JASS 2008 - Joint Advanced Student School, Saint Petersburg, 9. - 19.3.2008
Modelling and Simulation in Multidisciplinary Engineering

Numerical Simulation of the Performances of Centrifugal Pumps on a Numerical Test Rig

Lindenmüller W.
Technische Universität München,
Lehrstuhl für Fluidmechanik – Abteilung Hydraulische Anlagen und Maschinen,
D-85747 Garching Germany

Content

Numerical Simulation of the Performances of Centrifugal Pumps on a Numerical Test Rig

JASS 2008
W. Lindenmüller
Modelling and Simulations in Multidisciplinary Engineering
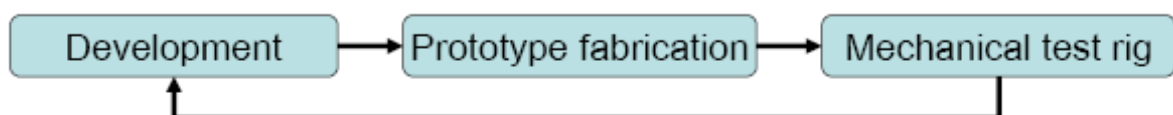TU München

Motivation

The Fluid Mechanics chair (FLM) of the Technischen Universität München (TUM) develops design, simulation and optimization tools to support the development process of turbo machinery. These tools are integrated in a software suite which is called **I**ntegrated **D**esign **S**uite (**IDS)**. The tool package is designed for small and medium businesses where only a medium budget for numerical tools available. Therefore the suite runs on a usual desktop computer with a dual or quad core CPU. The IDS system consists of three parts which can be divided into pre processor, solver and post processor.

The aim of the work is to proof the fitness and reliability of these tools as a numerical test rig. That means the performance data of the pump are investigated with the assistance of IDS suite. For the simulation only the CAD, fluid and performance data in design point of the centrifugal pump are available.

In future the development process could be improved to be more efficient. That means a change from a "classical" design approach to a **C**omputer **A**ided **E**ngineering (**CAE)** based design approach.
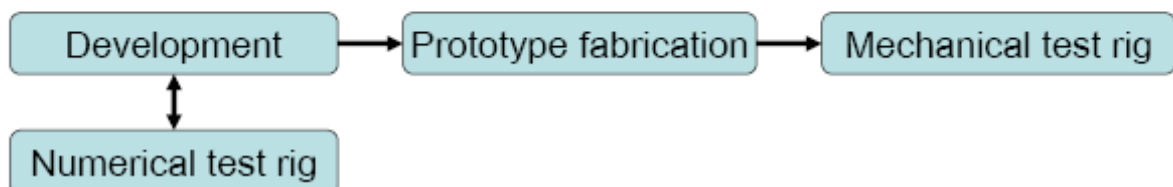
Features of the IDS suite:

- Geometry tools
- automatic grid generator
- Several solvers
  fluid
- "single click" post processing unit
- Automatic run system
- Block structured grids are needed

Needed inputs:

- Geometric data of the three
  main parts
- Physical conditions of the

- Design point data from the
  pump

Data of pump in design point

Volume flow   90 m³/h
Max. delivery head   18,35 m
Rotation frequency   1460 U/min
Reference radius   0,125 m
Number of runner blades 5

Nomenclature

φ dimensionless volume flow

ψ_t dimensionless total pressure

Q volume flow [m³/s]

r_{ref} reference radius [m]

u_{ref} circumferential velocity regarding reference radius [m/s]

Δp_t total pressure difference between inlet and outlet [N/m²]

p static pressure [N/m²]

ρ density of fluid [kg/m³]

y+   dimensionless wall distance

1. Preprocessing

The three main parts of a centrifugal pump

| Casing | Runner blade | Spiral |

Figure 1

Casing:
The casing is the main part of the pump. The inlet, impeller, spiral and driving motor are built into or onto the casing.

Runner blade:
The runner blade is the part which transfers the mechanical power from the drive shaft to the fluid. The number of blades has to be known in order to calculate the pump with the IDS system.

Spiral:
The spiral collects the fluid at the outlet of the impeller and leads it to the outlet of the spiral into the system which the pump is built in.

Every part of the pump has to be trimmed in order to mesh it with the grid generator. An important condition is that every part must have the same origin in the coordinate system.

The data file from the stream channel has to be saved as an *.igs file because the system can only import this file format. The *.igs format is an exchange format for CAD systems like the *.pdf is for letters. From this format the suite can only use points, lines and B-Splines. Therefore the meridian contour of the stream channel has to be reproduced by points.

The data file from the blade and spiral has to be a *.stl file. This file reproduces a shape by triangles.

Meridian contour

The points of the meridian contour is transformed from the (x,y,z) coordinates into cylindrical (r,z) coordinates. After the transformation the points are linked with B-Splines.
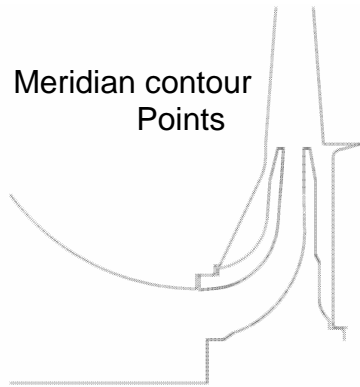
Meridian contour Points

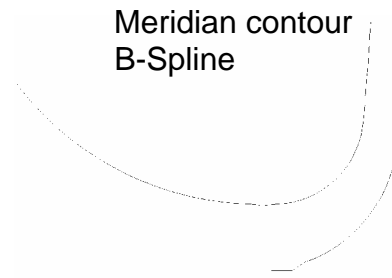Figure 2

Meridian contour B-Spline

Figure 3

Runner blade

After the B-Splines of the meridian contour are created geometric stream traces which consist also of B-Splines are created using a geometric interpolation algorithm. The blade is intersected with the stream traces and blade cuts are the results of is cutting process. The blade cuts are meshed by cubic splines after the blade grid is extruded to fully fit the meridian contour.
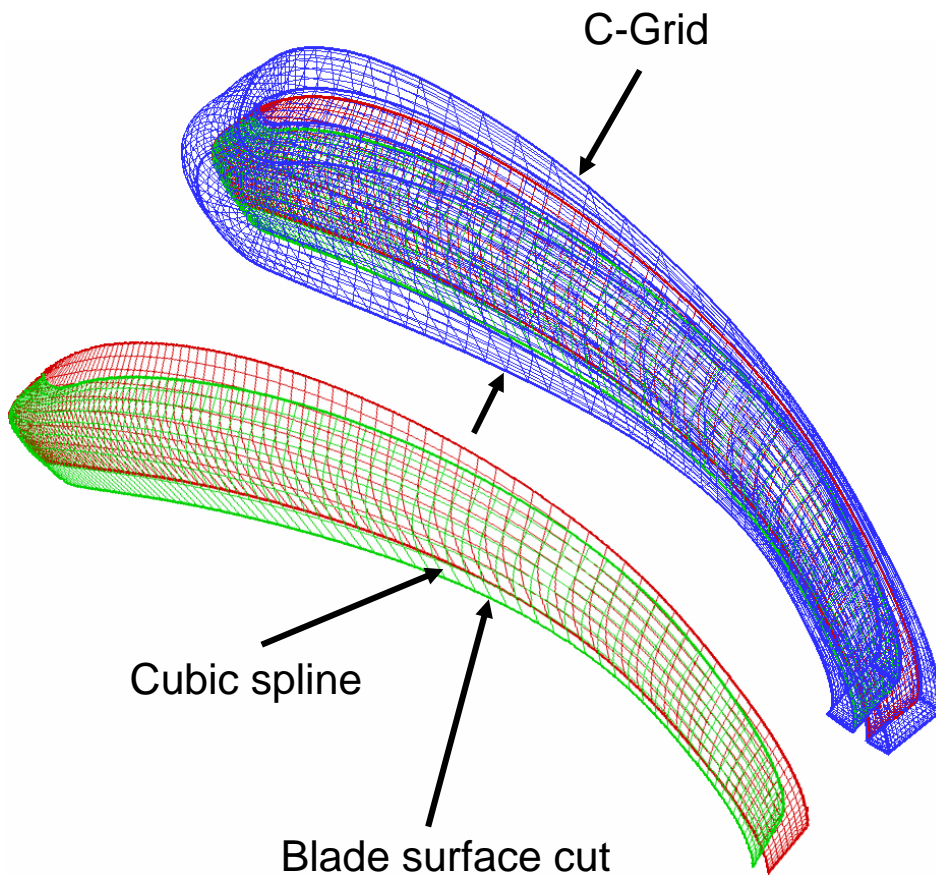

C-Grid

Cubic spline

Blade surface cut

Figure 4

Meshing the stream channel

The stream channel has to be divided into grid blocks. The blocks are meshed in 2D and the 2D – mesh is rotated to a 3D mesh around the z-axis.
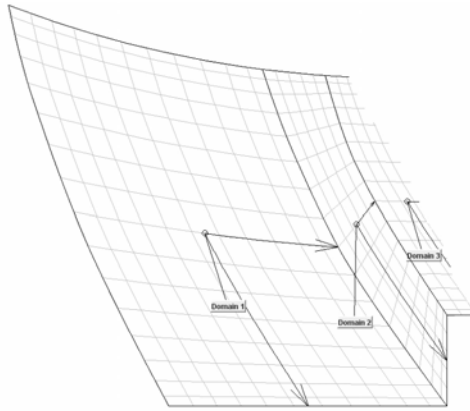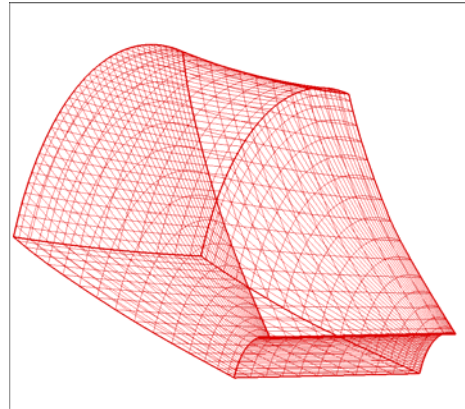

Figure 5

Figure 6

Meshing the spiral

The Spiral is cut into slices. These slices are structured in 2D and the slices will be connected with B-Splines. The grid is structured into grid blocks.
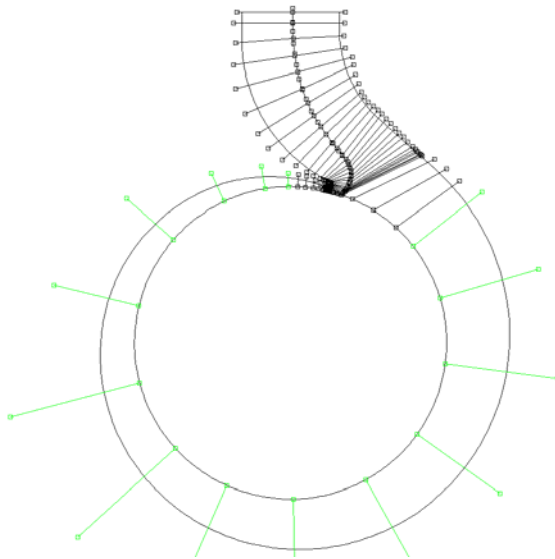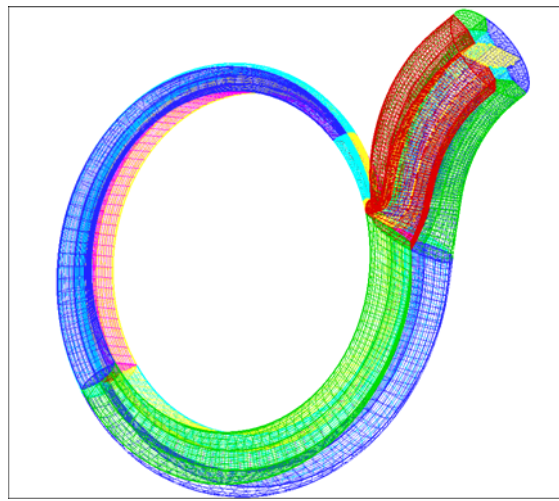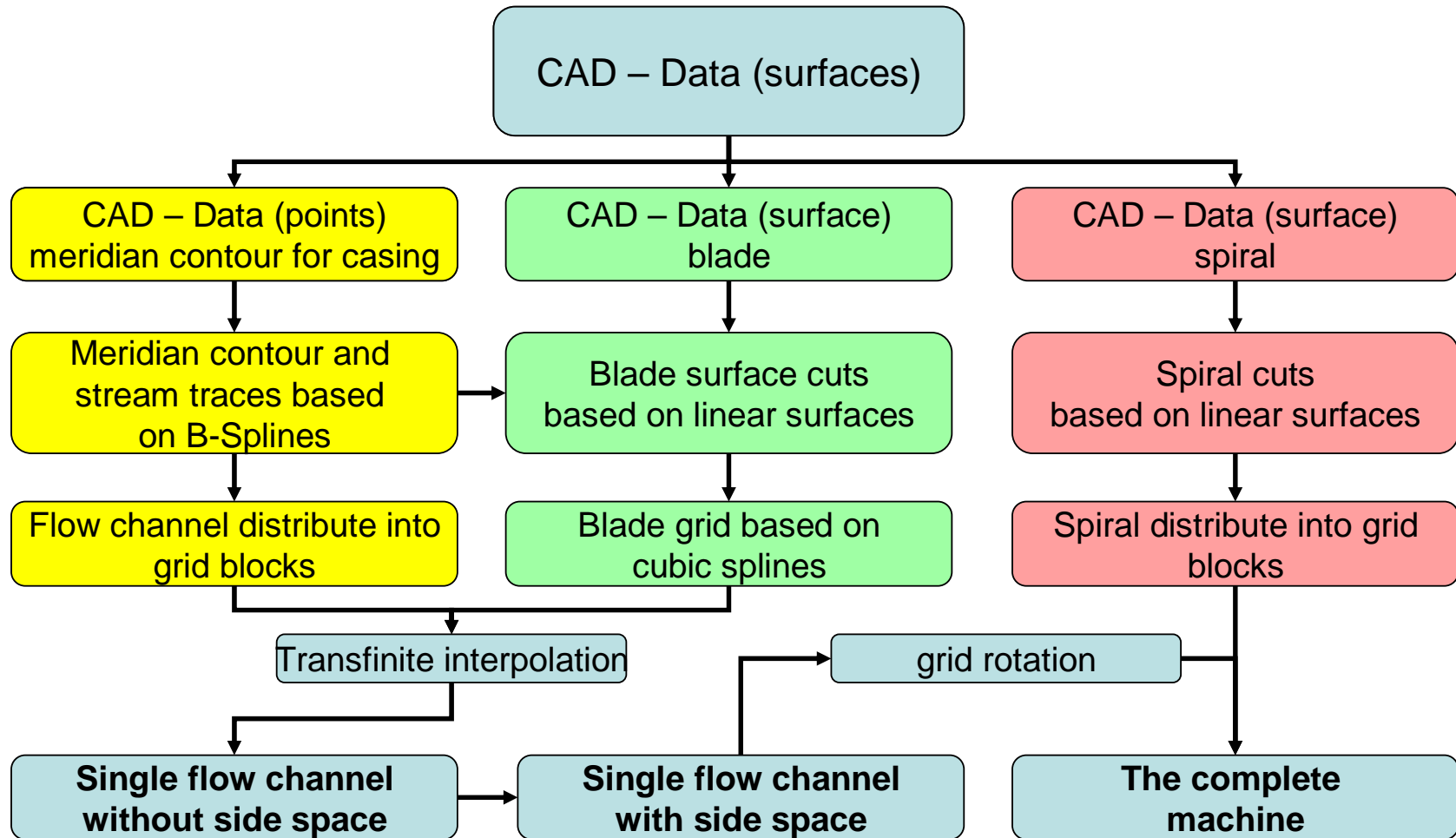

Figure 7

Figure 8

After the trimming and meshing process the separate parts are linked according to requirements in the different simulation approaches. There are 3 separate simulation approaches. In every approach another mesh of pump will be improved. Good simulation results depend on an optimized grid. The following diagram shows the different simulation approaches.

Geometric data

FLM

```
                        CAD – Data (surfaces)
```

CAD – Data (points) meridian contour for casing	CAD – Data (surface) blade	CAD – Data (surface) spiral
Meridian contour and stream traces based on B-Splines	Blade surface cuts based on linear surfaces	Spiral cuts based on linear surfaces
Flow channel distribute into grid blocks	Blade grid based on cubic splines	Spiral distribute into grid blocks

Transfinite interpolation

grid rotation

Single flow channel without side space → **Single flow channel with side space** → **The complete machine**

2. Simulation Approaches

Single flow channel without side space

The single flow channel without side space is a simulation which is resulted a optimized blade grid. The most important value for this improvement is the y+ value. A result of y+ optimization is a blade grid which is the base for every other simulation. If the calculation doesn't converge in the single channel calculation there is no chance to achieve calculation convergence in more complex simulations. In the pictures below one can see a grid and the influence of y+ values were shown.
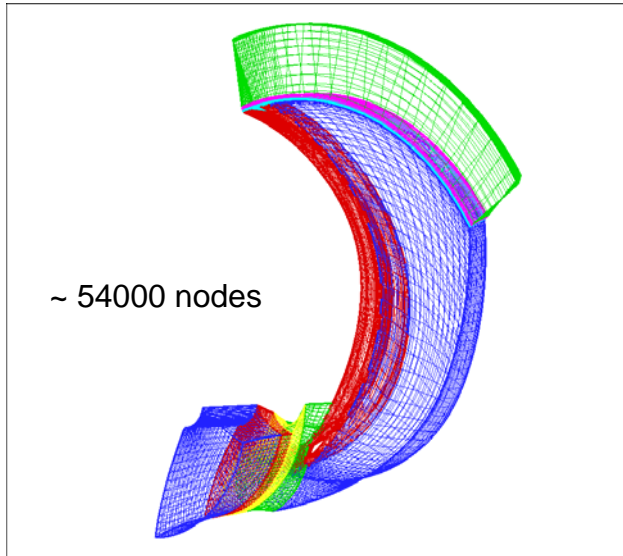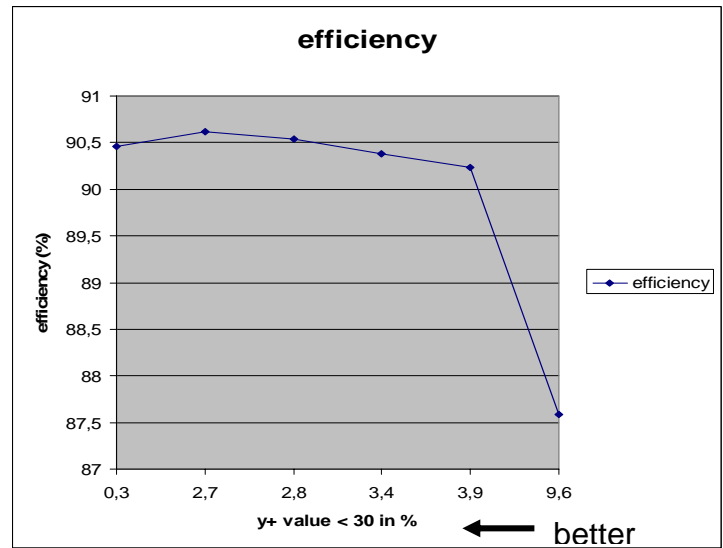

~ 54000 nodes

Figure 9

Figure 10

Single flow channel with side space

The single channel flow simulation is a simulation which shows the influence of the side space the performance of the pump. One of the important performance characteristic is the NPSH characteristic (**N**et **P**ositive **S**uction **H**ead). This characteristic shows which input pressure is required so no cavitation on the leading edge is caused. Another result is the loss of performance caused by side space flow.

$$\varphi = \frac{Q}{r_{ref}^2 \cdot \pi \cdot u_{ref}}$$

$$\Psi_t = \frac{2 \cdot \Delta p_t}{\rho \cdot u_{ref}^2}$$
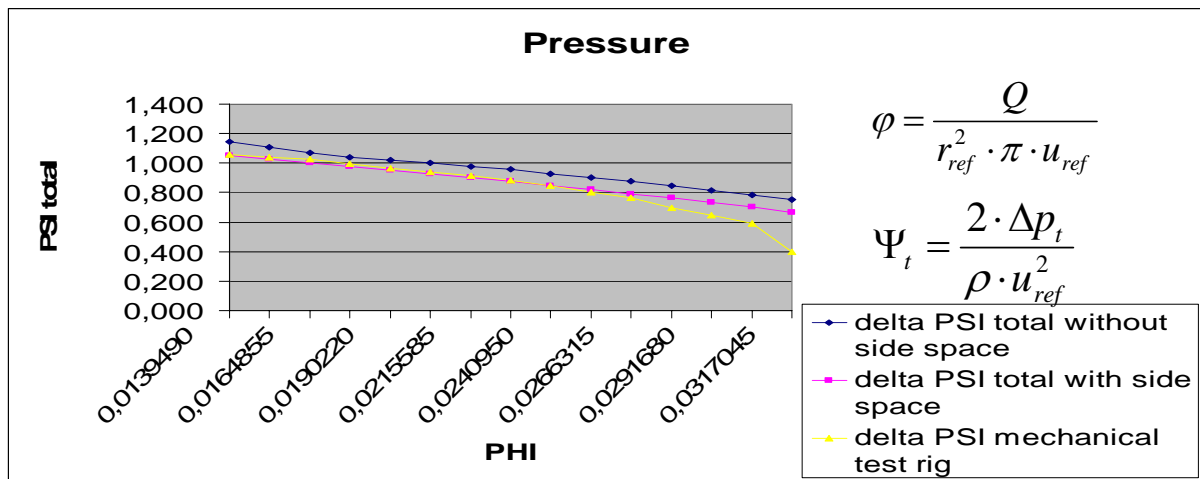
Figure 11

The diagram Figure 11 shows a good matching between the measurements on the mechanical and the numerical test rig. The pressure without side space is higher than with side space because there are no losses caused by the side space flow.

The complete machine

The complete machine simulation is a simulation which shows the influence of the spiral on the performance of the pump. In this simulation one can see the losses in the spiral. It is a frozen rotor simulation, i.e. a steady state calculation. Only the impeller (rotor) is moved in 9 degree steps until the next runner blade is on same position like the blade before. The result is the influence of the rotor position to the performance of the pump. This is shown in the diagram Figure 12 and Figure 13.
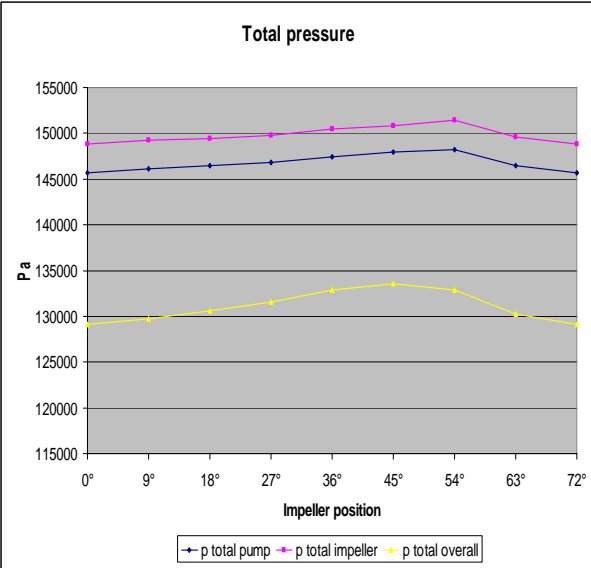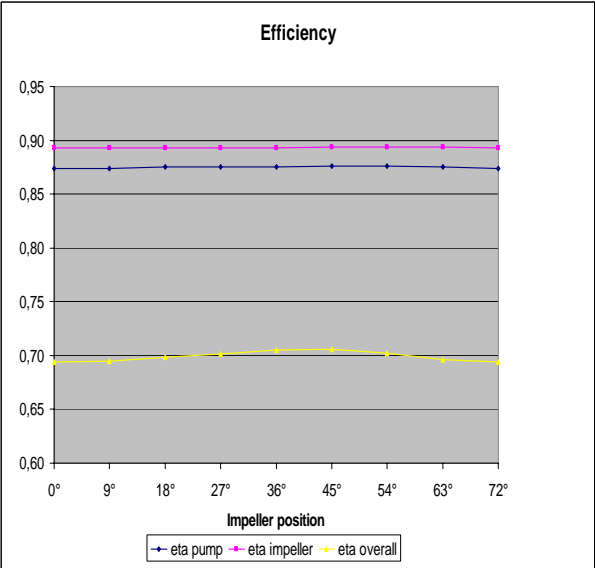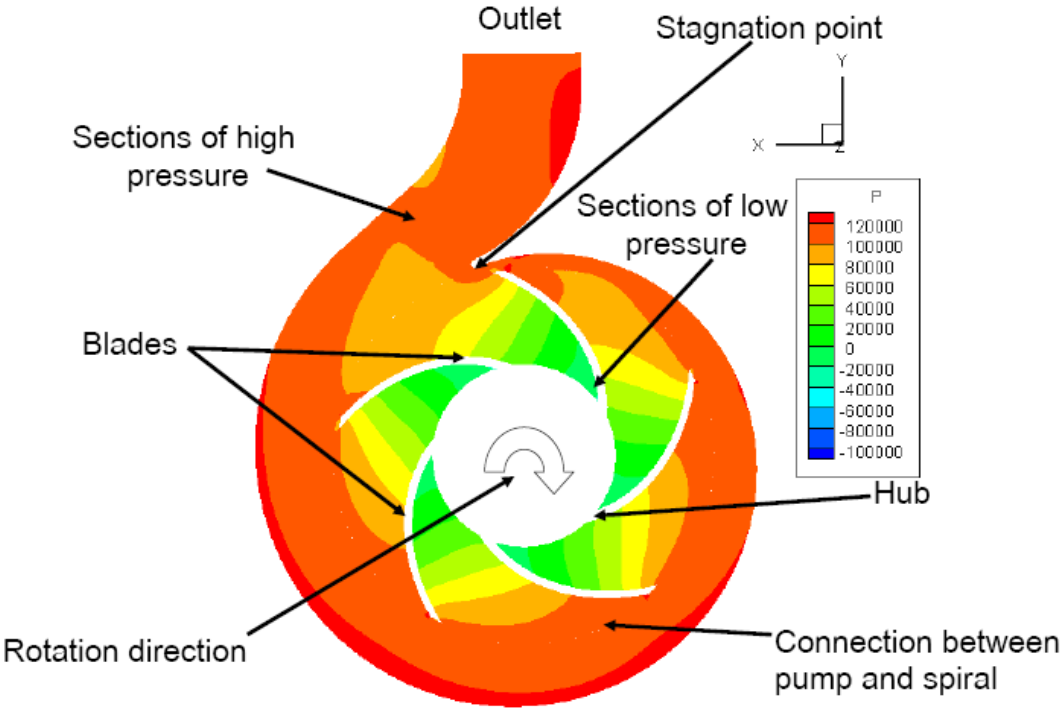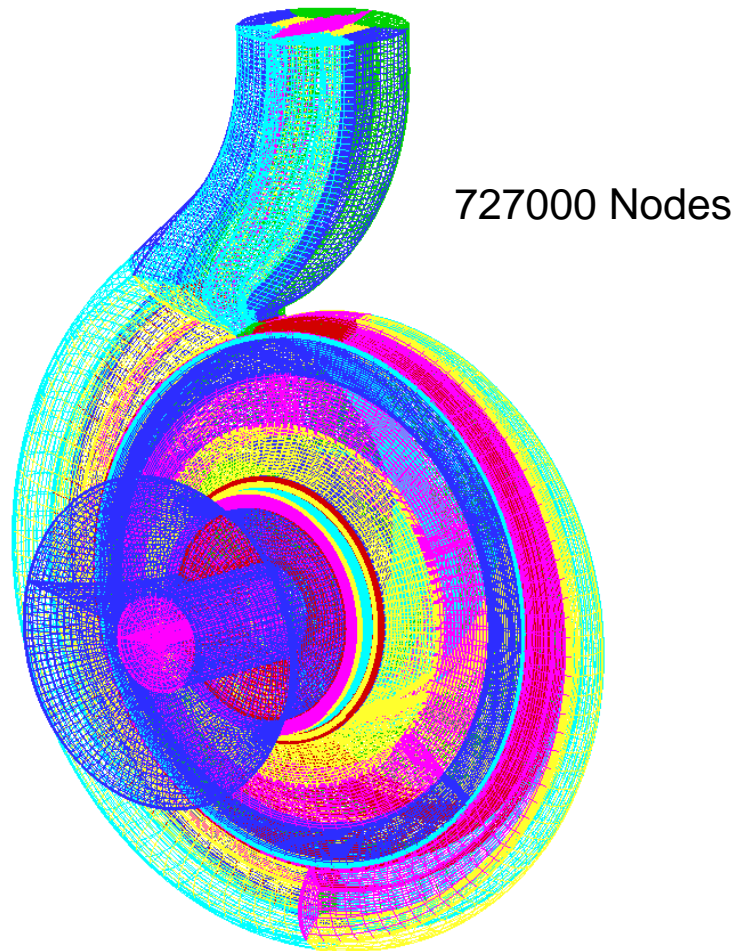

Figure 12

Figure 13

Figure 14

727000 Nodes

Figure 15

3. Conclusion

The Numerical Test Rig is a powerful software package containing full pre- and postprocessing tools and a fast solver-package.

The preprocessor contains a meshing tool which makes fast and high quality block structured meshing of geometry possible.

The solver-module produces very accurate and fast results also for highly complex turbulent and cavitating flows.

The automatized postprocessing allows the user to see and examine very fast the simulated results.