7}\right)^{1.11} + \frac{6.9}{Re}\right]}\right)^2 = \left(\frac{1}{-1.8log\left[\left(\frac{1.5E-6/0.0045[m]}{3.7}\right)^{1.11} + \frac{6.9}{9975}\right]}\right)^2$$

$$\cong 0.0313$$

$$Nu = \frac{(f/8)(Re-1000)Pr}{1 + 12.7(f/8)^{1/2}(Pr^{2/3} - 1)} = \frac{(0.0313/8)(9975-1000)6.98}{1 + 12.7(0.0313/8)^{1/2}(6.98^{2/3} - 1)} \cong 78.89$$

$$\therefore h = \frac{Nu \cdot k}{D_H} = \frac{(78.89)\left(0.6\left[\frac{W}{m \cdot K}\right]\right)}{0.0045[m]} = 10{,}518\left[\frac{W}{m^2 \cdot K}\right]$$

Or this?



CFD IN THE DESIGN OF SYNCHROTRON INSTRUMENTATION

Resolution of the Dilemma!

Chamber Aperture Temperature Comparison			
Method	Aperture Temp [z=0m] (°C)	Aperture End Temp [z=3.8m] (°C)	ΔT (°C)
Constant T_{ext}	20.03	20.07	0.07
Linear Approximation	20.00	20.07	0.07
CFD	20.00	20.09	0.09

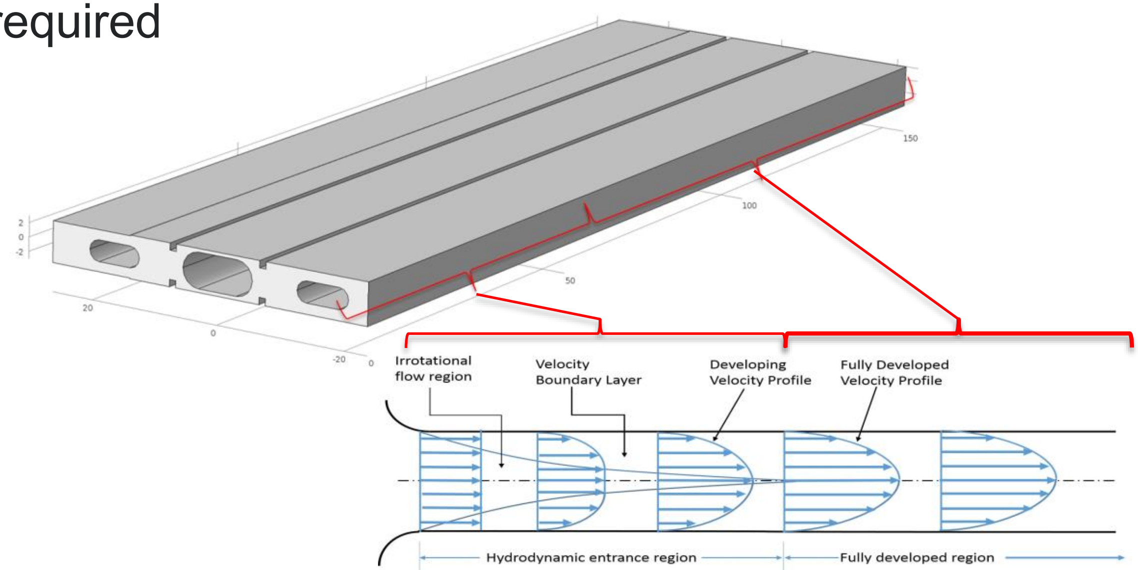
Solution Time			
Method	Solution Time (min)	Post Processing Time (min)	Total Time (min)
Constant T_{ext}	0.57	0	0.57
Linear Approximation	0.60	10	10.6
CFD	73.33	10	83.33

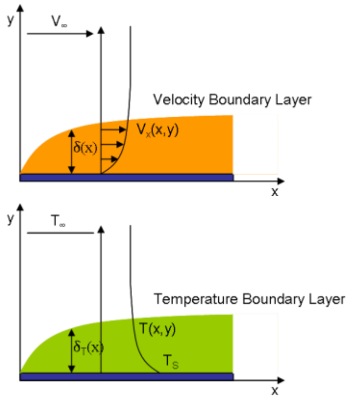
- CFD gave us
  - A more conservative estimate. The max. computed temperature was approximately 0.1% higher than with the analytical approach
  - A considerably longer computational time
    - 14620% longer when compared with constant temperature approach
    - 786% longer when compared with linear temperature approximation approach

CFD IN THE DESIGN OF SYNCHROTRON INSTRUMENTATION

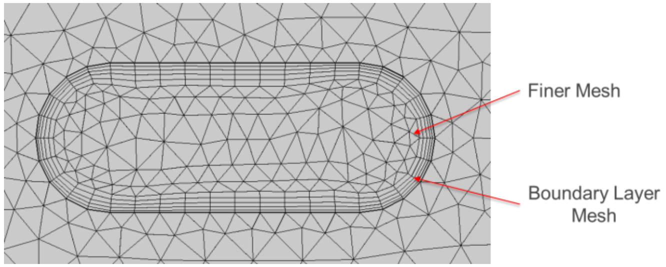
And all of this without hidden costs!

- A more detailed understanding of physics and more extensive homework is required





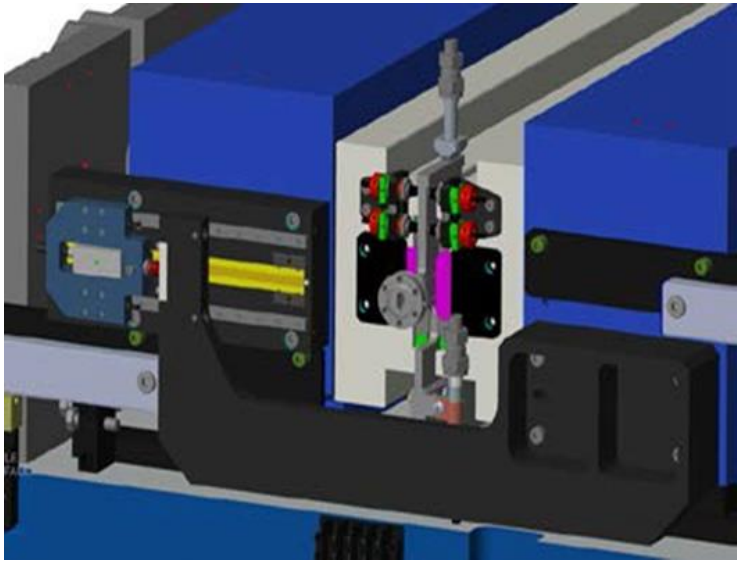
- A more intricate meshing required that is dependent on flow properties,



CFD IN THE DESIGN OF SYNCHROTRON INSTRUMENTATION

However…

- CFD would be a powerful tool to analyze the effect of natural convection on the temperature of the vacuum chamber embedded within the jaws of the undulator!



The chamber installed between the undulator jaws

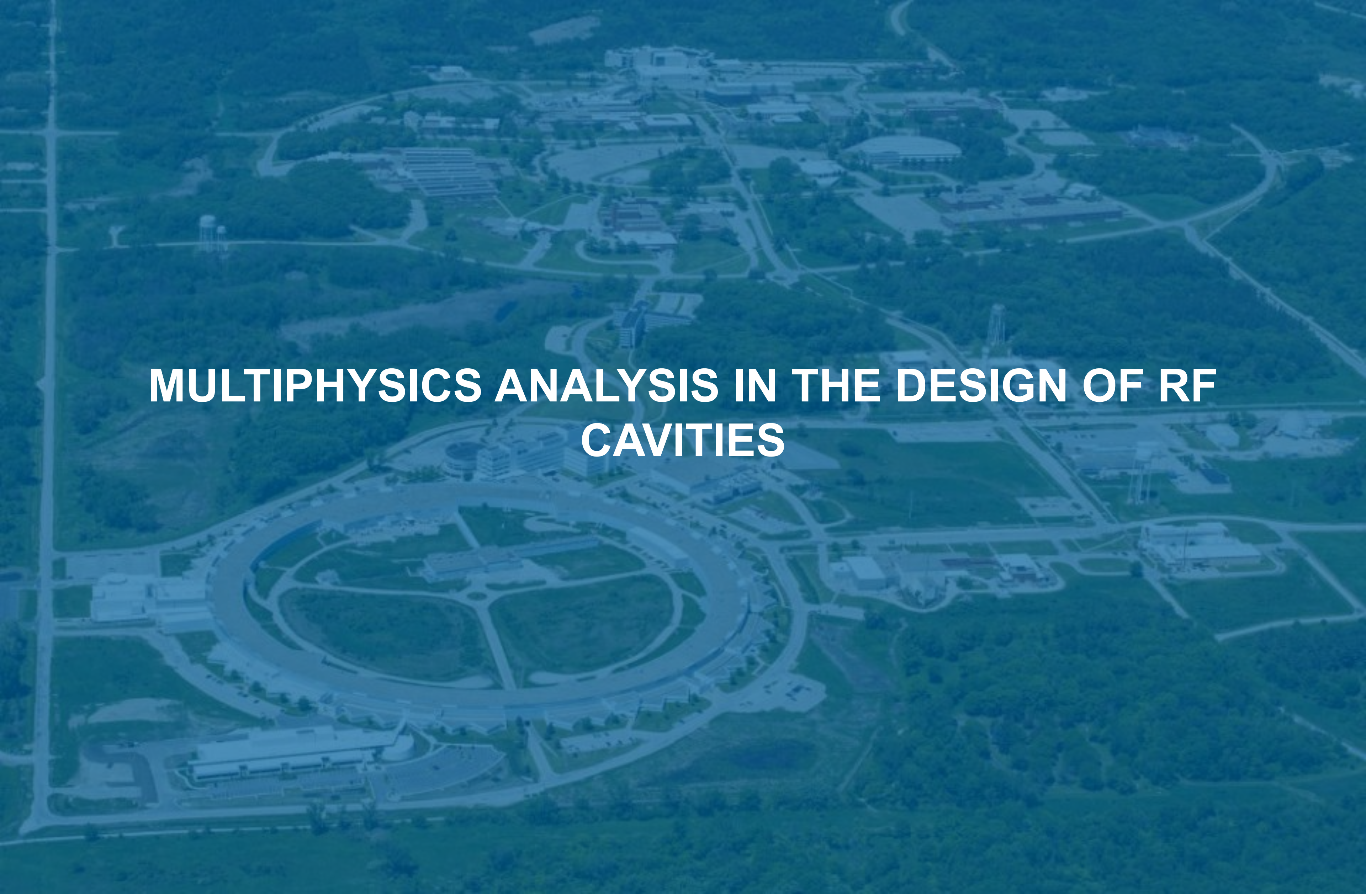
- The presented analysis proved to be an excellent example of validation of a new computing tool with a proven one!

CFD IN THE DESIGN OF SYNCHROTRON INSTRUMENTATION

It can definitely help, but don't get too excited!

- Do not use it to calculate heat transfer in straight cylindrical cooling channels
- Do not use it if you are not concerned with the change in coolant temperature,
- Use it when you are dealing with complex cooling geometries
- Use it when natural convection should not be neglected
- Invest in your computer hardware

MULTIPHYSICS ANALYSIS IN THE DESIGN OF RF CAVITIES



Argonne NATIONAL LABORATORY

MULTIPHYSICS ANALYSIS IN THE DESIGN OF RF CAVITIES

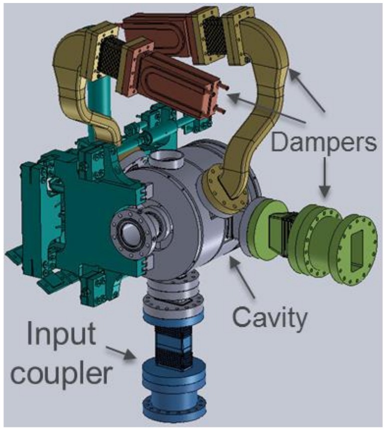
Ultimate tool in RF cavity design — Courtesy of Geoff Waldschmidt

- Objective
  - To integrate different analysis modules into a single analysis needed for RF cavity design
- Strategy
  - Create an analysis environment capable of modeling various physics:
    - Electromagnetic fields and thermal analysis
    - Fluid flow with heat transfer
    - Thermal distribution with structural analysis
    - Electromagnetically induced structural effects
    - Effects of thin deposition layers as well as material transition points
  - Preserve spatial distribution between physics models
  - Physics analyses can be interchanged and sequentially solved
  - Include the temperature dependence of physical and mechanical properties of materials in the analysis
  - Establish convergence criteria to ensure that each physics analysis converges in an iterative fashion
- Scope
  - The analysis is useful for static heat loads, dynamic heat loads, thermal equilibrium, structure deformation, material stresses, Lorentz force detuning, etc.

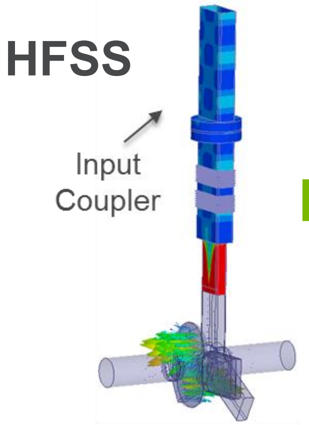
MULTIPHYSICS ANALYSIS IN THE DESIGN OF RF CAVITIES

FEA analysis of an RF cavity – an example

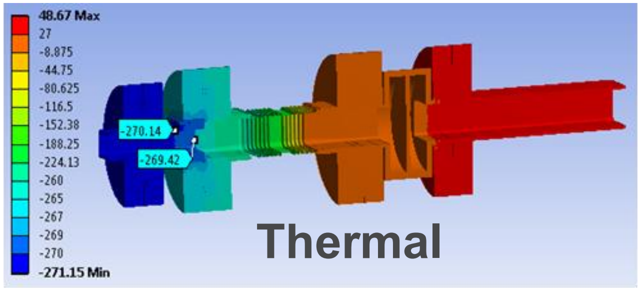
- A common analysis for accelerator applications to determine the effect of RF losses
- RF fields are generated and losses are calculated at a preliminary temperature
- An iterative solution is performed until a pre-determined convergence is achieved.
- A copper deposition layer can be modeled to improve electrical conductivity with limited thermal consequence for transition regions
- The Result of structural analysis with consequent deformation of the geometry can be fed back to an RF solver, if significant, to determine changes in the RF field magnitude and quality



Superconducting cavity geometry



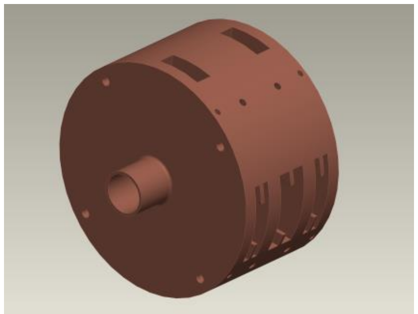
RF losses on input coupler



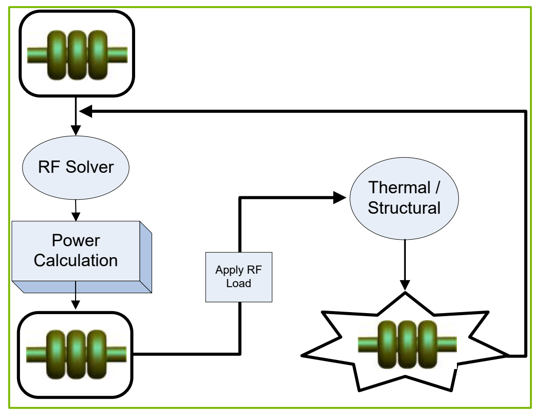
Thermal distribution on input coupler

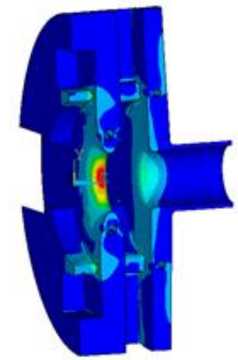
MULTIPHYSICS ANALYSIS IN THE DESIGN OF RF CAVITIES

Typical RF cavity design flow

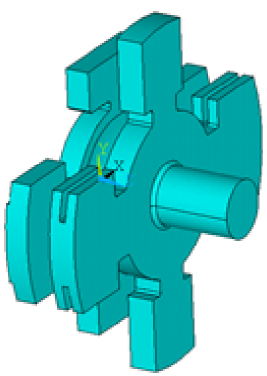


CAD geometry

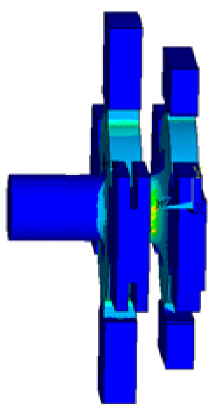




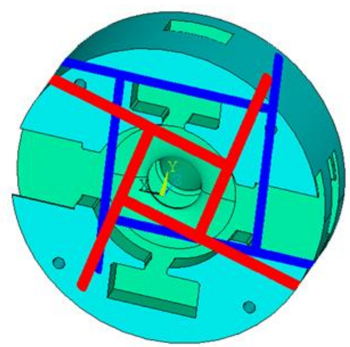
Mechanical Stresses



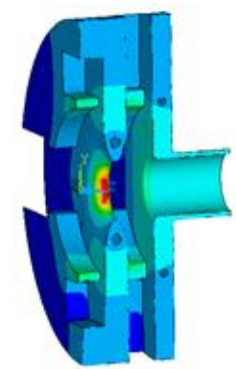
RF Volume



RF Losses



Cooling Network



Thermal Distribution

TOLERANCE ANALYSES OF QUADRUPOLE MAGNETS FOR THE ADVANCED PHOTON SOURCE UPGRADE

Argonne NATIONAL LABORATORY

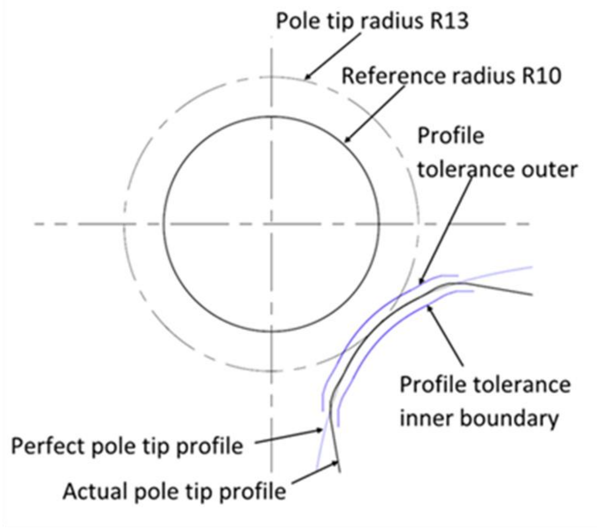
TOLERANCE ANALYSES OF QUADRUPOLE MAGNETS FOR THE ADVANCED PHOTON SOURCE UPGRADE

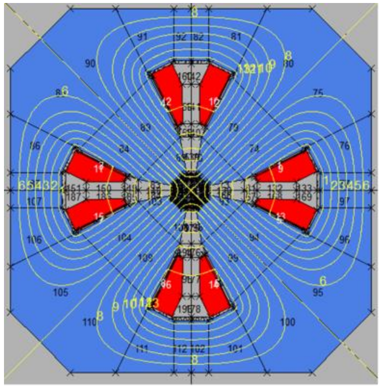
Courtesy of Jie Liu

- Objective
  - Calculate the allowed mechanical fabrication and assembly tolerances for storage ring magnets from given physics requirements
- Strategy
  - Hallbach analytical equations are used to calculate the allowed tolerances (range)
  - Opera 2D FEA software is used to determine the values and distribution of random errors (magnetic perturbations)
  - Lattice evaluation is performed to confirm that the errors are acceptable
  - A novel method is used to allocate the allowed manufacturing and assembly tolerances to part and subassembly levels
    - The mechanical tolerance stackup analysis is performed using a 3D tolerance analysis package

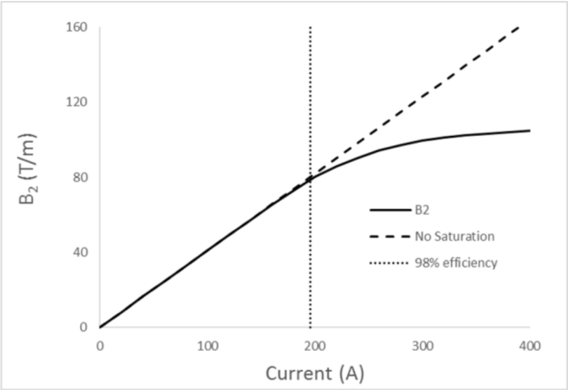
TOLERANCE ANALYSES OF QUADRUPOLE MAGNETS FOR THE ADVANCED PHOTON SOURCE UPGRADE

- Magnetic Tolerance Analysis
  - Opera 2D simulation package was used for magnetic simulations
    - The four pole tip profiles are allowed to vary between the outer and inner boundaries
    - Three different tolerance zone widths, ±15 µm, ±25 µm, and ±50µm were simulated. These zones include contributions from both machining and assembly errors
    - The examples of magnetic simulation results are given bellow





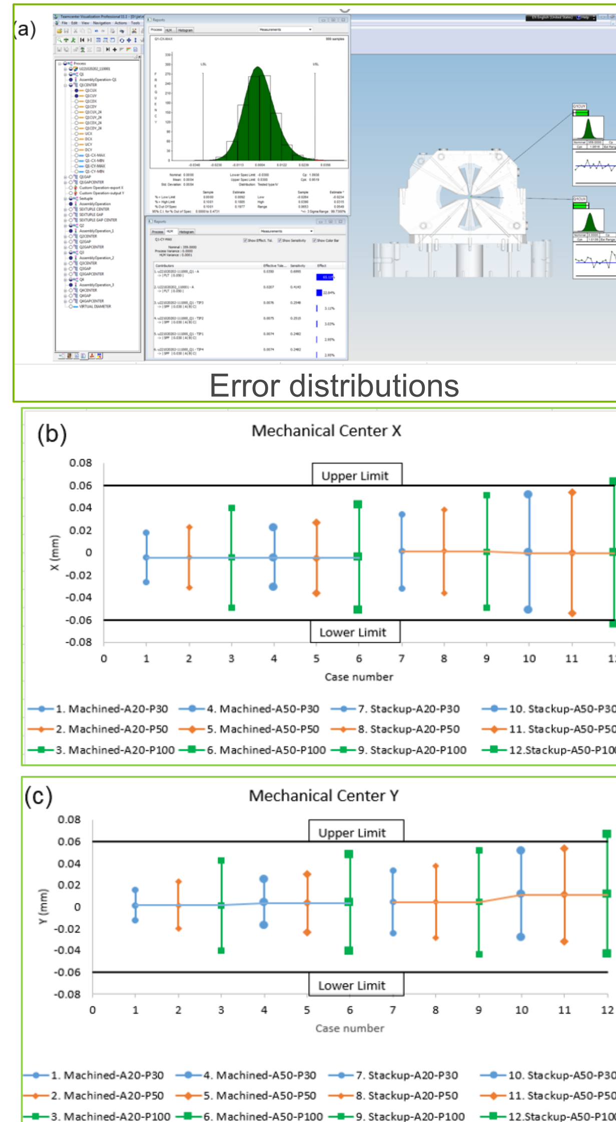
2D model with flux lines.

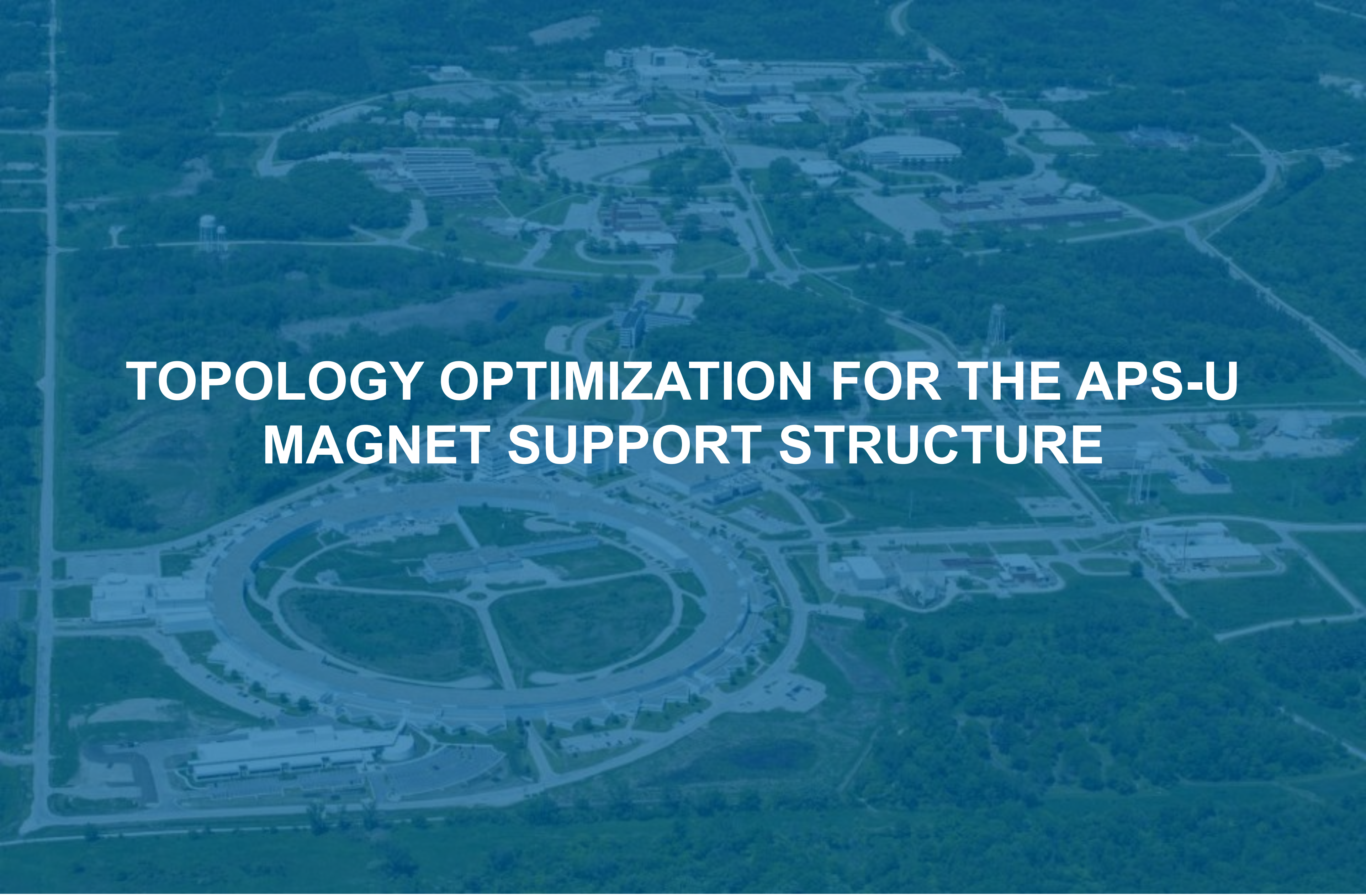


Field gradient versus current.

TOLERANCE ANALYSES OF QUADRUPOLE MAGNETS FOR THE ADVANCED PHOTON SOURCE UPGRADE

- Mechanical Tolerance Stackup Analysis
  - The “Teamcenter” variation analysis package was used to simulate the tolerances
    - The software randomly generates and assembles parts within a specified tolerance range using Monte Carlo simulation methods
  - Two scenarios were compared, one with the four pole tip profiles machined after assembly and one with them machined before assembly
    - It was found that with machining the pole tip profiles before assembly, the geometric tolerances on mounting surfaces need to be 20 μm or less
    - This will be expensive to meet
    - With machining of the magnet pole tip profiles after assembly, the tolerance can be relaxed to 50 μm without causing significant stackup errors





TOPOLOGY OPTIMIZATION FOR THE APS-U MAGNET SUPPORT STRUCTURE

TOPOLOGY OPTIMIZATION FOR THE APS-U MAGNET SUPPORT STRUCTURE

Courtesy of Zunping Liu

- Objective
  - Optimize the topology of the three-point semi-kinematic vertical mount for the magnet modules of the APS-U magnet lattice
    - Minimize the in-plane deflection at points along the beam path by minimizing strain energy
    - Maximize the frequency response for frequencies higher than 50 HZ
- Strategy
  - Use Genesys® Topology for ANSYS Mechanical (GTAM) software in a possibly multistep design optimization process

TOPOLOGY OPTIMIZATION FOR THE APS-U MAGNET SUPPORT STRUCTURE

Physics

- Optimization process
  - Find the set of **design variables values** that will optimize the **objective** function while satisfying all the **constraints**

Optimize:	$F(\mathbf{X})$
Subject to (Such that):	$g_j(\mathbf{X}) \leq 0$
	$X_i^{Lower} \leq X_i \leq X_i^{Upper}$

  - Optimization options:
    - Topology
    - Sizing
    - Topometry
    - Topography
    - Etc.
  - Constraint options
    - Mass
    - Strain energy
    - Inertia relief
    - Displacement
    - Frequency response
    - Contact
    - Etc.

TOPOLOGY OPTIMIZATION FOR THE APS-U MAGNET SUPPORT STRUCTURE

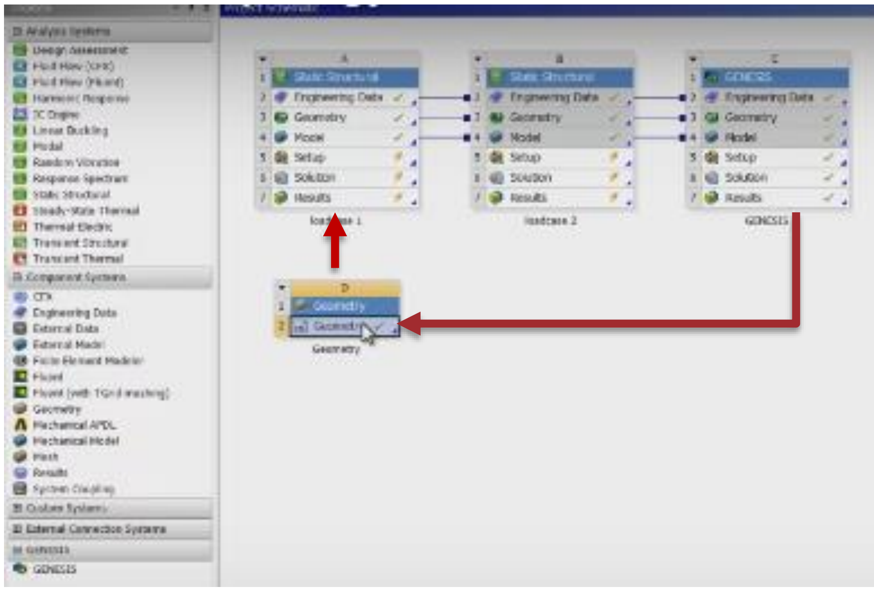
Homework

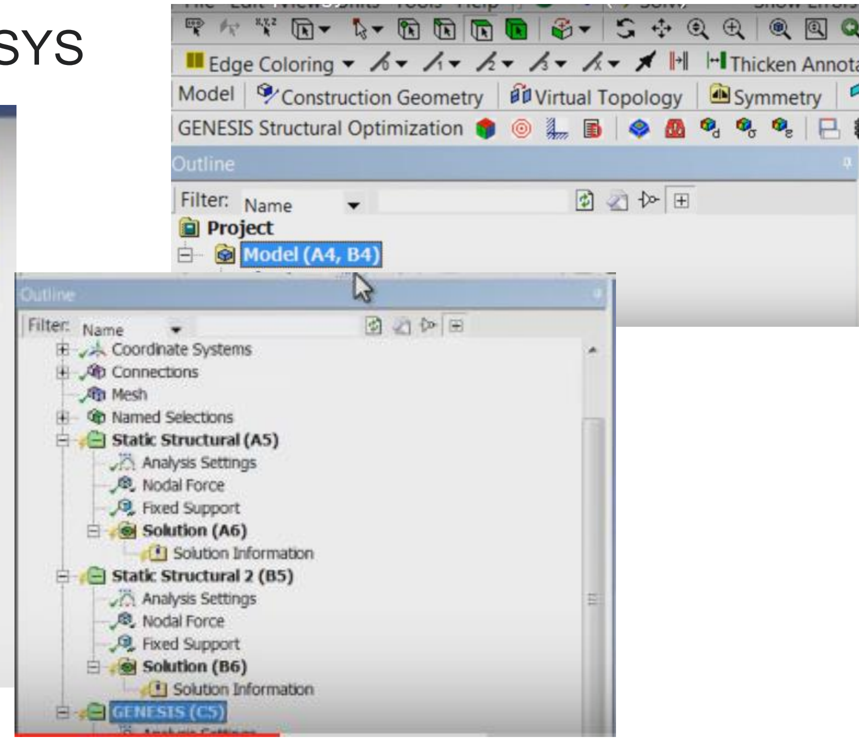
- Among the many commercially available packages:
  - Vanderplaats R&D products
    - Genesis
    - Design Studio for Genesis
    - Genesis Structural Optimization for ANSYS Mechanical (GSAM)
      - GTAM – Genesis Topology for ANSYS Mechanical
    - ESLDYNA – Structural Optimization for LS-DYNA
  - Dassault Systems product
    - Tosca Structure
  - Virtual.PYXIS
    - A topology optimization software
- Find the one that will best address the goals:
  - Minimize the in-plane deflection at points along beam path
  - Limit vibration amplitude to < 1nm for frequencies above 50 Hz

TOPOLOGY OPTIMIZATION FOR THE APS-U MAGNET SUPPORT STRUCTURE

GENESIS Topology for ANSYS Mechanical (GTAM)

- GTAM is seamlessly integrated with ANSYS

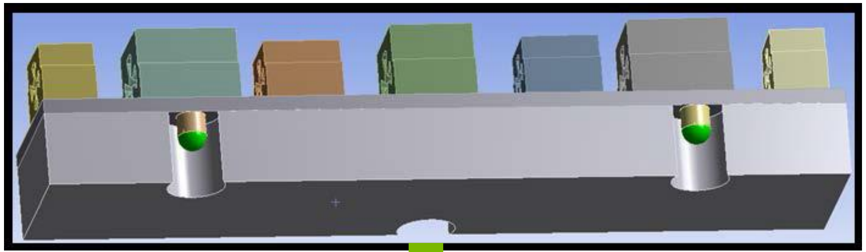




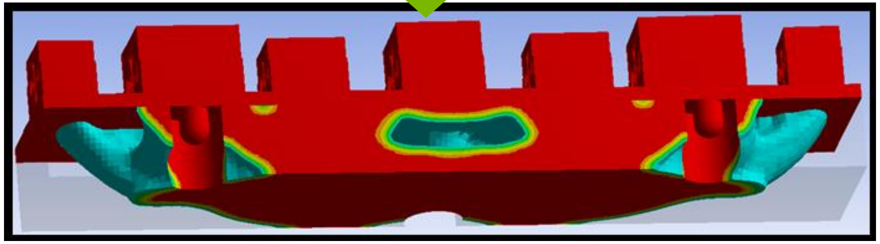
- Topology objectives and constrains can be defined using the command menu
  – Manufacturing constraints are one of the options
- Optimized geometry can be exported into the CAD package as an STL or IGES file

TOPOLOGY OPTIMIZATION FOR THE APS-U MAGNET SUPPORT STRUCTURE

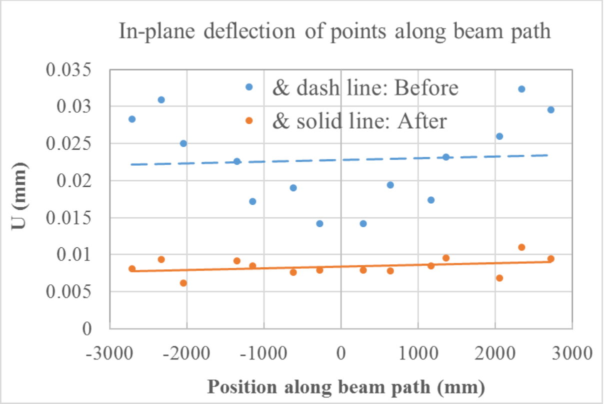
Results

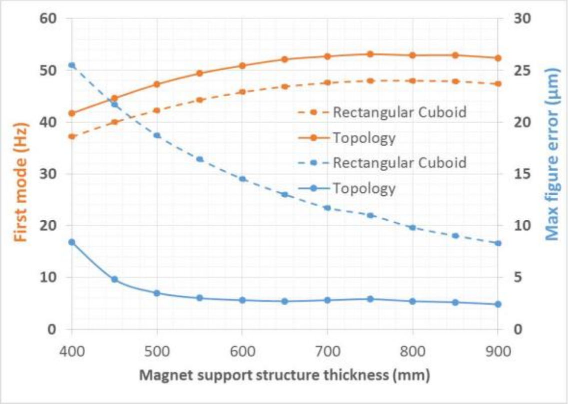


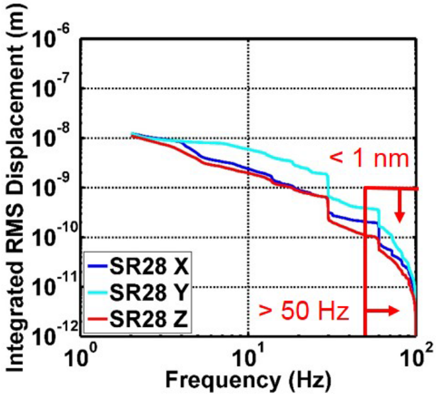
Initial geometry



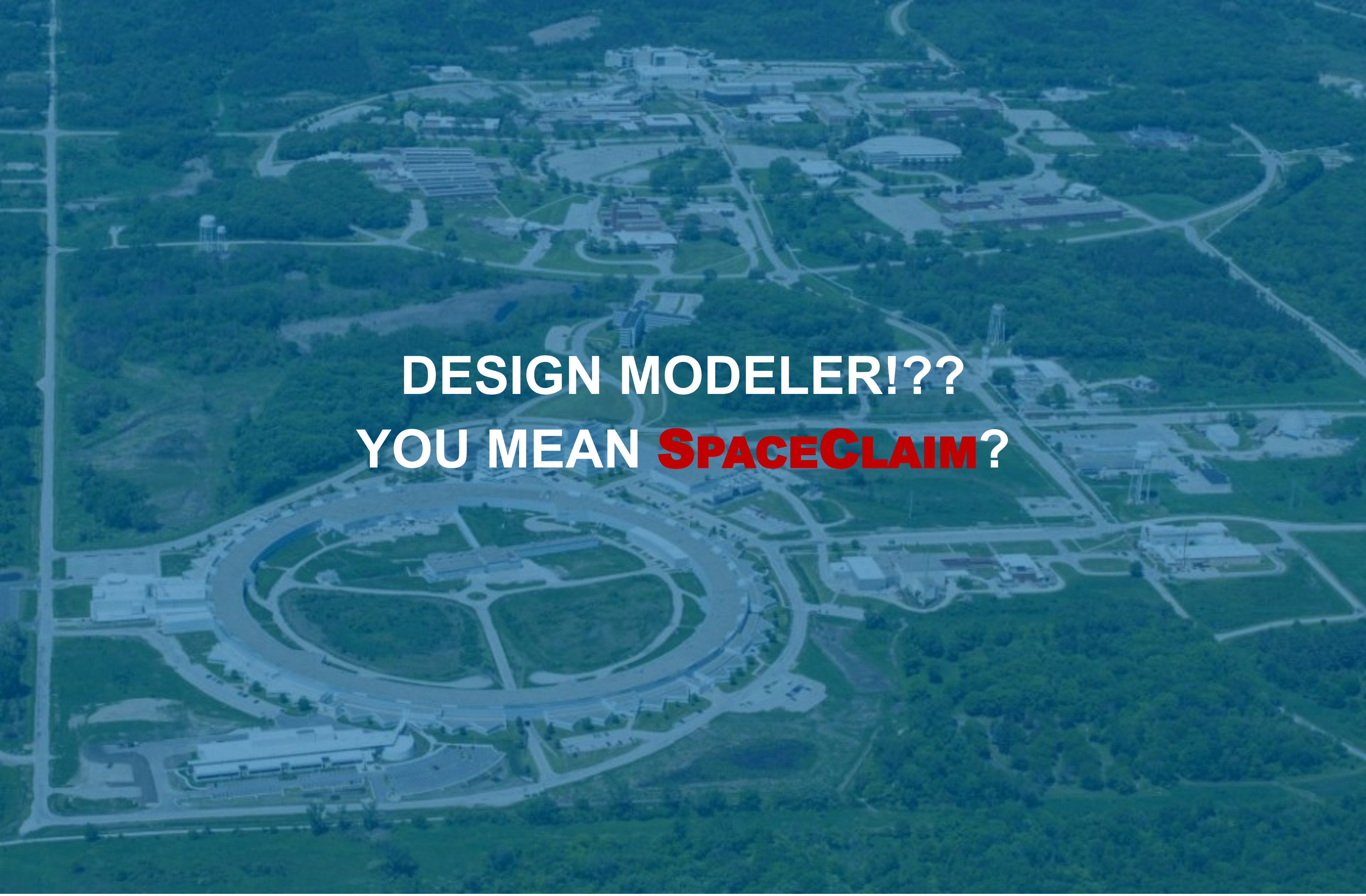
Optimized geometry







DESIGN MODELER!??
YOU MEAN SPACECLAIM?



Argonne NATIONAL LABORATORY

SpaceClaim – A REVOLUTION IN MODELING FOR FEA

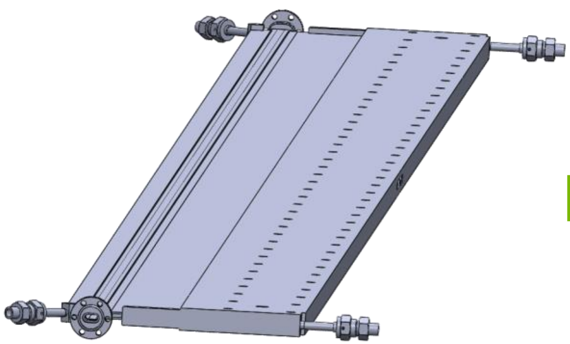
Feel the Space! (Courtesy of Jason Carter)

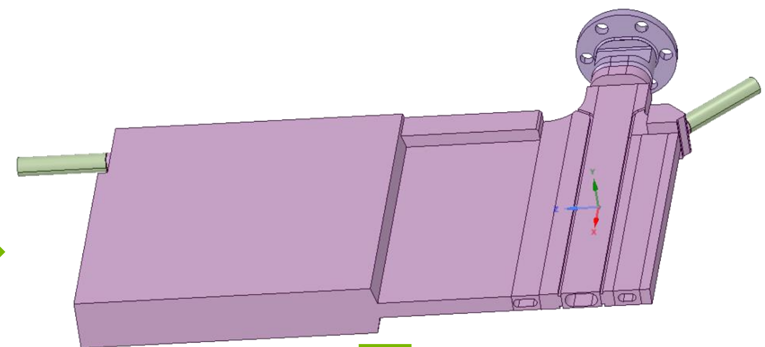
- SpaceClaim is a 3D modeling software with no history of operations
  - ‘Direct Modeling’ allows for designing loosely and quickly without worrying about the sequence of operations
- It manipulates geometry in an intuitive, easy, and very fast way
  - Previously very tedious operations like volume extraction are reduced to a single mouse click
- It is perfect for the preparation of models for use with analysis tools
  - Converts fully detailed 3D models aimed at creation of production drawings into models ready for FEA in matter of minutes
- SpaceClaim export/imports to/from all major CAD file types
  - Geometry can be created, edited or repaired without worrying about the underlying technology
- It will replace Design Modeler!

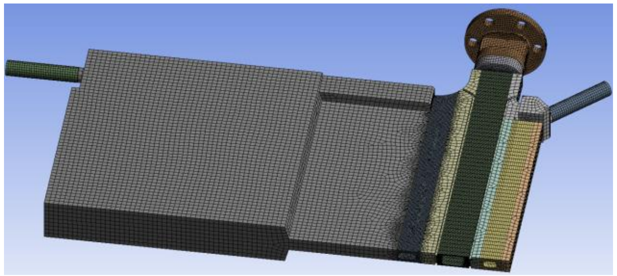
SPACECLAIM – A REVOLUTION IN MODELING FOR FEA

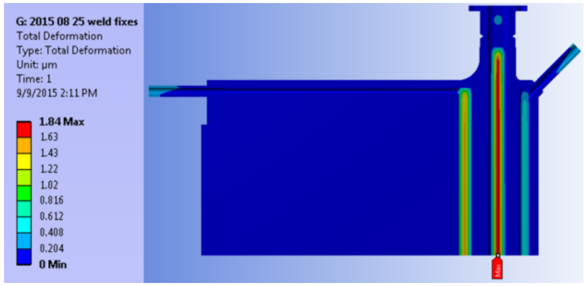
Examples of use

- LCLS-II vacuum chamber stress analysis
  - The ‘split body’ and ‘combine’ features were used to dice up a model to isolate high mesh density regions
  - This saves time in meshing the problem in ANSYS due to carefully controlled meshing rules





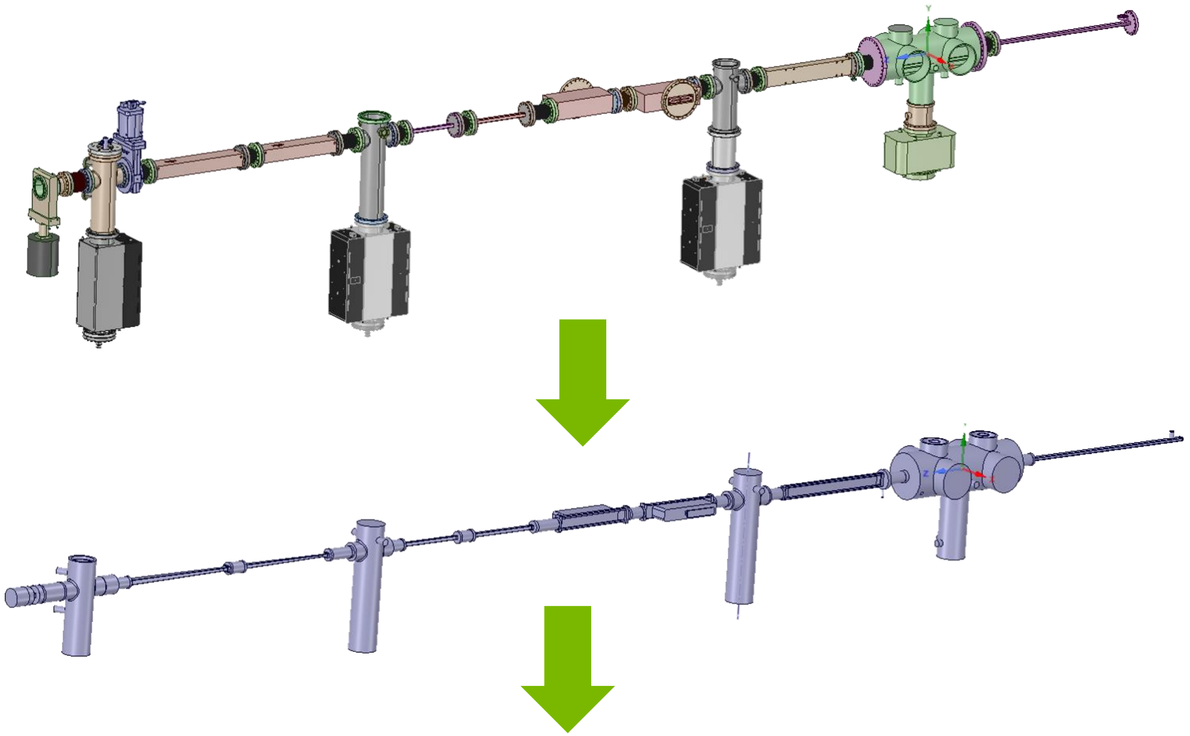




SpaceClaim – A Revolution in Modeling for FEA

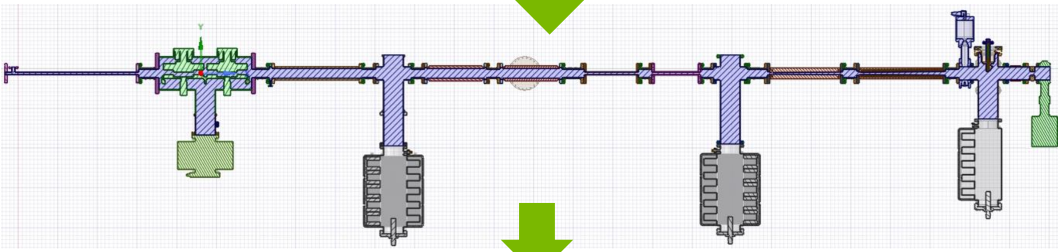
Examples of use

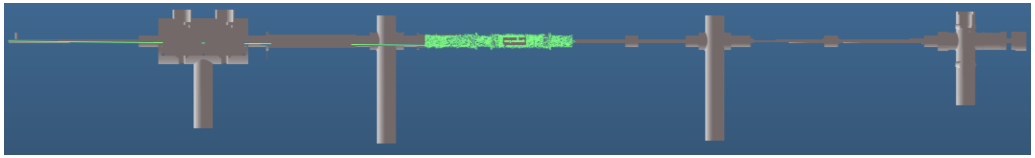
- Capturing the complex conductance of front end vacuum system design, e.g. helping in the optimization of the size of ion pumps
  - Using volume extraction to define vacuum volume
  - Using the vacuum volume as the input in MolFlow

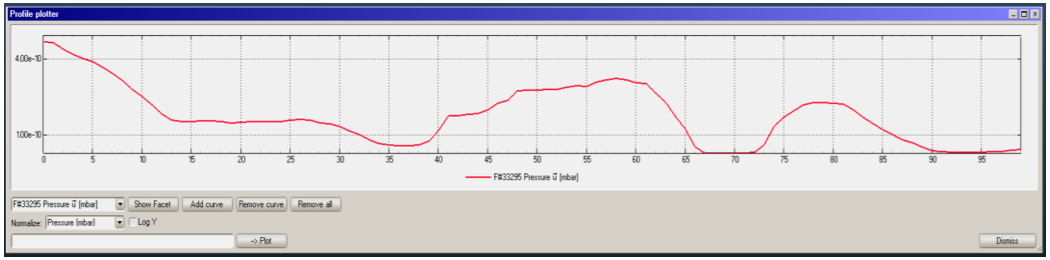


SPACECLAIM – A REVOLUTION IN MODELING FOR FEA

Examples of use—continued



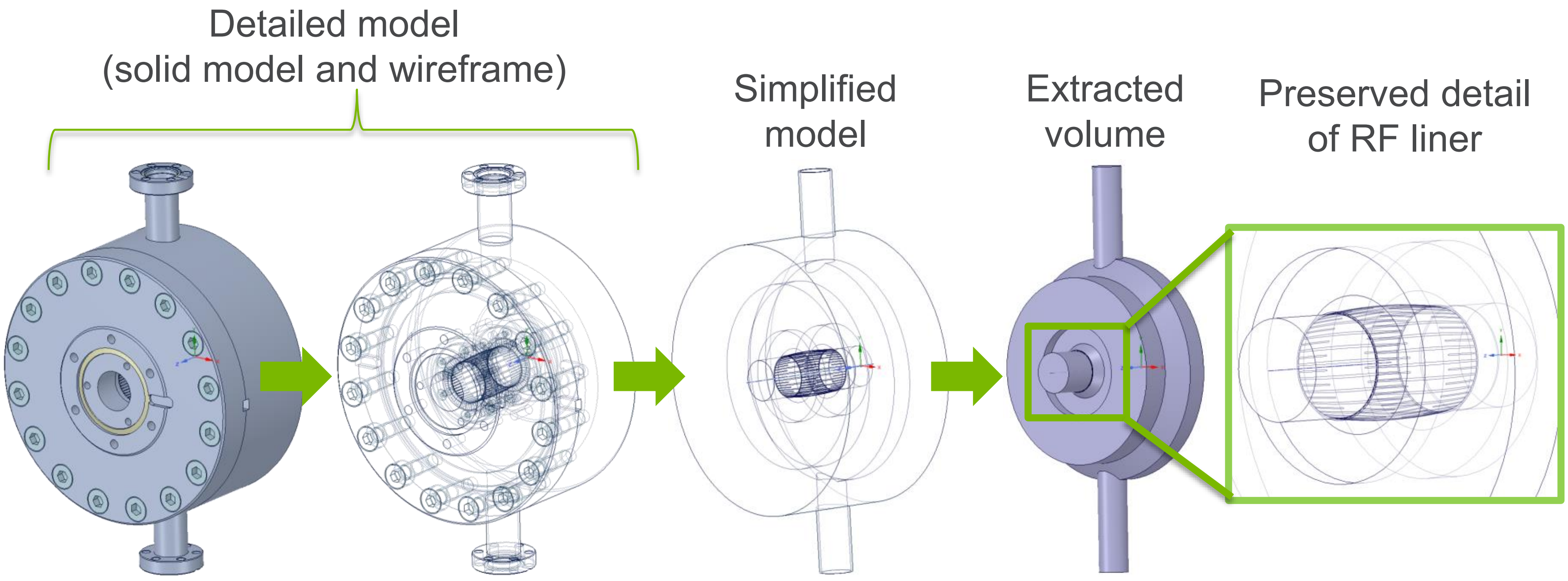




SPACECLAIM – A REVOLUTION IN MODELING FOR FEA

Examples of use

- Simplifying the gate valve liner model so it can be used for impedance calculations
  - While maintaining complexity of gate valve liner, key area to study



MODELING OF ACOUSTIC LEVITATION SUPPORT

Argonne NATIONAL LABORATORY

MODELING OF ACOUSTIC LEVITATION SUPPORT

An example of right and wrong use of CFD (in same analysis)

- Objective
  - To understand the physics of acoustic levitation
    - To develop sample supports that will minimally interfere with the examined sample
    - This is of particular interest in the investigation of liquid samples where the presence of a container is otherwise necessary
- Strategy
  - Use coupled Frequency Domain Study and CFD
    - Frequency Domain Study generates pressure profiles (steady state condition)
    - CFD computes interaction between the pressure waves and particles
    - Computed results were validated by experiment
    - An Acoustic horn was used to levitate particles of water mist

MODELING OF ACOUSTIC LEVITATION SUPPORT

Physics

- Computational details

Frequency domain study

A harmonic solution $p(x,t) = p(x)e^{i\omega t}$

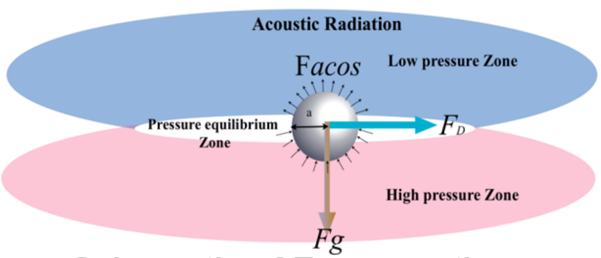
The Helmholtz pressure equation $$\nabla\left(-\frac{1}{\rho_o}(\nabla p)\right) - \frac{\omega^2}{\rho_o c^2}p = 0$$

Fluid velocity in terms of pressure $p = -K_0 \nabla \cdot \mathbf{u}$

Acoustic Levitator

Transient condition Particle Tracing (CFD)

$$\frac{d(m_p \mathbf{v})}{dx} = \boldsymbol{F}_D + \boldsymbol{F}_g + \boldsymbol{F}_{acos}$$



Schematic of Forces acting on water droplets

Gravity Force: $\boldsymbol{F}_g = \left(\frac{\rho_{p-\rho}}{\rho_p}\right) m_p \boldsymbol{g}$

Drag Force: $\boldsymbol{F}_D = \left(\frac{1}{\tau_p}\right) m_p(\mathbf{u}\text{-}\boldsymbol{v}_{in})$

Acoustic Force: $\boldsymbol{F}_{acos} = -\nabla U_{radiation}$

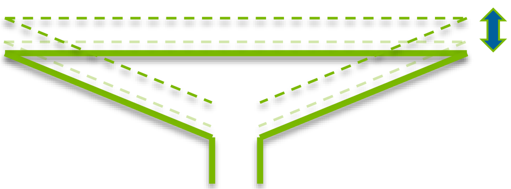
MODELING OF ACOUSTIC LEVITATION SUPPORT

Can FEA really model this physics??

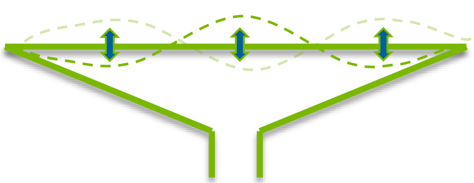
MODELING OF ACOUSTIC LEVITATION SUPPORT

Homework

- Originally, rigid body motion of the acoustic horn was assumed in the model



- Experimental investigation, using camera with 100 000 frames per second, indicated that the motion was more like this:

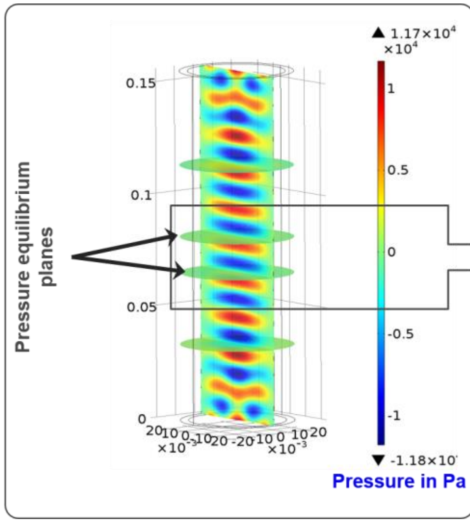


- The reintroduction of more realistic vibrational motion improved the accuracy of pressure wave calculations

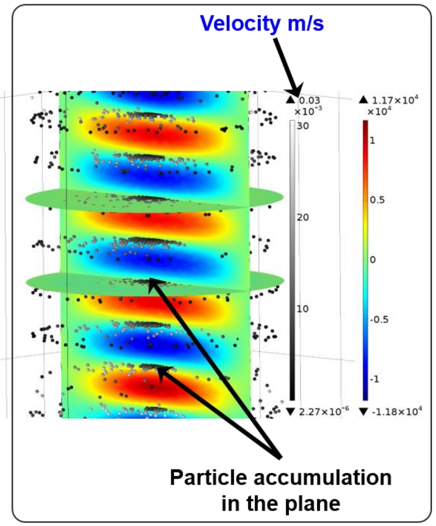
MODELING OF ACOUSTIC LEVITATION SUPPORT

Results

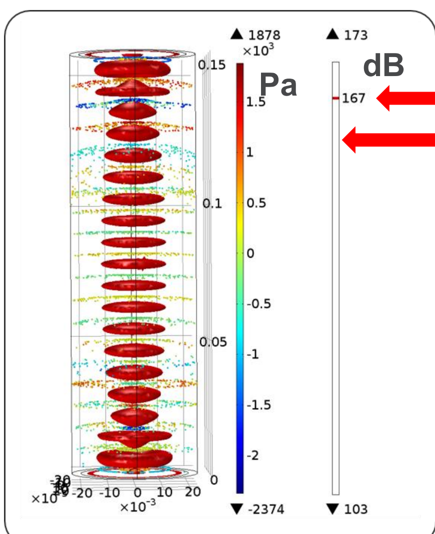
Frequency Domain Analysis



CFD particle tracing



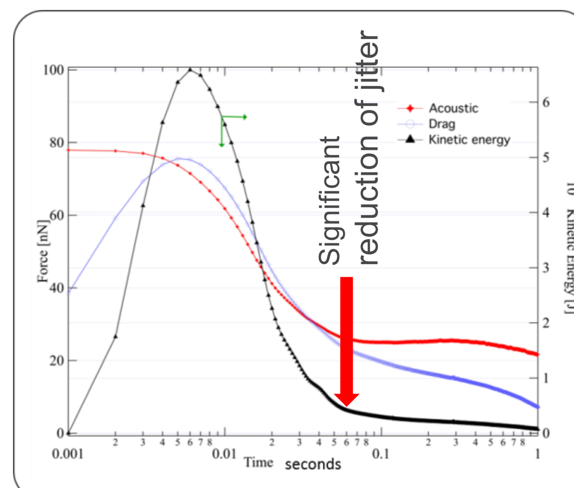
Sound level and pressure



Experiment sound level 167 dB (luckily at 22.5 kHz, but your dog wouldn't like it)

Jet engine noise at 30m - 140dB

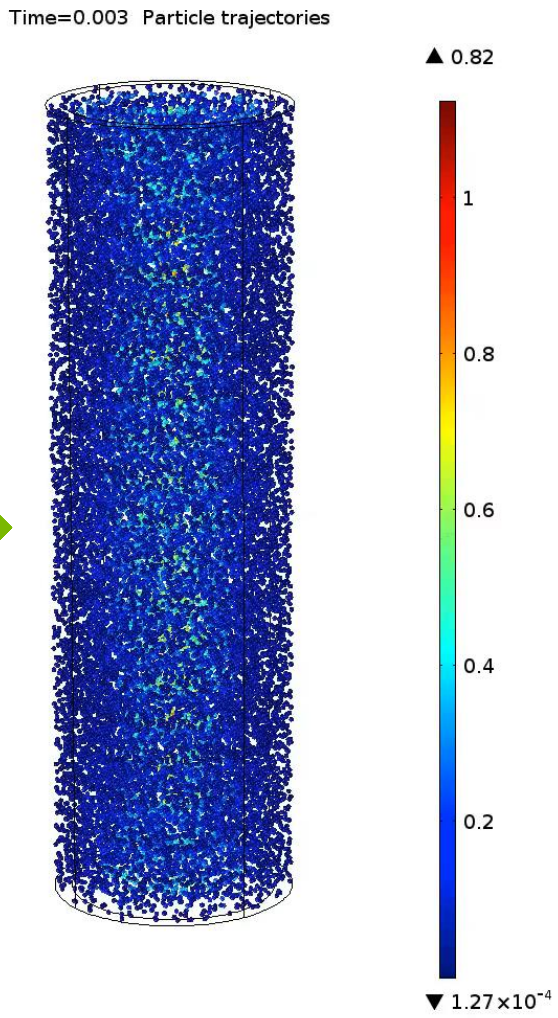
Forces acting on a particle



Argonne NATIONAL LABORATORY

MODELING OF ACOUSTIC LEVITATION SUPPORT

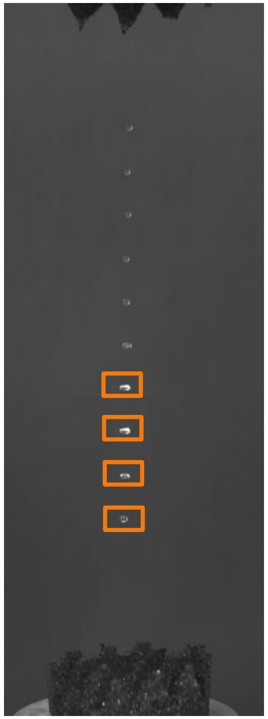
Experimental validation (qualitative)



MODELING OF ACOUSTIC LEVITATION SUPPORT

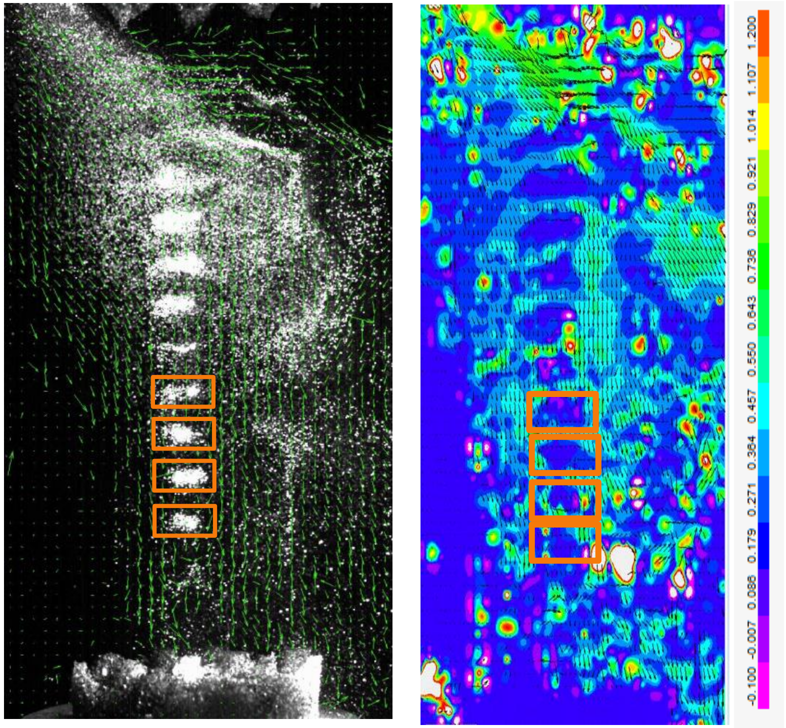
Experimental validation (quantitative)

Experiment

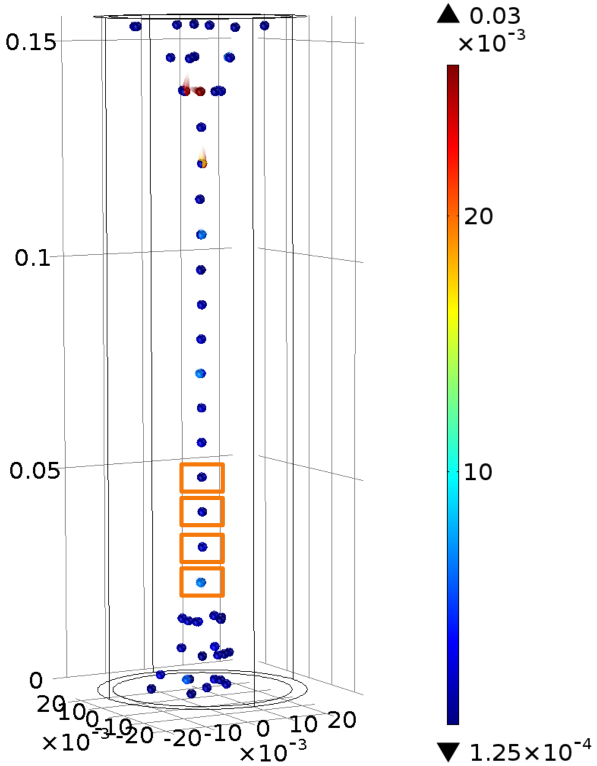


Characterization

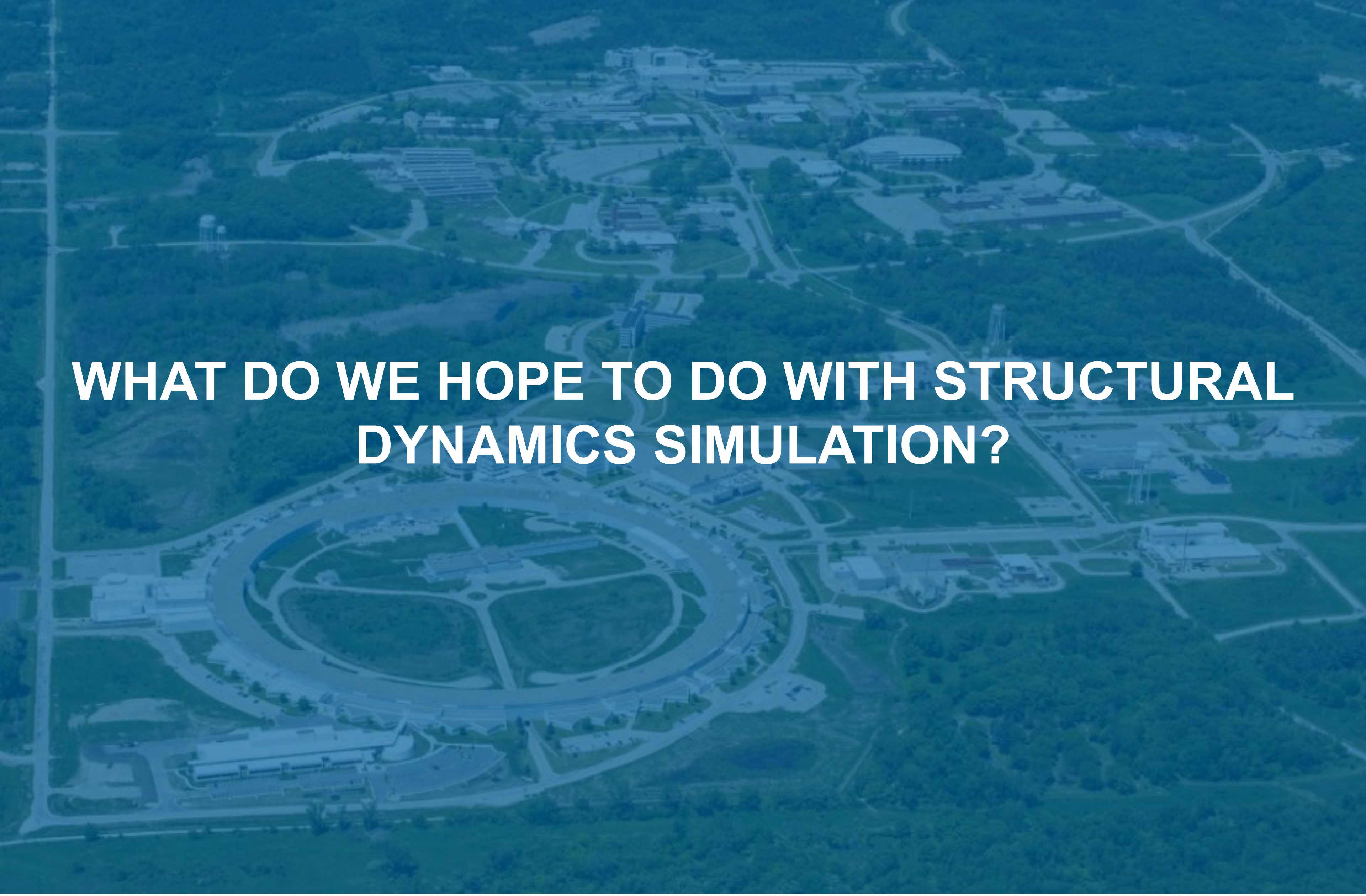
Laser velocimetry



Simulation



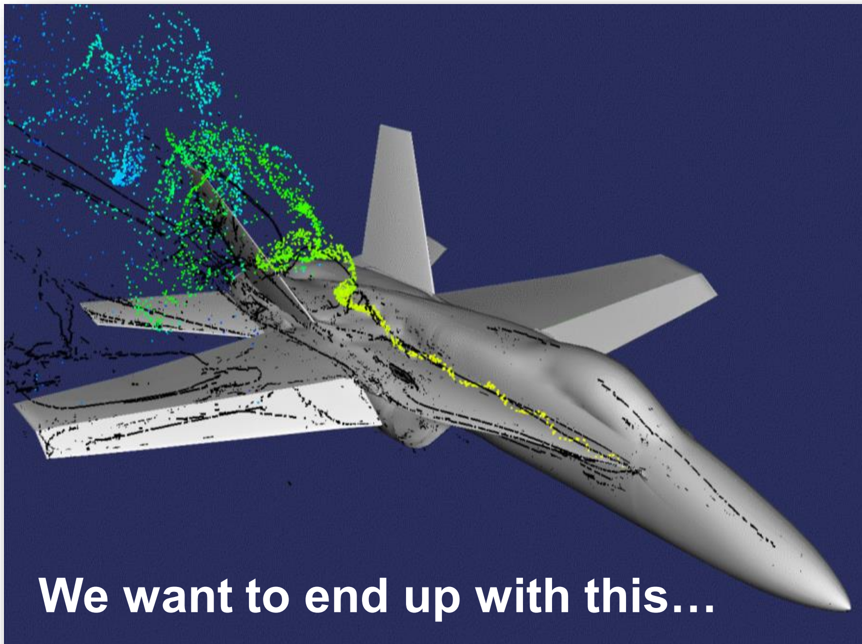
Argonne NATIONAL LABORATORY



WHAT DO WE HOPE TO DO WITH STRUCTURAL DYNAMICS SIMULATION?

WHAT DO WE HOPE TO DO WITH STRUCTURAL DYNAMICS SIMULATION?

Motivation: Evaluate design against requirements

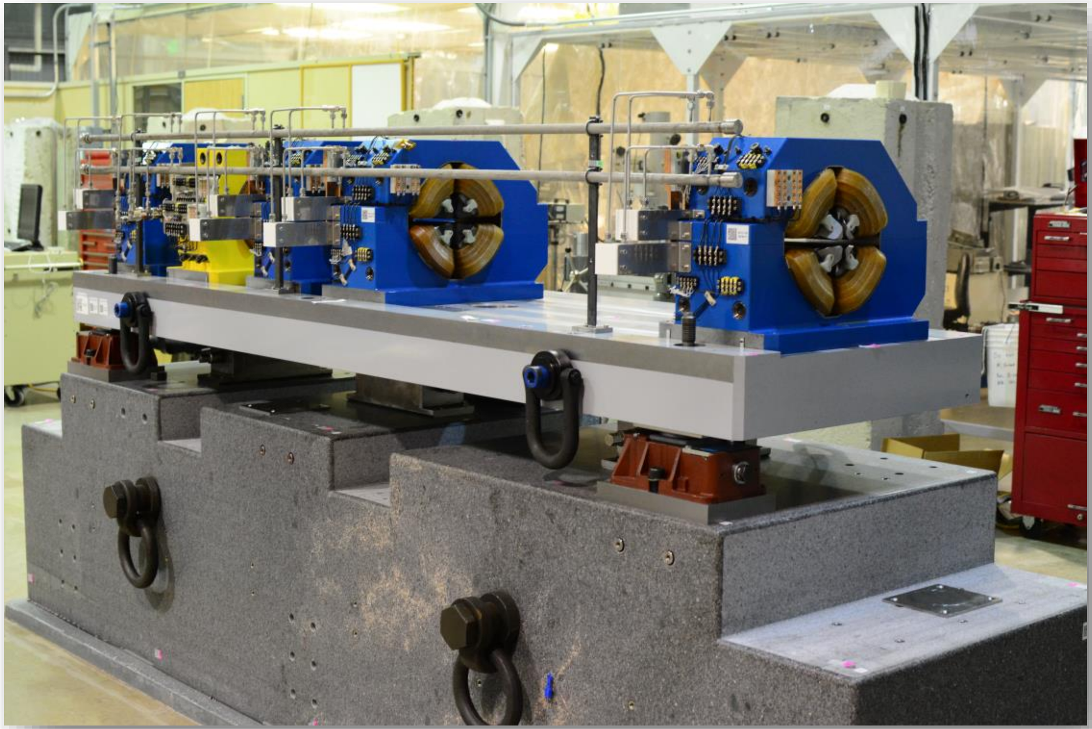


Use analysis to accurately predict performance.

Not this…

STRUCTURAL DYNAMICS FOR SYNCHROTRON COMPONENTS

Motivation: Evaluate design against requirements



This can be challenging because:

- behavior of interest may depend upon hard-to-determine characteristics, or
- component of interest may be hard to model.

CONSIDER THE FOLLOWING PROBLEMS…

Will "typical" FE modeling techniques be a good predictor?

- Static deflection of a girder
  - Dependent mostly upon material properties and geometry
  - Both of which can be well characterized
  - Typical structural FEA techniques can be good predictor of behavior
- Modal analysis of a magnet support assembly
  - Dependent upon material properties and geometry, but
  - Also highly dependent upon interfaces and sub-structure behavior.
  - These can be hard to characterize
  - Typical structural FEA techniques can be unreliable predictor of behavior
- Vibration response of magnet support assembly (random vib., harmonic, transient)
  - Dependent upon all the things mentioned above, plus
  - Highly dependent upon the *damping*
  - In fact, damping "knob" can be "turned" to have results say "anything"
  - Typical structural FEA techniques can be unreliable predictor of behavior

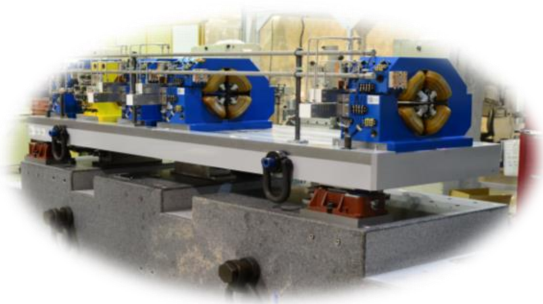
So…what can be done to improve quality?

A COUPLE OF POSSIBLE WORKFLOWS…

1. **Design, build prototype, test prototype, tune FE model to match**
   - Pro: relatively straight forward
   - Cons:
     - Need to go all the way to full prototype
     - Results only applicable to narrow family of closely-related designs
     - *Post hoc*
2. **Design, component property. ID, analyze design space, *verification* test**
   - Pros:
     - Building block approach…information applicable to *any* design
     - Smaller scale tests to generate property information
     - *A priori*
   - Con: may need specialized testing knowledge

EXAMPLE: SUPPORT/ALIGNMENT MECHANISM

Proposed workflow



What is critical component?

Component test

- Natural frequencies
- Modal shapes

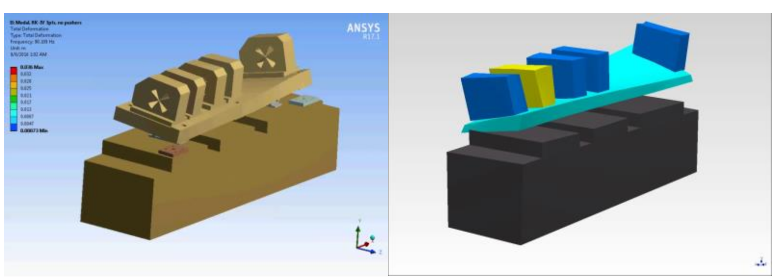
$$K_J = \begin{bmatrix} K_{XX} & K_{XY} & K_{XZ} & K_{X\theta x} & K_{X\theta y} & K_{X\theta z} \\ & K_{YY} & K_{YX} & K_{Y\theta x} & K_{Y\theta y} & K_{Y\theta z} \\ \vdots & & K_{ZZ} & K_{Z\theta x} & K_{Z\theta y} & K_{Z\theta z} \\ & & & K_{\theta X} & K_{\theta x\theta y} & K_{\theta x\theta z} \\ \vdots & \iddots & & & K_{\theta y} & K_{\theta y\theta z} \\ & \cdots & & \cdots & & K_{\theta Z} \end{bmatrix}_{sym}$$

Component properties

- Stiffness
- Damping

Verification test

FEA



Experiment

Details: See Preissner talk 10:50 Wednesday

MANY, MANY THANKS TO

JIE LIU

ZUNPING LIU

JASON LURCH

JASON CARTER

JEREMY NUDELL

CURT PREISSNER

KAMLESH SUTHAR

GEOFF WALDSCHMIDT

Argonne NATIONAL LABORATORY

QUESTIONS?