OpenPOWER™

OpenFOAM on POWER8

Dr. Markus Bühler

POWER **A**cceleration and **D**esign **C**enter
IBM Deutschland Research & Development GmbH, Boeblingen

buehler@de.ibm.com

IBM

Agenda

OpenFOAM Overview

The Benchmarks

Performance Results

OpenFOAM with GPU

Summary

OpenFOAM Overview

OpenFOAM Overview

- **O**pen Source **F**ield **O**peration and **M**anipulation
- C++ Toolbox for Simulation of Mechanical Problems
  - Solvers
  - pre-/post processing utils
  - MPI parallelized
  - NO OpenMP
  - NO vector intrinsics
- Widely used in industry and academics
  - Computational Fluid Dynamics (CFD)
    - Aerodynamics simulations
    - Pharmaceutical industry
  - Electromagnetism
  - Combustion
  - ...
- Many different modules
  - simpleFoam – static analysis
  - pisoFoam – dynamic analysis
- Known to be memory bandwidth limited

OpenFOAM

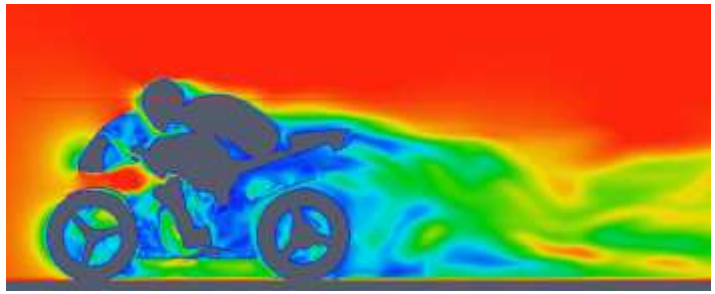
http://www.openfoam.com

The Benchmarks

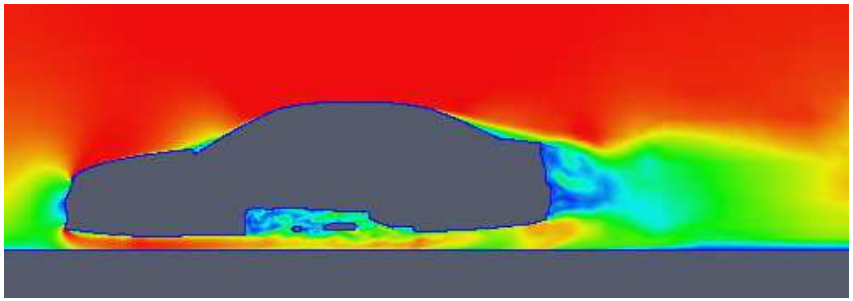
OpenFOAM on POWER8

- OpenFoam 2.3.0 Compiled on x86 and POWER8
  - Only minor configuration changes to compile on POWER8
  - Optimizations see next slide

- Benchmark 1: Motorbike Example
  - provided with OpenFoam examples: incompressible/simpleFoam
  - Different problem sizes by changing grid: 1k – 100M points



- Benchmark 2: Car
  - Car model
  - Morphed from two real cars
  - Only few problem sizes



Hardware Configurations

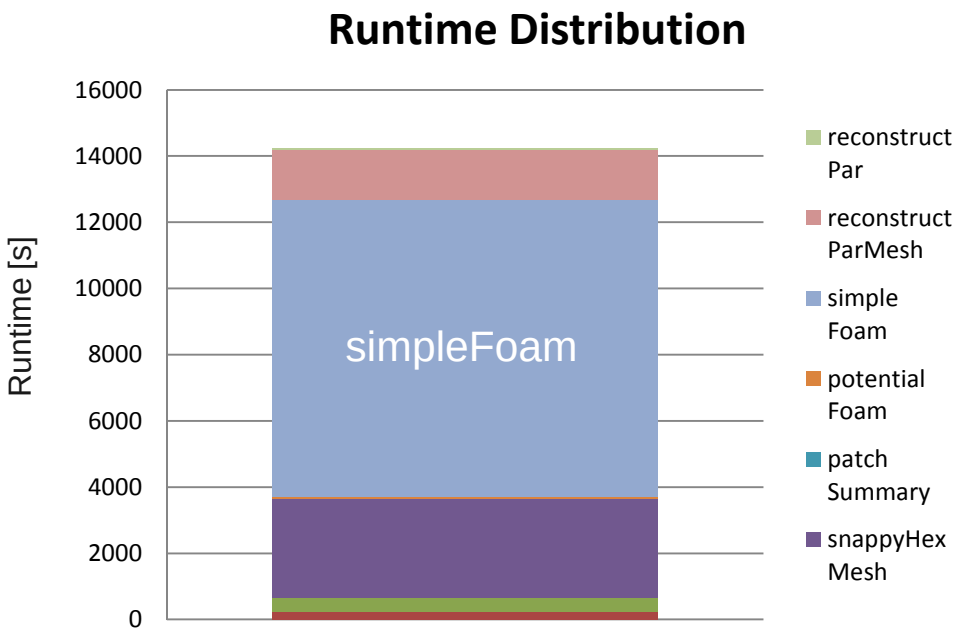
All values per socket

	IBM	Intel Ivybridge	Intel Haswell
Processor	P8	E7-4890V2	E5-2699V3
Clock	3.4GHz	2.8Ghz	2.3GHz
Cores / Threads (max)	10 / 80	15 / 30	18/36
Memory	128 GB	96 GB	384 GB
Memory BW	230 GB/s	85 GB/s	68 GB/s
Caches L1 / L2 / L3 / L4	640k / 5M / 80M / 128M	480k / 3.75M / 37.5M / n/a	576k / 4.6M / 45M / n/a
Linux	Ubuntu 14.04	Red Hat 6.5	Red Hat 7.1
Compiler	GCC 4.8.2	GCC 4.7.2	GCC 4.8.3

MotorBike Benchmark Overview (1)

9 Steps

1. surfaceFeatureExtract
2. blockMesh
3. decomposePar
4. snappyHexMesh
5. patchSummary
6. potentialFoam
7. **simpleFoam**
8. reconstructParMesh
9. reconstructPar



- simpleFoam dominates runtime
- other steps´not always required

➢ concentrate on simpleFoam

MotorBike Benchmark Overview (2)

OpenFoam Example

① Start

② blockMesh

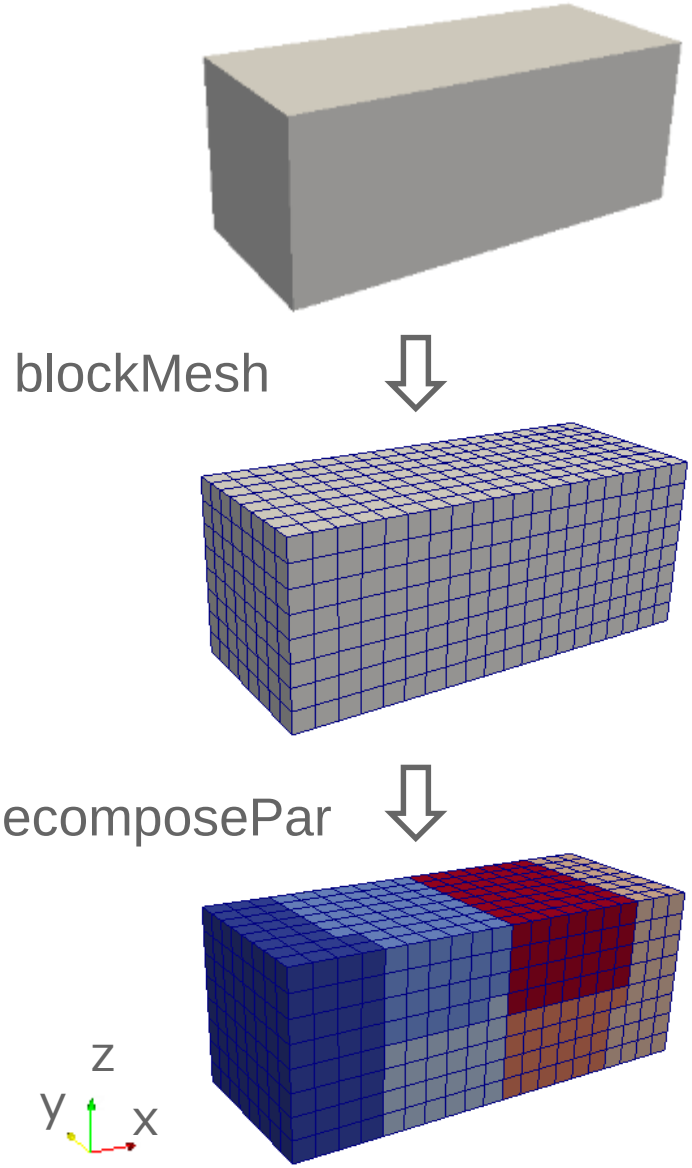
Mesh the 3D space

- 20 x 8 x 8 = 1280 cells (default)

  …
- 857 x 343 x 343 = 100M cells

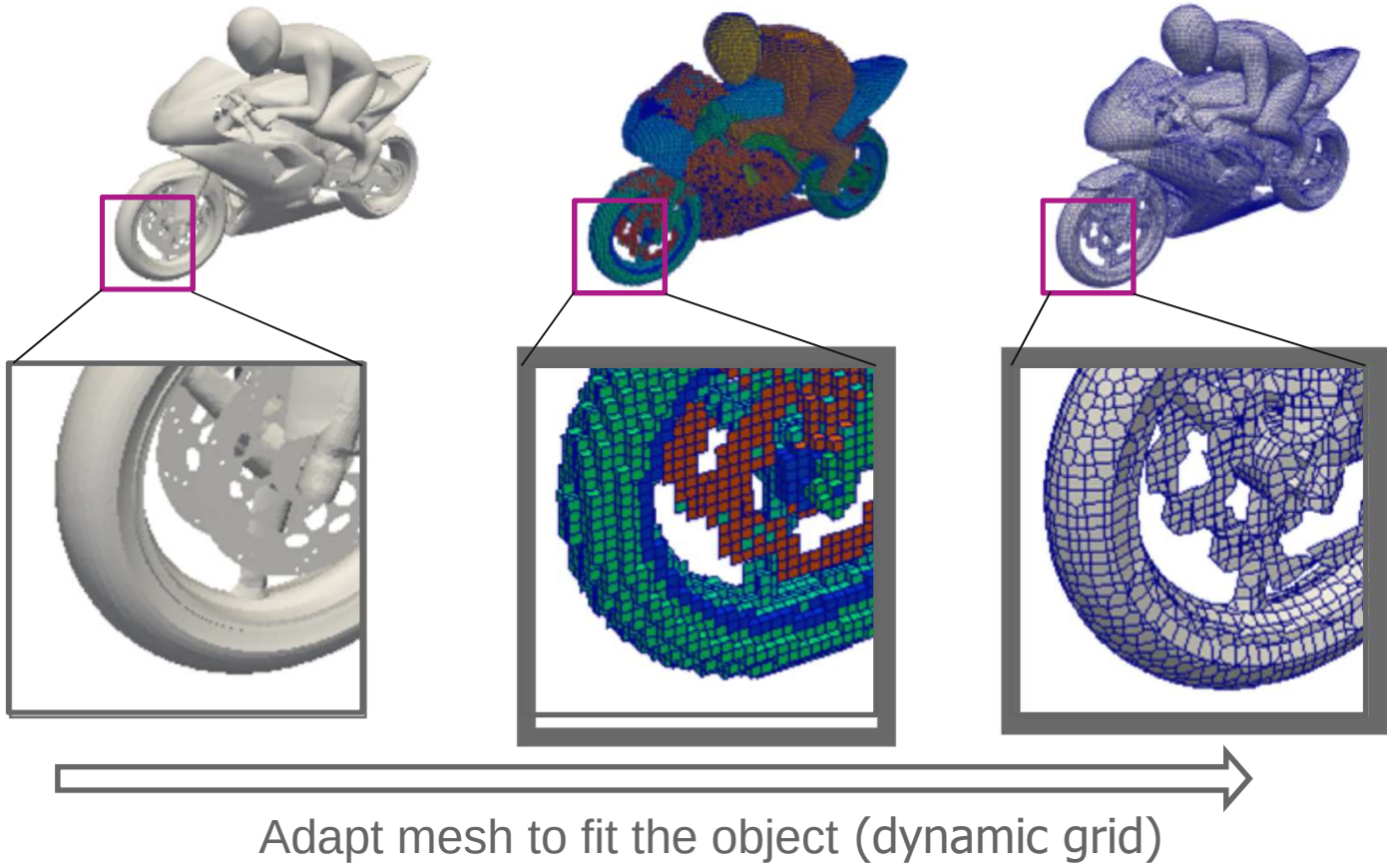
③ decomposePar

➢ Divide into submeshes



MotorBike Benchmark Overview (3)

④ snappyHexMesh

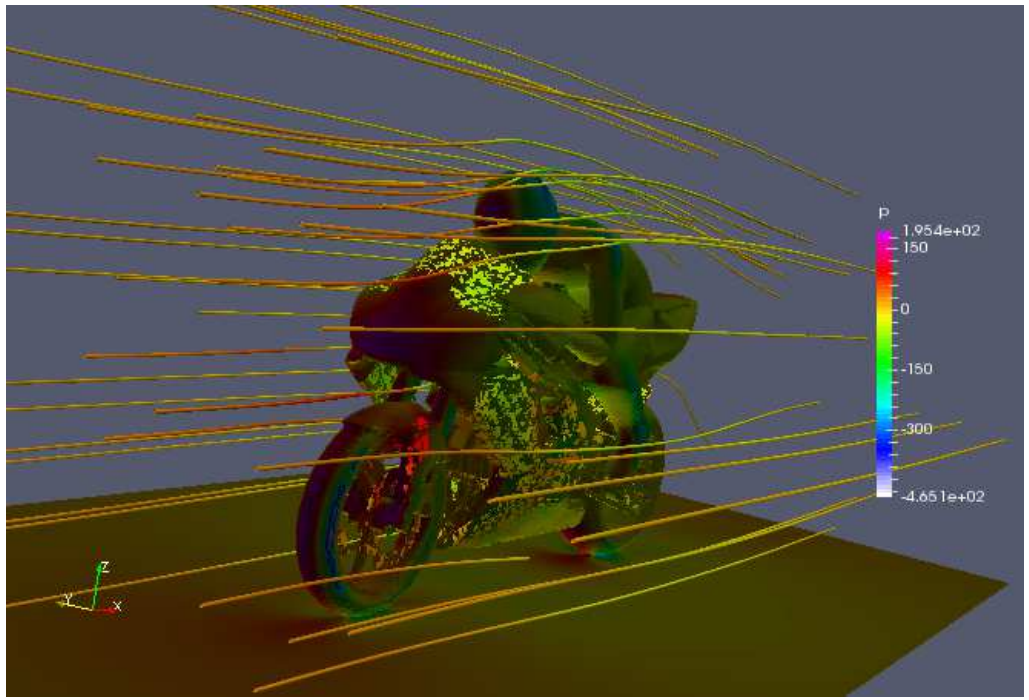


(Reference) http://www.rccm.co.jp/icem/pukiwiki/index.php?SnappyHexMesh

MotorBike Benchmark Overview (4)

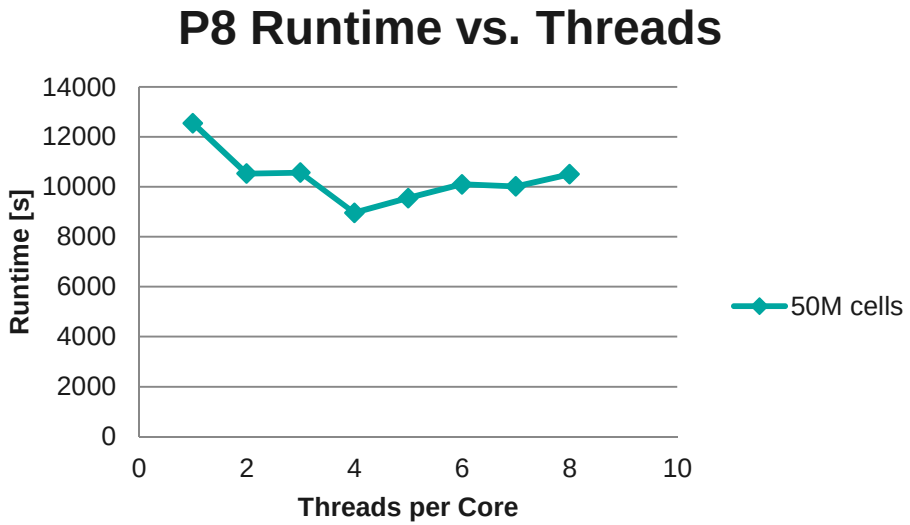
⑦ simpleFoam

- One of the main solver included in OpenFOAM.
- **S**emi-**I**mplicit **M**ethod for **P**ressure-**L**inked **E**quation: calculate velocity and pressure by iterative calculation
- Benchmark: run 500 iterative steps
- outcome: aerodynamic coefficients



Optimizations for POWER8

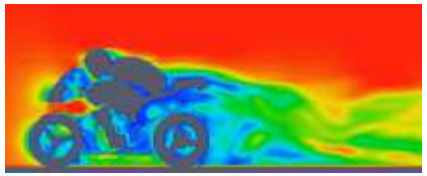
- Compiled with –O3
- Link with tc_malloc instead of standard malloc
  - simpleFoam: almost no impact
  - recomposeParMesh: up to 2x speedup
- Threads per Core
  - P8 offers up to 8 threads per core
  - Runtime impact



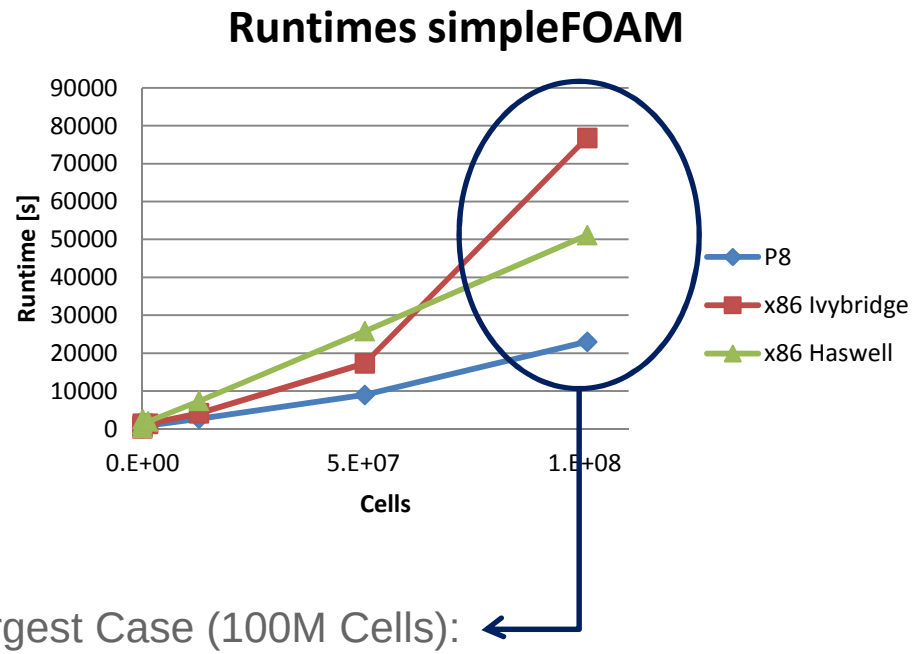
  - chose 4 threads per core unless stated otherwise
  - x86: little impact on runtime → use 1 thread per core

Performance Results

MotorBike Performance



- Running different problem sizes on P8, Ivybridge and Haswell
- Best threads/socket setting used for every data point



Conclusion

POWER8

- 1.5-3x faster than x86
- likely due to
  - Memory Bandwidth: 3x
  - Caches: 2x
  - SMT

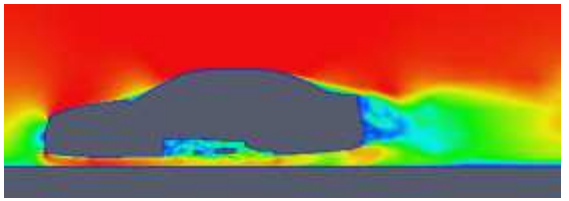
Largest Case (100M Cells):

- Ivybridge seems to reach memory limit (96GB), although not swapping
- Similar but less obvious effect on P8 (128 GB)
- Haswell box has more memory (256GB) → linear behaviour

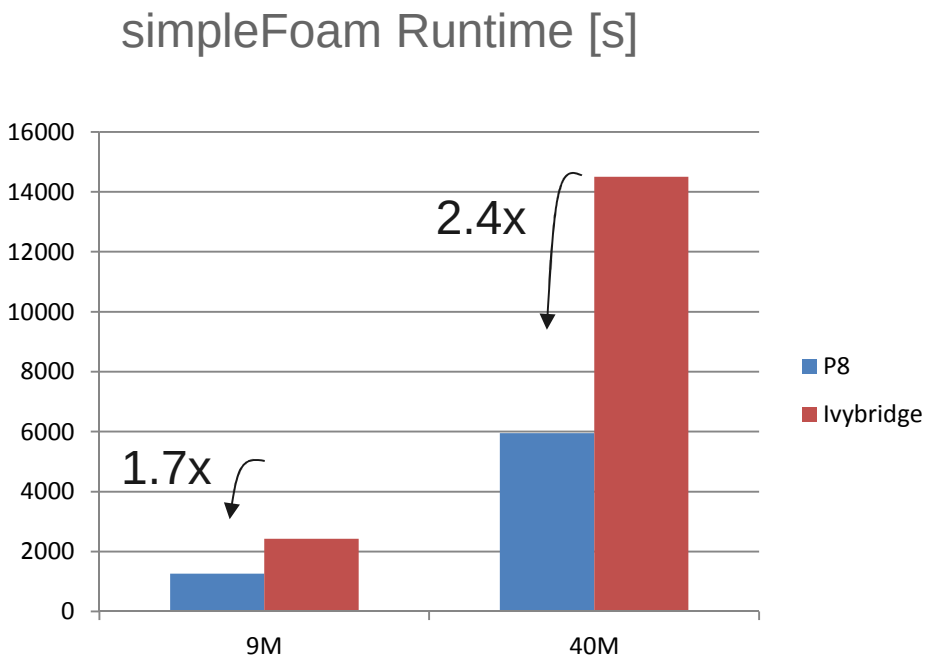
Smaller Cases

- show memory bandwidth dependence

Car Performance



Industrial Testcase

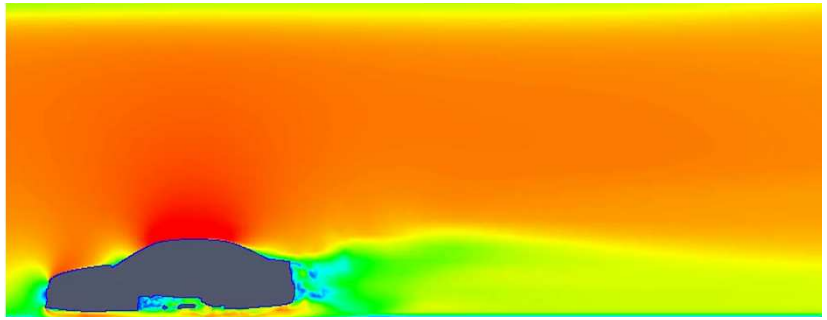


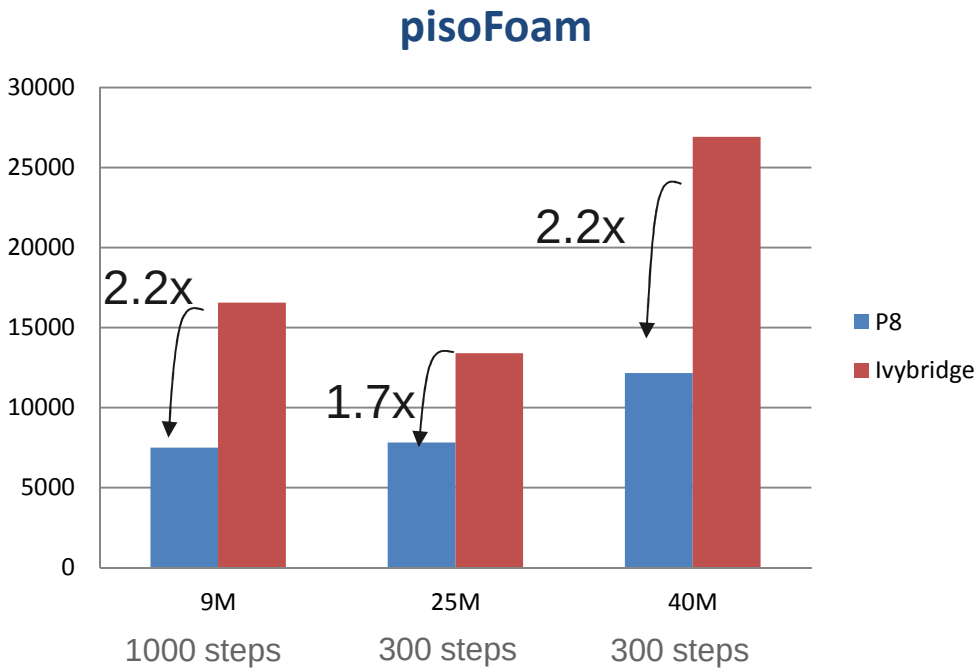
Conclusion

- Similar behaviour than motorBike
  - ➢ speedup 1.5 – 2.5x
  - ➢ larger advantage for P8 for larger cases

Dynamic Simulation

- pisoFoam: Simulates transient behaviour
- Runtimes even longer than simpleFoam
  - ➢ hinders industrial use



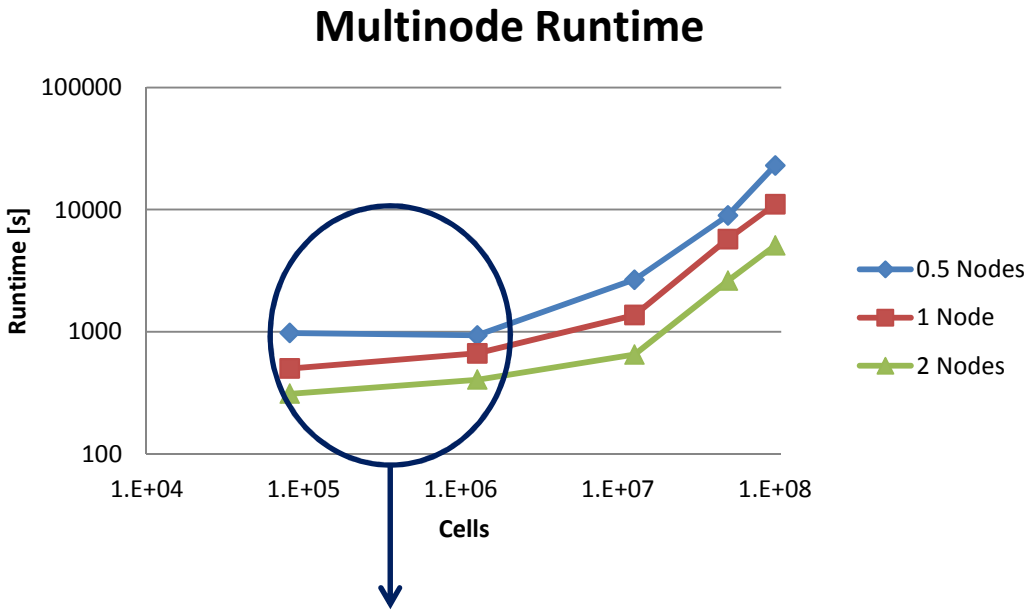


Conclusion

- Similar speedup as simpleFoam

Scaling

- All previous runs on single socket
- Multinode runs with
  - 2 sockets per node
  - 2 threads/socket
  - OpenMPI 1.8.3
  - Infiniband EDR with 100GB/s



Adaptive mesh increases number of cells

Conclusion

- good scaling
  - more nodes to be run in the future

OpenFOAM on GPU

OpenFOAM on GPU

AeroFluidX: GPU plugin for OpenFOAM

- Developed by FluiDyna
- Ported to POWER8 in collaboration with PADC Boeblingen
- Implements solver & matrix assembly
  - MPI communication by host
  - I/O and data preparation by OpenFOAM
  - Based on AmgX from NVIDIA
- 95% of total runtime on GPU → runtime indepedent from host
- Still under development
- First results indicate performance of 1 K80 ~ 1 P8 socket

	P8	K80
mem. bandwidth [GB/s]	230	2x240
GPU memory [GB]	n/a	2x12
CPU-GPU bandwidth [GB/s]	16	

Future Systems

CPU	GPU	Improvement
~constant	1TB/s	2x
n/a	32	1.3x
80		5x

Summary

Summary & Outlook

- POWER8: up to 3x better performance than x86 on typical benchmarks
  - Stationary – simpleFoam
  - Instationary – pisoFoam
    - ➢ Memory bandwidth
    - ➢ Cache sizes
    - ➢ SMT
- OpenFOAM scales well on P8 with up to 2 nodes
  - ➢ more tests with larger cluster ongoing
- GPU acceleration with POWER8 possible
  - ➢ Becomes interesting with future systems
  - ➢ Better concepts possible with Pascal and NVLINK
- Looking into further optimizations with XLC
- Evaluation with new 822LC

Acknowledgments

- Tsuyoshi Kamenoue, IBM Japan: Motorbike bechmark overview
- Peter Kaltstein, IBM R&D Germany: Multinode simulations
- Björn Landmann, Fluidyna: AerofluidX port and GPU simulations

OpenFOAM (V2.4.0) Command list

```
root@s824l01:~/OpenFOAM/OpenFOAM-2.4.0/platforms/linuxPPC64leGccDPOpt/bin# ls
Co                                  combinePatchFaces                   foamToEnsight                       nonNewtonianIcoFoam                 rhoSimpleFoam                       surfaceHookUp
DPMFoam                             compressibleInterDyMFoam            foamToEnsightParts                  objToVTK                            rhoSimplecFoam                      surfaceInertia
LTSInterFoam                        compressibleInterFoam               foamToGMV                           orientFaceZone                      rotateMesh                          surfaceLambdaMuSmooth
LTSReactingFoam                     compressibleMultiphaseInterFoam     foamToStarMesh                      pPrime2                             sammToFoam                          surfaceMeshConvert
LTSReactingParcelFoam               createBaffles                       foamToSurface                       particleTracks                      sample                              surfaceMeshConvertTesting
Lambda2                             createExternalCoupledPatchGeometry  foamToTetDualMesh                   patchAverage                        scalarTransportFoam                 surfaceMeshExport
MPPICFoam                           createPatch                         foamToVTK                           patchIntegrate                      selectCells                         surfaceMeshImport
Mach                                createTurbulenceFields              foamUpgradeCyclics                  patchSummary                        setFields                           surfaceMeshInfo
PDRFoam                             datToFoam                           foamUpgradeFvSolution               pdfPlot                             setSet                              surfaceMeshTriangulate
PDRMesh                             decomposePar                        gambitToFoam                        pimpleDyMFoam                       setsToZones                         surfaceOrient
Pe                                  deformedGeom                        gmshToFoam                          pimpleFoam                          shallowWaterFoam                    surfacePointMerge
Q                                   dnsFoam                             icoFoam                             pisoFoam                            simpleFoam                          surfaceRedistributePar
R                                   driftFluxFoam                       icoUncoupledKinematicParcelDyMFoam  plot3dToFoam                       simpleReactingParcelFoam            surfaceRefineRedGreen
SRFPimpleFoam                       dsmcFieldsCalc                      icoUncoupledKinematicParcelFoam     polyDualMesh                        singleCellMesh                      surfaceSplitByPatch
SRFSimpleFoam                       dsmcFoam                            ideasUnvToFoam                      porousInterFoam                     smapToFoam                          surfaceSplitByTopology
XiFoam                              dsmcInitialise                      insideCells                         porousSimpleFoam                    snappyHexMesh                       surfaceSplitNonManifolds
adiabaticFlameT                     electrostaticFoam                   interDyMFoam                        postChannel                         solidDisplacementFoam               surfaceSubset
adjointShapeOptimizationFoam        engineCompRatio                     interFoam                           potentialFoam                       solidEquilibriumDisplacementFoam    surfaceToPatch
ansysToFoam                         engineFoam                          interMixingFoam                     potentialFreeSurfaceDyMFoam         sonicDyMFoam                        surfaceTransformPoints
applyBoundaryLayer                  engineSwirl                         interPhaseChangeDyMFoam             potentialFreeSurfaceFoam            sonicFoam                           temporalInterpolate
applyWallFunctionBoundaryConditions enstrophy                           interPhaseChangeFoam                probeLocations                      sonicLiquidFoam                     tetgenToFoam
attachMesh                          equilibriumCO                       kivaToFoam                          ptot                                splitCells                          thermoFoam
autoPatch                           equilibriumFlameT                   laplacianFoam                       reactingFoam                        splitMesh                           topoSet
autoRefineMesh                      execFlowFunctionObjects             magneticFoam                        reactingParcelFilmFoam              splitMeshRegions                    transformPoints
blockMesh                           expandDictionary                    mapFields                           reactingParcelFoam                  sprayEngineFoam                     twoLiquidMixingFoam
boundaryFoam                        extrude2DMesh                       mdEquilibrationFoam                 reconstructPar                      sprayFoam                           twoPhaseEulerFoam
boxTurb                             extrudeMesh                         mdFoam                              reconstructParMesh                  star3ToFoam                         uncoupledKinematicParcelFoam
buoyantBoussinesqPimpleFoam         extrudeToRegionMesh                 mdInitialise                        redistributePar                     star4ToFoam                         uprime
buoyantBoussinesqSimpleFoam         faceAgglomerate                     mergeMeshes                         refineHexMesh                       steadyParticleTracks                viewFactorsGen
buoyantPimpleFoam                   financialFoam                       mergeOrSplitBaffles                 refineMesh                          stitchMesh                          vorticity
buoyantSimpleFoam                   fireFoam                            mhdFoam                             refineWallLayer                     streamFunction                      vtkUnstructuredToFoam
cavitatingDyMFoam                   flattenMesh                         mirrorMesh                          refinementLevel                     stressComponents                    wallFunctionTable
cavitatingFoam                      flowType                            mixtureAdiabaticFlameT              removeFaces                         subsetMesh                          wallGradU
cfx4ToFoam                          fluent3DMeshToFoam                  modifyMesh                          renumberMesh                        surfaceAdd                          wallHeatFlux
changeDictionary                    fluentMeshToFoam                    moveDynamicMesh                     rhoCentralDyMFoam                   surfaceAutoPatch                    wallShearStress
checkMesh                           foamCalc                            moveEngineMesh                      rhoCentralFoam                      surfaceBooleanFeatures              wdot
chemFoam                            foamDataToFluent                    moveMesh                            rhoLTSPimpleFoam                    surfaceCheck                        writeCellCentres
chemkinToFoam                       foamDebugSwitches                   mshToFoam                           rhoPimpleDyMFoam                    surfaceClean                        writeMeshObj
chtMultiRegionFoam                  foamFormatConvert                   multiphaseEulerFoam                 rhoPimpleFoam                       surfaceCoarsen                      yPlusLES
chtMultiRegionSimpleFoam            foamHelp                            multiphaseInterDyMFoam              rhoPimplecFoam                      surfaceConvert                      yPlusRAS
coalChemistryFoam                   foamInfoExec                        multiphaseInterFoam                 rhoPorousSimpleFoam                 surfaceFeatureConvert               zipUpMesh
coldEngineFoam                      foamListTimes                       netgenNeutralToFoam                 rhoReactingBuoyantFoam              surfaceFeatureExtract
collapseEdges                       foamMeshToFluent                    noise                               rhoReactingFoam                     surfaceFind
```