



2019

Kathrin Wendl – NUMECA Ingenieurbüro

**Advanced Grid Generation for Impeller and Volute by Coupling
NUMECA's Meshing Tools with CAESES**



1

Grid Generation in the Optimisation Workflow

2

Grid Generation for CFD Simulation

3

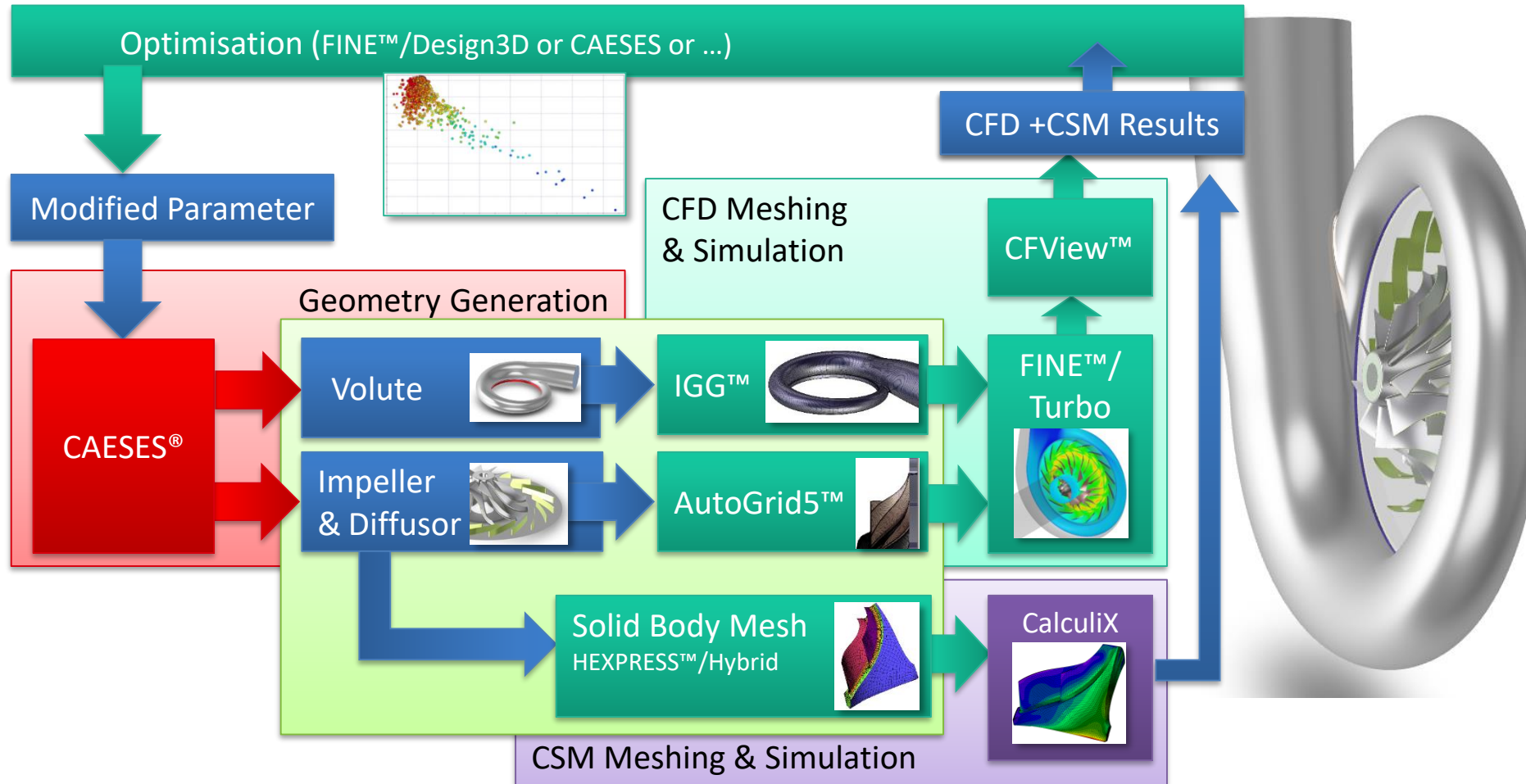
Grid Generation for CSM Simulation

4

Summary

Grid Generation in the Optimisation Workflow

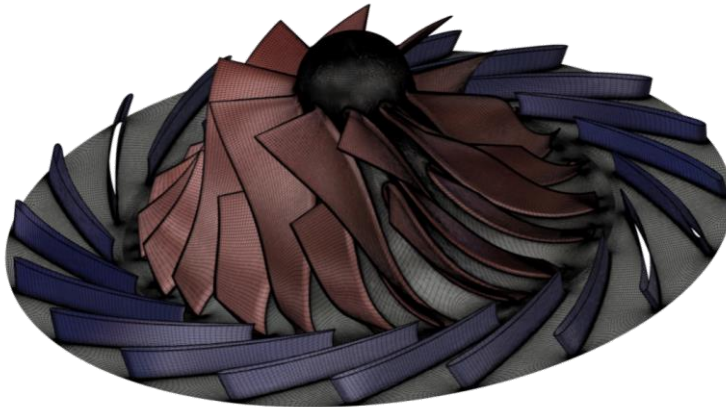
Optimisation Workflow for Compressor with Volute



Meshing requirements for optimisation:

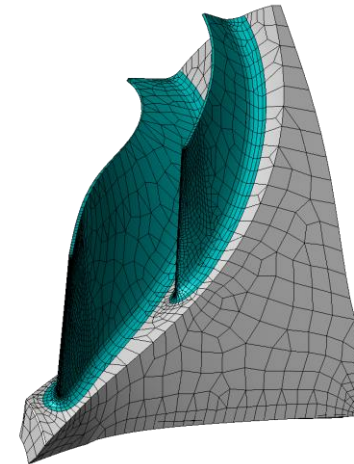
- Robust geometry generation for the whole design space
- High mesh quality for reliable simulation
- Automatized coupling between geometry and grid generation
- CFD Simulation: block-structured hexahedral mesh
- CSM Simulation: pure tetrahedral mesh

When good aerodynamic designs fail regarding geometry or grid generation issues,
the optimisation could be directed to a wrong direction!



Structured Hexahedral Meshing
Tool: Autgrid5™ & IGG™

- Block-structured grid generator
- All turbomachinery configurations
- Advanced smoothing algorithms for highest quality
- Full batch and scripting capability

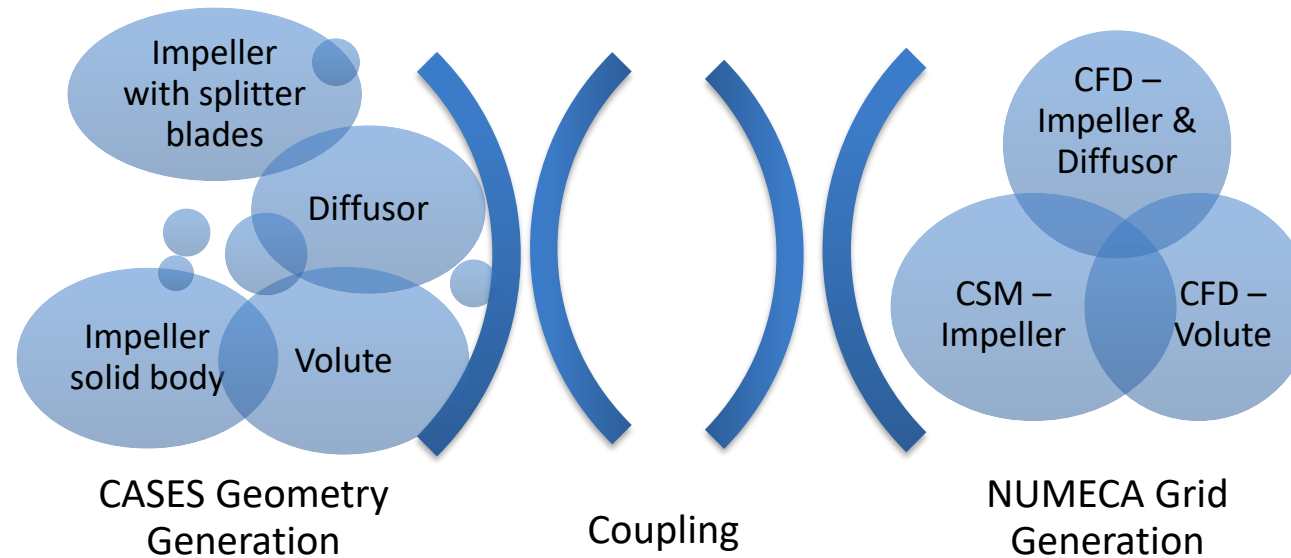


Unstructured Hexahedral-dominant Meshing
Tool: HEXPRESS™/Hybrid

- Hex-dominant unstructured grid generator
- Highly complex geometries
- Volume-to-Surface or Surface-to-Volume approach
- Mesh generation based on a text file

Meshing Tasks:

- CFD meshing impeller with splitter blade and vaned diffusor
- CFD meshing volute
- CSM meshing solid body impeller



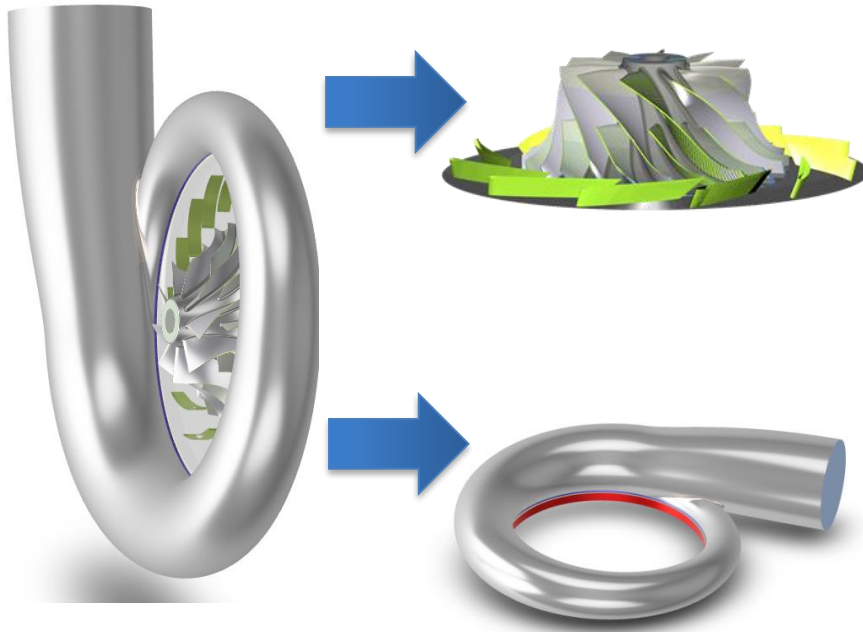


2

Grid Generation for CFD Simulation

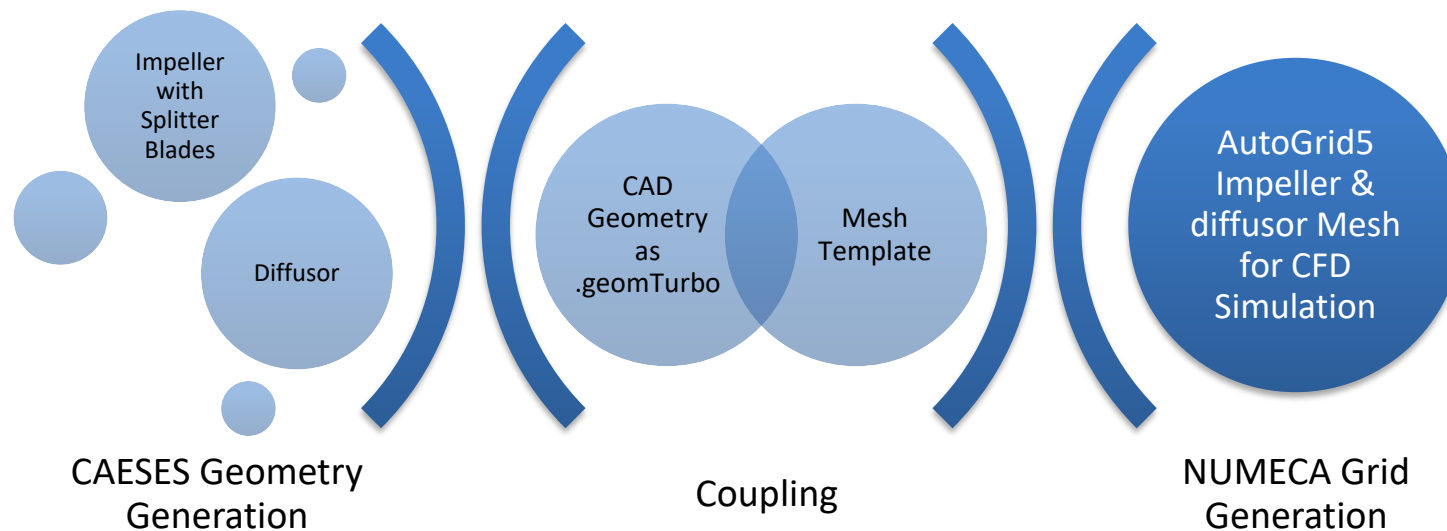


Block structured hexahedral multi-grid meshes:



- Impeller and diffuser
 - Multi-row mesh
 - One passage of the impeller
 - 360° mesh of the diffuser
- Volute
 - Complete meshed

- Coupling CAESES and AutoGrid5™
 - Geometry as .geomTurbo file
 - Mesh template
- Preparation for the optimisation
 - Parametric modelling of impeller and diffusor in CAESES
 - Manual meshing of first parameterised design
 - Grid convergence study
 - Robustness tests for geometry and mesh generation



CFD Meshing Impeller

CFD Meshing Volute

CAESES® - impeller & vaned diffusor

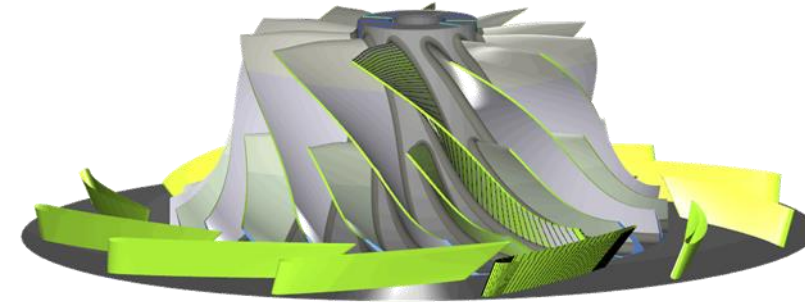
Parametric geometry generation

Export as .geomTurbo

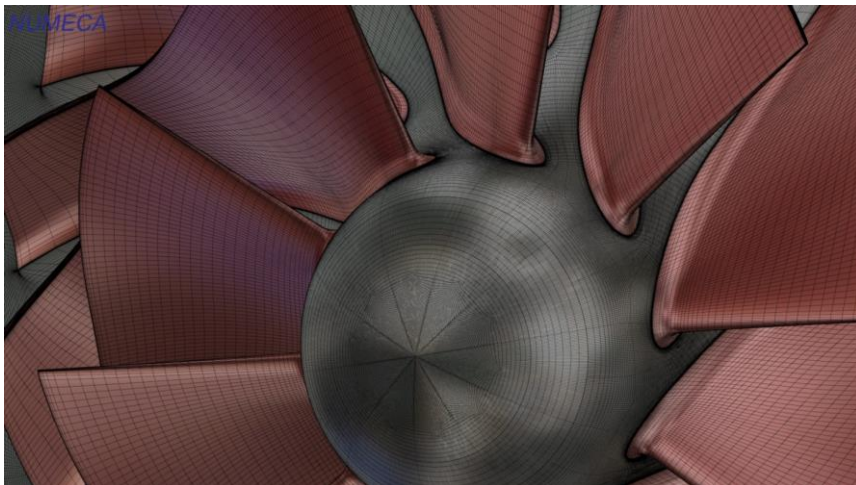
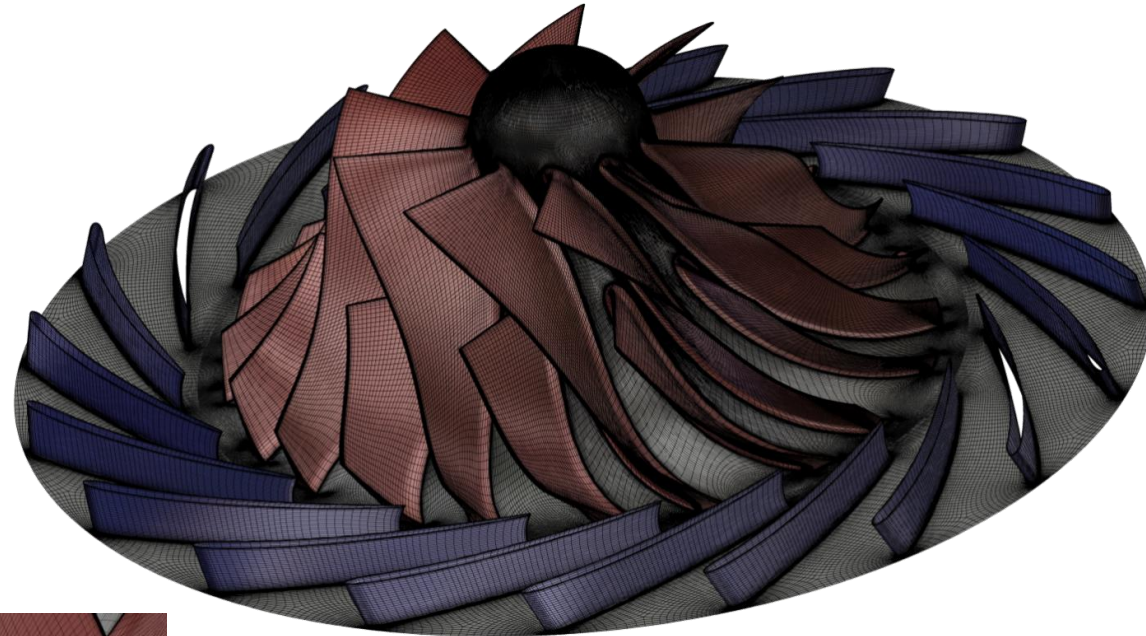
AutoGrid5™ - block structured meshing

Mesh generation in batch mode

Mesh export



- Impeller with splitter blade
- Multistage
- Bulb
- Fillets
- Blunt trailing edge

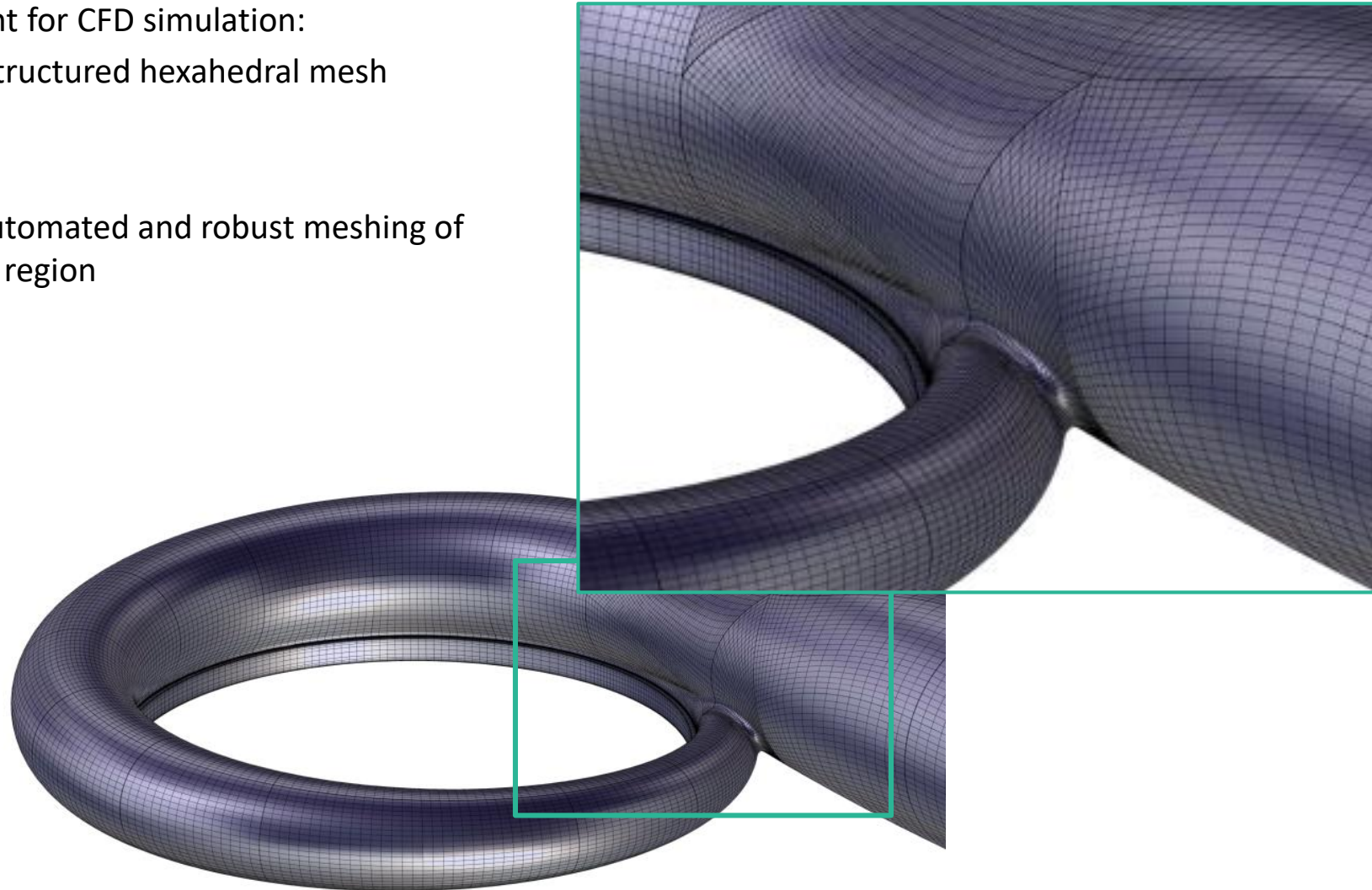


Requirement for CFD simulation:

- Block structured hexahedral mesh

Challenge:

- Fully automated and robust meshing of tongue region



Requirement for CFD simulation:

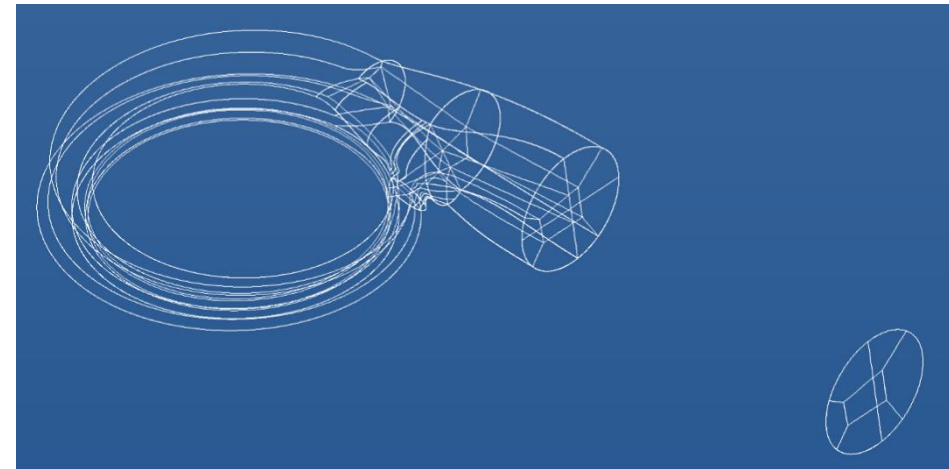
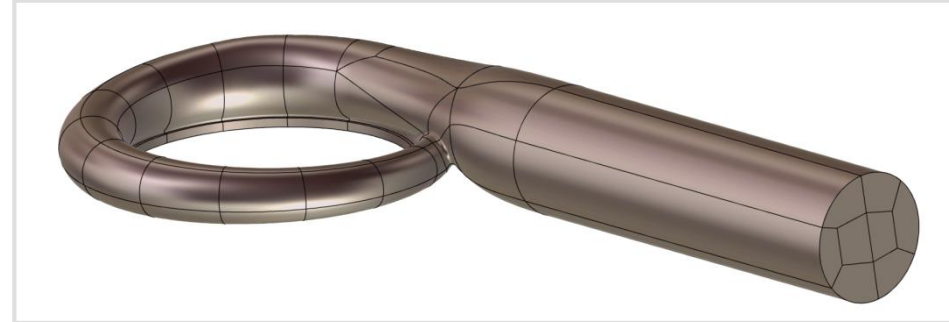
- Block structured hexahedral mesh

Challenge:

- Fully automated and robust meshing of tongue region

Solution:

- Block structure
 - Two ring section
 - Transition block
- Additional curves for blocking
 - Parametrisation in CAESES
 - Export as Parasolid geometry
- Coupling CAESES and IGG™
 - Geometry and additional curves as Parasolid
 - Script base multi-block meshing



CFD Meshing Impeller

CFD Meshing Volute

CAESES® - volute & support geometry
for blocking

Parametric geometry generation

Export as Parasolid

IGG™ - script based meshing

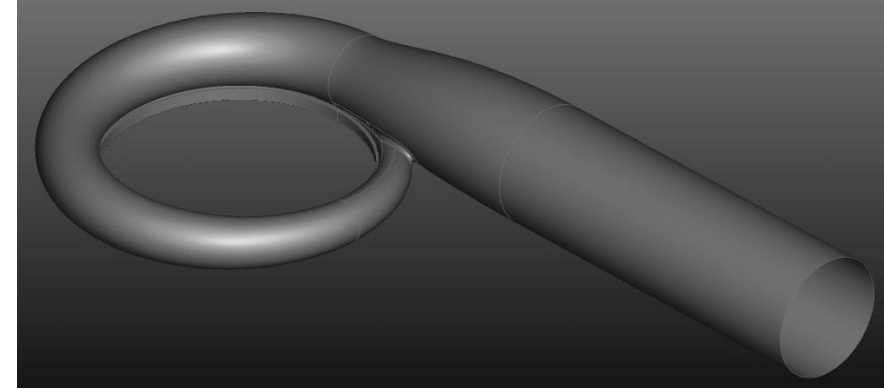
Geometry import as Parasolid

Blocking

Optimisation of grid point distribution

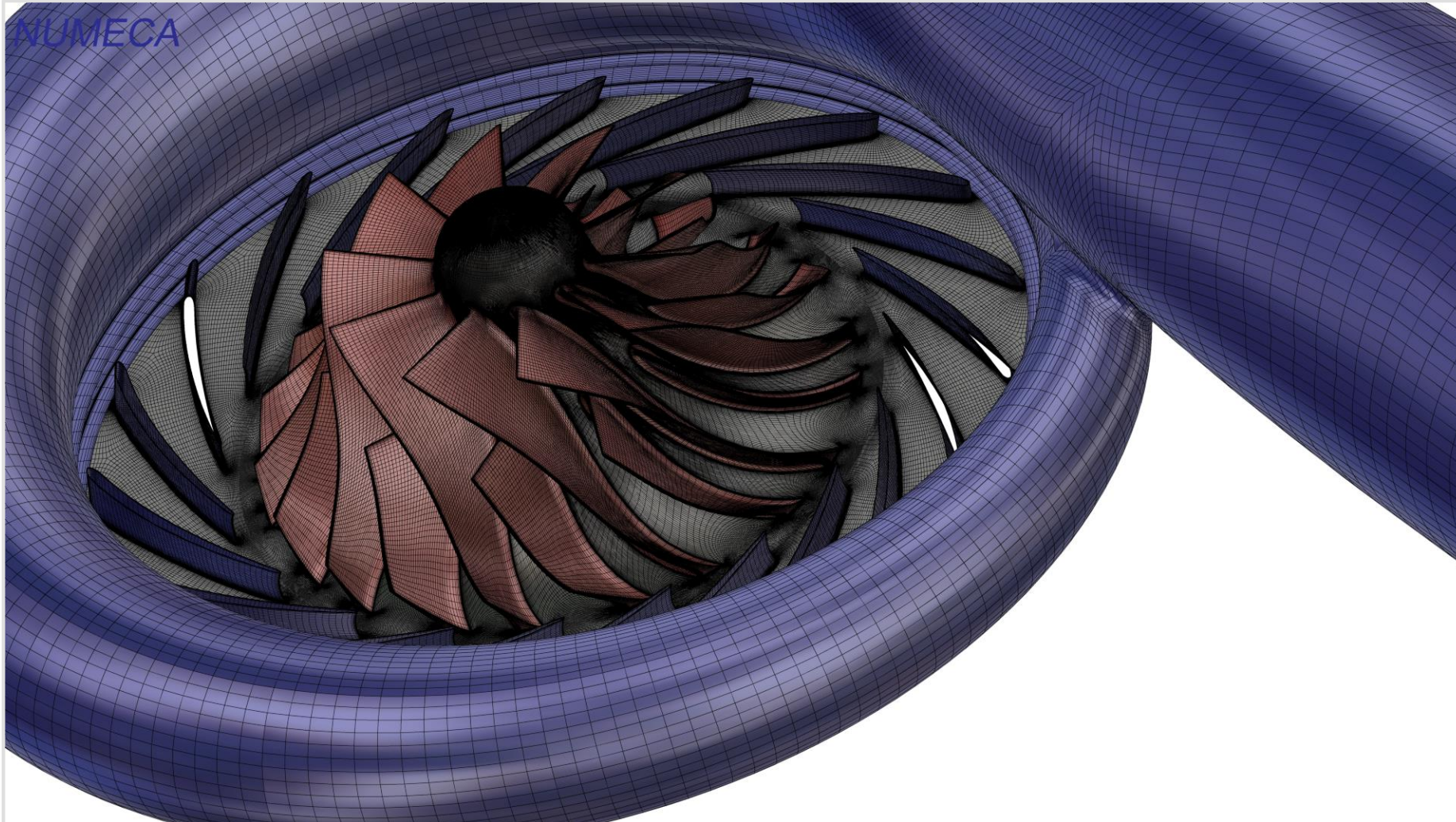
Mapping on surface

Mesh export



CFD Meshing Impeller

CFD Meshing Volute



Block structured hexahedral mesh - impeller with splitter blades, vaned diffuser and volute



3

Grid Generation for CSM Simulation



Requirements & Solutions :

- Fully automated periodic matching meshing of one passage
 - Parametrisation of periodic segment in CAESES
- CSM Solver require only tetrahedral elements
 - Two meshing steps:
 - Hexahedral dominant mesh
 - Transfer of elements to tetrahedrons
- Minimum amount of cells but:
 - Accurate leading edge and fillet capturing
 - More than one cell per blade thickness
 - Surface-to-Volume approach
- Coupling CAESES and HEXPRESS™/Hybrid
 - Geometry segment as Parasolid
 - Configuration file

```
INFILENAMES
1
Impeller_solid.x_t

OUTFILENAME
S2V-mix.spb.gz

#Option für CAD import bei infilenames
CADIMPORTPARAM GREAT

GENERATES2VMESH
#####S2VPARAM
S2VPARAM GENERATEVOLUMEMESH MIXED
S2VPARAM EXPANSIONRATIO 1.3
S2VPARAM CURVATURE 10
S2VPARAM MAXASPECTRATIO 4
OPTIPARAMETER 4

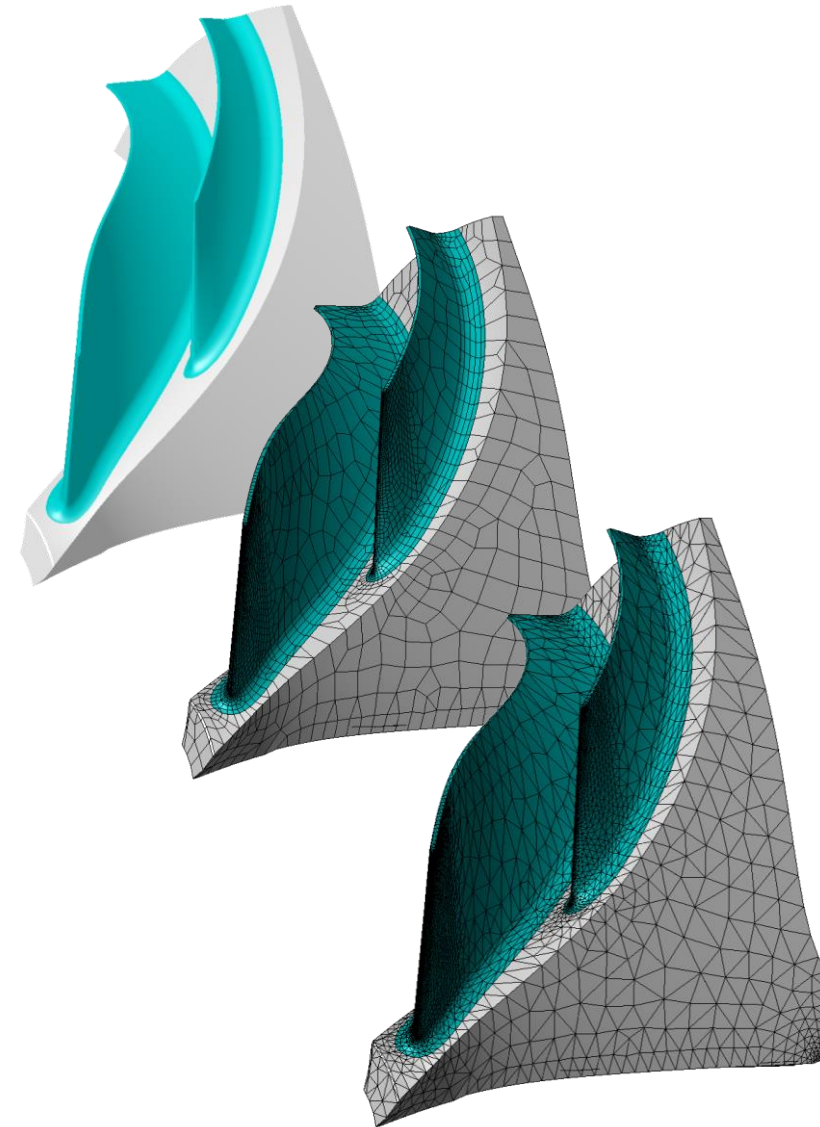
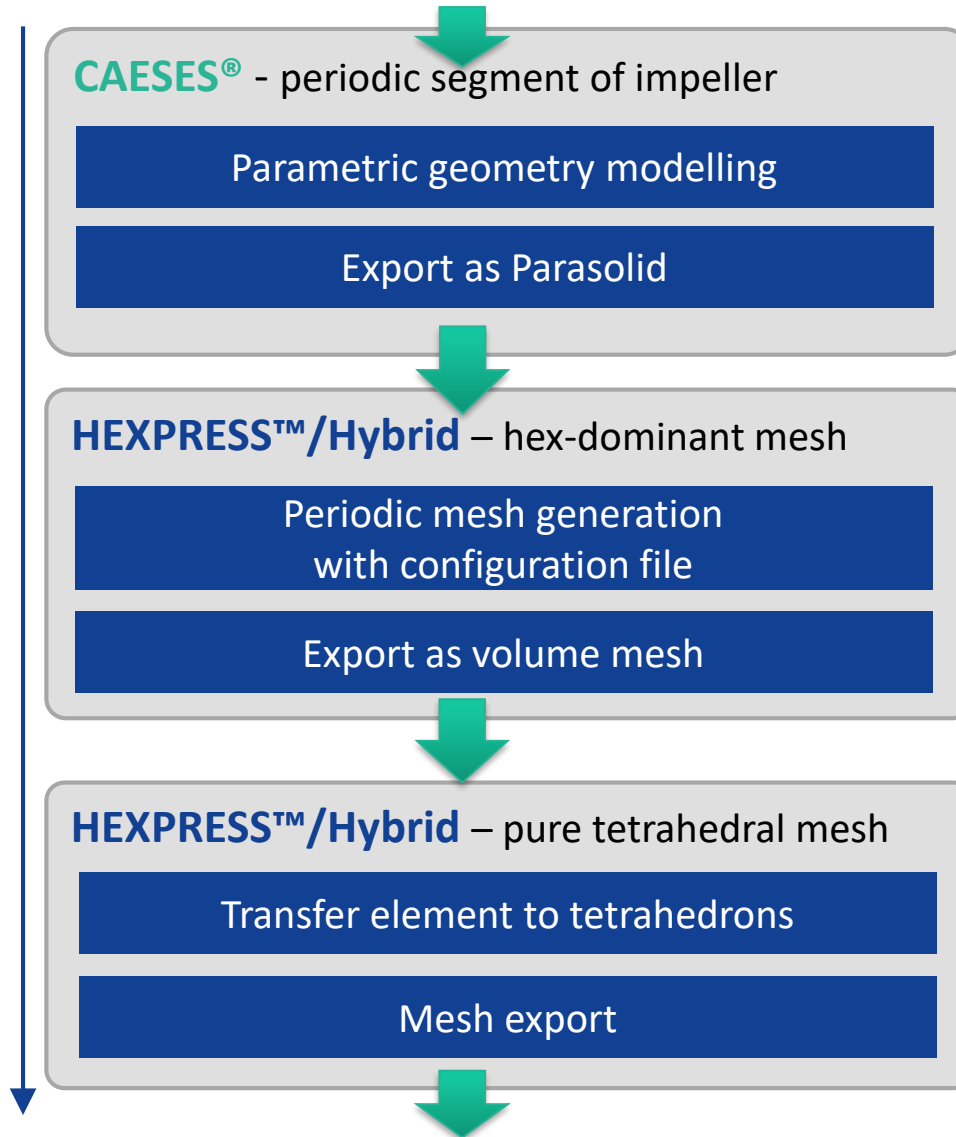
USEBINFILEVERSION 11
SUPPRESSPREFIX
BASEH 0.004
#NUMBNDLAYERS 1

LOCALREFINEMENTDEFINITION nodes_drill1 1 1
LOCALREFINEMENTDEFINITION nodes_wheel_c2_ax 1 1
LOCALREFINEMENTDEFINITION *|leading_edge_fillet* 2 1

#Periodic Impeller = 9
PERIODICCONNECTIONROTATE
1
nodes_wheel_ri
1
nodes_wheel_le
0 0 0
0 0 1
9
```

Grid Generation for CSM Simulation

Coupling for the Optimisation Workflow



Advanced Grid Generation for Impeller, Diffusor and Volute

Mesh Details

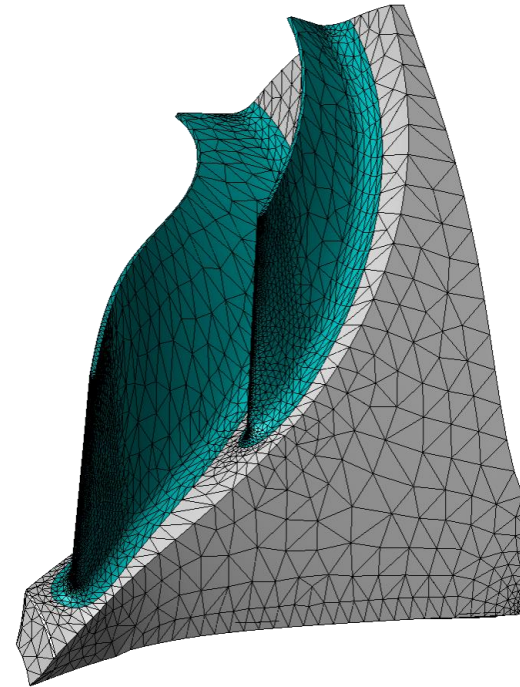
Copyright
NUMECA
Ingenieurbüro

19



Features

- Only tetrahedral elements
- Periodic matching
- Accurate leading edge and fillet capturing
- More than one cell per blade thickness

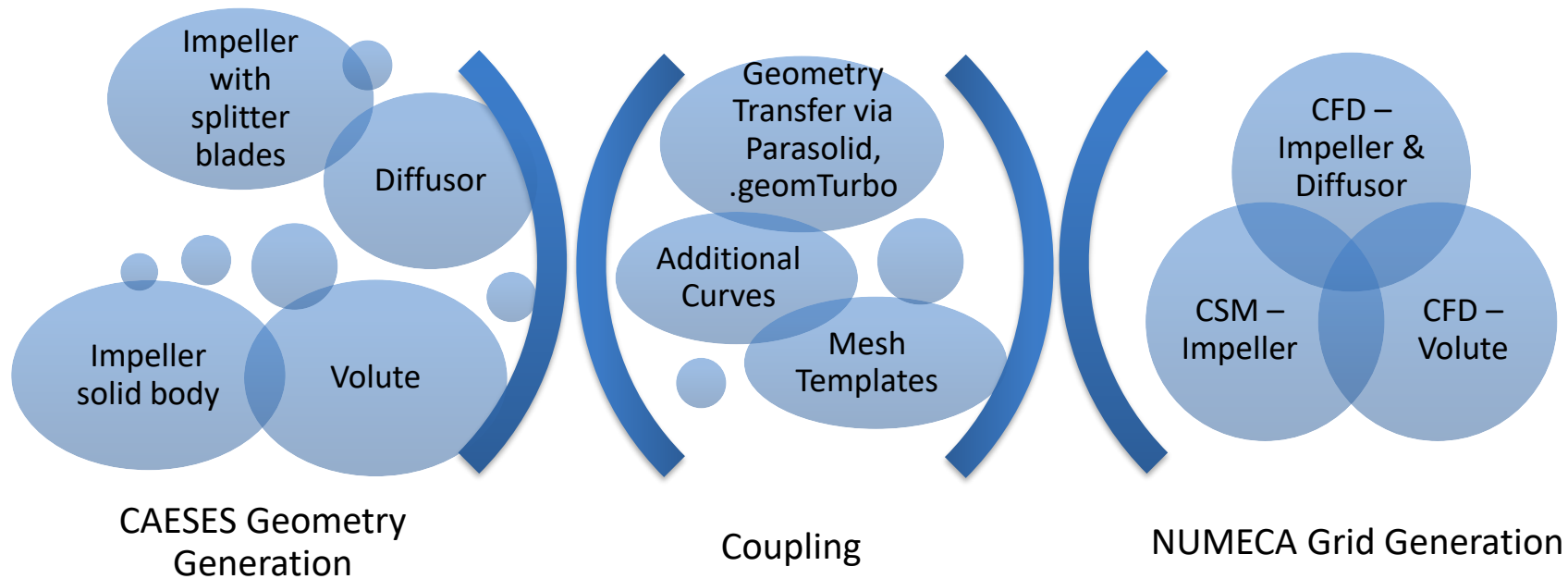


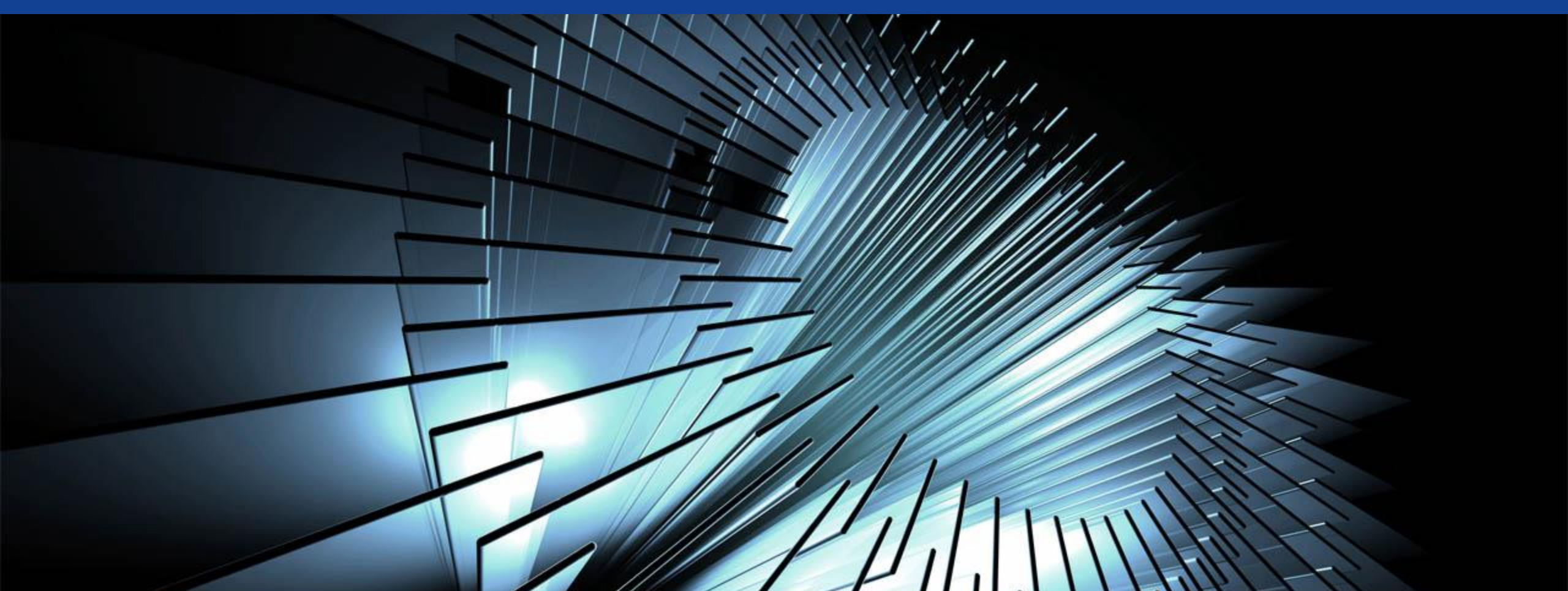
4

Summary



- Robust geometry generation with CAESES
- Automatized and robust coupling
- High quality grids for CFD and CSM simulation





2019

End of Presentation

Thank you for your attention