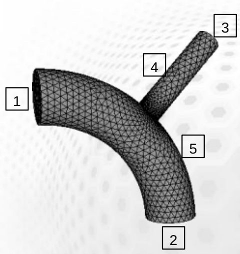
PATC – Elmer FEM

Parallel Introductory Example: Flow through a pipe

CSC-IT Center for Science Ltd.
Elmer Team

Descritpion of the Problem

- Flow through pipe junction
- Boundary conditions:
  1. v_{in} = 1 cm/s
  2. Normal outflow
  3. v_{in} = 1 cm/s
  4. no-slip (u_i = 0 m/s)
  5. Same as 4

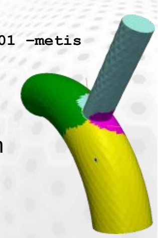


Mesh Partitioning

- Mesh was produced in cm → needs scaling
- Mesh is serial → needs partitioning

```
ElmerGrid 2 2 mesh -scale 0.01 0.01 0.01 -metis
  4 2 -out mesh_part
```

- This uses METIS-library (PartGraphRecursive) to split mesh



Solver input file

```
Header
  Mesh DB "." "mesh_part"
End
Simulation
  Coordinate System  = "Cartesian 3D"
  Simulation Type ="Steady"
  Output Intervals = Integer 1
  Post File = File "parallel_flow.ep"
  Output File = File "parallel_flow.result"
  max output level = Integer 4
End
```

Solver Input File

```
Solver 1
  Equation = "Navier-Stokes"
  Optimize Bandwidth = Logical True
  Linear System Solver = Iterative
  Linear System Iterative Method = "BiCGStab"
  Linear System Max Iterations = 500
  Linear System Convergence Tolerance = 1.0E-05
  Linear System Abort Not Converged = True
  Linear System Preconditioning = "ILU1"
  Linear System Residual Output = 1
  Stabilization Method = Stabilized
  Nonlinear System Convergence Tolerance = Real 1.0E-04
  Nonlinear System Max Iterations = Integer 30
  Nonlinear System Newton After Iterations = Integer 3
  Nonlinear System Newton After Tolerance =  Real 1.0E-03
End
```

Solver input file

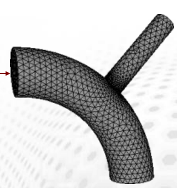
```
Body 1
  Name = "fluid"
  Equation = 1
  Material = 1
  Body Force = 1
  Initial Condition = 1
End
Equation 1
 Active Solvers(1) = 1
 Convection = Computed
End
```

Solver input file

```
Initial Condition 1
  Velocity 1 = Real 0.0
  Velocity 2 = Real 0.0
  Velocity 3 = Real 0.0
  Pressure = Real 0.0
End

Body Force 1
  Flow BodyForce 1 = Real 0.0
  Flow BodyForce 2 = Real 0.0
  Flow BodyForce 3 = Real 0.0
End
```

Solver input file

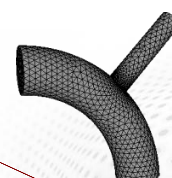
```
Material 1
  Density = Real 1000.0
  Viscosity = Real 1.0
End

Boundary Condition 1
  Name = "largeinflow"
  Target Boundaries  = 1
  Normal-Tangential Velocity = True
  Velocity 1 = Real -0.01
  Velocity 2 = Real 0.0
  Velocity 3 = Real 0.0
End
```



Outward pointing normal

Solver input file

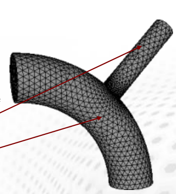
```
Boundary Condition 2
  Name = "largeoutflow"
  Target Boundaries  = 2
  Normal-Tangential Velocity = True
  Velocity 2 = Real 0.0
  Velocity 3 = Real 0.0
End
Boundary Condition 3
  Name = "smallinflow"
  Target Boundaries  = 3
  Normal-Tangential Velocity = True
  Velocity 1 = Real -0.01
  Velocity 2 = Real 0.0
  Velocity 3 = Real 0.0
End
```



Outward pointing normal

Solver input file

```
Boundary Condition 4
  Name = "pipewalls"
  Target Boundaries(2)  = 4 5
  Normal-Tangential Velocity = False
  Velocity 1 = Real 0.0
  Velocity 2 = Real 0.0
  Velocity 3 = Real 0.0
End
```



Parallel run

- Save the sif-file with name *parallel_flow.sif*
- Write the name of the sif-file into ELMERSOLVER_STARTINFO
- Commands for SLURM (eg. vuori.csc.fi):

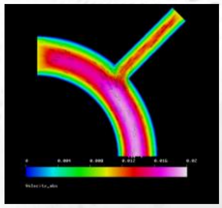
```
module switch PrgEnv-pgi PrgEnv-gnu
module load elmer/latest
sallocate -n 4 --ntasks-per-node=4
--mem-per-cpu=1000 -t 00:10:00 -p
interactive
srun ElmerSolver_mpi
```

- On an usual MPI platform:

```
mpirun -np 4 ElmerSolver_mpi
```

Combining the results

- Change into the mesh directory
- Run ElmerGrid to combine results
  ElmerGrid 15 3 parallel_flow
- Launch ElmerPost
- Load **parallel_flow.ep**



Adding heat transfer

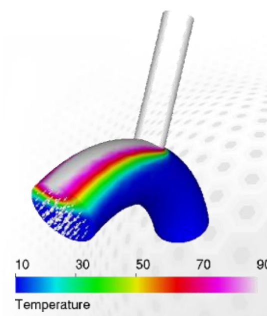
```
Solver 2
  Exec Solver = Always
  Equation = "Heat Equation"
  Procedure = "HeatSolve" "HeatSolver"
  Steady State Convergence Tolerance = Real 3.0E-03
  Nonlinear System Max Iterations = Integer 1
  Nonlinear System Convergence Tolerance = Real 1.0e-6
  Nonlinear System Newton After Iterations = Integer 1
  Nonlinear System Newton After Tolerance = Real 1.0e-2
  Linear System Solver = Iterative
  Linear System Max Iterations = Integer 500
  Linear System Convergence Tolerance = Real 1.0e-6
  Stabilization Method = Stabilized
```

Adding heat transfer

```
  Linear System Use Hypre = Logical True
  Linear System Iterative Method = BoomerAMG
  BoomerAMG Max Levels = Integer 25
  BoomerAMG Coarsen Type = Integer 0
  BoomerAMG Num Functions = Integer 1
  BoomerAMG Relax Type = Integer 3
  BoomerAMG Num Sweeps = Integer 1
  BoomerAMG Interpolation Type = Integer 0
  BoomerAMG Smooth Type = Integer 6
  BoomerAMG Cycle Type = Integer 1
End
```

Adding heat transfer

```
Material 1
  …
  Heat Capacity = Real 1000.0
  Heat Conductivity = Real 1.0
End
Boundary Condition 1
  …
  Temperature = Real 10.0
End
Boundary Condition 3
  …
  Temperature = Real 90.0
End
```



Postprocessing wit Paraview

- Paraview: open source multi-platform data analysis and visualization application by Kitware
- Essentially the GUI to VTK
- Elmer's ResultOutputSolve solver provides VTK unstructured data files: VTU and PVTU (parallel)

Output for ParaView

```
Solver 3
  Equation = "Result Output"
  Procedure = "ResultOutputSolve" "ResultOutputSolver"
  Exec Solver = After Saving
  Output File Name = String "flowtemp"
  Output Format = Vtu
  Show Variables = Logical True
  Scalar Field 1 = Pressure
  Scalar Field 2 = Temperature
  Vector Field 1 = Velocity
End
```

ParaView