

ANSYS®



ANSYS
CONVERGENCE
CONFERENCES

2016

ANSYS中国技术大会
中国·上海

ANSYS在航空发动机设计仿真中的应用

杨帆/ 应用工程师

ANSYS

目录

1. 航空发动机与ANSYS
2. ANSYS在叶轮机通流设计中的应用
3. ANSYS在燃烧设计中的应用
4. ANSYS在热端部件热防护设计中的应用
5. ANSYS在二次空气系统设计中的应用
6. ANSYS在零部件强度振动校核中的应用
7. ANSYS在转子动力学校核中的应用
8. ANSYS在鸟撞与包容性设计中的应用
9. ANSYS在发动机其他设计中的应用
10. 多物理场仿真在发动机设计中的应用
11. 总结

ANSYS在航空发动机领域的客户



GE Aviation



Rolls-Royce®

Honeywell



Pratt & Whitney



SAFRAN
AEROSPACE · DEFENCE · SECURITY

BAE SYSTEMS



Williams International

ANSYS and General Electric Unlock Future Innovation with New Collaboration Agreement

Joint technology collaboration agreement extends 20-year relationship to bring next-generation of products to life

May 07, 2014, 07:27 ET from ANSYS, Inc.

English ▼



PITTSBURGH, May 7, 2014 /PRNewswire/ -- ANSYS (NASDAQ: [ANSS](#)) and GE Aviation are deepening their long-standing strategic relationship by establishing a new joint technology collaboration agreement that will help solve future engineering challenges and drive product development processes in a world of smart products and big data. Both organizations recognize the value of cutting-edge research and are jointly aligned in their desire to spur innovation by creatively applying simulation to the manufacturing and industrial space.

ANSYS and GE Aviation will work together over a range of applications to establish forward-looking analysis techniques that leverage expertise from both parties. In the first project under this agreement, ANSYS and GE Aviation will investigate industry data to create new engineering best practices associated with the accurate analysis of some of GE's core industrial products. Projects may include simulation of system-level product performance, enhanced by live data, to predict overall system efficiency and maximize product life.

GE currently leverages ANSYS engineering simulation technology across its product portfolio to create best-in-class products ranging from aircraft engines to high-fidelity MRI scanners. This new joint agreement highlights the commitment of both organizations to stay at the forefront of innovation and productivity.

"We have employed the power of ANSYS simulation for over two decades to confidently design and analyze our products and quickly bring them to market," said Christopher Lorence, general manager of engineering technologies at GE Aviation. "This new strategic agreement will allow us to work more closely with ANSYS and push the boundaries of engineering even further."

"GE is an industry leader that has built its reputation on providing world-class products and services like advanced jet engines and power systems," said Jim Cashman, ANSYS president and CEO. "Our long-standing relationship with GE Aviation provided a strong foundation for this new collaborative agreement, and we're confident that the resulting insight will further solidify GE's status as leader in innovation in the products and markets it serves. We

Journalists and Bloggers

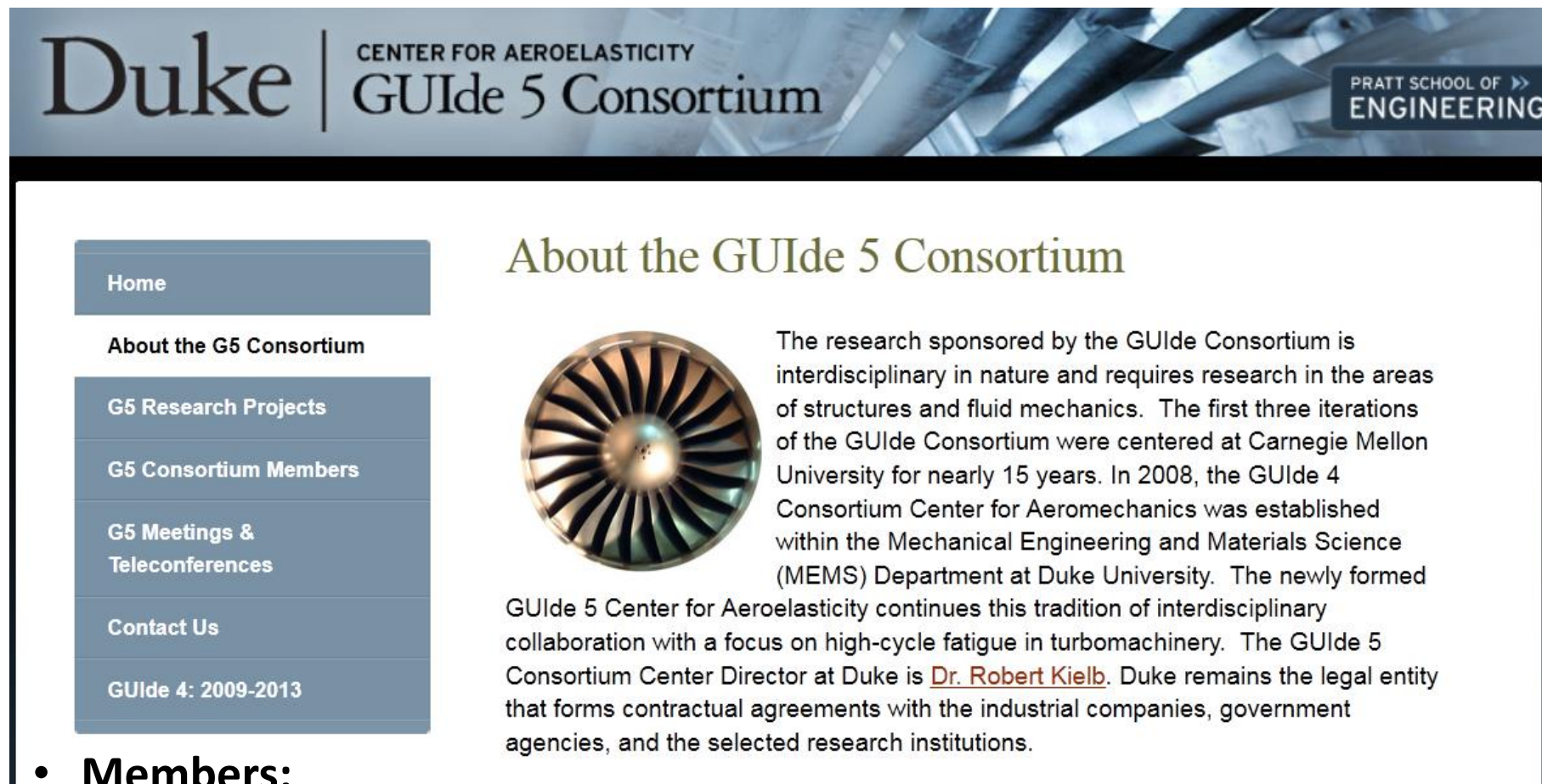
The news you need, when you need it.



Join PR Newswire for [Journalists](#) to access all of the free services designated to make your job easier.

In need of subject matter experts for your story? [Submit a free ProfNet request](#) and find the sources you need.

[LEARN MORE](#)



The screenshot shows the website for the Duke University Guide 5 Consortium. The header features the Duke University logo, the text "CENTER FOR AEROELASTICITY", "GUIDe 5 Consortium", and "PRATT SCHOOL OF ENGINEERING". A navigation menu on the left includes links to Home, About the G5 Consortium, G5 Research Projects, G5 Consortium Members, G5 Meetings & Teleconferences, Contact Us, and GUIDe 4: 2009-2013. The main content area is titled "About the GUIDe 5 Consortium" and features a photograph of a turbine engine component. The text describes the interdisciplinary nature of the research, its history at Carnegie Mellon University, and its current focus on high-cycle fatigue in turbomachinery at Duke University. It also mentions Dr. Robert Kielb as the Consortium Center Director.

About the GUIDe 5 Consortium

The research sponsored by the GUIDe Consortium is interdisciplinary in nature and requires research in the areas of structures and fluid mechanics. The first three iterations of the GUIDe Consortium were centered at Carnegie Mellon University for nearly 15 years. In 2008, the GUIDe 4 Consortium Center for Aeromechanics was established within the Mechanical Engineering and Materials Science (MEMS) Department at Duke University. The newly formed GUIDe 5 Center for Aeroelasticity continues this tradition of interdisciplinary collaboration with a focus on high-cycle fatigue in turbomachinery. The GUIDe 5 Consortium Center Director at Duke is [Dr. Robert Kielb](#). Duke remains the legal entity that forms contractual agreements with the industrial companies, government agencies, and the selected research institutions.

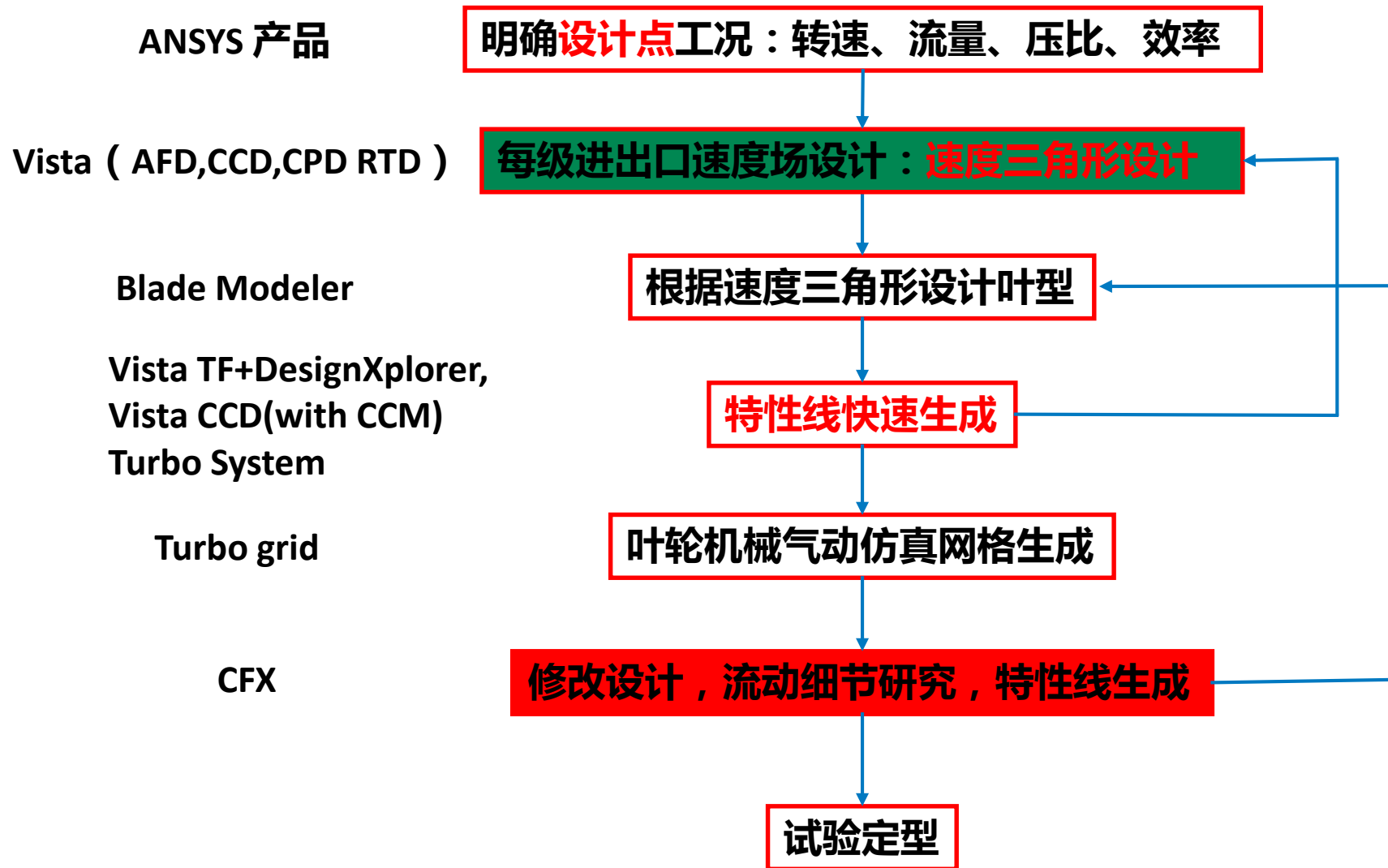
- **Members:**
 - Industry: Rolls Royce, GE, Honeywell, Siemens, IHI, MHI, Doosan, PW, **ANSYS**
 - Government: NASA Glenn, NASA Marshall, USAF AFRL
 - Research: Duke U., Purdue U., U. Michigan, Texas A&M U.

目录

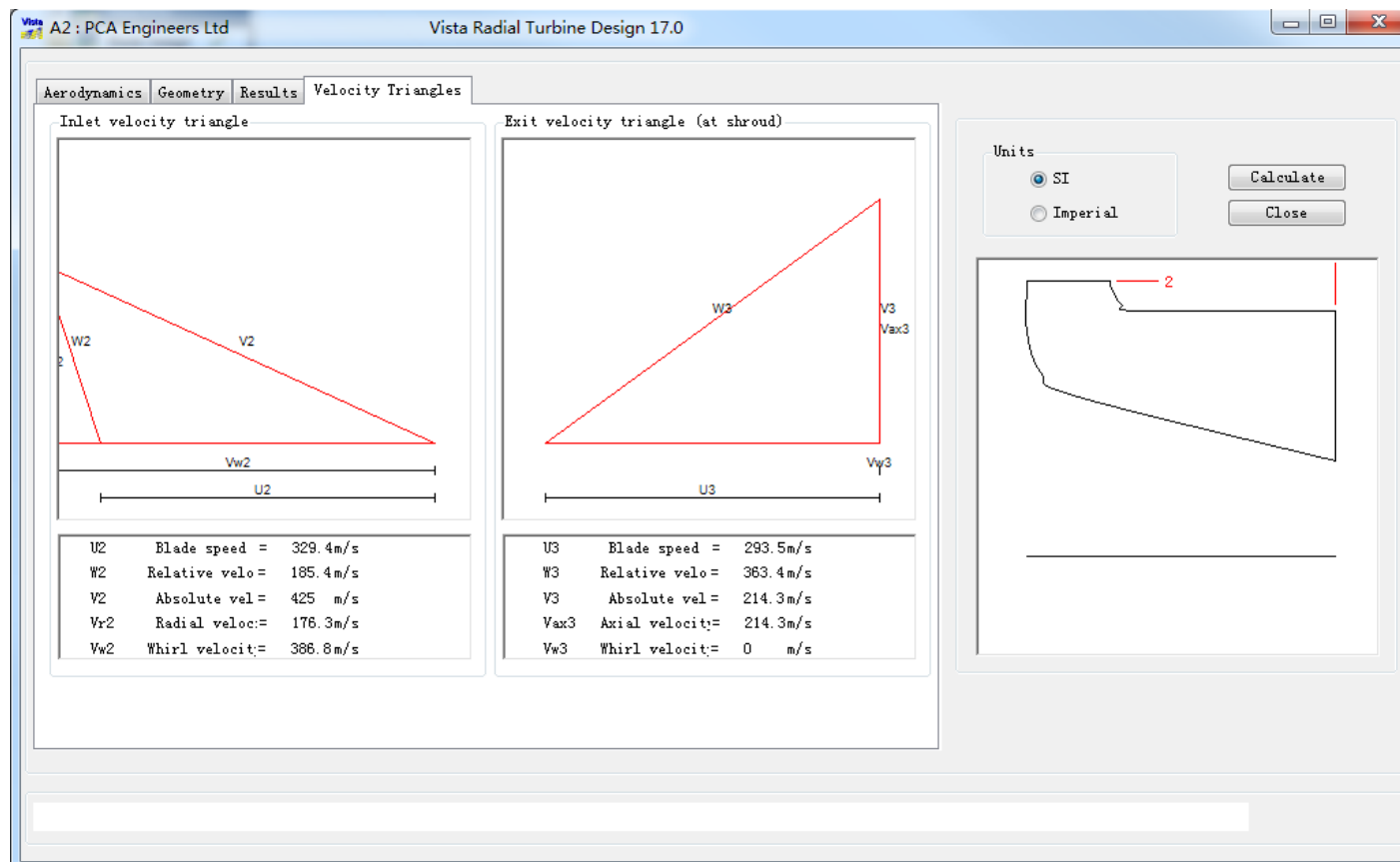
1. 航空发动机与ANSYS
2. ANSYS在叶轮机通流设计中的应用

3. ANSYS在燃烧设计中的应用
4. ANSYS在热端部件热防护设计中的应用
5. ANSYS在二次空气系统设计中的应用
6. ANSYS在零部件强度振动校核中的应用
7. ANSYS在转子动力学校核中的应用
8. ANSYS在鸟撞与包容性设计中的应用
9. ANSYS在发动机其他设计中的应用
- 10.多物理场仿真在发动机设计中的应用
- 11.总结

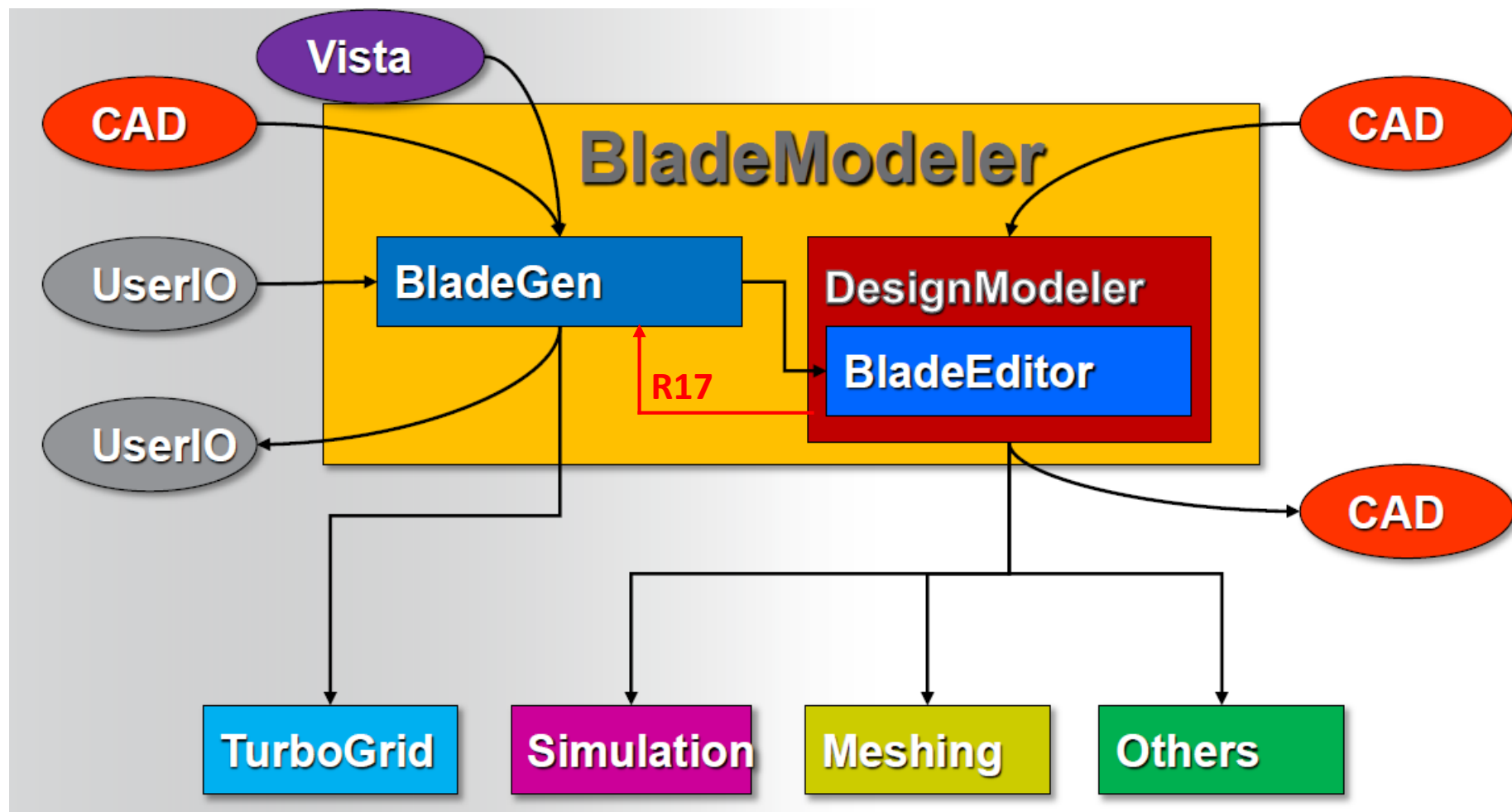
风扇、压气机、涡轮通流设计流程



速度三角形设计结果



ANSYS的叶片造型工具



设计指标

速度三角形

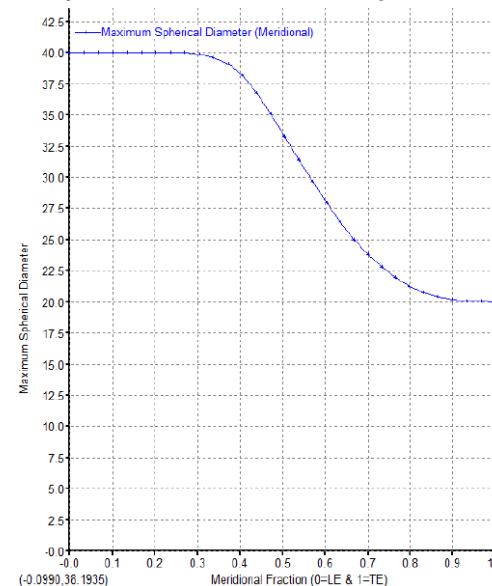
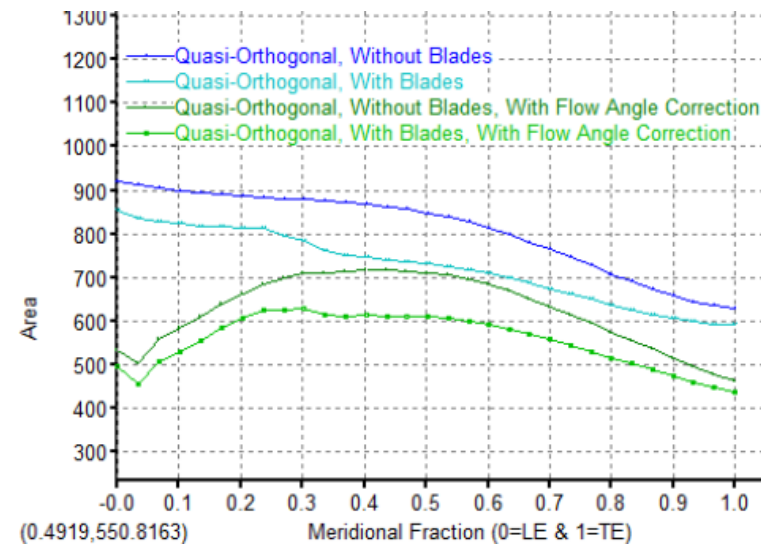
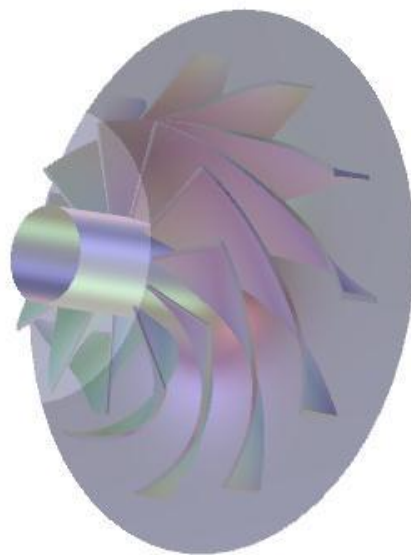
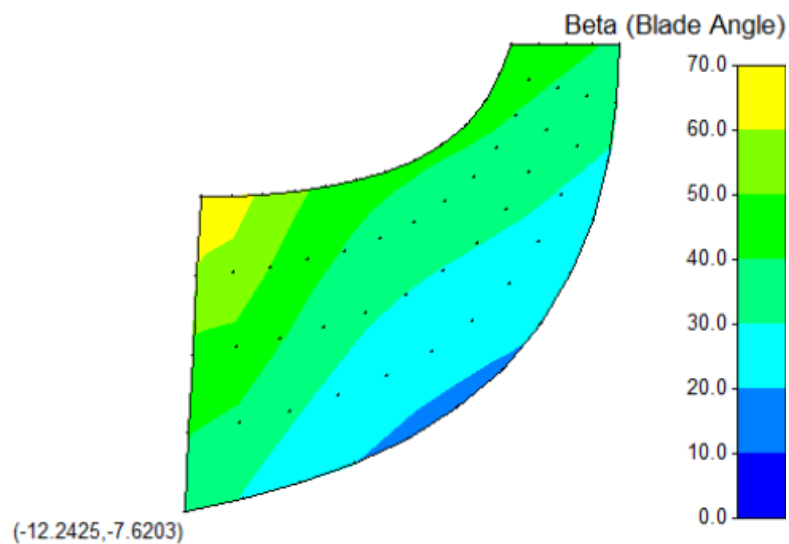
叶型设计

特性曲线快速生成

网格

细节分析

BladeGen



设计指标

速度三角形

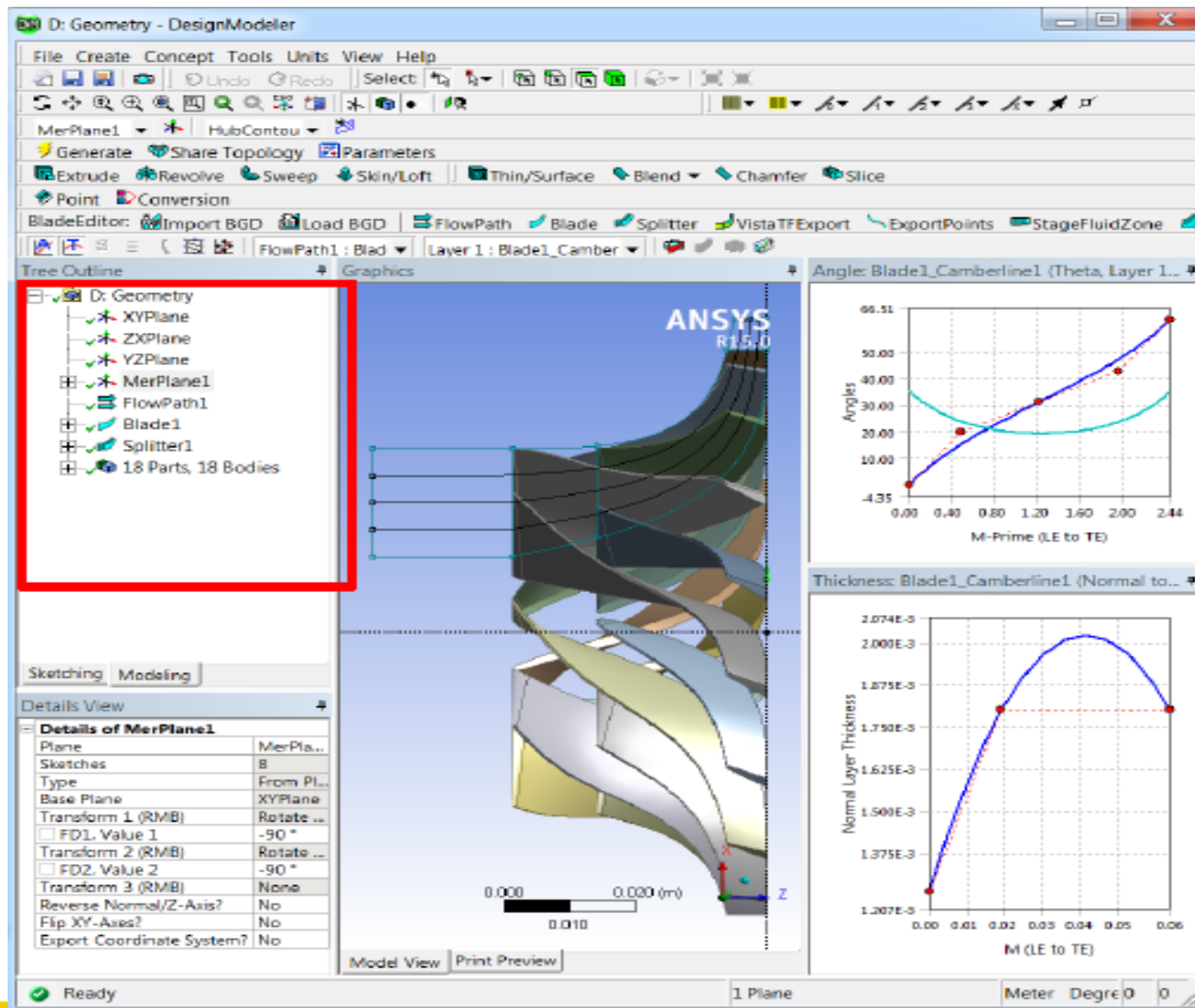
叶型设计

特性曲线快速生成

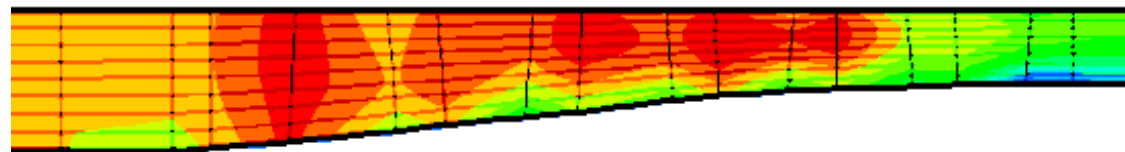
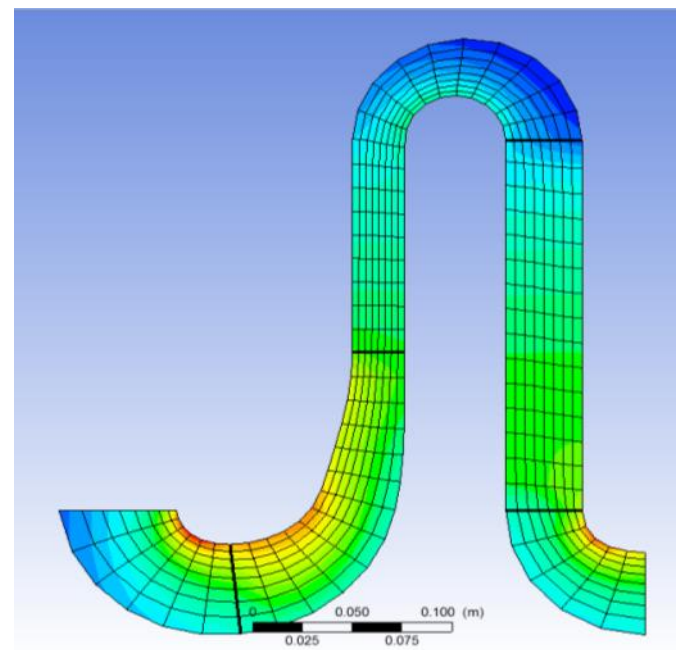
网格

细节分析

BladeEditor



- Integration of through-flow solver
 - VISTA-TF
 - PCA Engineers, Lincoln
- Rapid blade row analysis
- Complementary to CFD simulation
 - Flow path optimisation
 - Initial condition



Vista TF + DesignXplorer

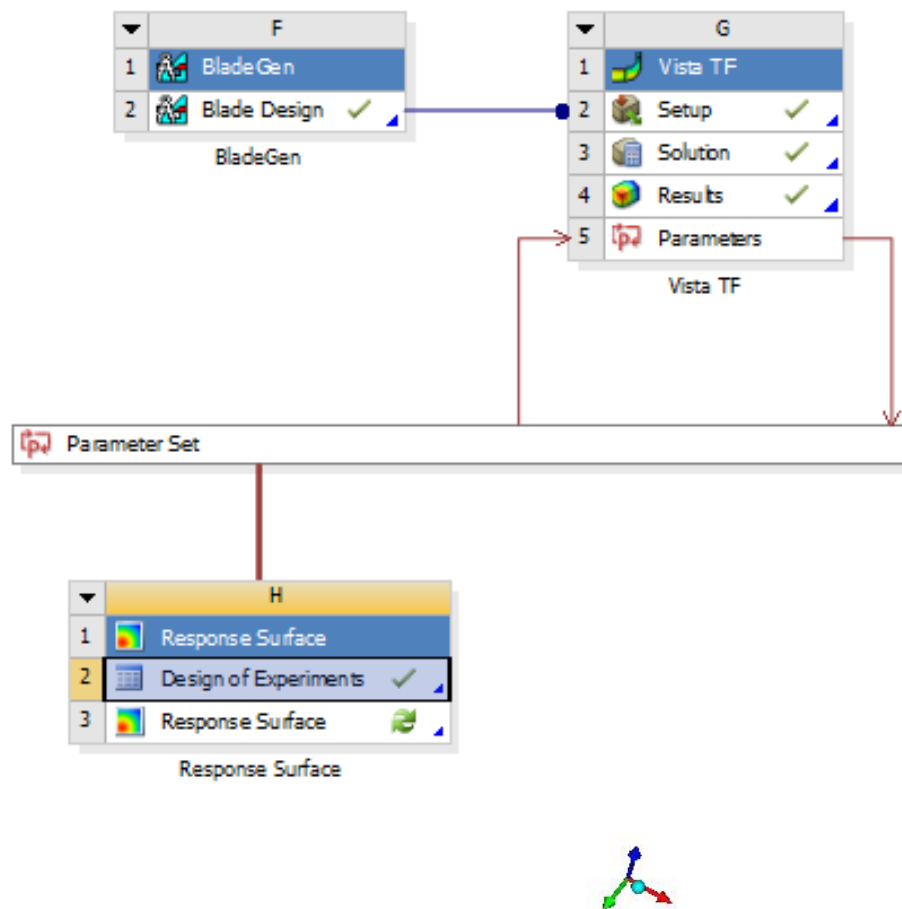
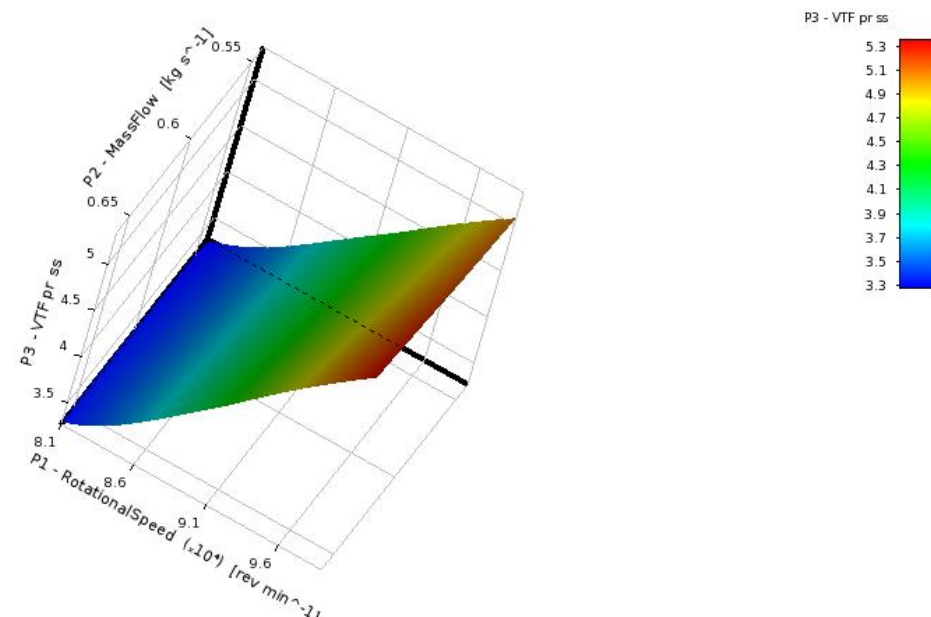
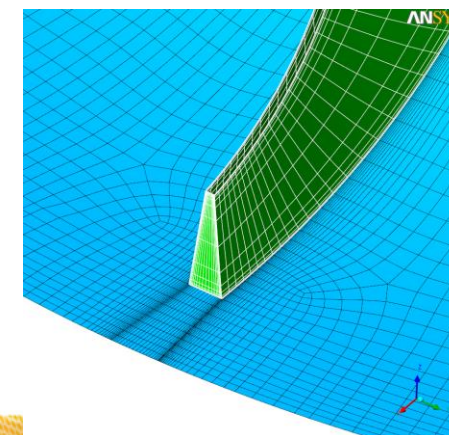
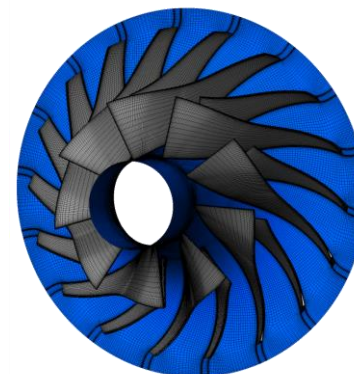
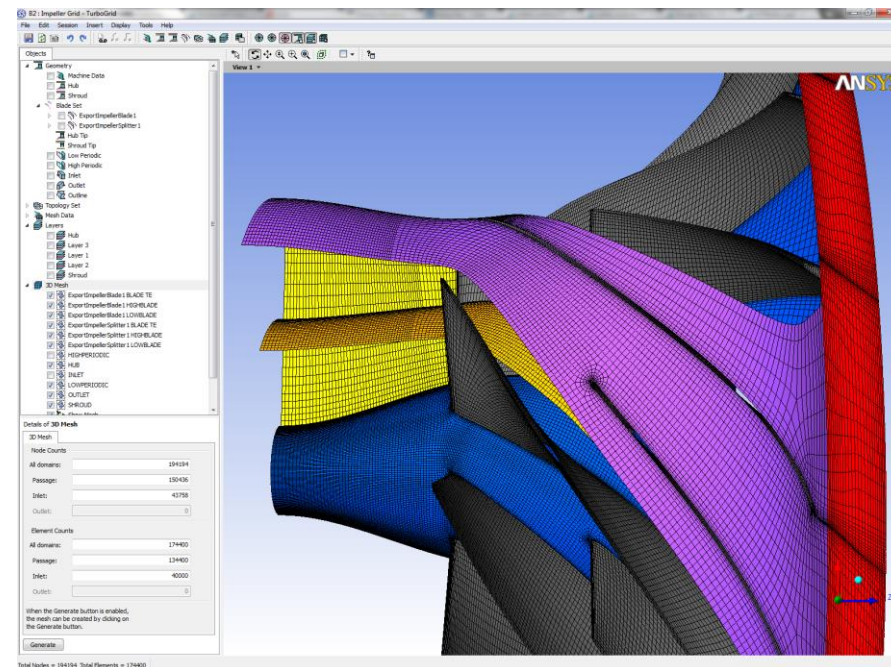
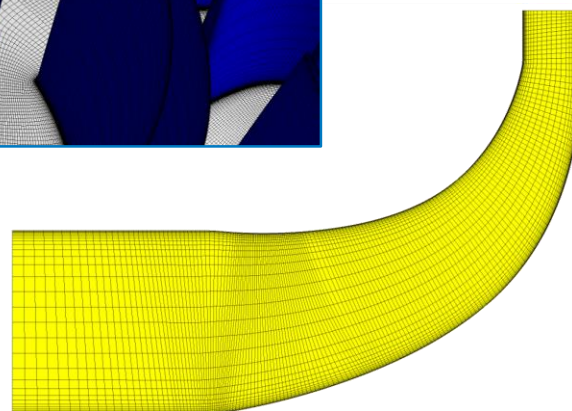
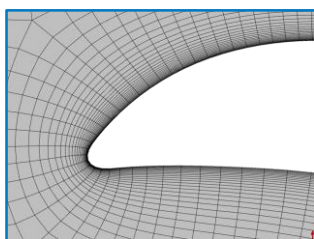
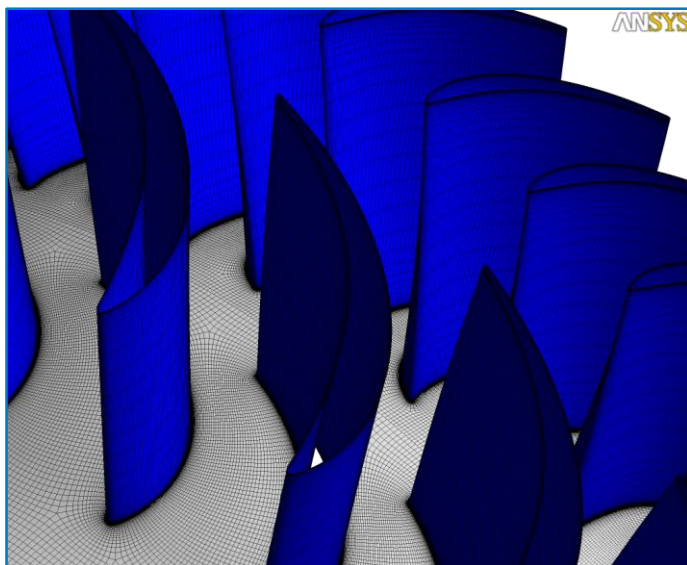


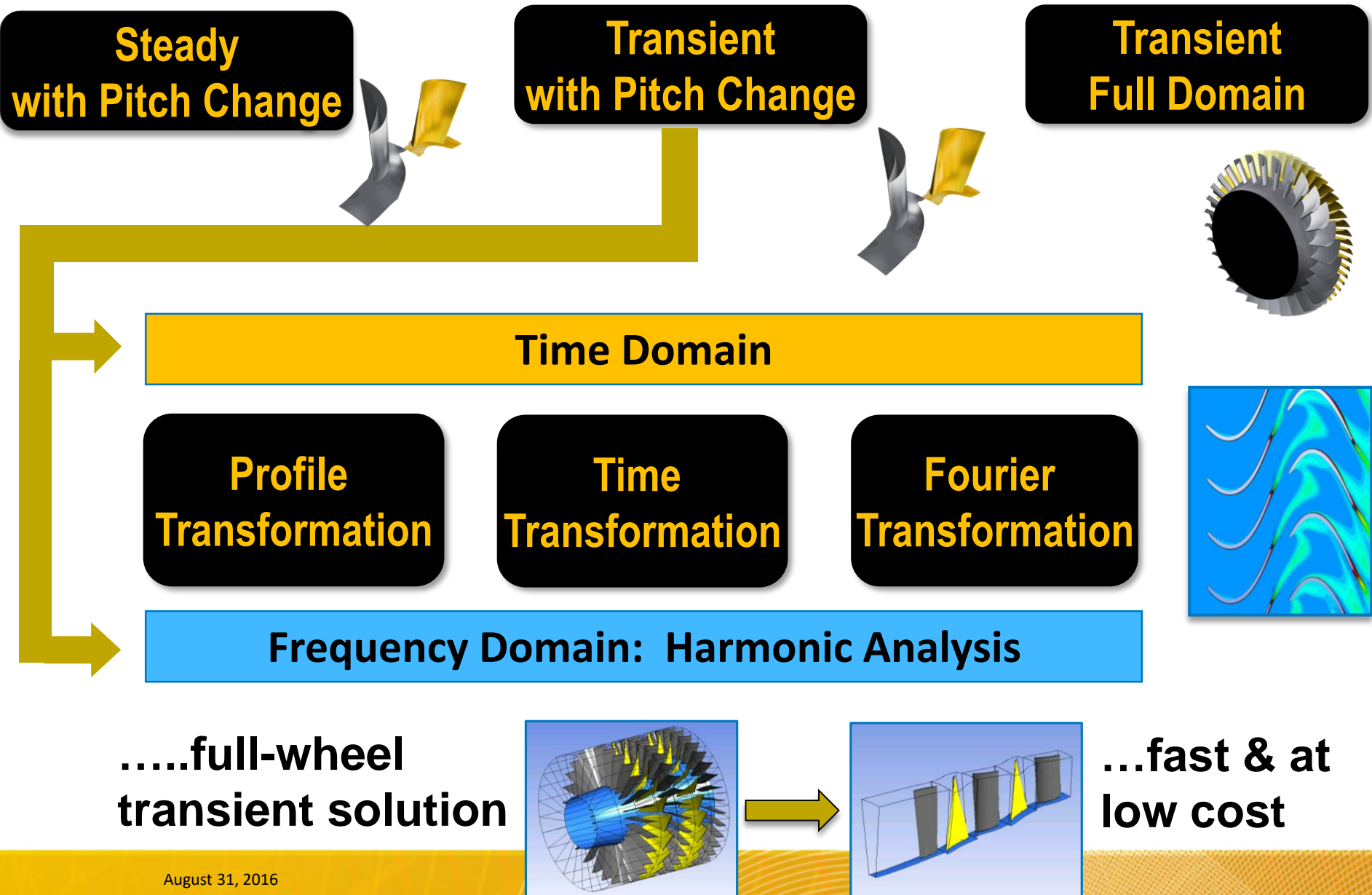
Table of Schematic H2: Design of Experiments (Central Composite Design : Auto Defined)

	A	B	C	D
1	Name	P1 - RotationalSpeed (rev min ⁻¹)	P2 - MassFlow (kg s ⁻¹)	P3 - VTF pr ss
2	1 DP 0	90000	0.6	4.1605
3	2	81000	0.6	3.3039
4	3	99000	0.6	5.2048
5	4	90000	0.54	4.0788
6	5	90000	0.66	4.2362
7	6	81000	0.54	3.2695
8	7	99000	0.54	5.0736
9	8	81000	0.66	3.2943
10	9	99000	0.66	5.3507



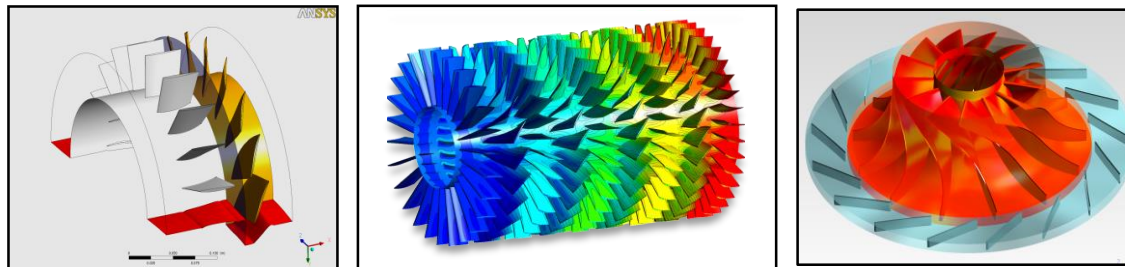
- High quality hexahedral grids
- Repeatable & scalable
- Axial, radial, mixed flow geometries



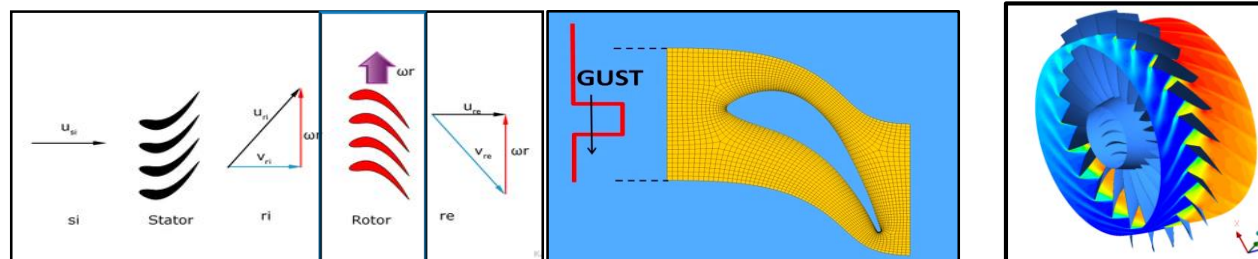


ANSYS TBR 应用场景

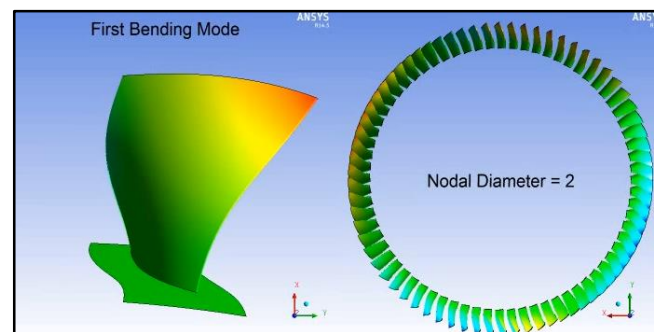
- 转静相互作用
 - 单级& 多级
 - 轴流 & 径向



- 进口扰动
 - 阵风分析
 - 进气畸变

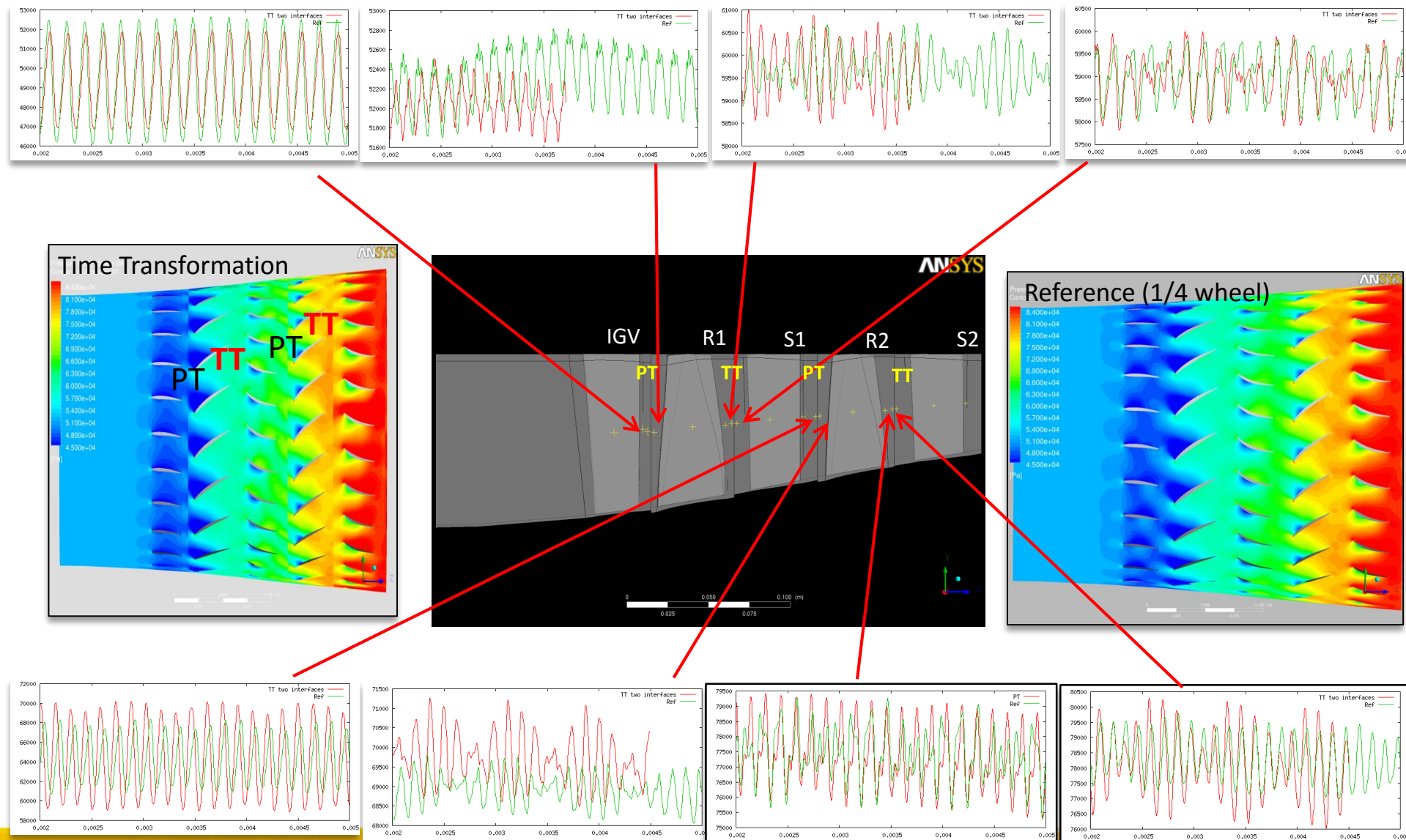


- 流固耦合
 - 叶片颤振
 - 受迫响应



Modified Hannover Axial Compressor

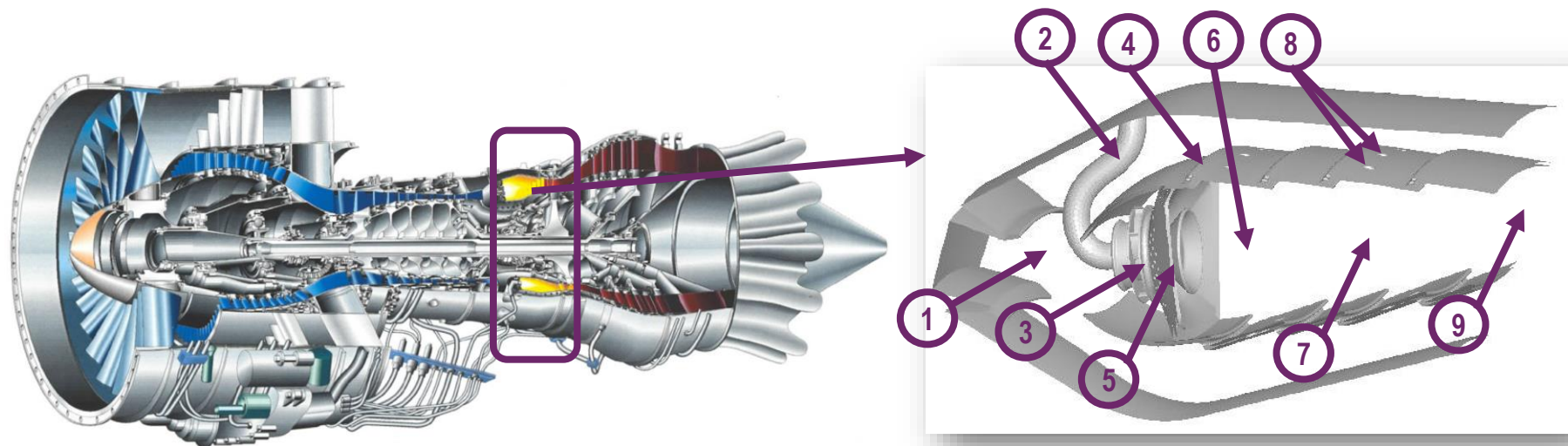
PT + 2 TT vs. Full Domain



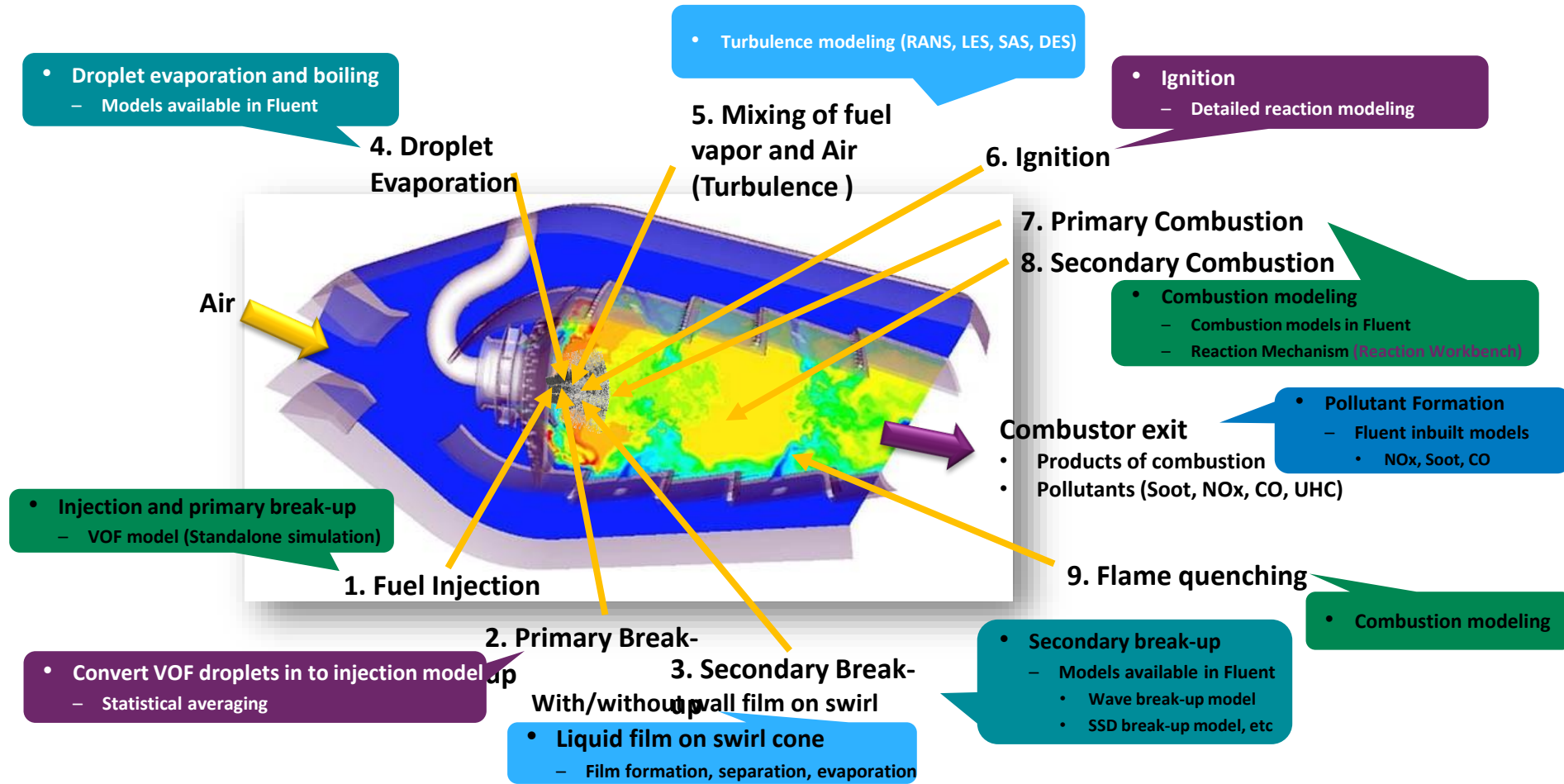
目录

1. 航空发动机与ANSYS
2. ANSYS在叶轮机通流设计中的应用
3. ANSYS在燃烧设计中的应用
4. ANSYS在热端部件热防护设计中的应用
5. ANSYS在二次空气系统设计中的应用
6. ANSYS在零部件强度振动校核中的应用
7. ANSYS在转子动力学校核中的应用
8. ANSYS在鸟撞与包容性设计中的应用
9. ANSYS在发动机其他设计中的应用
10. 多物理场仿真在发动机设计中的应用
11. 总结

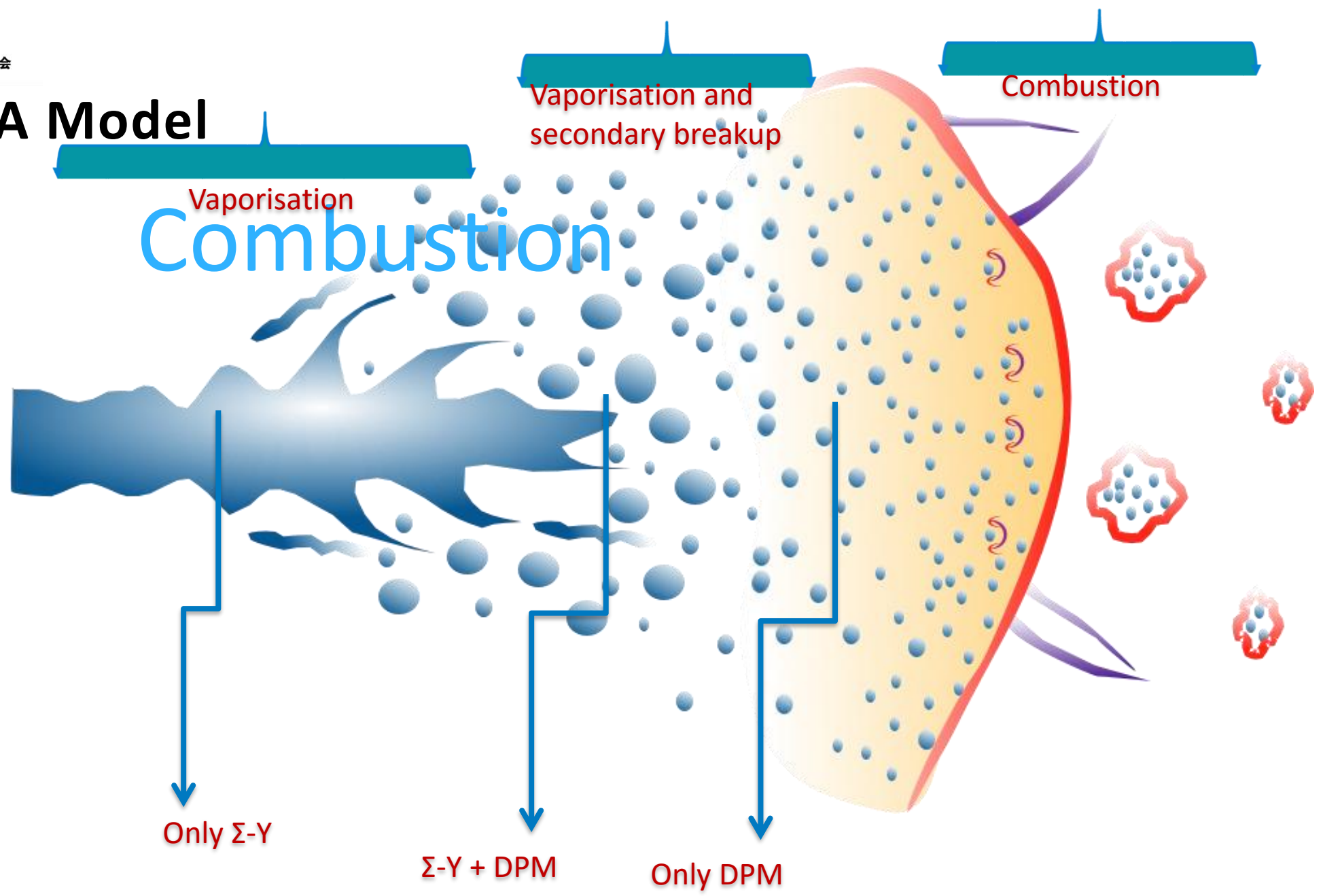
燃烧室介绍



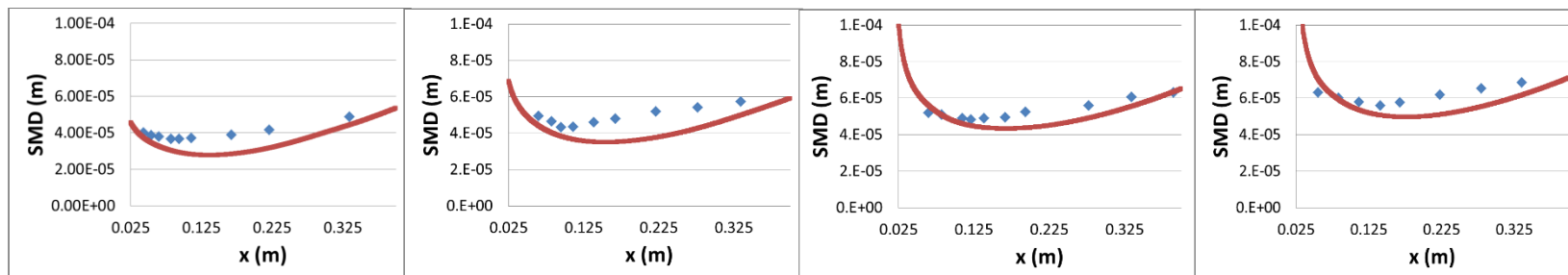
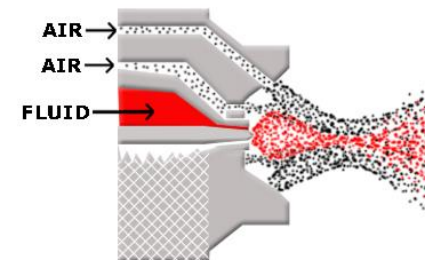
- 1: 扩压器
- 2: 供油管
- 3: 旋流器
- 4: 火焰筒
- 5: 旋流杯
- 6: 主燃区
- 7: 次燃区或掺混区
- 8: 掺混孔
- 9: 燃烧室出口



ELSA Model



仿真与试验的对比结果



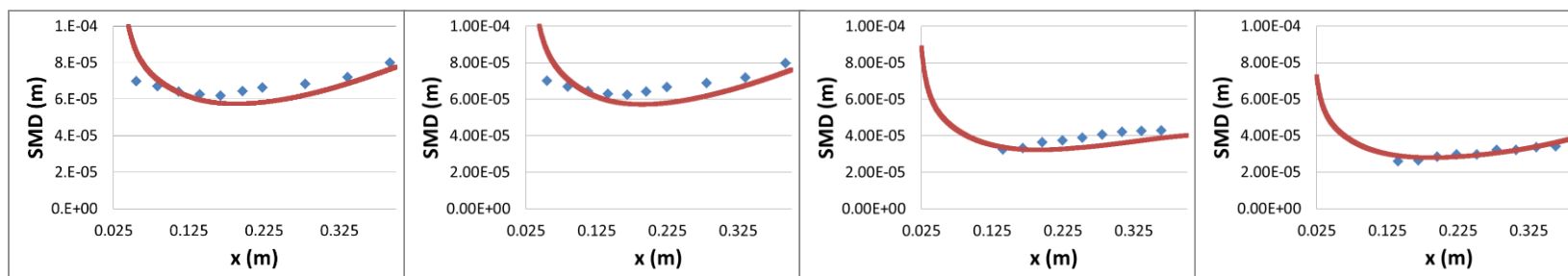
- $U_g = 140$ m/s and
- $U_{liq} = 0.13$ m/s

$U_g = 140$ m/s and
 $U_{liq} = 0.2$ m/s

$U_g = 140$ m/s and
 $U_{liq} = 0.31$ m/s

$U_g = 140$ m/s and
 $U_{liq} = 0.43$ m/s

◆ Experimental
■ Fluent



$U_g = 140$ m/s and
 $U_{liq} = 0.55$ m/s

$U_g = 140$ m/s and
 $U_{liq} = 0.5$ m/s

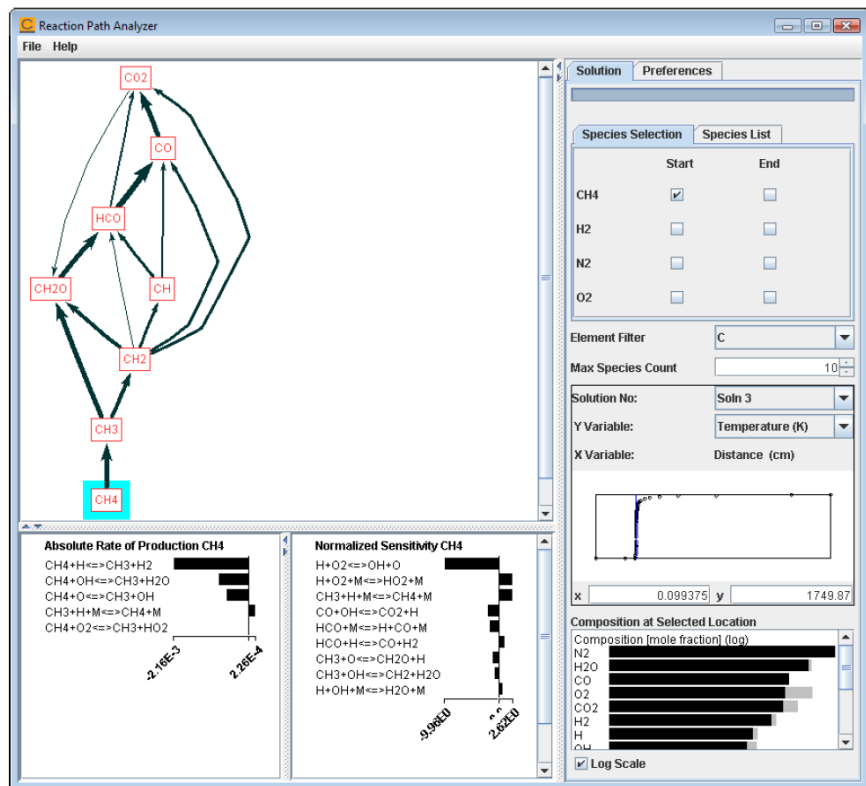
$U_g = 205$ m/s and
 $U_{liq} = 0.5$ m/s

$U_g = 225$ m/s and
 $U_{liq} = 0.5$ m/s

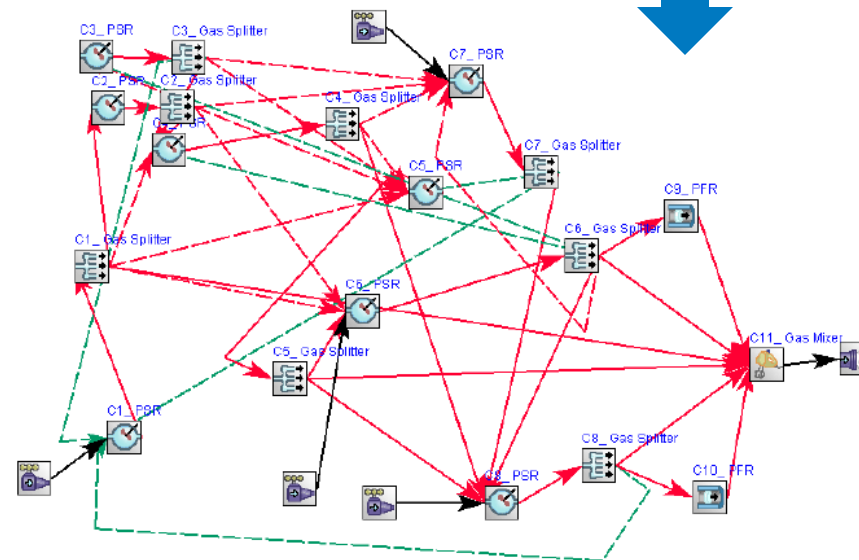
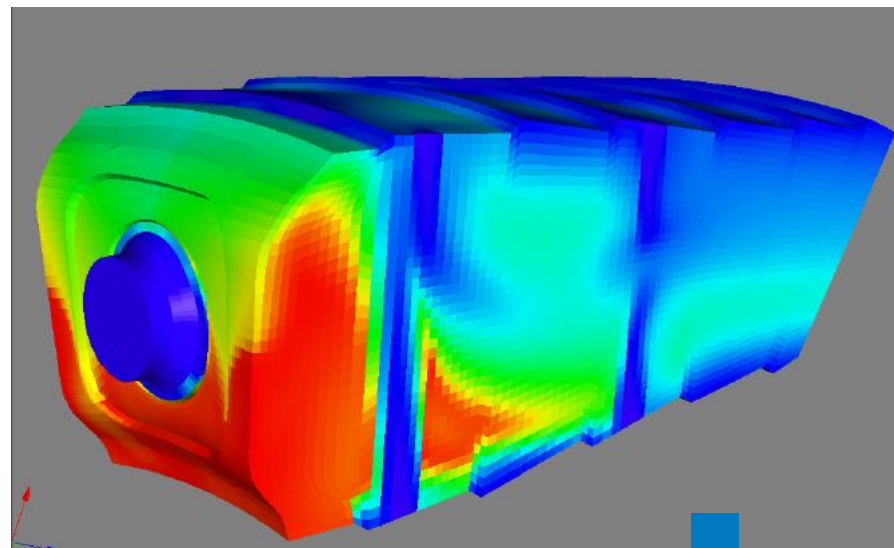
- Comparison of Sauter mean diameter data at symmetry axis obtained from simulations with experimental reporting of Lasheras et. al. (1998)

		Flow Configuration		
		Premixed Combustion	Non-Premixed Combustion	Partially Premixed Combustion
Chemistry	Fast Chemistry	Finite Rate/Eddy Dissipation Model (Species Transport)		
		Premixed Combustion Model Reaction Progress Variable	Non-Premixed Equilibrium Model Mixture Fraction	Partially Premixed Model Reaction Progress Variable + Mixture Fraction
		Flamelet Generated Manifold Model		
		Finite Rate Chemistry	Laminar Finite Rate Model	
	Eddy-Dissipation Concept (EDC) Model			
Composition PDF Transport Model (Eulerian/Lagrangian)				

Reaction Design的产品



Chemkin Pro

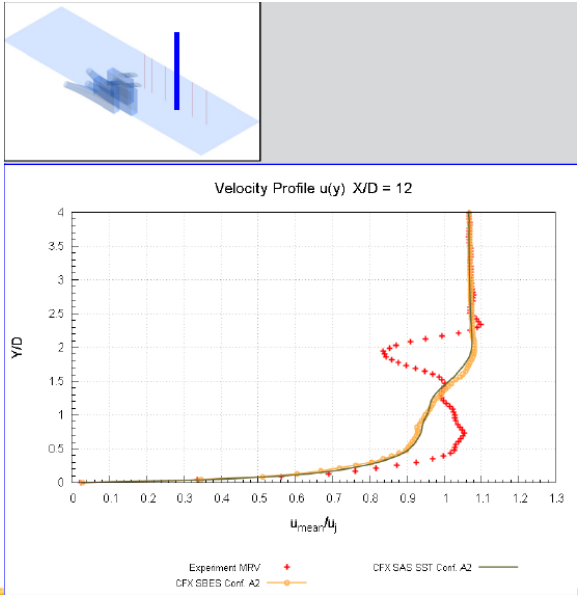
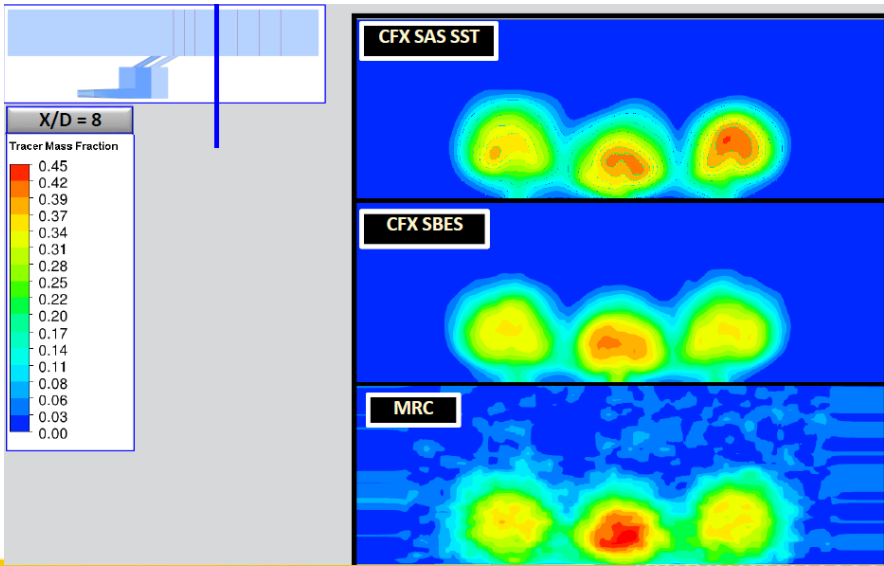
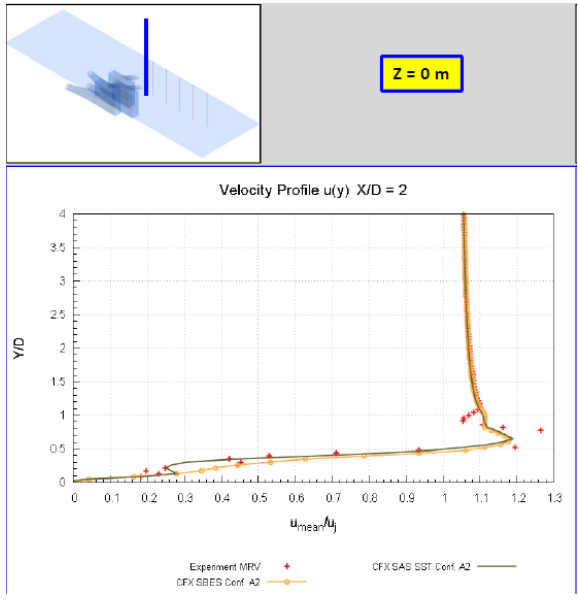
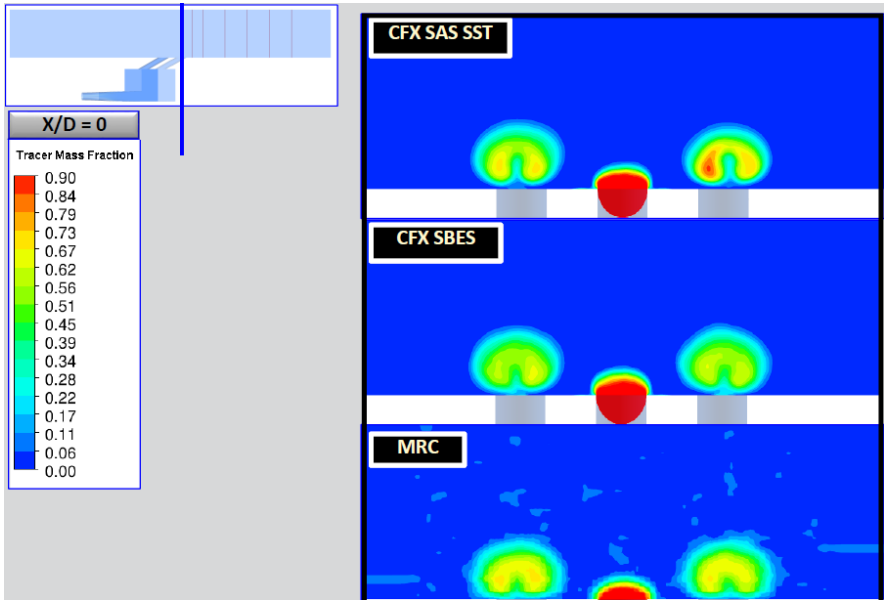


Energico

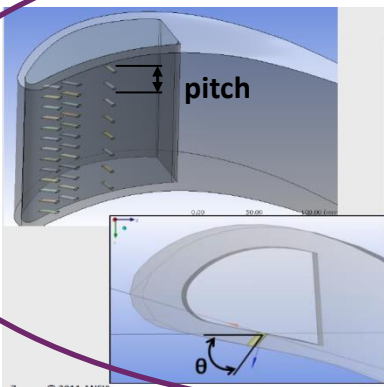
目录

1. 航空发动机与ANSYS
2. ANSYS在叶轮机通流设计中的应用
3. ANSYS在燃烧设计中的应用
4. ANSYS在热端部件热防护设计中的应用
5. ANSYS在二次空气系统设计中的应用
6. ANSYS在零部件强度振动校核中的应用
7. ANSYS在转子动力学校核中的应用
8. ANSYS在鸟撞与包容性设计中的应用
9. ANSYS在发动机其他设计中的应用
10. 多物理场仿真在发动机设计中的应用
11. 总结

机理研究

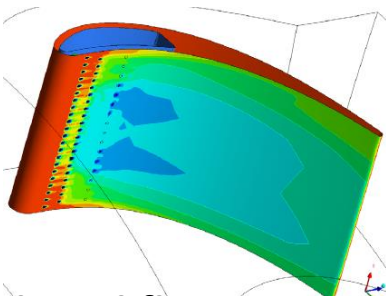


气膜孔设计优化



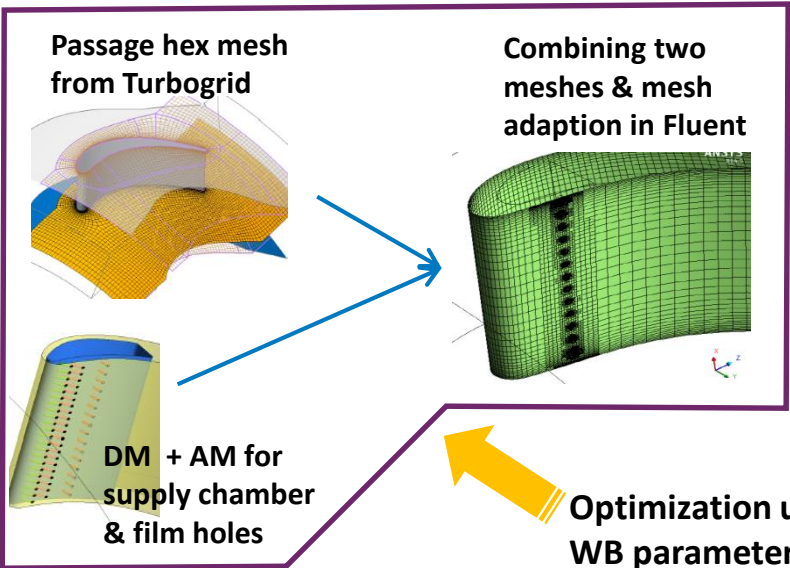
Parameters

- 1) Hole diameter
- 2) Pitch
- 3) Angle of releasing coolant
- 4) Flow through hole
- 5) Parameters related to flaring of hole near exit



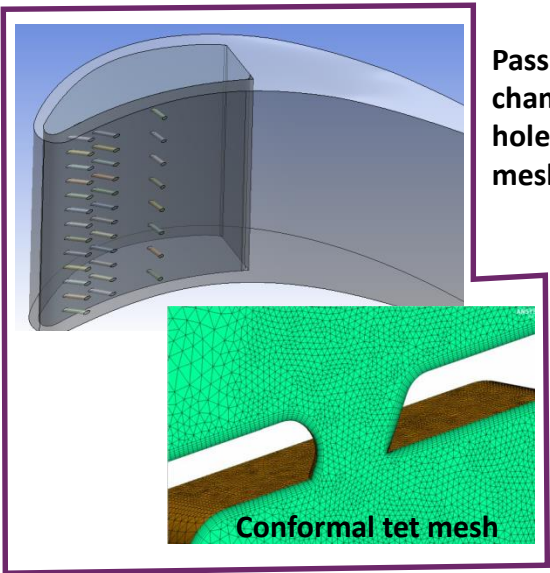
Automated workflow & use of ANSYS workbench for optimization of film cooling effectiveness

Method 1



Optimization using WB parameters

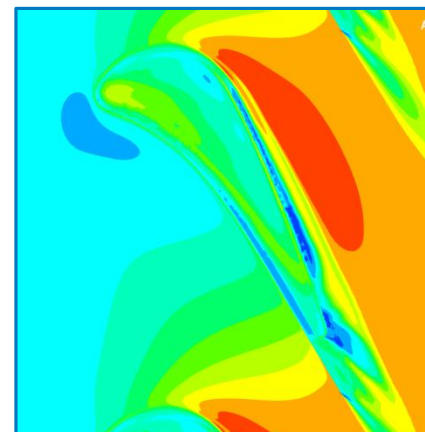
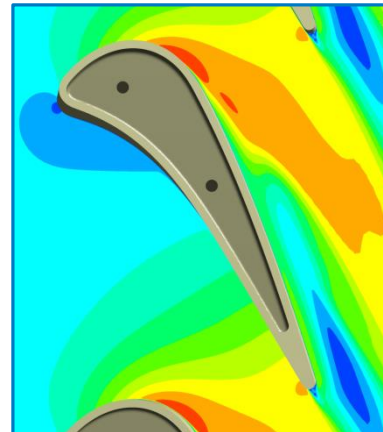
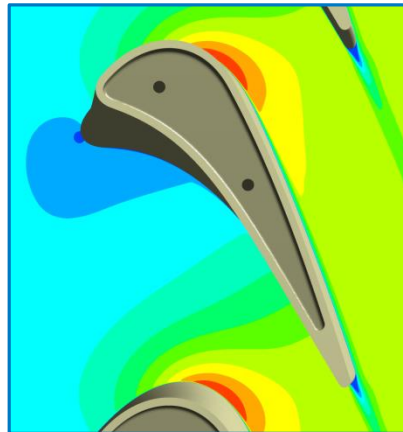
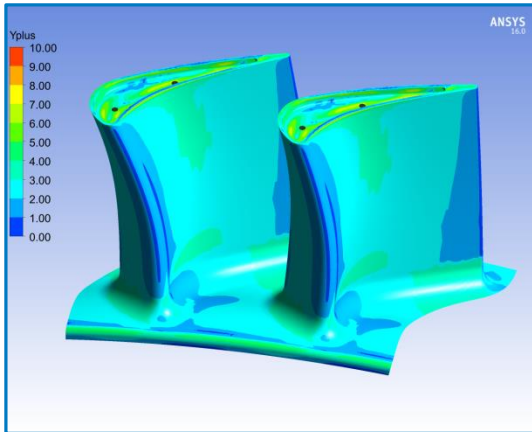
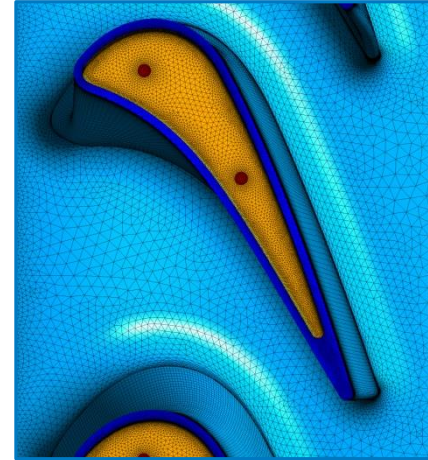
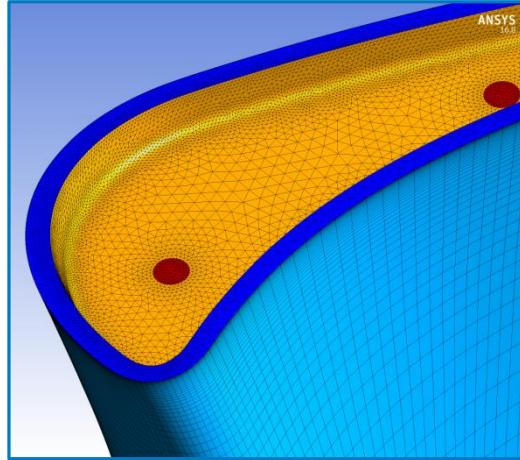
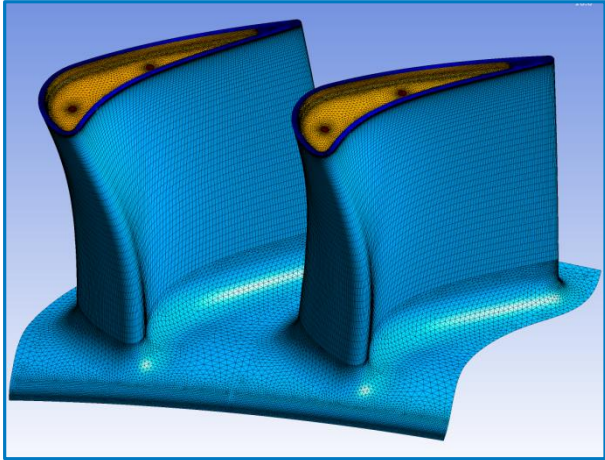
Method 2



Optimization using WB parameters

Cooled Turbine Blade Meshing

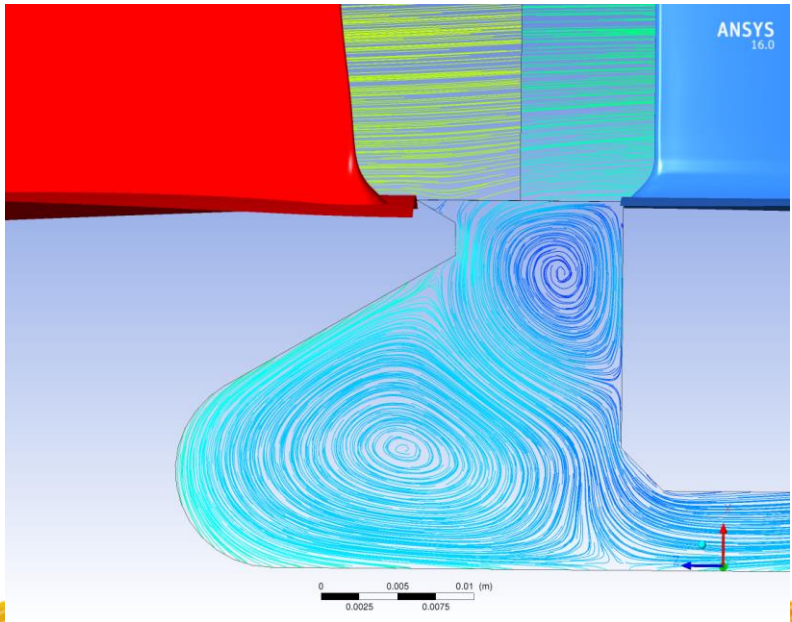
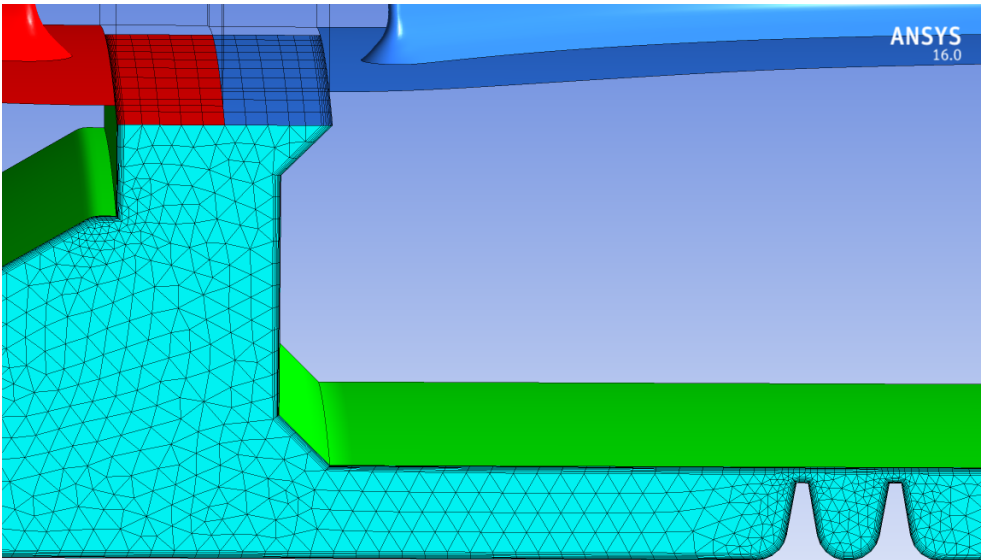
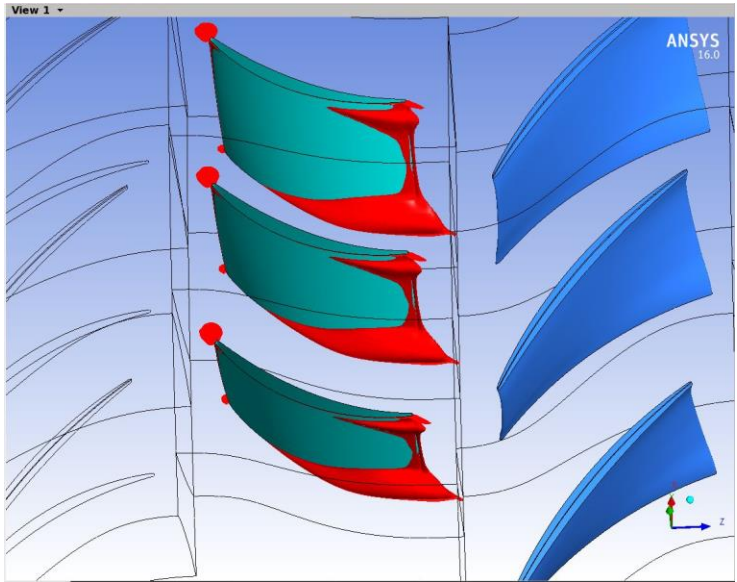
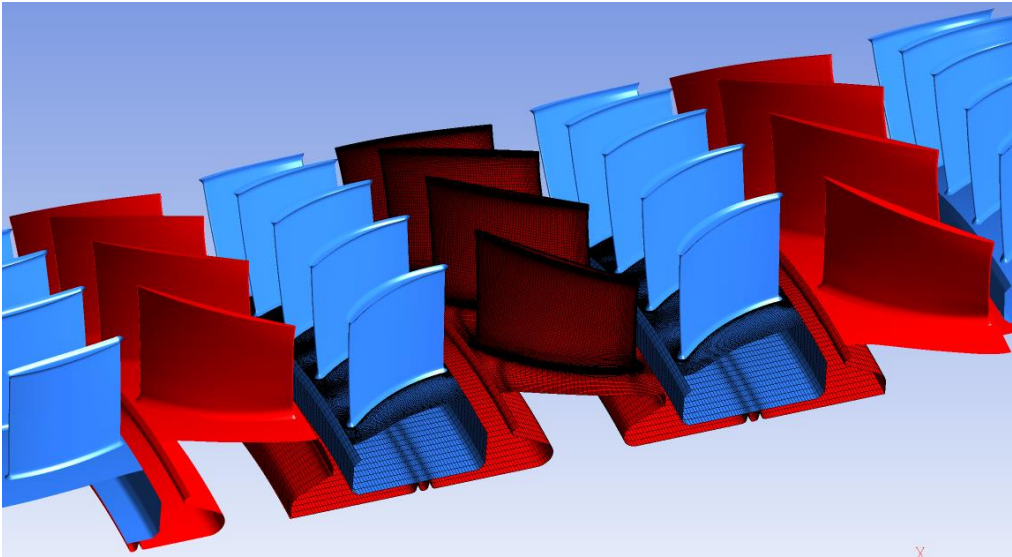
- Hybrid meshing with ANSYS tools
 - Geometric flexibility + High Quality



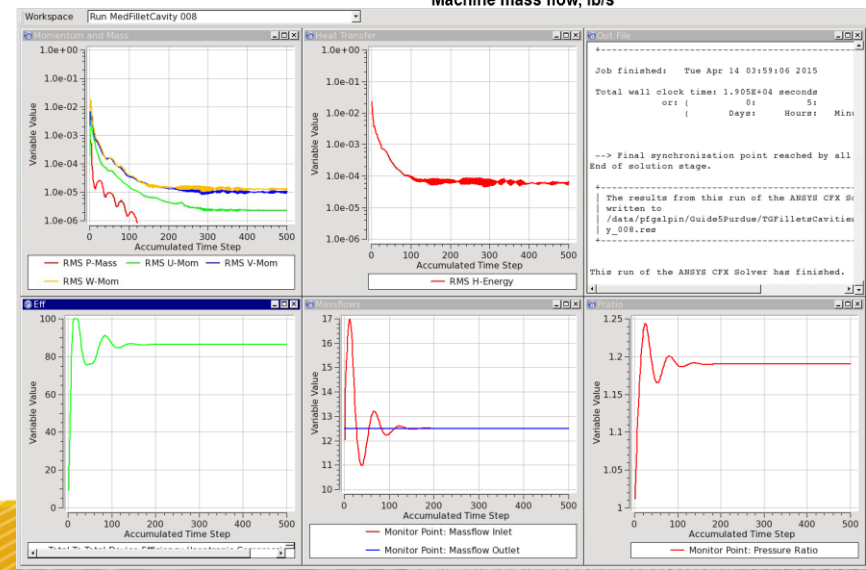
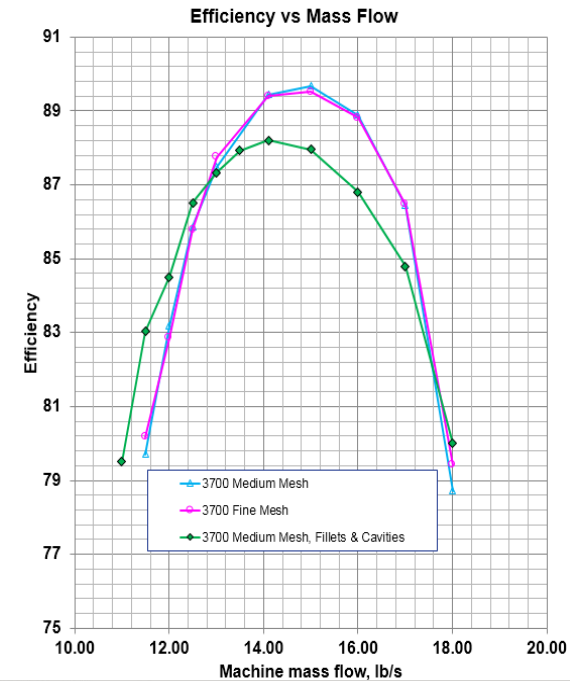
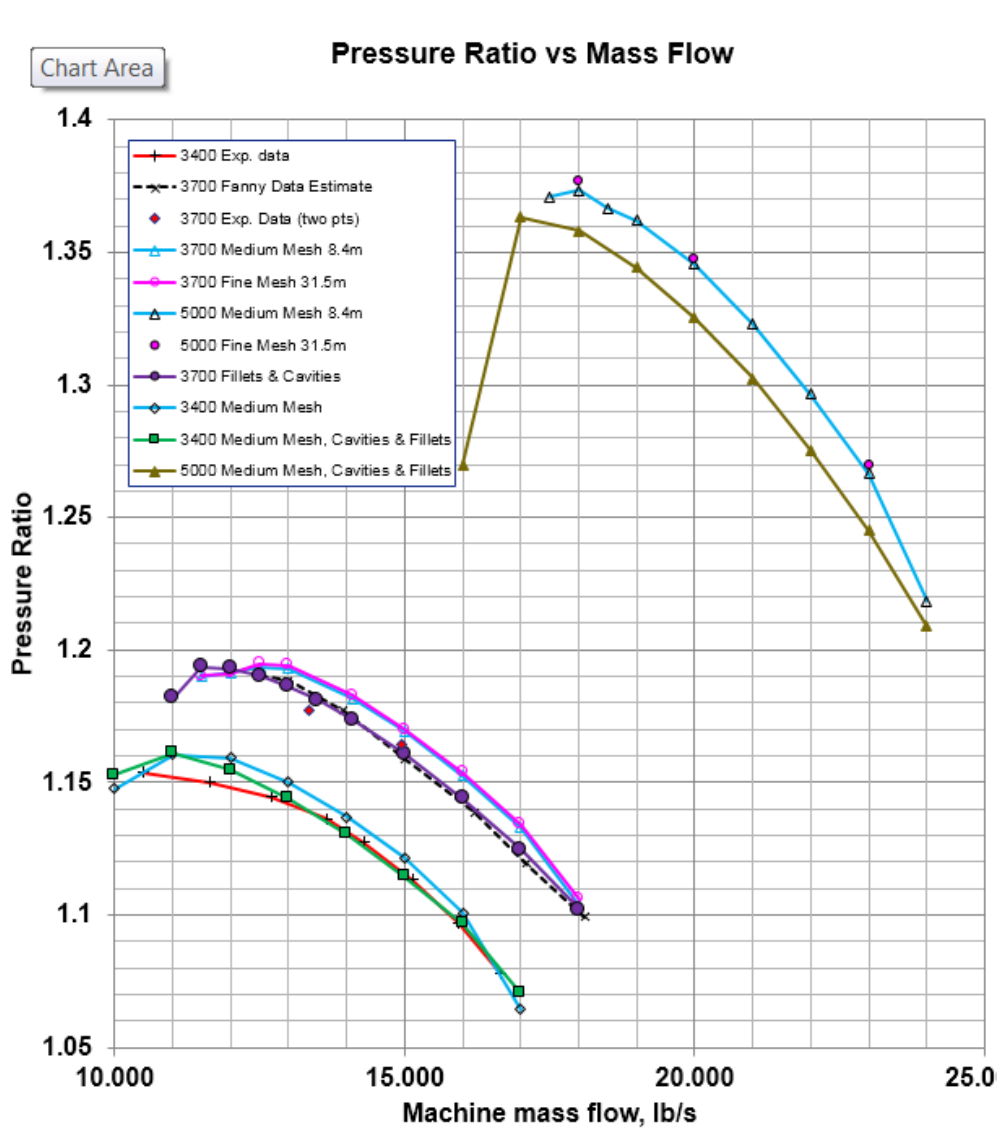
目录

1. 航空发动机与ANSYS
2. ANSYS在叶轮机通流设计中的应用
3. ANSYS在燃烧设计中的应用
4. ANSYS在热端部件热防护设计中的应用
5. ANSYS在二次空气系统设计中的应用
6. ANSYS在零部件强度振动校核中的应用
7. ANSYS在转子动力学校核中的应用
8. ANSYS在鸟撞与包容性设计中的应用
9. ANSYS在发动机其他设计中的应用
10. 多物理场仿真在发动机设计中的应用
11. 总结

ANSYS Preliminary Analysis

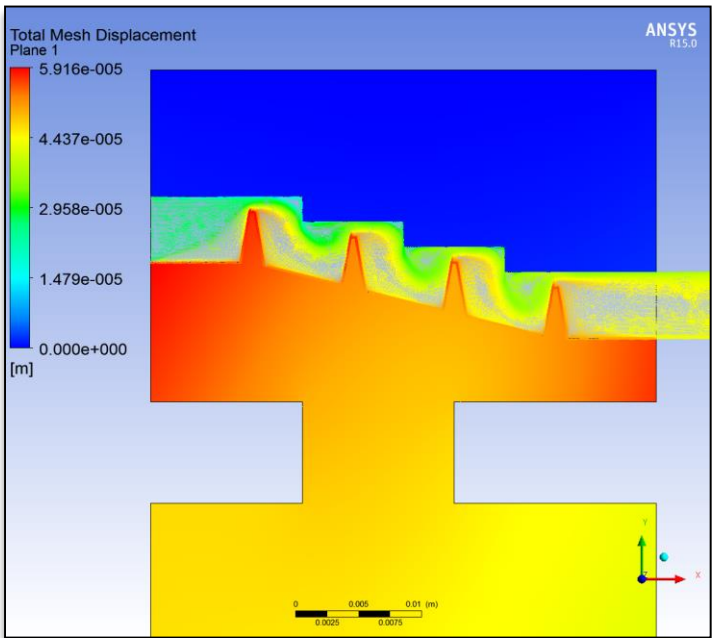


ANSYS Preliminary Analysis

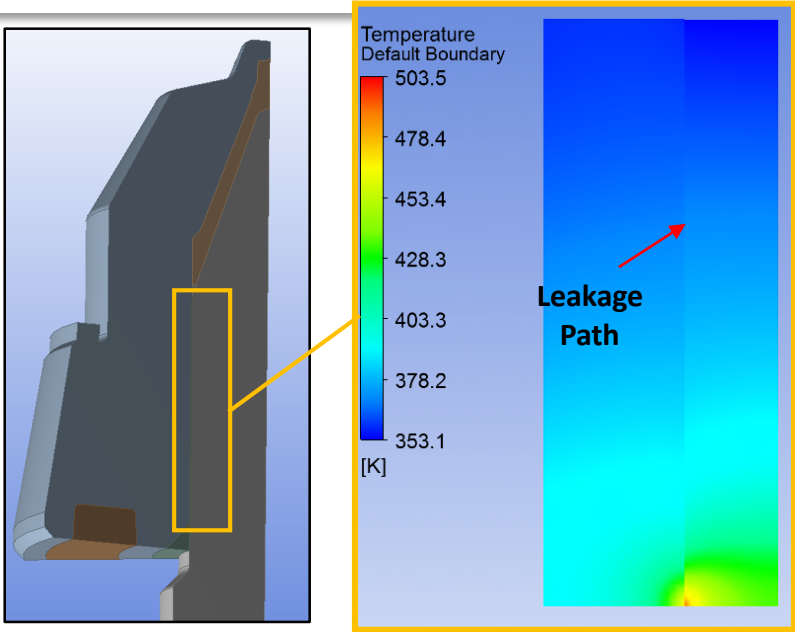
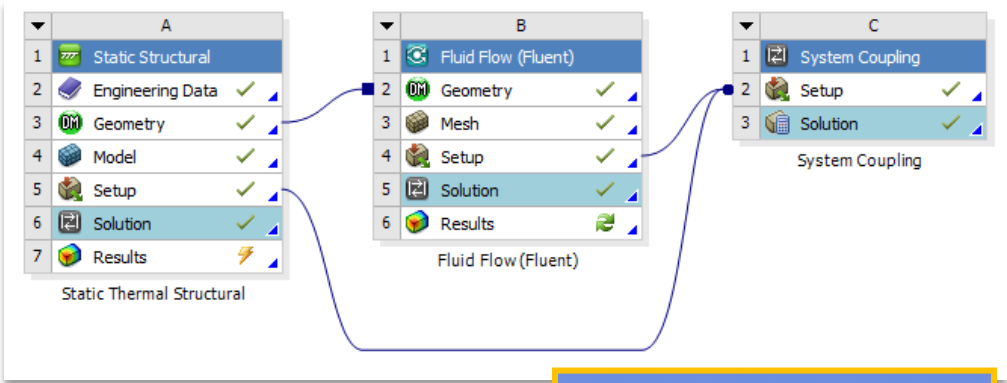


2-way Thermal+Structural – Fluids

Force-Displacement and Thermal coupling



Labyrinth seal. The seal gap depends on the structural rotational forces, fluid pressure loads and thermal stresses caused in part by the hot fluid.



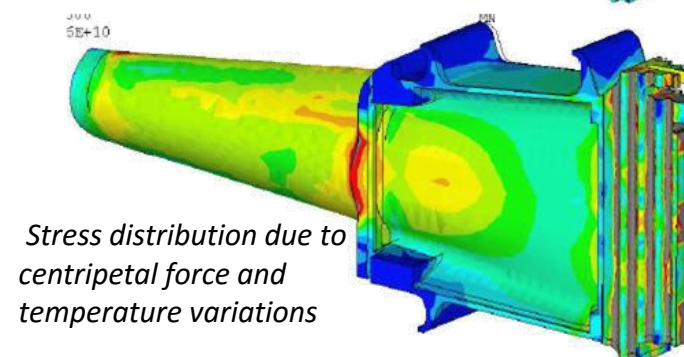
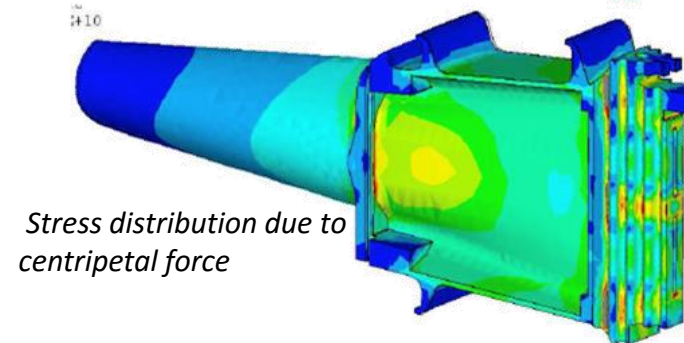
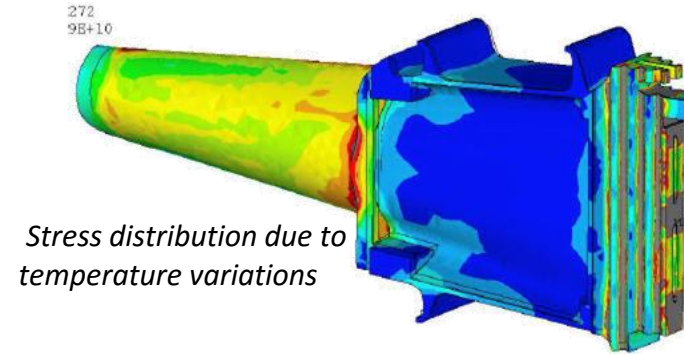
Fuel injector leakage path solved using Fluent and Coupled Field Elements in Mechanical

目录

1. 航空发动机与ANSYS
2. ANSYS在叶轮机通流设计中的应用
3. ANSYS在燃烧设计中的应用
4. ANSYS在热端部件热防护设计中的应用
5. ANSYS在二次空气系统设计中的应用
6. ANSYS在零部件强度振动校核中的应用
7. ANSYS在转子动力学校核中的应用
8. ANSYS在鸟撞与包容性设计中的应用
9. ANSYS在发动机其他设计中的应用
10. 多物理场仿真在发动机设计中的应用
11. 总结

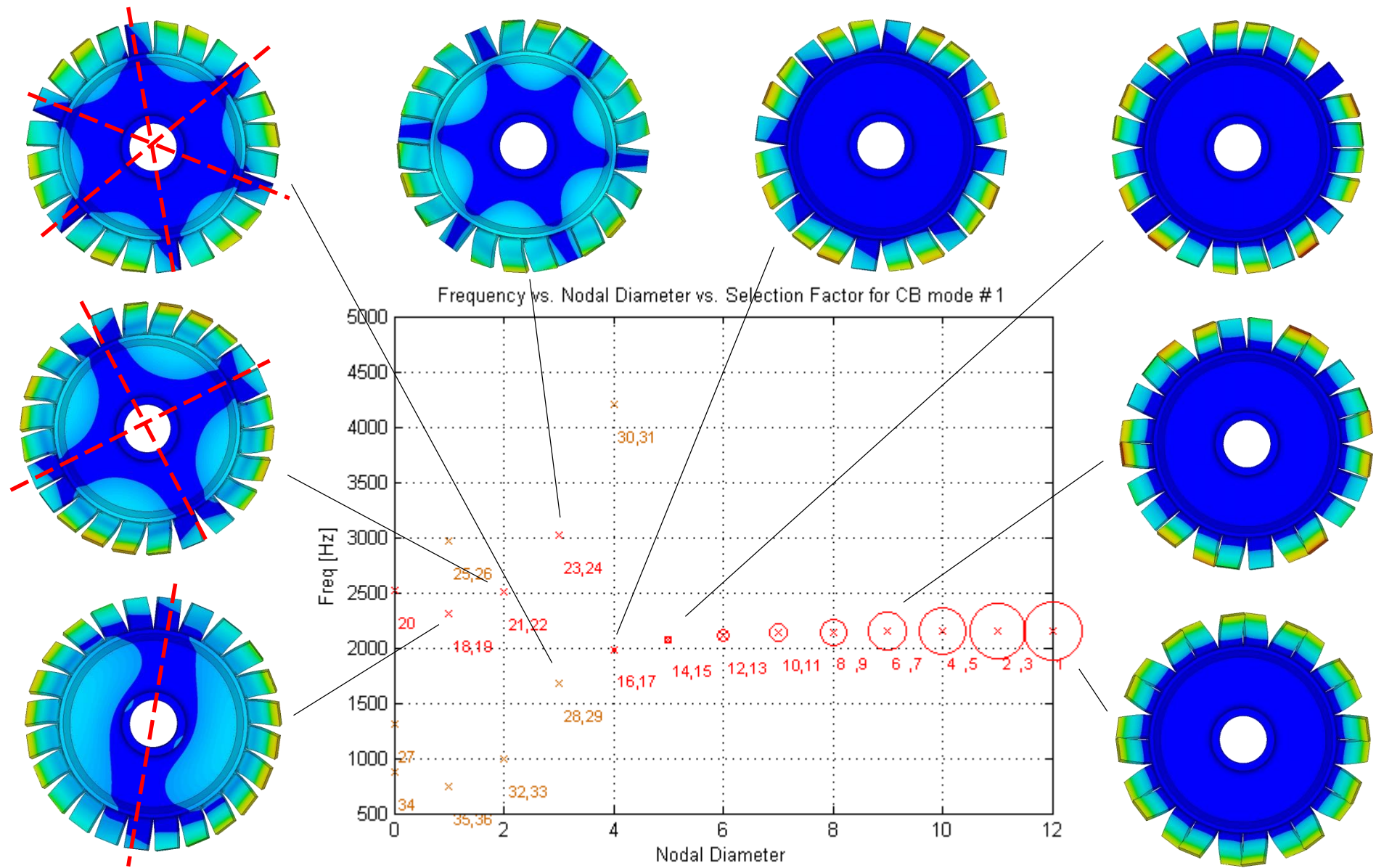
Multiphysics Simulation Captures Gas Turbine Blade Deformation

- **GOAL:** Ensure blades can withstand thermal loading *and* centripetal force
 - First stage blades in 40MW industrial gas turbines experience high temperatures and must meet performance expectations even if they deform
- Transferred thermal load results from CFD transient analyses to Mechanical to capture deformations and stress
- **KEY RESULTS:**
 - Understanding of stress distributions → *extended blade life and reduced downtime*
 - Informed about the effects of pressure and thermal forces separately and together → *informed trade-off decisions about materials and welds possible*



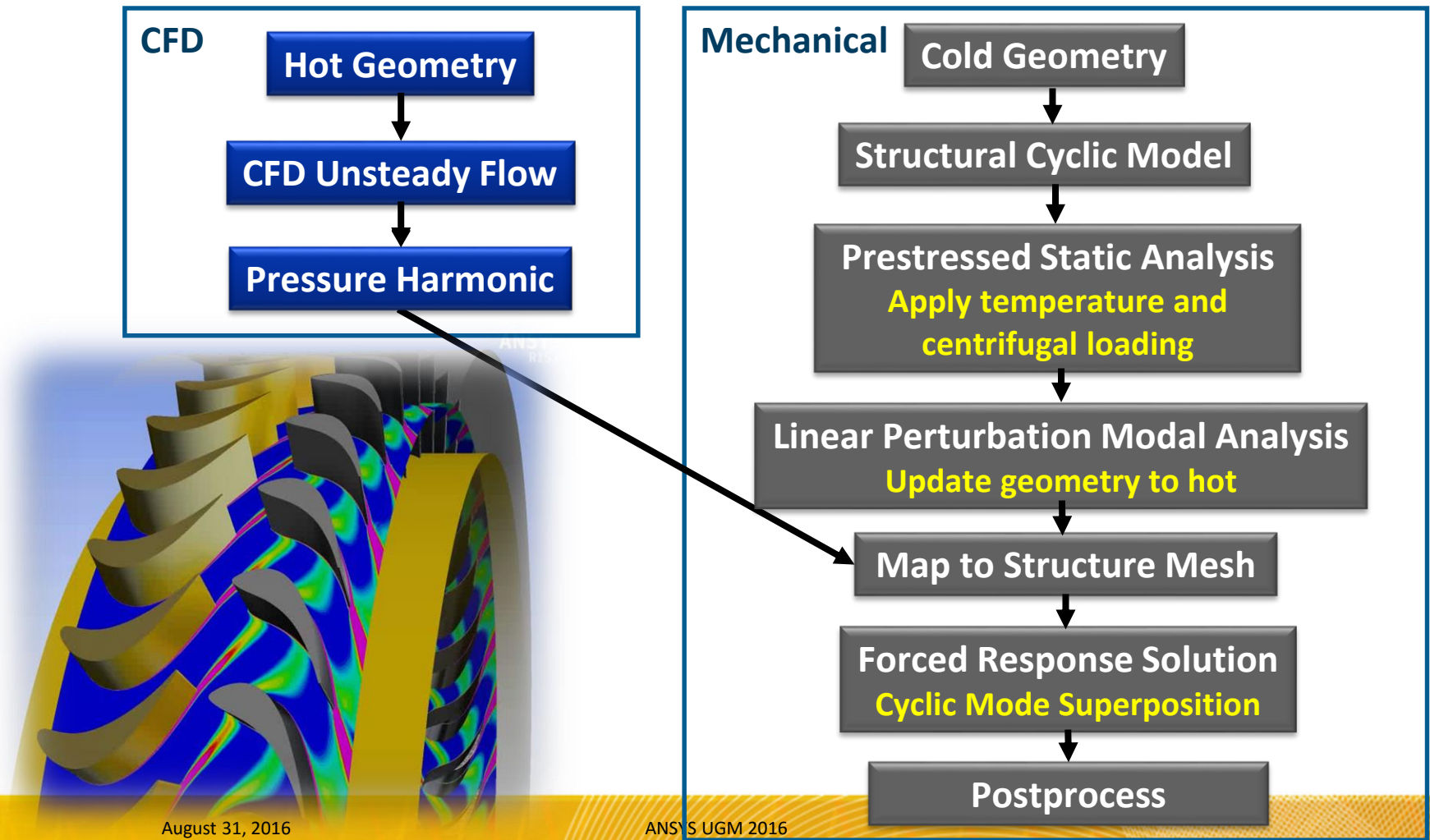
Courtesy Wood Group Heavy Industrial Turbines AG

Cyclic HI's, Nodal Diameters, EO

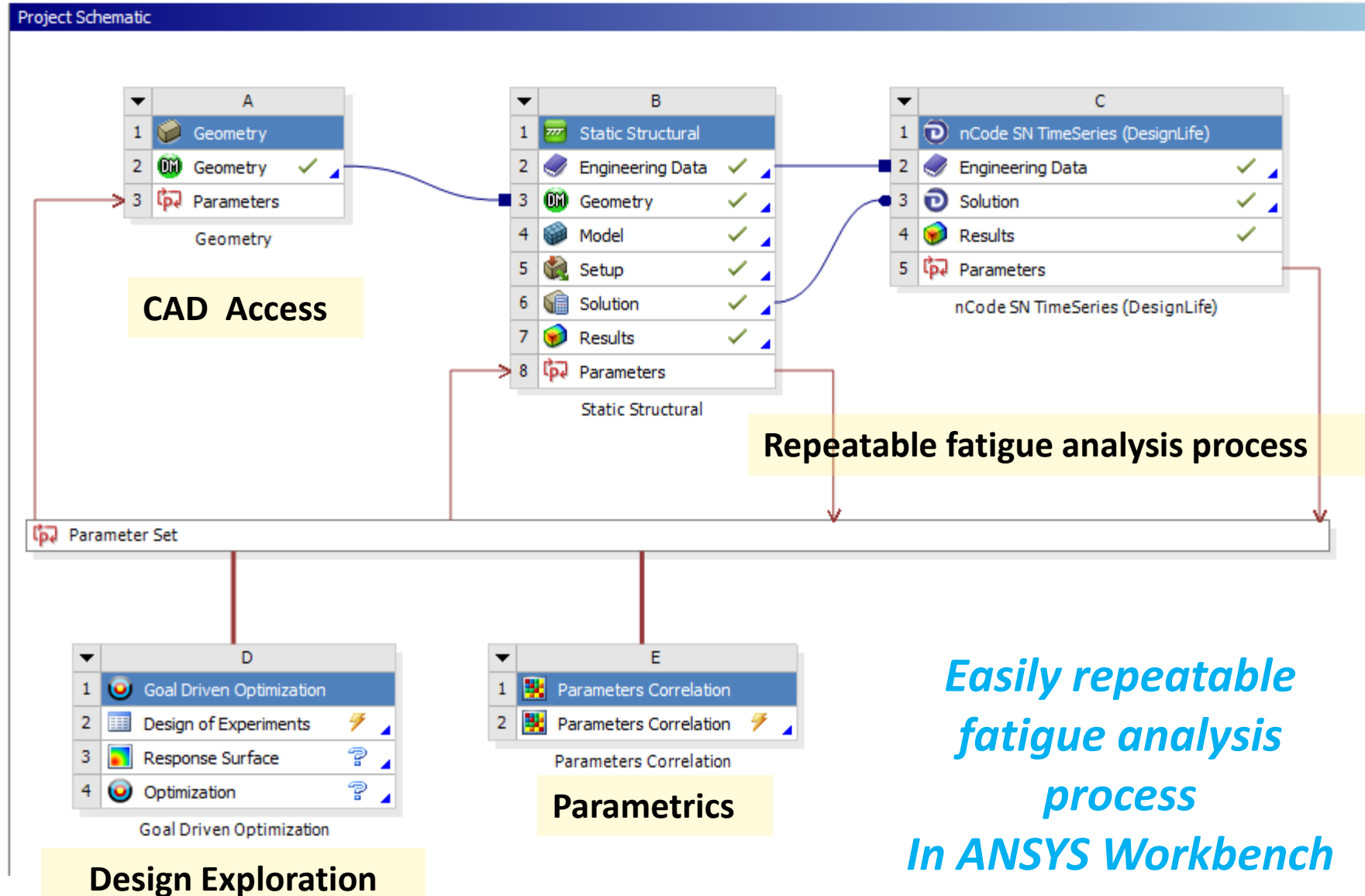


Forced Response

Determine blade response due to excitation from neighboring blade rows to calculate fatigue life. Harmonic pressure loads mapped from CFD results.



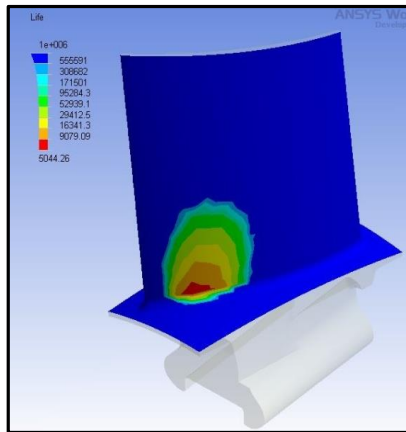
Start to End Automation & Optimization



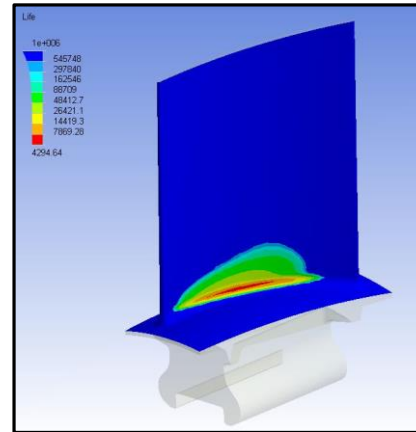
Turbine Blade Life Optimization

Increased Life of Components

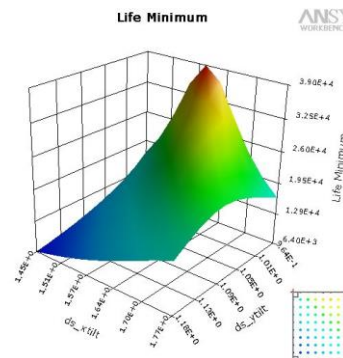
- Goal: optimize the minimum life for a titanium turbine blade based on stress and fatigue analysis



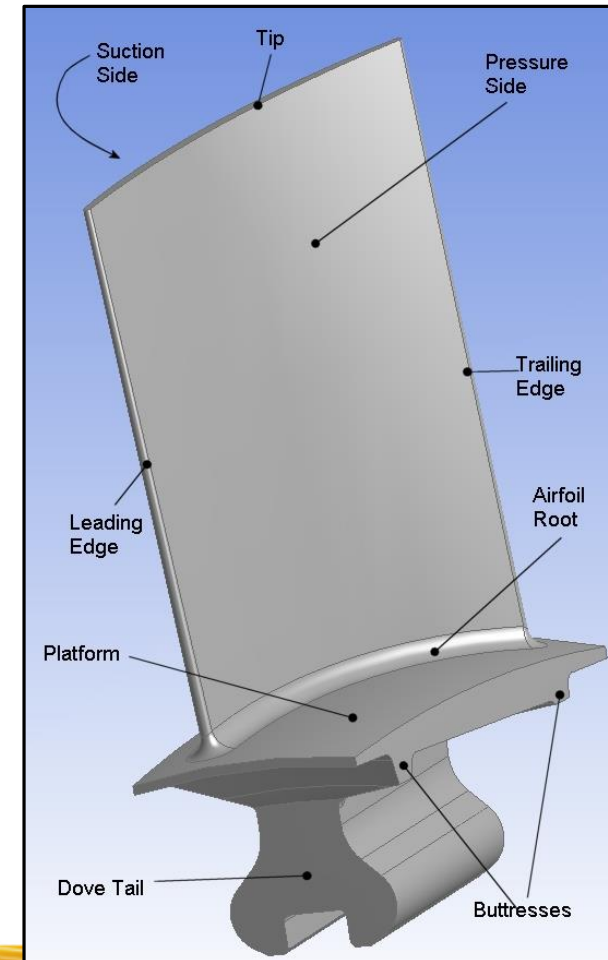
Initial Design



Optimized Design

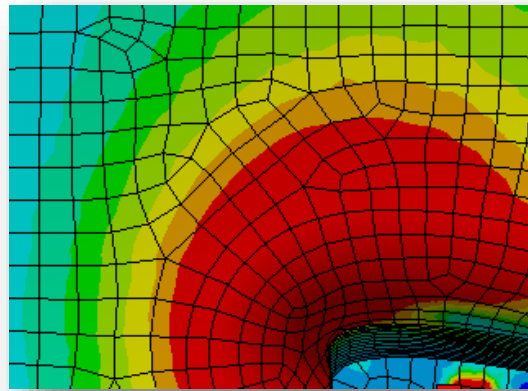


CAE tools are essential for optimizing component life

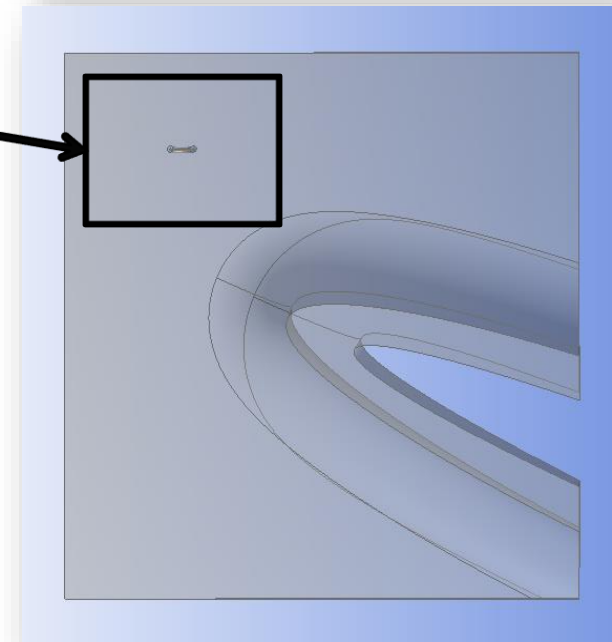
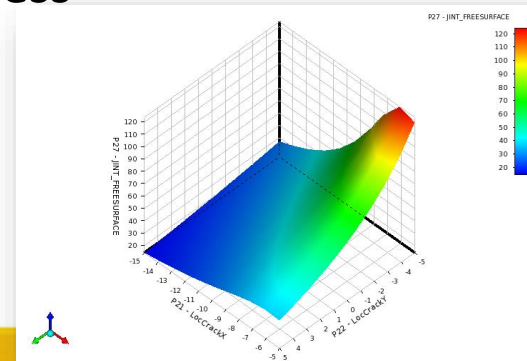
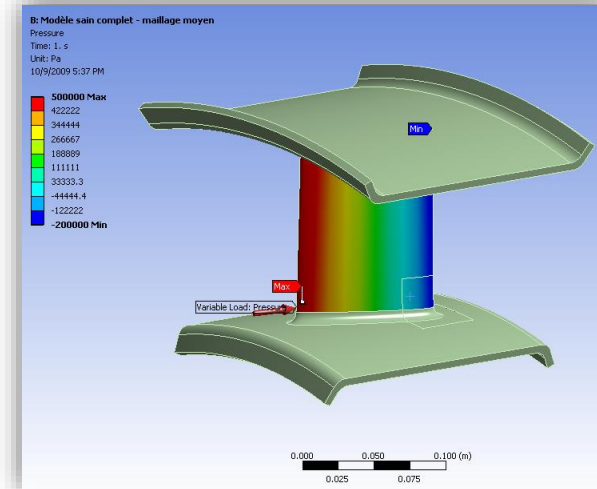


Turbine Crack Propagation Increased Life of Components

- Goal: Identify critical crack locations and model state of stress



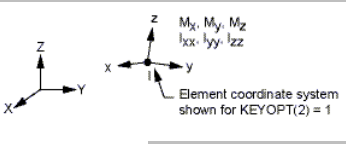
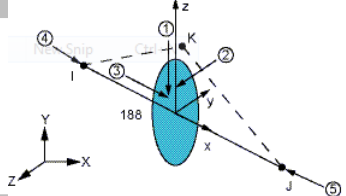
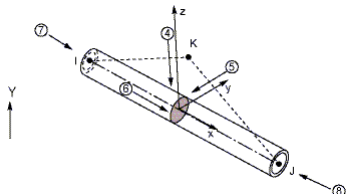
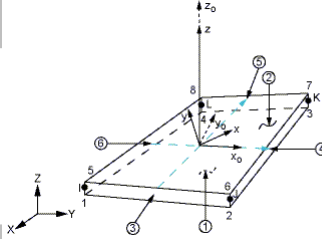
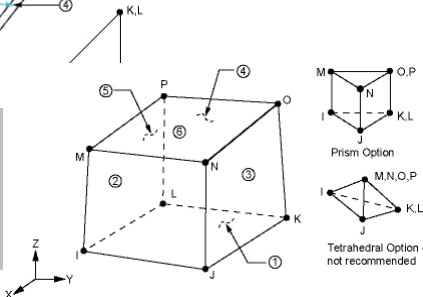
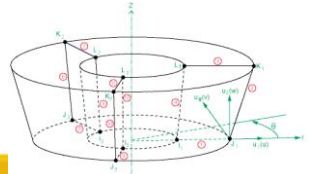
Equivalent Stress

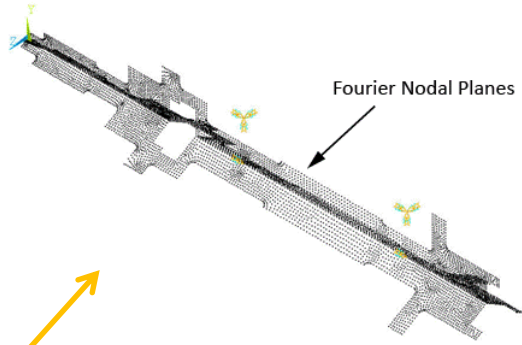


目录

1. 航空发动机与ANSYS
2. ANSYS在叶轮机通流设计中的应用
3. ANSYS在燃烧设计中的应用
4. ANSYS在热端部件热防护设计中的应用
5. ANSYS在二次空气系统设计中的应用
6. ANSYS在零部件强度振动校核中的应用
7. ANSYS在转子动力学校核中的应用
8. ANSYS在鸟撞与包容性设计中的应用
9. ANSYS在发动机其他设计中的应用
10. 多物理场仿真在发动机设计中的应用
11. 总结

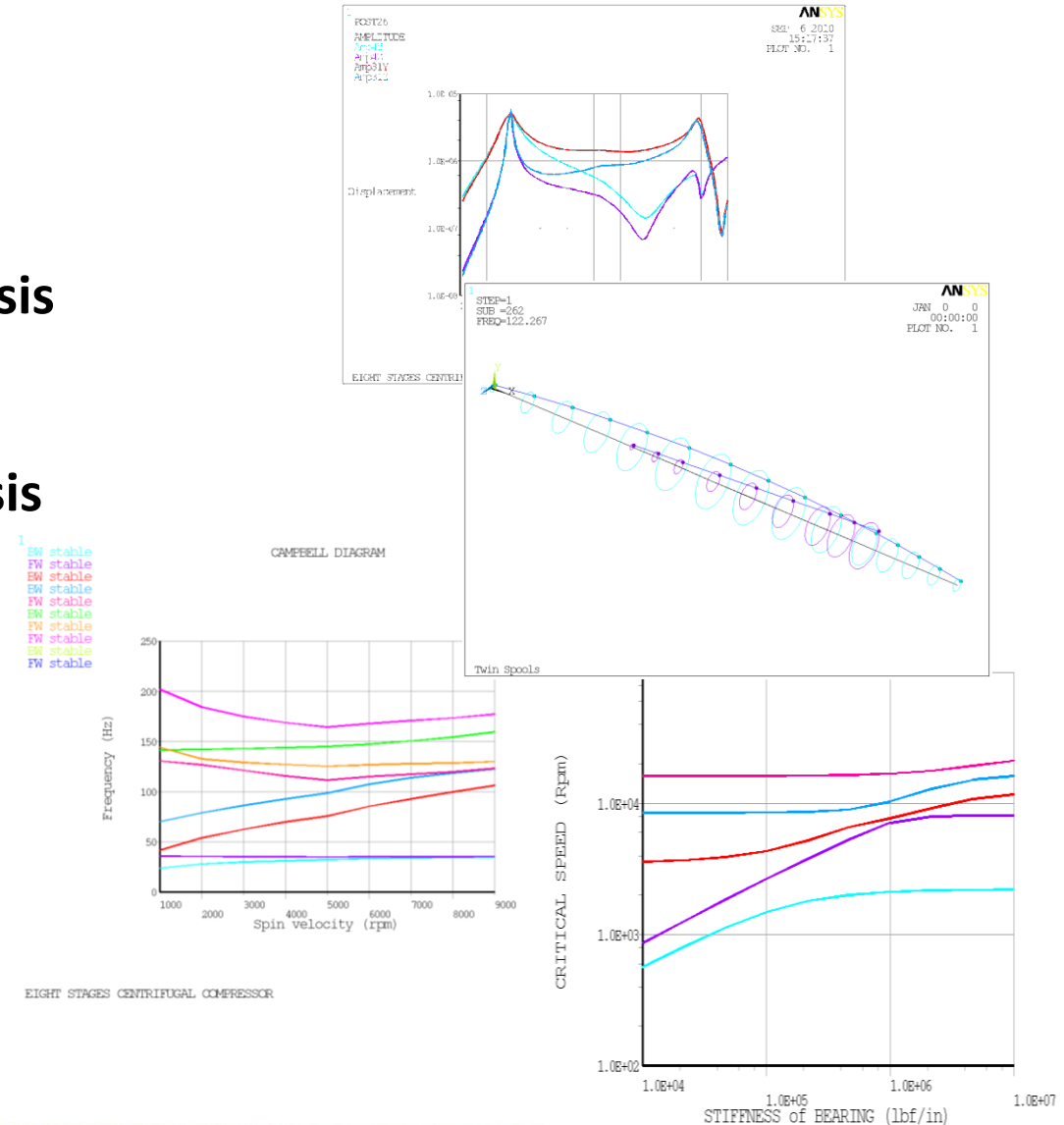
Structural Elements for Rotating Parts

Element Type		Detail
Structural Mass		MASS21
3D Beam		BEAM188 BEAM189
3D Pipe		PIPE288 PIPE289
Structural Shell		SHELL181 SHELL281
3D Structural Solid		SOLID185 SOLID186 SOLID187
General Axisymmetric Solid		SOLID272 SOLID273



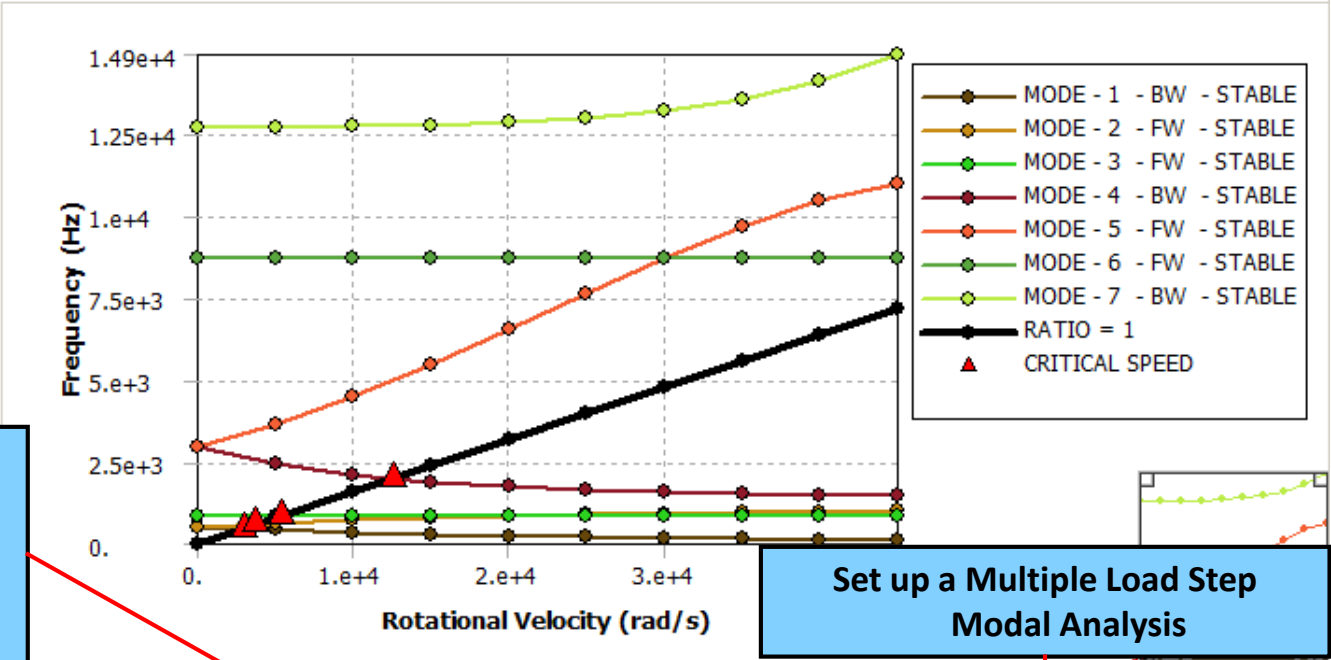
Features pertaining to Rotordynamics

- Static prestress analysis
- Damped and Undamped Modal analysis
- Harmonic (unbalance) response
- Linear and non-linear transient analysis
- Orbit plots
- Campbell diagram
- Whirl animation
- Critical Speed Map
- Multi-spool simulation



Modal Analysis – Campbell Diagram

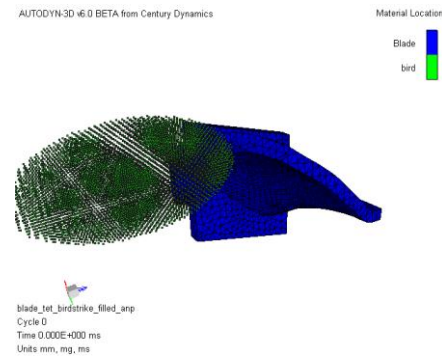
Campbell Diagram



Mode	Whirl Direction	Mode Stability	Critical Speed	0.	5000.	10000
1.	BW	STABLE	2955.5	526.27	431.73	351.36
2.	FW	STABLE	3774.2	526.48	624.79	714.91
3.	FW	STABLE	5381.8	856.54	856.54	856.54
4.	BW	STABLE	12605	2933.6	2447.1	2118.3
5.	FW	STABLE	NONE	2942.4	3620.9	4482.6
6.	FW	STABLE	NONE	8716.7	8716.7	8716.7
7.	BW	STABLE	NONE	12686	12695	12721

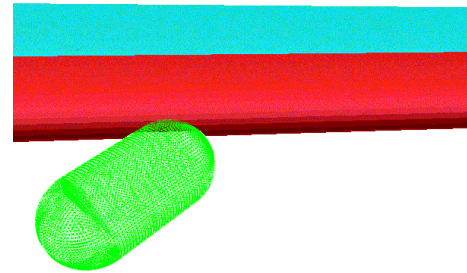
目录

1. 航空发动机与ANSYS
2. ANSYS在叶轮机通流设计中的应用
3. ANSYS在燃烧设计中的应用
4. ANSYS在热端部件热防护设计中的应用
5. ANSYS在二次空气系统设计中的应用
6. ANSYS在零部件强度振动校核中的应用
7. ANSYS在转子动力学校核中的应用
8. ANSYS在鸟撞与包容性设计中的应用
9. ANSYS在发动机其他设计中的应用
10. 多物理场仿真在发动机设计中的应用
11. 总结

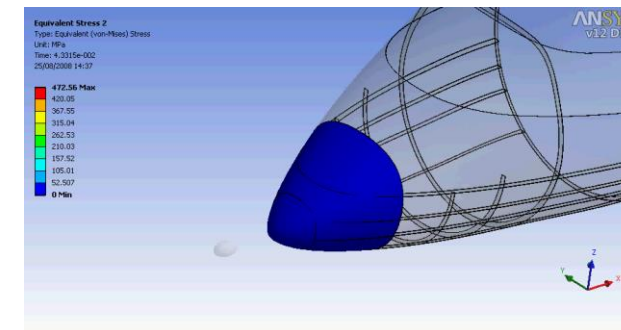


Bird strike on a turbine blade section

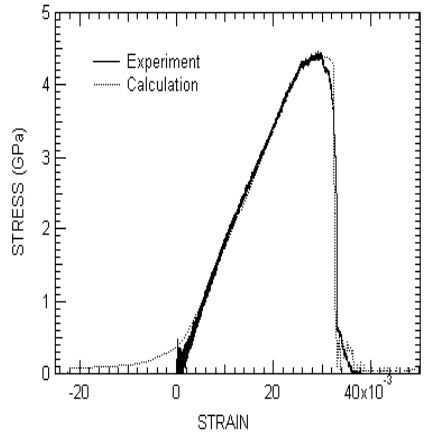
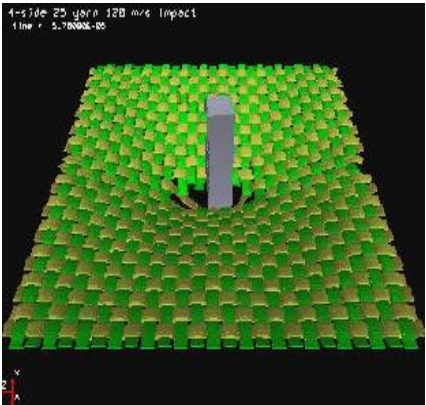
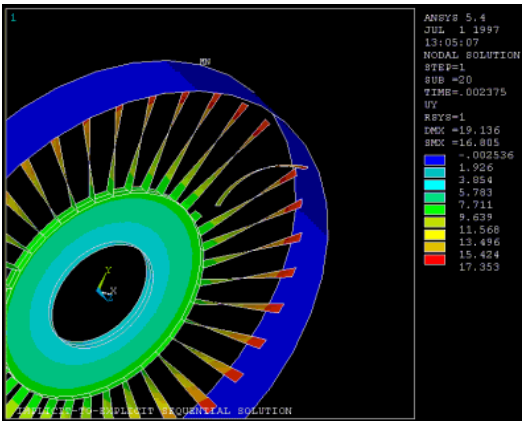
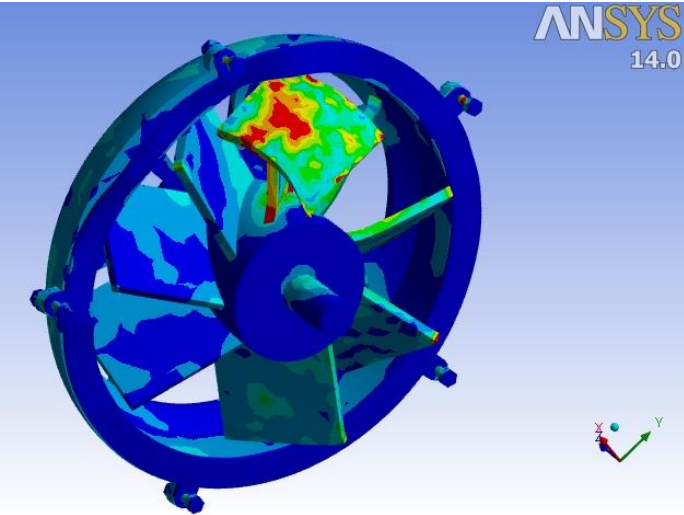
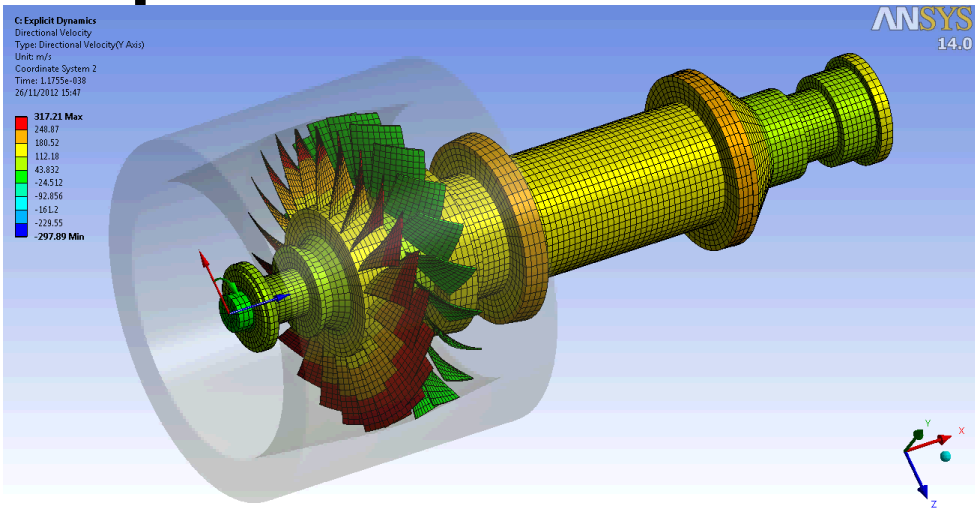
Bird strike on a composite wing



Bird strike on a radome



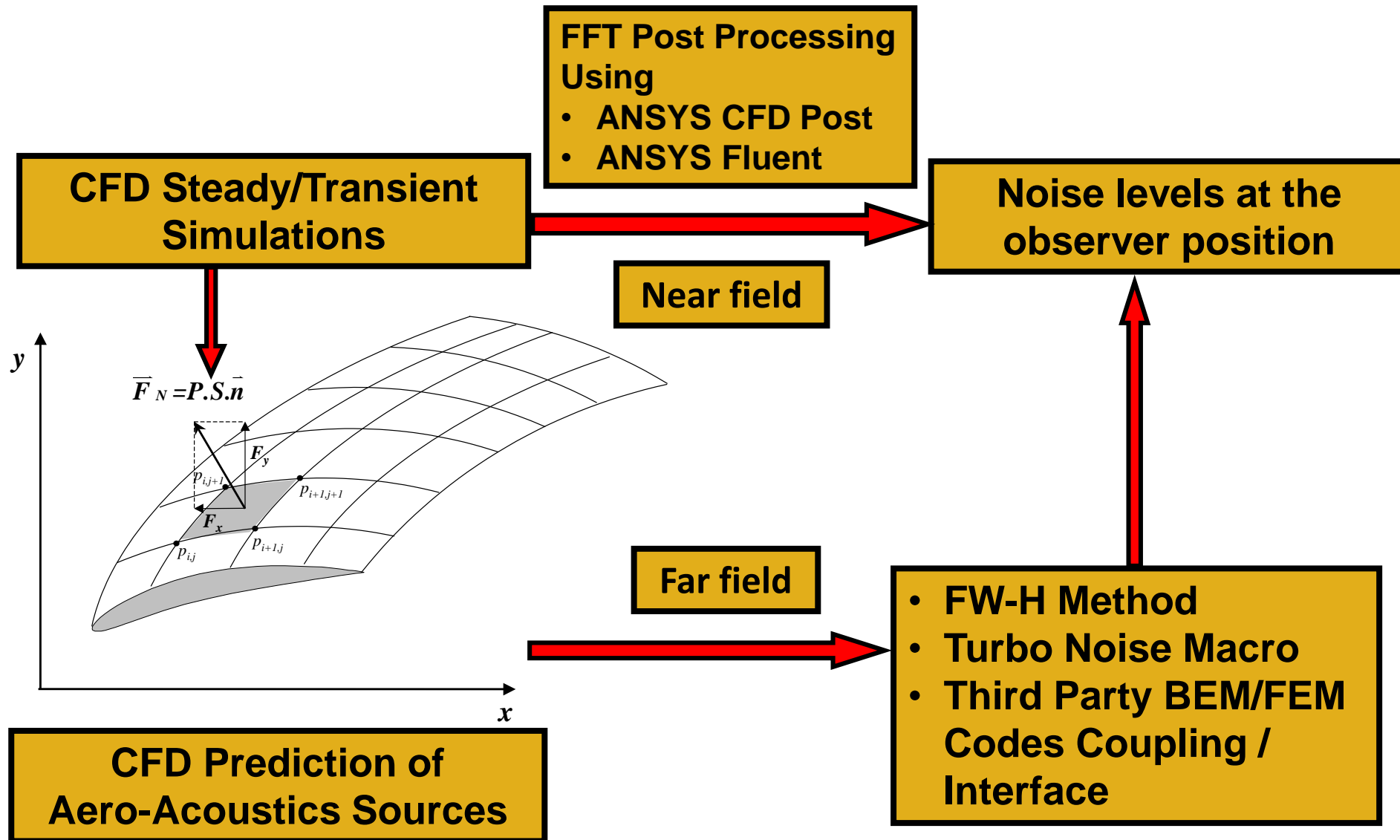
ANSYS Explicit - Blade out



目录

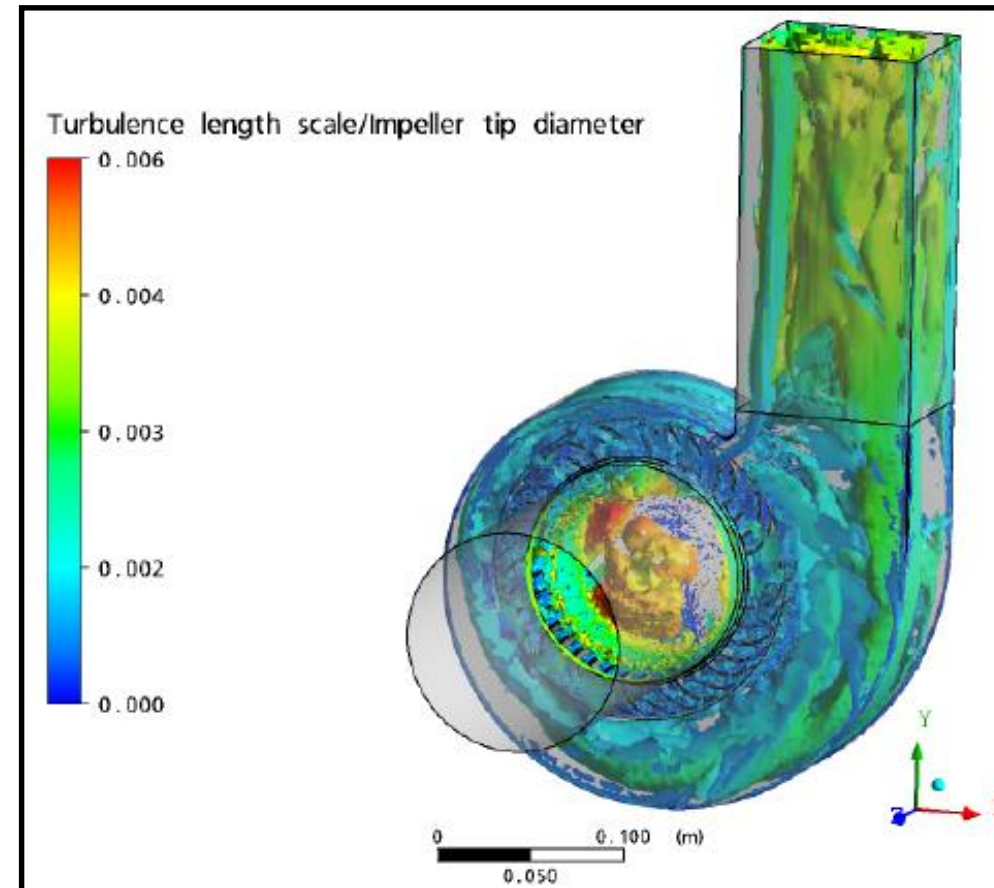
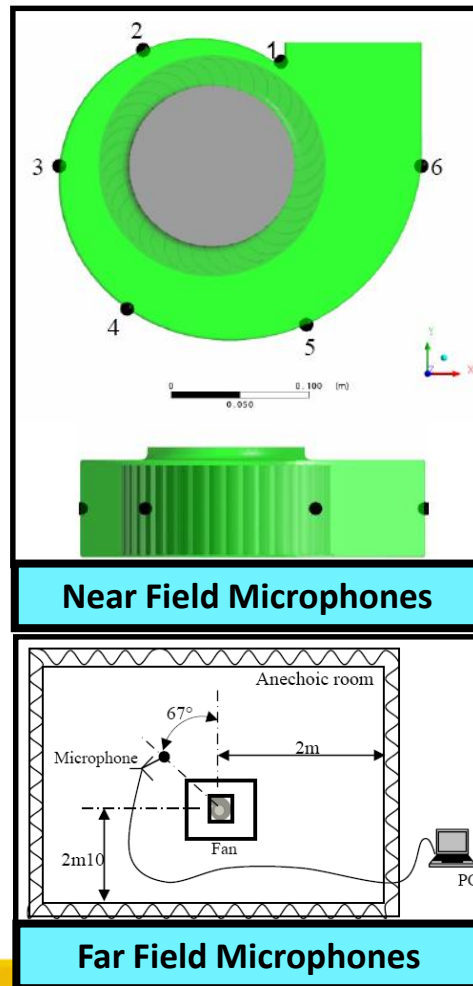
1. 航空发动机与ANSYS
2. ANSYS在系统设计中的应用
3. ANSYS在叶轮机通流设计中的应用
4. ANSYS在燃烧设计中的应用
5. ANSYS在热端部件热防护设计中的应用
6. ANSYS在二次空气系统设计中的应用
7. ANSYS在零部件强度振动校核中的应用
8. ANSYS在转子动力学校核中的应用
9. ANSYS在鸟撞与包容性设计中的应用
10. ANSYS在发动机其他设计中的应用
11. 多物理场仿真在发动机设计中的应用
12. 总结

ANSYS CFD : Aeroacoustics Simulation Capabilities



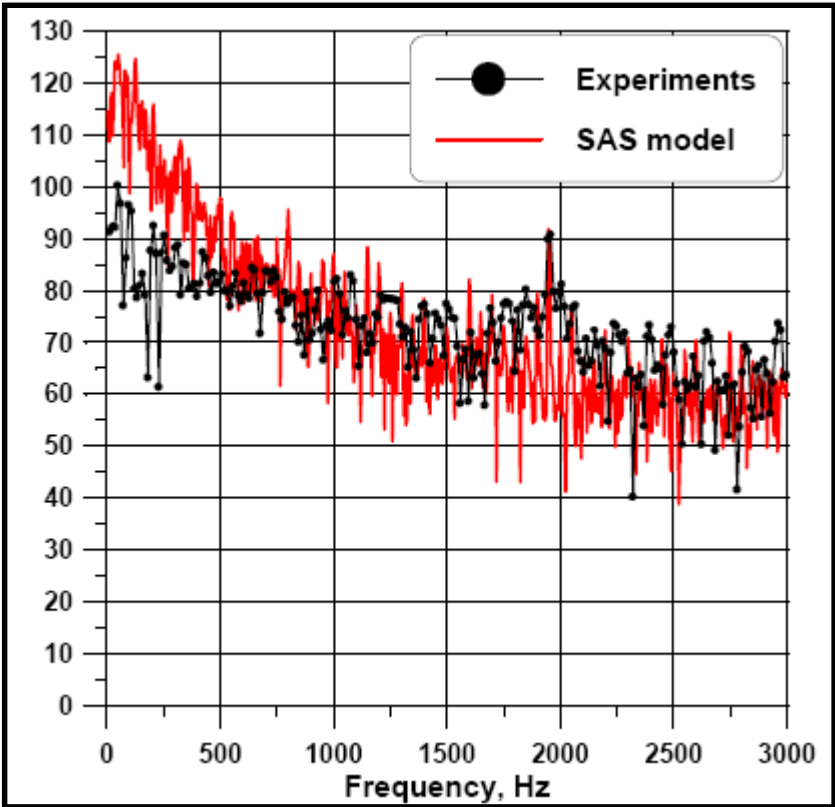
Aeroacoustics Modeling of a Centrifugal Fan Using CFX : CAA

- Aeroacoustics Modeling of a Centrifugal Fan Using ANSYS CFX

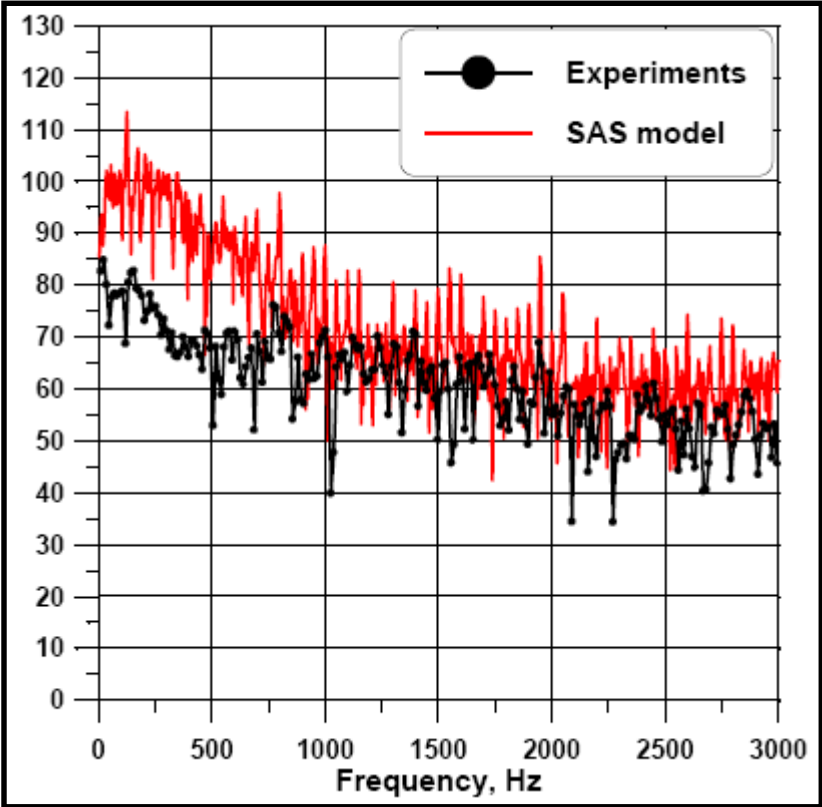


Aeroacoustics Modeling of a Centrifugal Fan Using CFX : CAA

- Aeroacoustics Modeling of a Centrifugal Fan Using ANSYS CFX



Microphone #1



Microphone #4

Near Field Noise Prediction

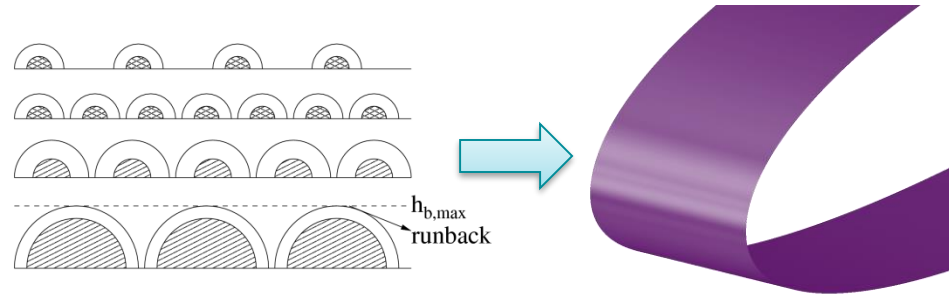
And has been used to re-certify aircraft with major regulatory bodies without the need for icing tunnel studies



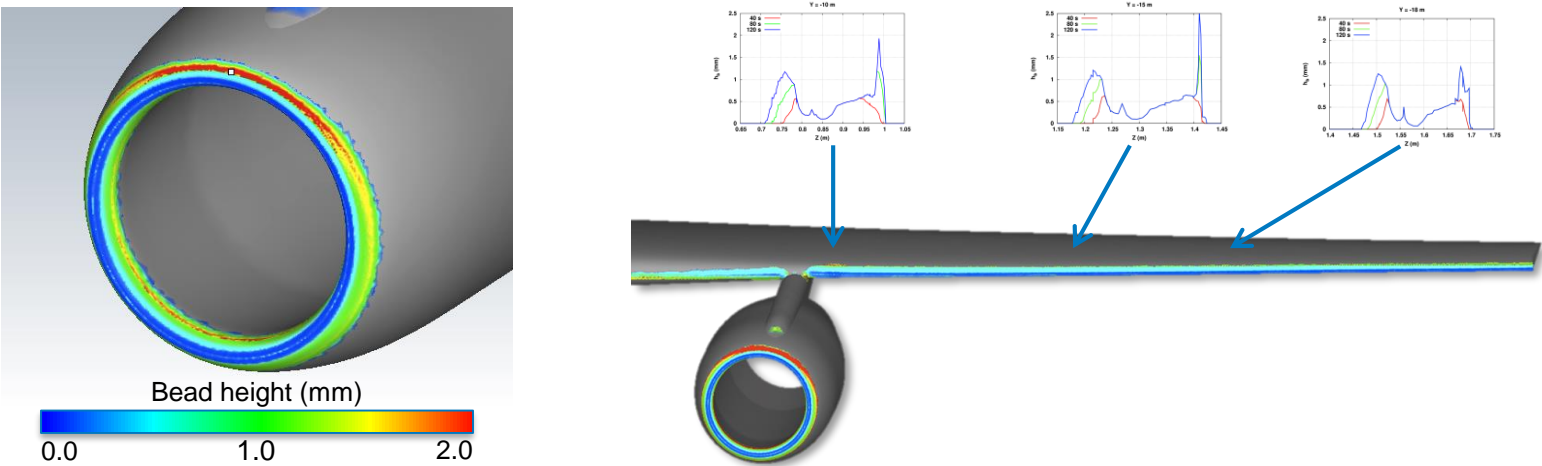
Key technical feature advantages include:

Ice roughness prediction without heuristics using a proprietary beading model

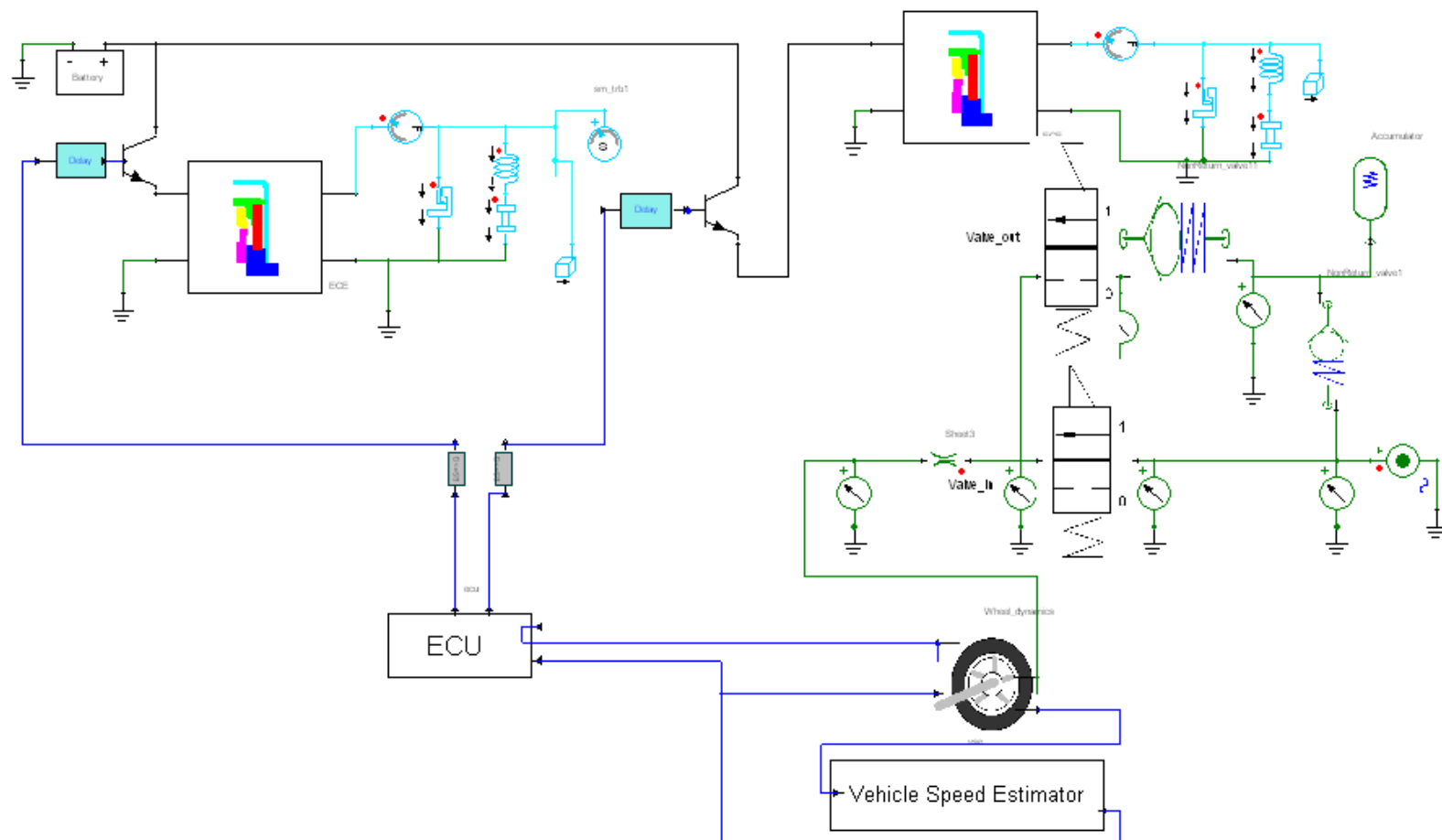
- Impinging droplets form small beads on the surface.
- Beads start to coalesce once maximum surface coverage is reached.
- Bead growth stops when the average bead height hits the local limit.
- The excess liquid part of the beads runback.



Predicts 3D spatial and temporal variation of ice roughness on nacelle and wing



液压系统分析



液压元件仿真

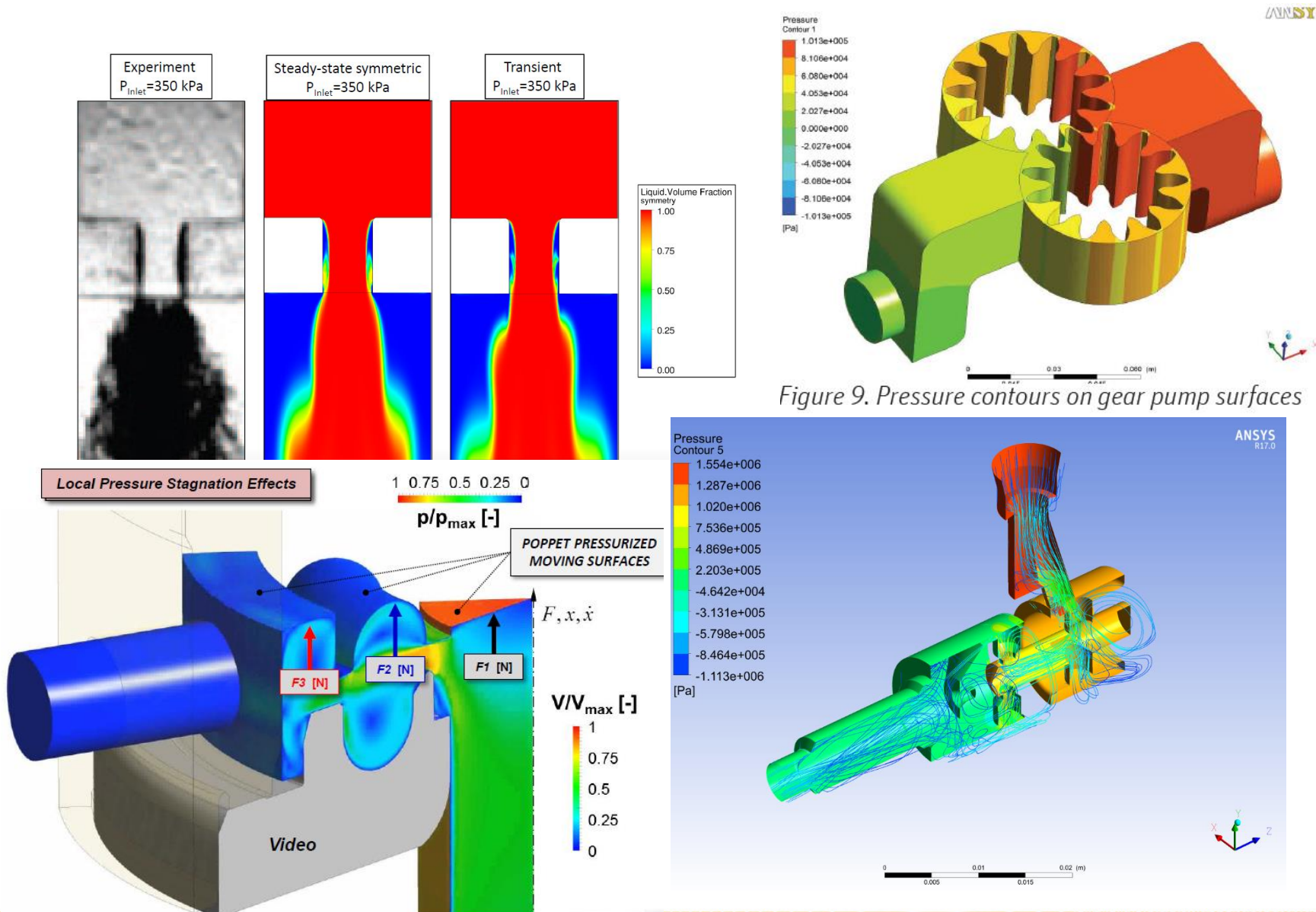
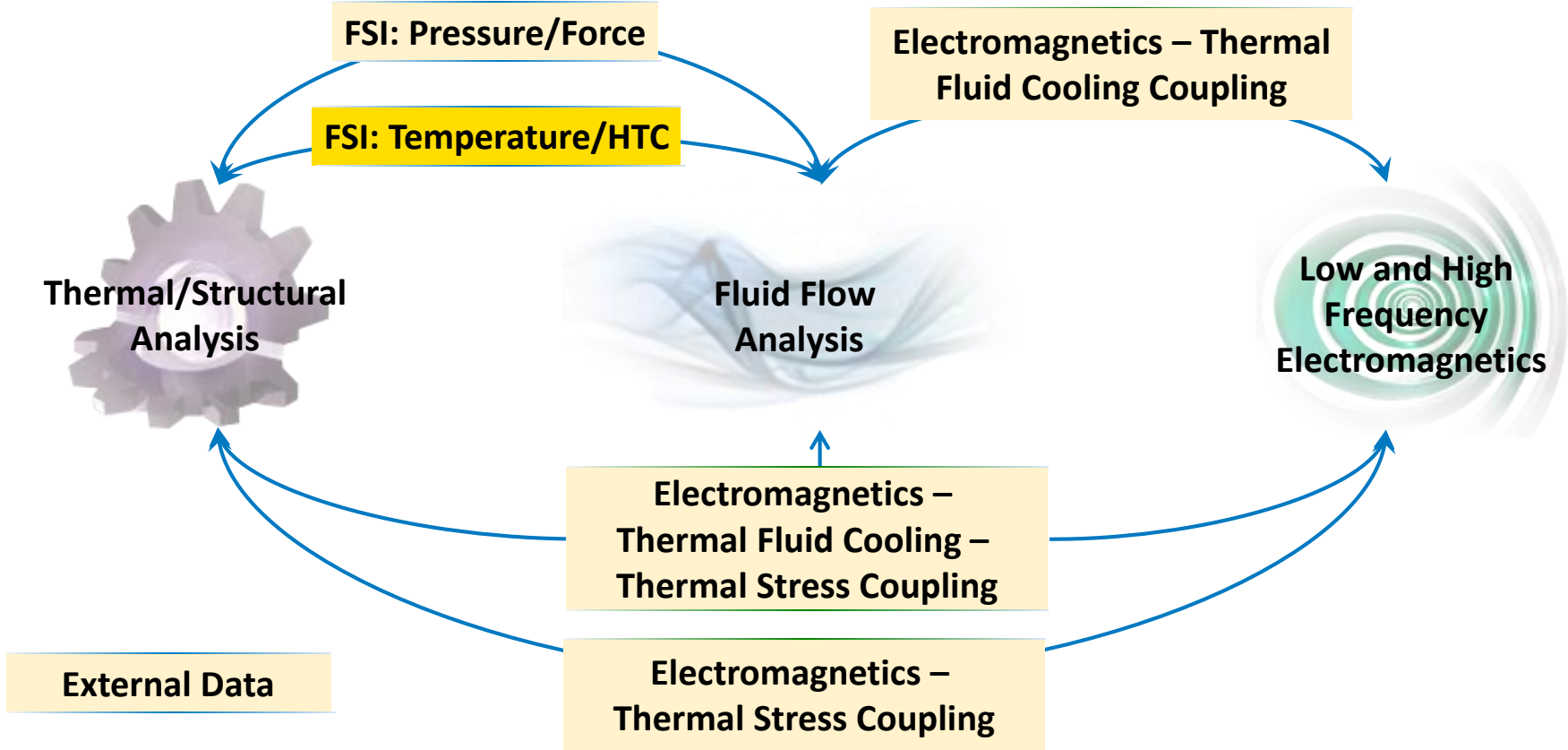


Figure 9. Pressure contours on gear pump surfaces

目录

1. 航空发动机与ANSYS
2. ANSYS在系统设计中的应用
3. ANSYS在叶轮机通流设计中的应用
4. ANSYS在燃烧设计中的应用
5. ANSYS在热端部件热防护设计中的应用
6. ANSYS在二次空气系统设计中的应用
7. ANSYS在零部件强度振动校核中的应用
8. ANSYS在转子动力学校核中的应用
9. ANSYS在鸟撞与包容性设计中的应用
10. ANSYS加工仿真
11. ANSYS在发动机其他设计中的应用
12. 多物理场仿真在发动机设计中的应用
13. 总结

ANSYS Multiphysics ‘At A Glance’

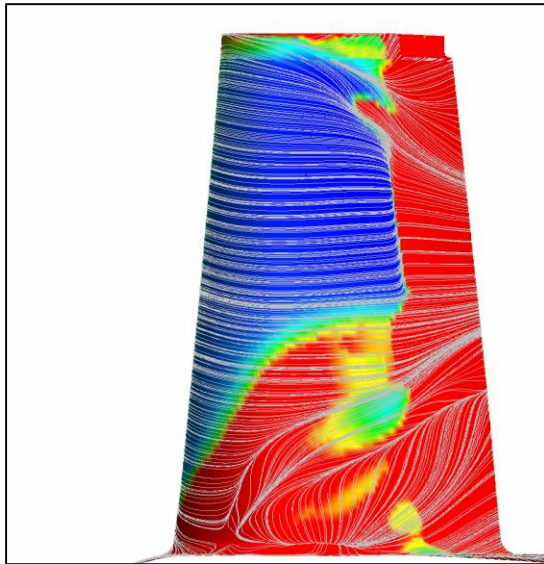


Axial Turbine Blade FSI Modelling

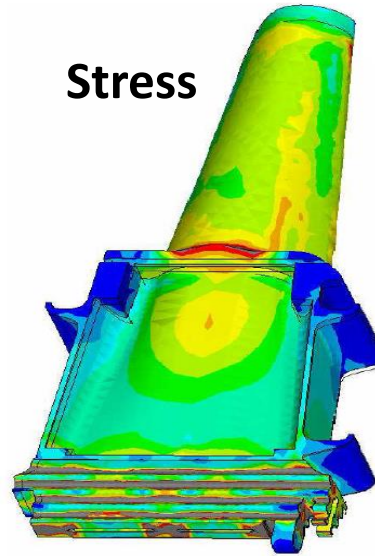
Increased Life of Components

- **Goal:**
 - Compute thermal, mechanical and life properties due to thermal loading from the hot gas path

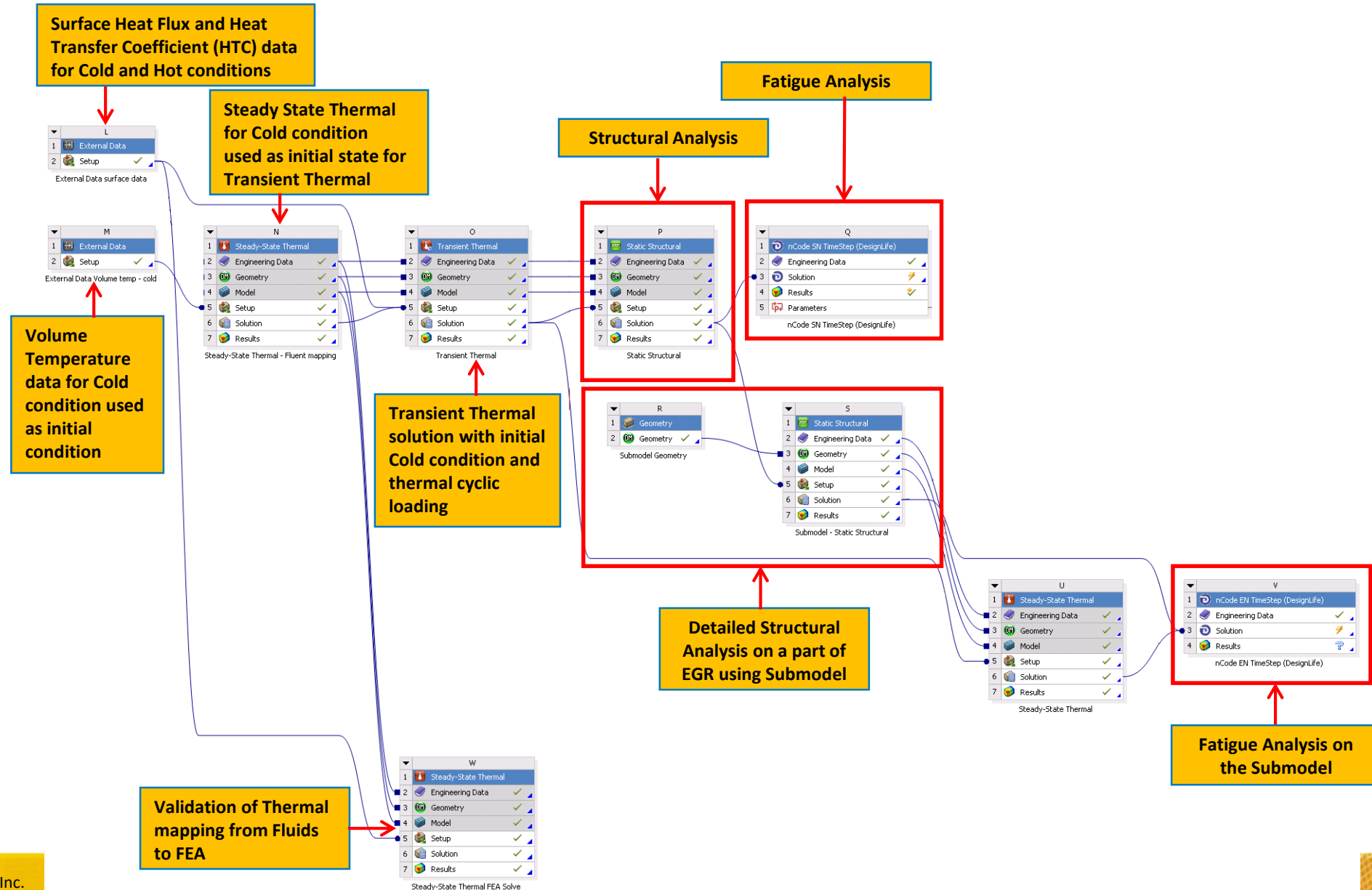
Transitional Turbulence CFD Simulation



Stress

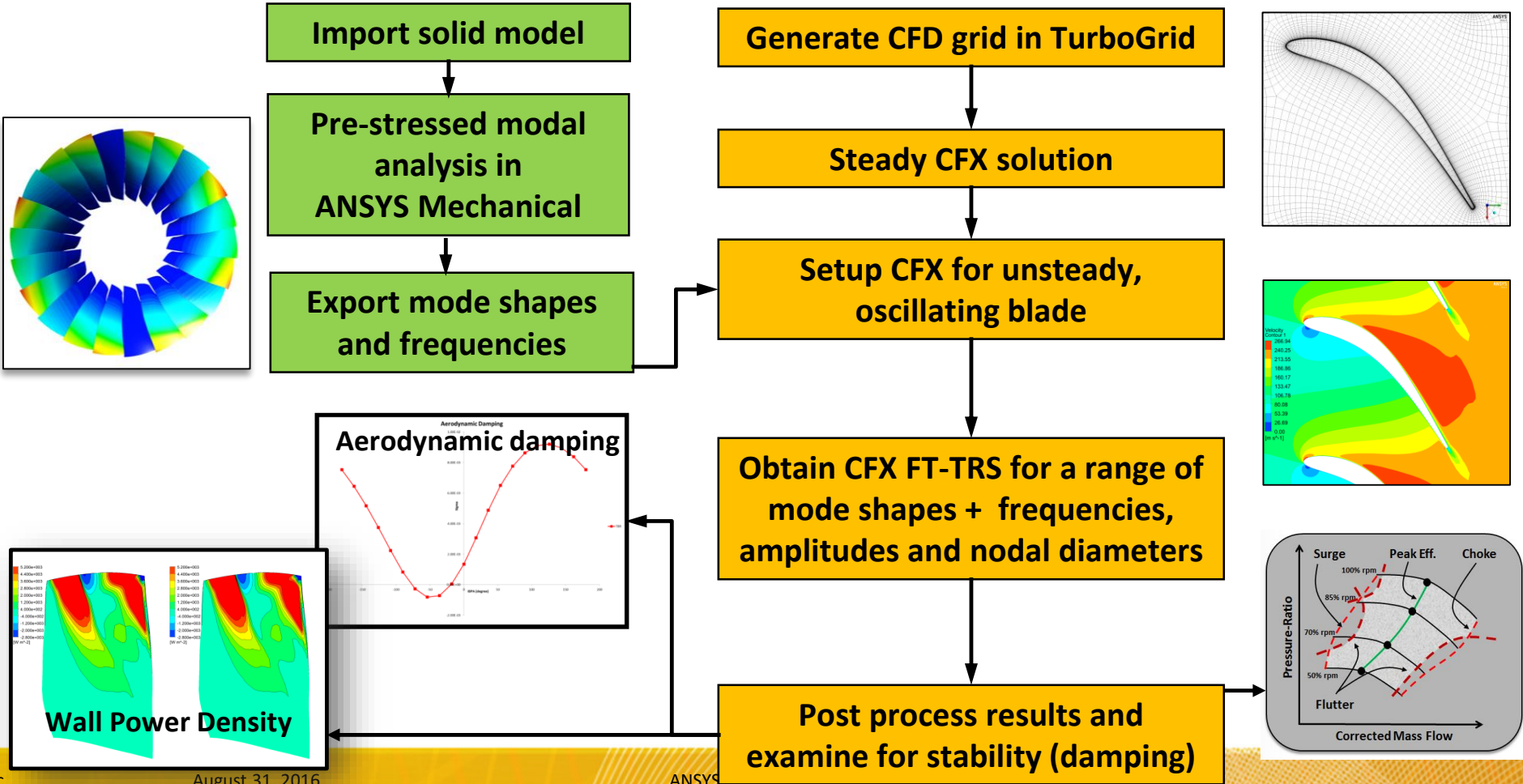


ANSYS Workbench Project Schematic

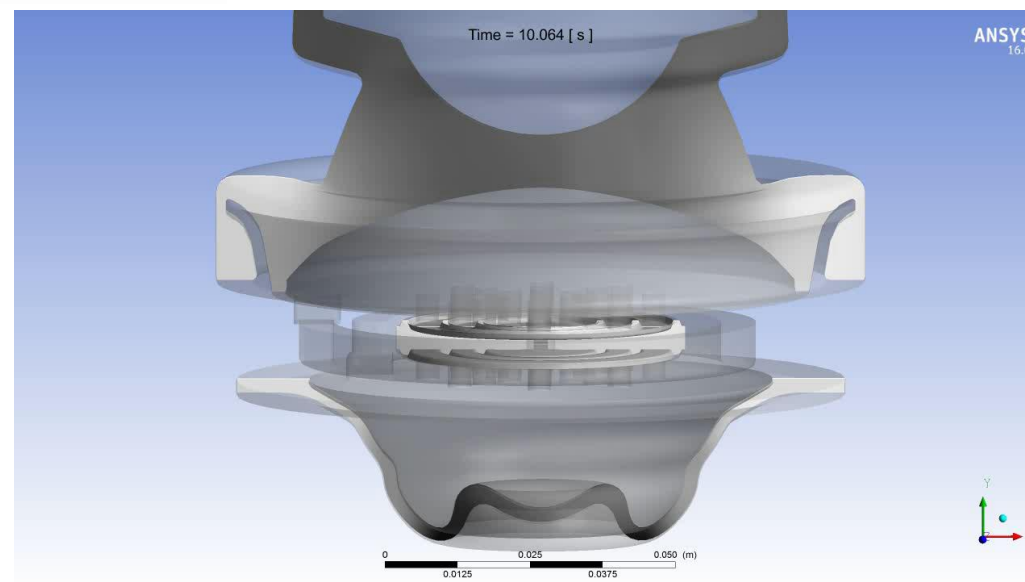
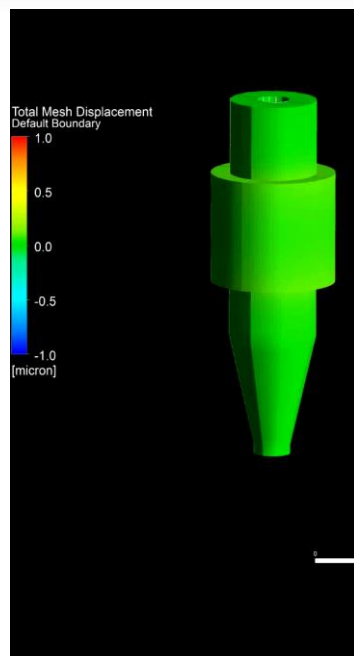
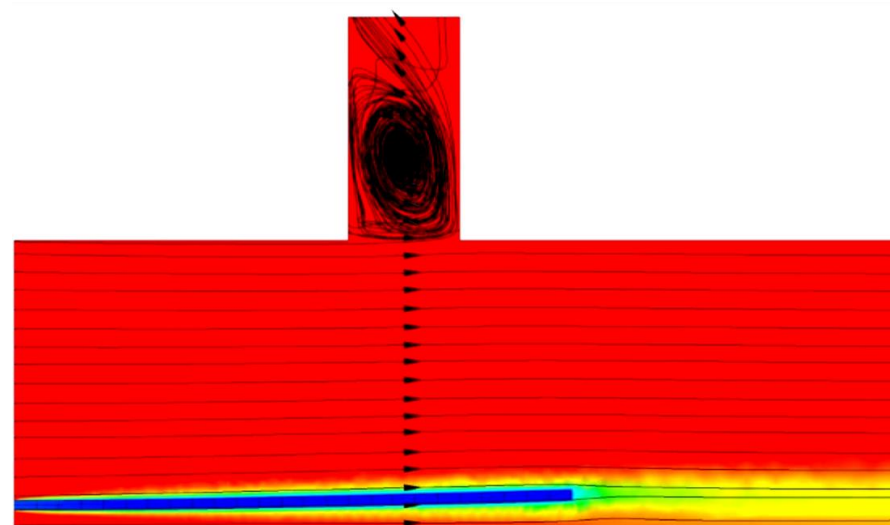
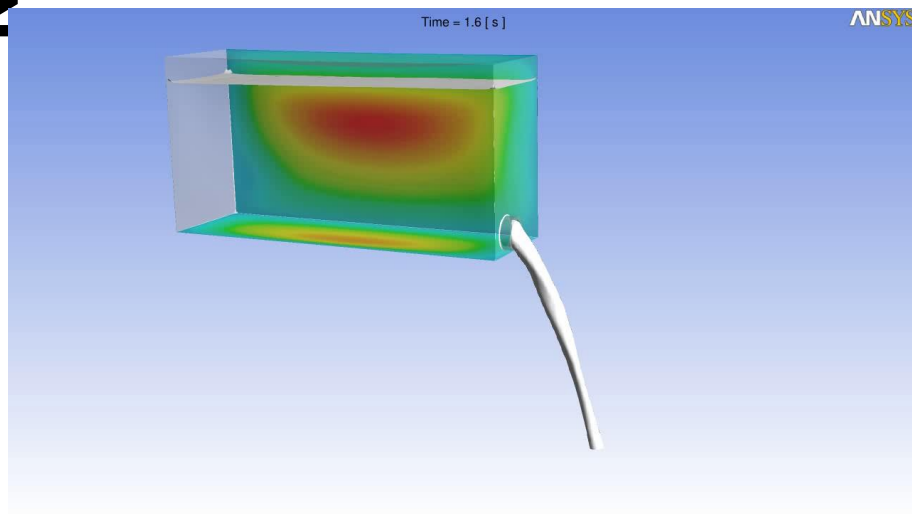


Blade Flutter

Determine if the blade could enter self-sustained harmful vibration (flutter), due to cyclic loading when its undergoing vibration at natural frequency



例子



总结

1. 航空发动机与ANSYS
2. ANSYS在叶轮机通流设计中的应用
3. ANSYS在燃烧设计中的应用
4. ANSYS在热端部件热防护设计中的应用
5. ANSYS在二次空气系统设计中的应用
6. ANSYS在零部件强度振动校核中的应用
7. ANSYS在转子动力学校核中的应用
8. ANSYS在鸟撞与包容性设计中的应用
9. ANSYS在发动机其他设计中的应用
10. 多物理场仿真在发动机设计中的应用

ANSYS®



ANSYS
CONVERGENCE
CONFERENCES

2016

ANSYS中国技术大会

中国·上海

感谢聆听



ANSYS-China