Thrusters

Numerical Analysis of Flow Around a Thruster

Norbert W. H. Bulten

Wärtsilä Propulson, Netherlands

October 17-18, 2006

Return to Session Directory

Numerical Analysis of Flow around a Thruster

Norbert Bulten
18 October 2006

© Wärtsilä

WÄRTSILÄ

Topics

- General introduction about thrusters
  - Application of nozzles
  - Performance predictions issues
- Research method: CFD
  - State-of-art numerical techniques
  - Experience at Wartsila with CFD
  - Basic priciples of CFD
- Results of analyses
  - Propeller-HR-nozzle configuration
  - Complete thruster-unit
- Conclusions

© Wärtsilä

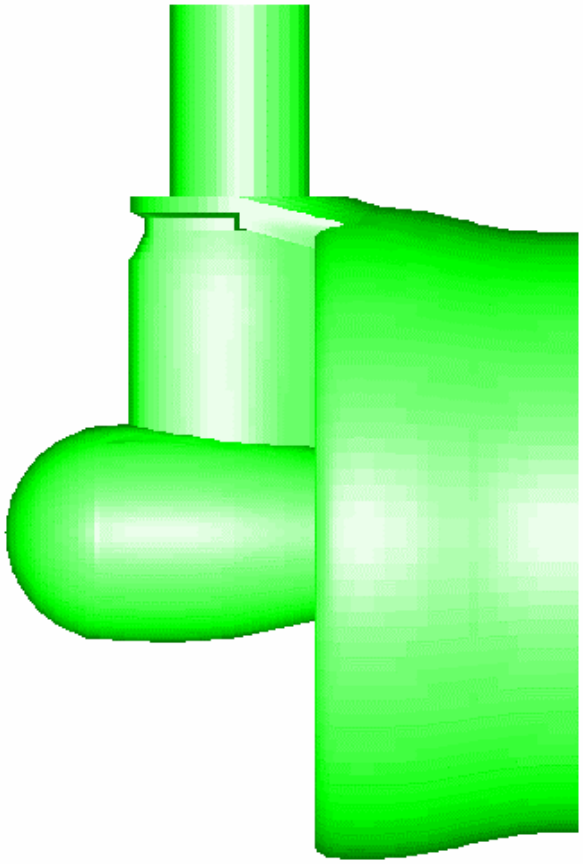
General thruster introduction

- Thrusters are applied for:
  - Dynamic positioning
  - Heavy lift vessels
  - Offshore support vessels
  - Cable laying vessels
  - Tugs

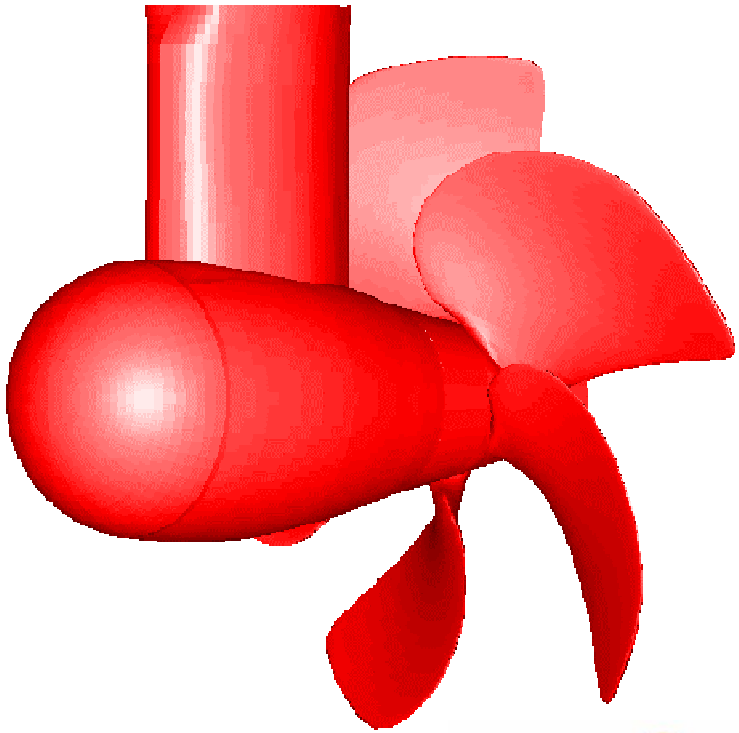
© Wärtsilä

General thruster introduction

- Thrusters can be equipped with or without nozzle



Lips HR-nozzle



Open propeller

© Wärtsilä

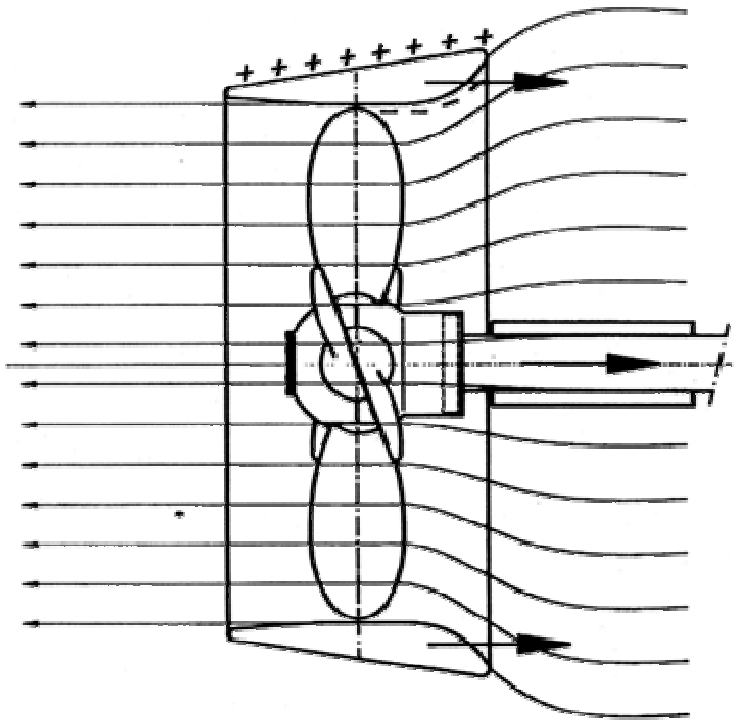
Application of nozzles

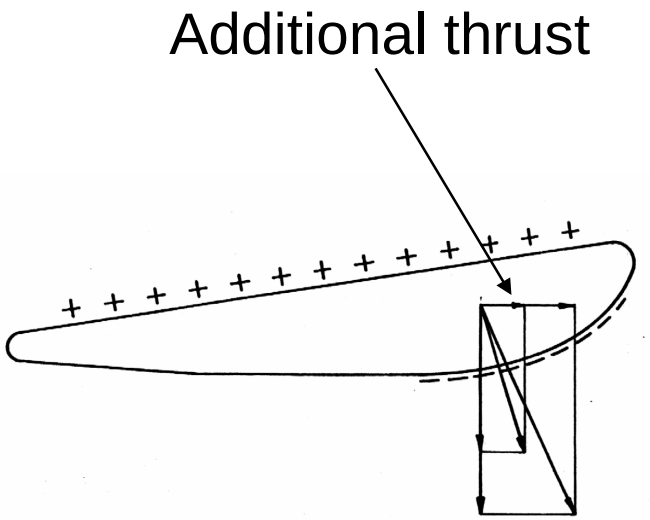
- Nozzles are applied to increase thrust at low ship speed. Typical examples of low speed operation are:
  - Bollard pull
  - Dynamic positioning
  - Fishing
  - Heavy lift vessels

© Wärtsilä

Application of nozzles

- Basic principle of nozzle: section acts similar as wing section
- Pressure distribution around nozzle changes due to propeller action

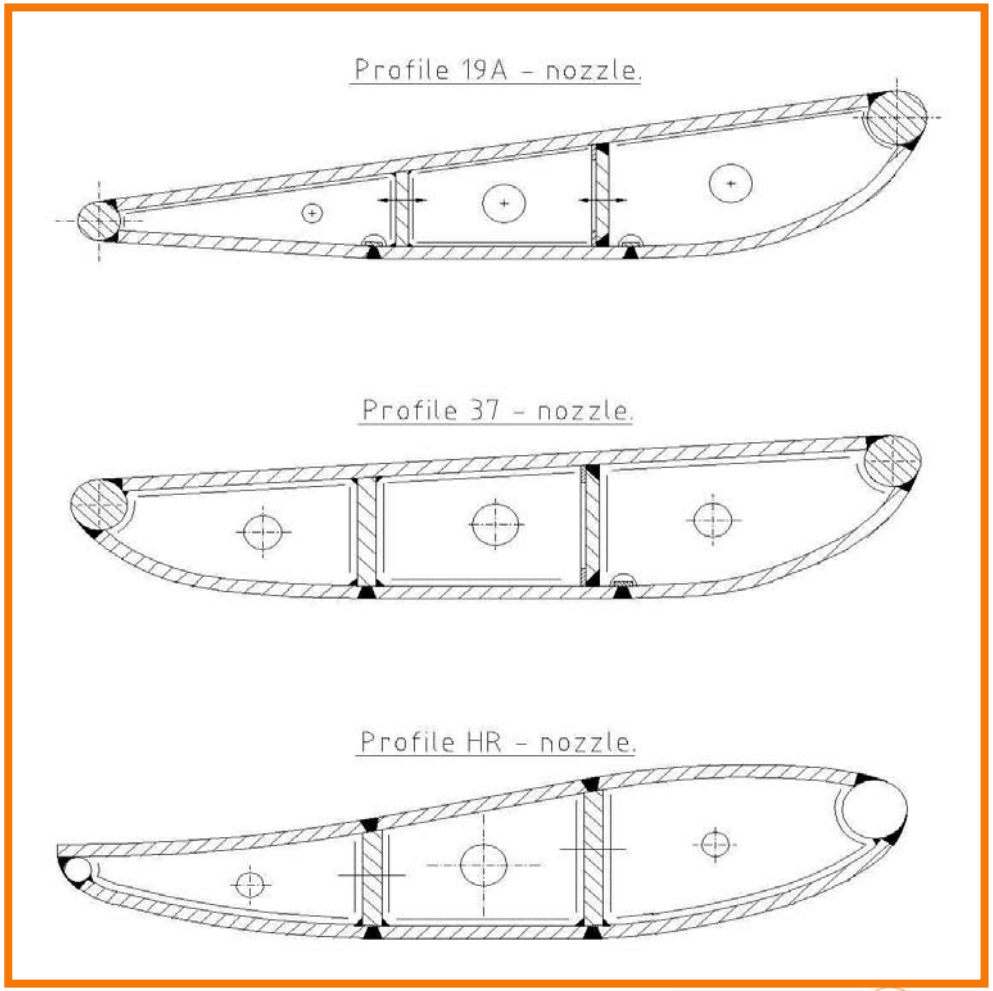




© Wärtsilä

Typical nozzle types

- Cross-sectional profile of nozzles:
  - 19A
  - 37
  - LIPS-HR
- Full scale measurements show 8% bollard pull improvement for HR compared to 19A



© Wärtsilä

Performance prediction of nozzles

- In general, open propeller performance is based on model scale experiments.
- Full scale performance is based on measured data in combination with scaling methods (ITTC'78)
- Scaling of ducted propellers (=propeller + nozzle configuration) not well understood
- New nozzle designs, like Lips HR-nozzle, do not behave like old 19A nozzles.
- Advanced numerical method (CFD) is applied to predict full scale performance of propeller with HR-nozzle

© Wärtsilä

WÄRTSILÄ

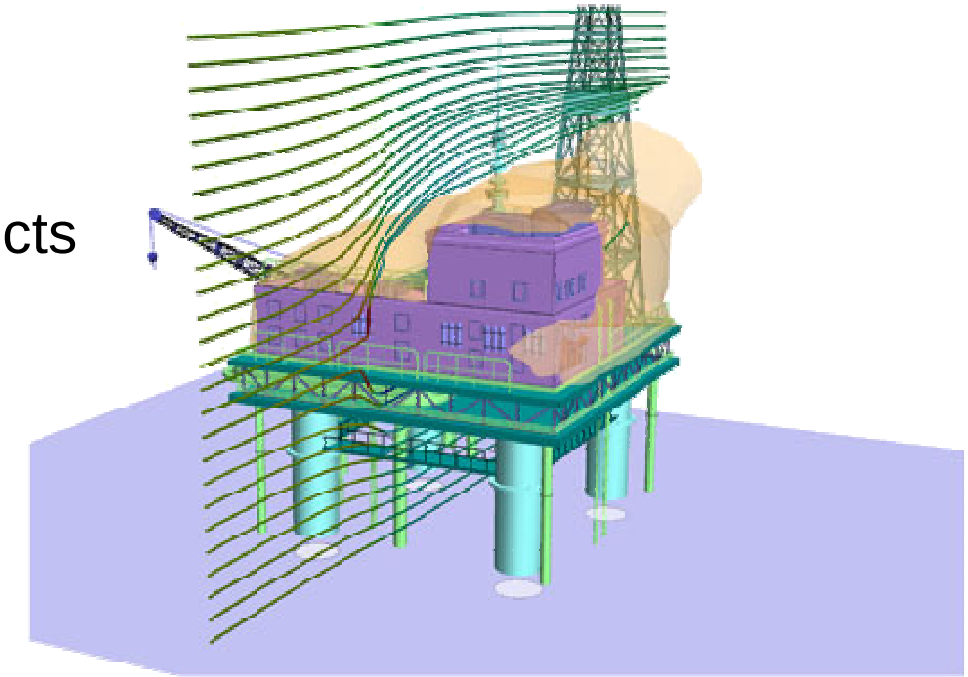
Performance prediction of thrusters

- Conventional thruster performance is based on model scale measurements.
- Scaling of thruster performance is more complex than ducted propeller, due to strut and thruster house
- Full scale performance of thruster unit can be based directly on state-of-the-art numerical calculations.

© Wärtsilä

Introduction to CFD

- CFD = Computational Fluid Dynamics
- State-of-the-Art numerical method to calculate flow field in three-dimensions
  - Velocity
  - Pressure distributions
- Method takes scaling effects into account
- Applications in industry:
  - Formula 1
  - Marine
  - Oil and gas
  - ….
  - …



Fire and smoke dispersion

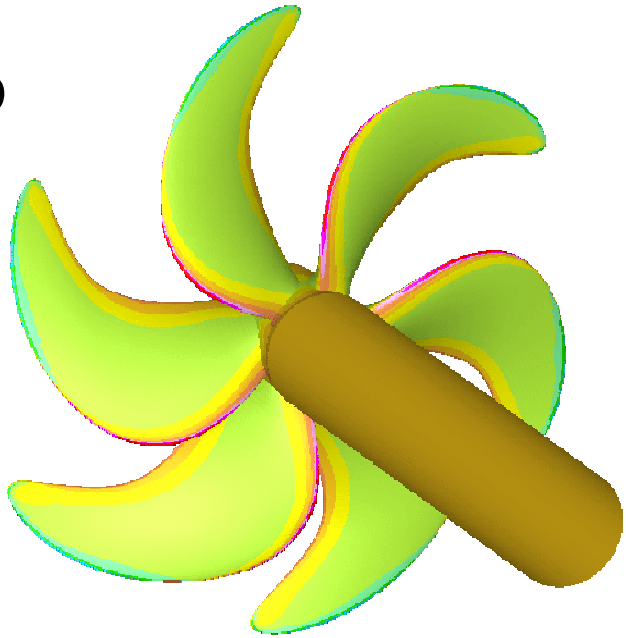
(www.cd-adapco.com)

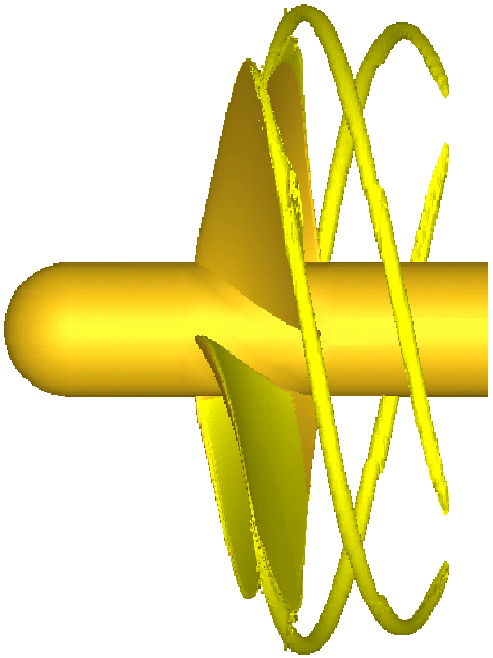
© Wärtsilä

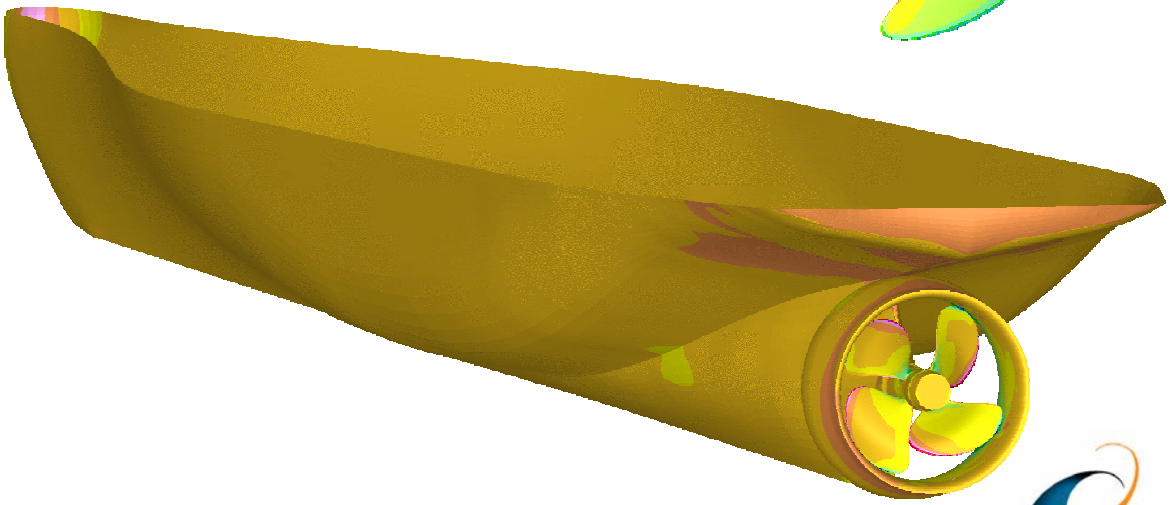
WÄRTSILÄ

Examples of CFD results

- Experience at Wartsila Propulsion with CFD
  - Propellers
  - Thrusters
  - Waterjets
  - Propeller – hull interaction







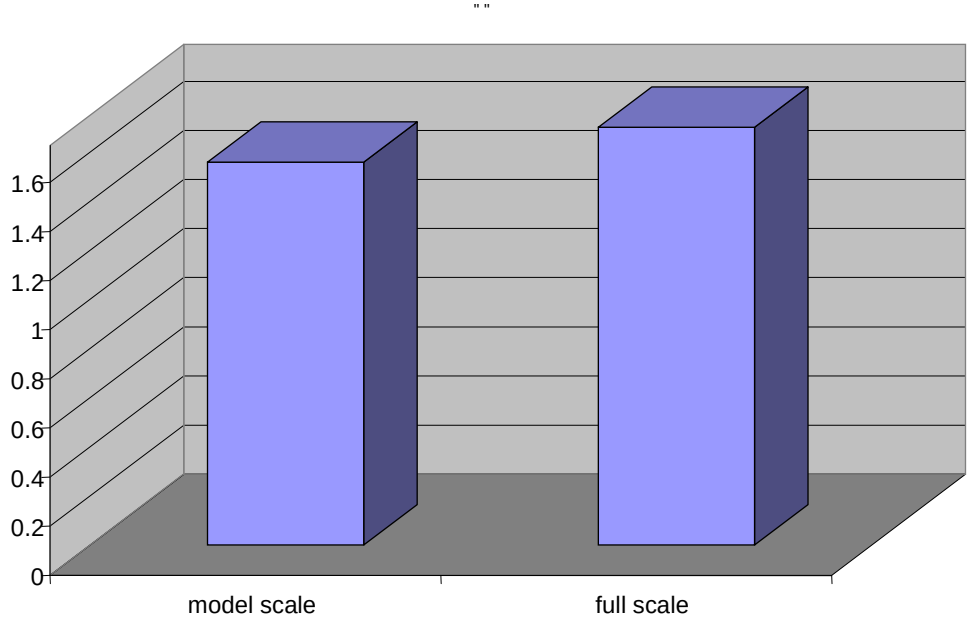
Basic principles of CFD

- Definition of numerical domain around geometry of interest (for example thruster unit)
- Numerical domain is split in large number of small cells
  -> grid or mesh with about 1.000.000 cells
- Velocity and pressure are solved in each cell
- Results of velocity field and pressure distribution are analyzed. This provides among others forces on the thruster:
  - Propeller thrust
  - Nozzle thrust
  - Resistance of thruster house

© Wärtsilä

Results of HR-nozzle analyses

- Comparison of model scale and full-scale calculations
- Open water efficiency is not suitable for bollard pull analysis
- Merit coefficient is used instead: $mc = \frac{\left(K_{tt}/\pi\right)^{3/2}}{K_q}$
- Results for Ka5-75 propeller show 9% increase



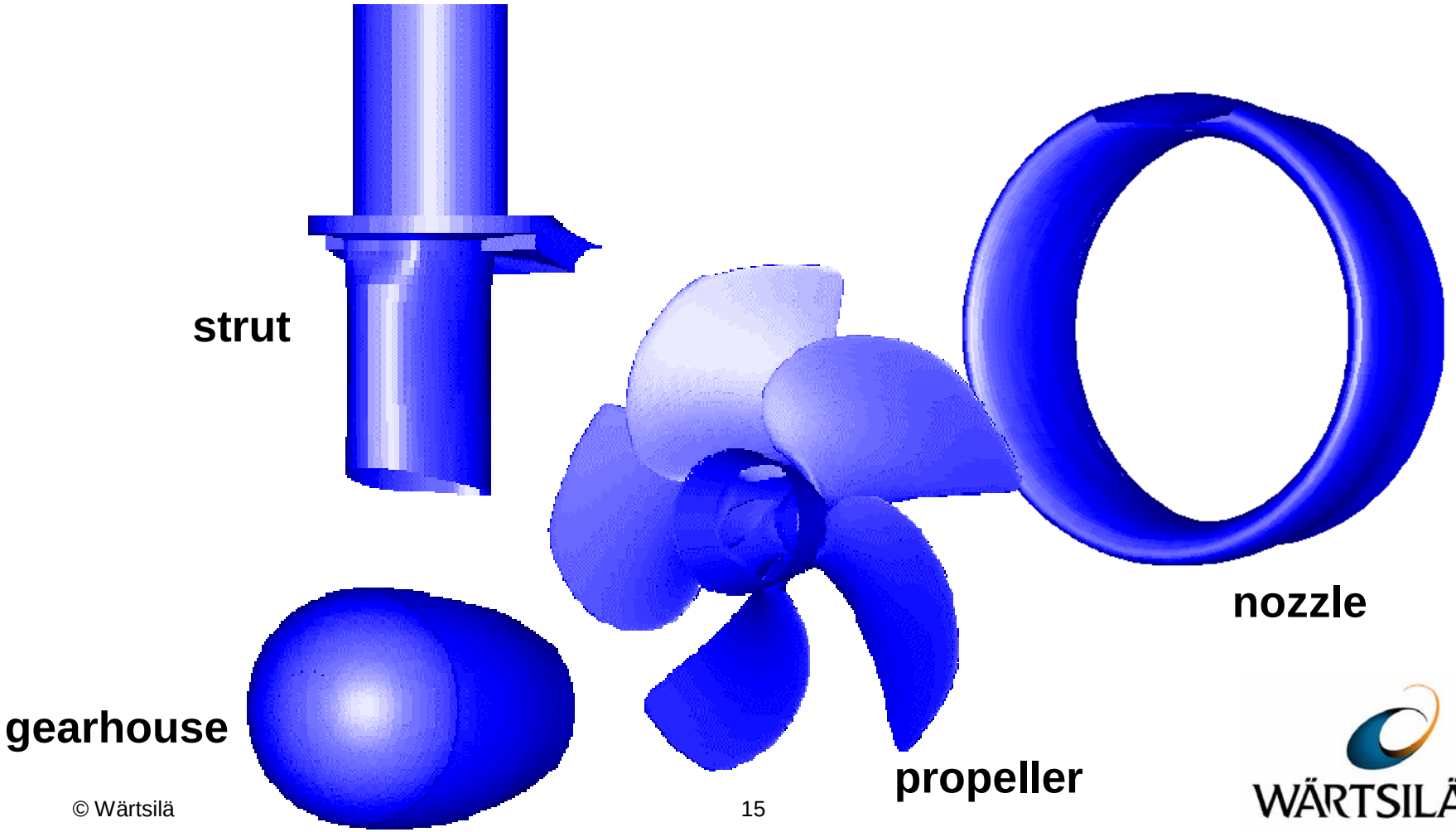
Results of thruster-unit analyses

- Pressure distribution along thruster unit

© Wärtsilä

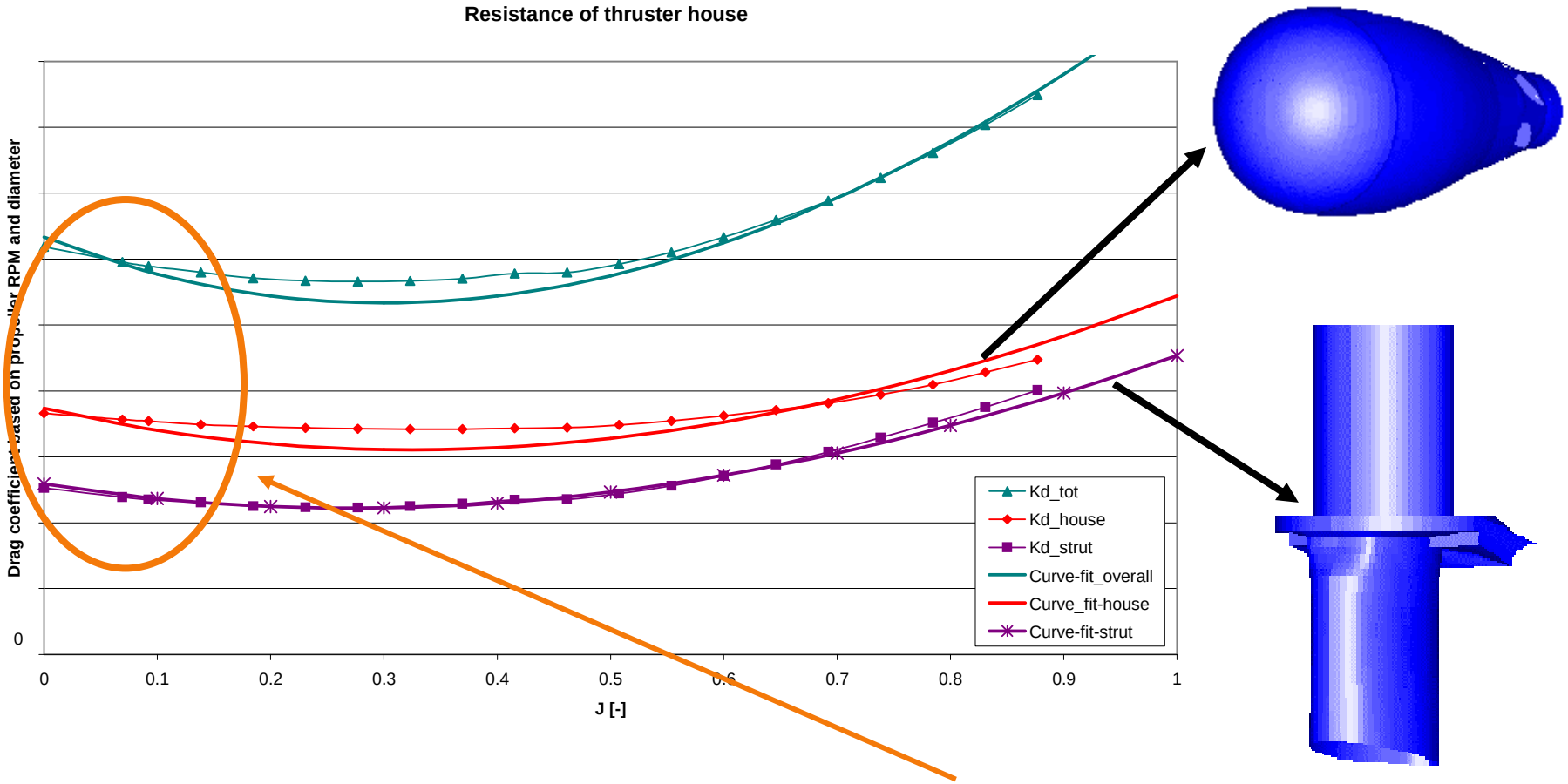
Results of thruster-unit analyses

- Thrust and resistance of thruster-unit is analyzed for all parts separately:



© Wärtsilä

Drag of thruster house



- Resistance of thruster house is present at low J values

Conclusions

- Bollard pull performance of HR-nozzle at full scale is significantly better than on model scale
- Drag of a thruster-house is present both at low ship speed and high ship speed. Though, the cause is different.
- The state-of-the-art numerical method of CFD is suitable to predict the full scale performance of ducted propellers and complete thruster units.

© Wärtsilä