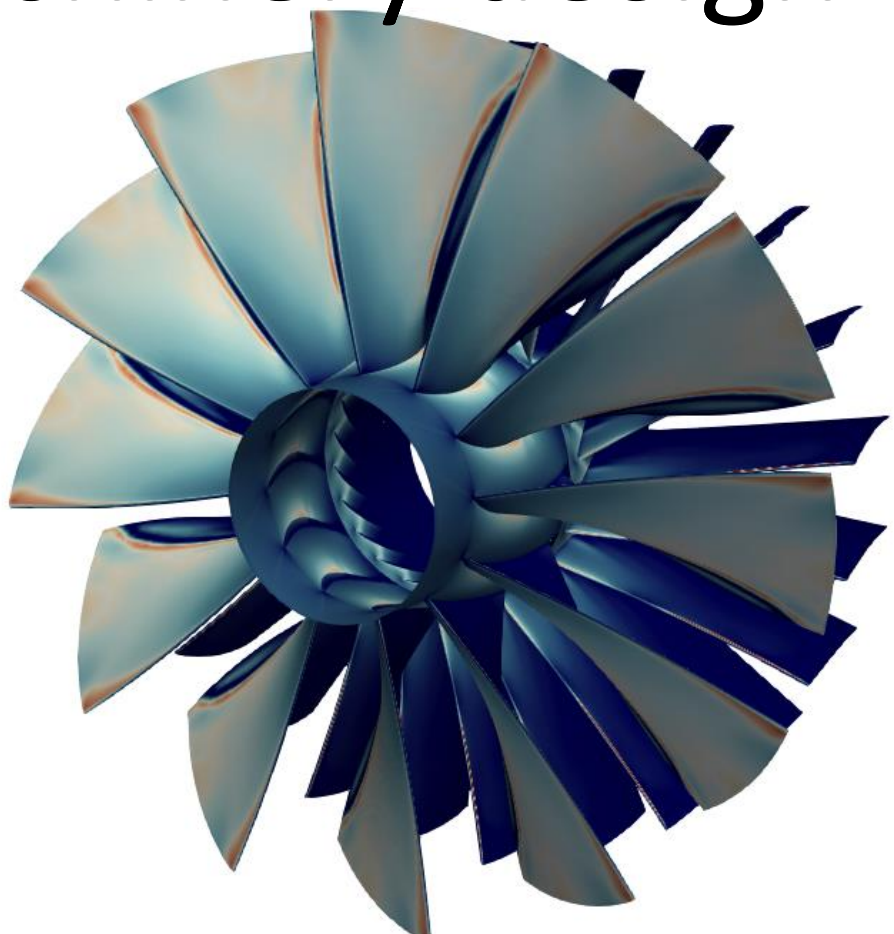
Multall turbomachinery design tutorial



TA: Pietro Colonna
Matteo Pini
Acknowledgements: John Denton, Pedro García Gozalez , Antonio Rubino, Peter Onodi, Pablo Garrido.

TU Delft

Overview

- The **Multall-open** software package is an open-source Fortran code for turbomachinery design based on the 3D, multistage, Navier-Stokes solver Multall.
- User manuals, documents on theoretical background are available on the BS package.
- This document includes information on the installation and use of the software, as well as some examples.
- Additional notes are included below the slides, marked with an asterisk (*)

- Most important take away:

  READ THE DOCUMENTATION

TU Delft

Multall Turbomachinery Design

- The design software consists of 3 programs:
  - Meangen – Meanline design
  - Stagen – Geometry generation and meshing
  - Multall – CFD
- Source code, executables and manuals are available on Brightspace (AE4206 Turbomachinery – Content > Software).

TU Delft

THE SYSTEM CONSISTS OF 3 LINKED PROGRAMS ALL WRITTEN IN FORTRAN

MEANGEN

Performs a 1D calculation to obtain the velocity triangles.
Sets the annulus boundaries.
Generates initial blade shapes and twists them in a prescribed way.
Writes an input file for STAGEN.

STAGEN

Generate the blade shapes.
Stacks them and combines them into stages.
Writes an input file for MULTALL

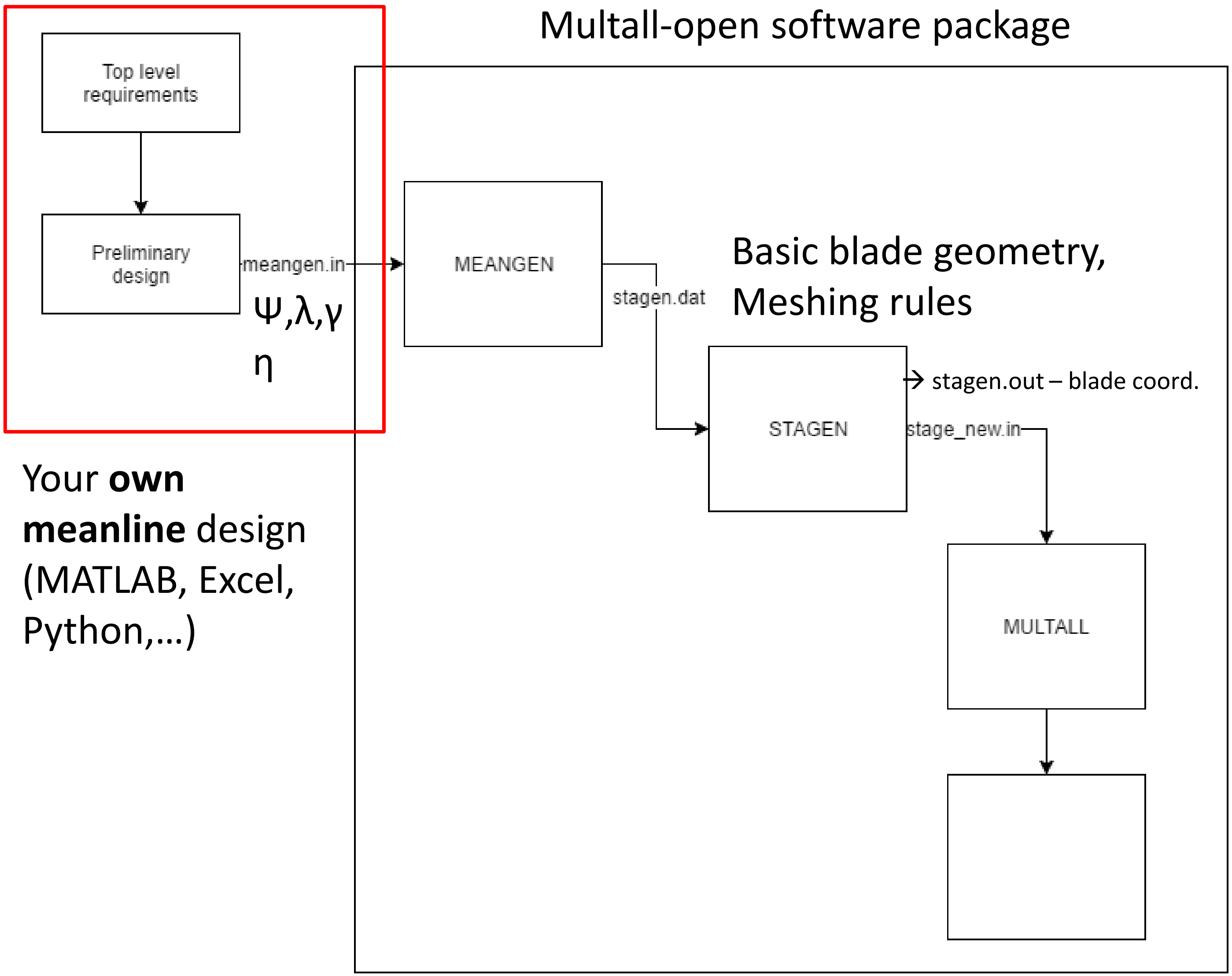
MULTALL

Performs a 3D multistage calculation to predict the detailed flow pattern and overall performance.

TU Delft

Source: Denton: *J. Denton*: MULTALL – AN OPEN-SOURCE, CFD BASED, TURBOMACHINERY DESIGN SYSTEM

Data exchange with meanline design performed elsewhere



Your **own meanline** design (MATLAB, Excel, Python,…)

How to **install** Multall-open software package

Install Multall-open software on Windows-based machines

1. Download the software as FORTRAN (.f) files.
2. Follow the steps of the MinGW-gfortran.pdf (available in this folder) to install the gfortran compiler on your command line.
3. Open the meangen-17.4.f, stagen-18.1.f and multall-open-20.9.f files with a text editor, search for *dev/tty* and comment the line that includes it by adding ***!*** to the line. E.g.:

```
!OPEN(UNIT=5,  FILE= '/dev/tty’)
```

4. Open the commall-open-20.9 file with a text editor and change the value marked in red in the first lines to around 30 (Windows presents a memory limitation, this number should not be higher than a certain limit)*:

```
PARAMETER(ID=30,JD=2500,KD=30,MAXKI=82,NRS=21,I
G1=32,&JG1=1000,KG1=41)
```

Install Multall-open software on Windows-based machines

1. Navigate to the folder in which you have the software (Multall\Source) using cd in the command line.
2. Type the following (in different lines):

```
gfortran meangen-17.4.f -o meangen-17.4.exe
gfortran stagen-18.1.f -o stagen-18.1.exe
gfortran multall-open-20.9.f -o multall-open-20.9.exe
```

3. The executable files should appear in the folder in which the .f files are.
4. If there is no *intype* file, create one and write *N* on it.
5. Check the correct operation of the software using the test cases.

TU Delft

Install Multall-open software on macOS/Linux-based machines (1)

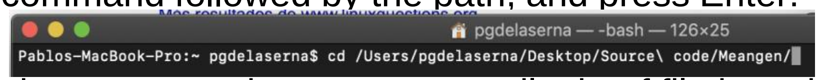
- First of all, you'll need to have installed Fortran on your machine.
  - For macOS:
    1. Install Xcode (available here https://itunes.apple.com/us/app/xcode/id497799835?mt=12)
    2. Install the Xcode command-line tools
       - Open Terminal (Applications > Utilities > Terminal) and type **xcode-select --install** press Enter and follow the dialog boxes.
    3. Download and install Gfortran (from https://github.com/fxcoudert/gfortran-for-macOS/releases).

       Select the version based on your macOS release!
  - For Ubuntu:
    1. Open Terminal and type **sudo apt-get install gfortran** and press Enter.

TU Delft

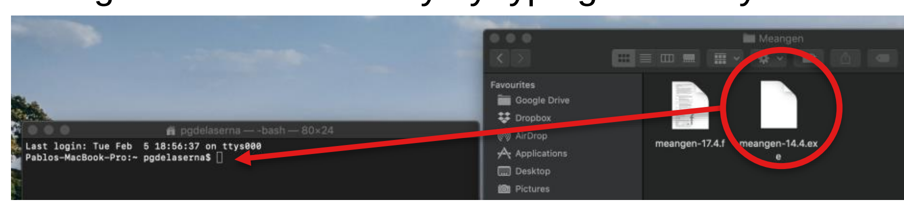
Install Multall-open software on macOS/Linux-based machines (2)

- Now, we need to compile the files (*meangen-17.4.f*, *multall-open-17.5.f* and *stagen-17.2.f*, located in the "*Source code*" folder) and create executables:
    1. Open Terminal.
    2. Navigate to the folder where the .f files that you want to compile are located by using the **cd** command followed by the path, and press Enter:



    3. Once you are in the correct path, you can compile the .f file by using the following command: **gfortran -o meangen-17.4.exe meangen-17.4.f**

    4. After a few seconds, a new file (.exe) will appear in the folder where the .f file was located.
    5. In order to execute this file, **you need to run it from the command window (Terminal)**. You can do this by dragging and dropping the .exe file to the Terminal window and pressing Enter or alternately by typing manually the path of the executable:



- The steps followed can be repeated for the other files (*multall-open-17.5.f* and *stagen-17.2.f*), where the Folder and File names in steps 2 and 3 will need to be changed accordingly,

TU Delft

How to **run** Multall-open software package

Meangen

1. Run meangen-17.4.exe by double-clicking on the file. The following message should appear.

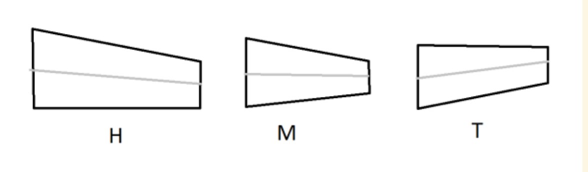
```
                WELCOME TO MEANGEN

THIS IS AN INTERACTIVE PROGRAM FOR THE ONE-DIMENSIONAL
              DESIGN OF AXIAL TURBOMACHINES.

ANSWER THE QUESTIONS AS THEY APPEAR ON THE SCREEN
AND THE PROGRAM WILL WRITE A DATA SET FOR THE
BLADE GEOMETRY PROGRAM "STAGEN" WHICH IN TURN WILL
GENERATE A 3D DATASET FOR "MULTALL-OPEN".

*******************************************************
*
*******************************************************
*
 INPUT FROM SCREEN OR FILE ?
 ANSWER "S"  or  "F" .
```

2. If answer S, the software will ask you to provide all the design parameters in the command line.
3. If answer F, type the name of your own generated .in file, a text document with all the required design parameters (see next slide, recommended method).

Meangen - Input example

```
C                        TURBO_TYP,"C" FOR A COMPRESSOR,"T" FOR A TURBINE
AXI                      FLO_TYP FOR AXIAL OR MIXED FLOW MACHINE
   287.500     1.400     GAS PROPERTOES, RGAS, GAMMA
     1.000   300.000     POIN,  TOIN
    3                    NUMBER OF STAGES IN THE MACHINE
M                        CHOICE OF DESIGN POINT RADIUS, HUB, MID or TIP
    5000.000             ROTATION SPEED, RPM
      50.000             MASS FLOW RATE, FLOWIN.
A                        INTYPE, TO CHOOSE THE METHOD OF DEFINING THE VELOCITY TRIANGLES
  0.700  0.600  0.400    REACTION, FLOW COEFF., LOADING COEFF.
A                        RADTYPE, TO CHOOSE THE DESIGN POINT RADIUS
       0.500             THE DESIGN POINT RADIUS
       0.050       0.040 BLADE AXIAL CHORDS IN METRES.
       0.250       0.500 ROW GAP  AND STAGE GAP
   0.00000   0.02000     BLOCKAGE FACTORS, FBLOCK_LE,  FBLOCK_TE
       0.900             GUESS OF THE STAGE ISENTROPIC EFFICIENCY
   5.000   5.000         ESTIMATE OF THE FIRST AND SECOND ROW DEVIATION ANGLES
  -2.000  -2.000         FIRST AND SECOND ROW INCIDENCE ANGLES
   1.00000               BLADE TWIST OPTION, FRAC_TWIST
n                        BLADE ROTATION OPTION , Y or N
  88.000  92.000         QO ANGLES AT LE  AND TE OF ROW 1
  92.000  88.000         QO ANGLES AT LE  AND TE OF ROW 2
n                        DO YOU WANT TO CHANGE THE ANGLES FOR T
y                        IFSAME_ALL, SET = "Y" TO REPEAT THE LAST STAGE INPUT TYPE AND VELOCITY TRIANGLES, SET = "C" TO CHANGE INPUT TYPE.
   0.02000   0.04000     BLOCKAGE FACTORS, FBLOCK_LE,  FBLOCK_TE
n                        DO YOU WANT TO CHANGE THE ANGLES FOR THIS STAGE ? "Y" or "N"
y                        IFSAME_ALL, SET = "Y" TO REPEAT THE LAST STAGE INPUT TYPE AND VELOCITY TRIANGLES, SET = "C" TO CHANGE INPUT TYPE.
   0.04000   0.06000     BLOCKAGE FACTORS, FBLOCK_LE,  FBLOCK_TE
n                        DO YOU WANT TO CHANGE THE ANGLES FOR THIS STAGE ? "Y" or "N"
Y                        IS OUTPUT REQUESTED FOR ALL BLADE ROWS ?
Y    ROTOR No.   1 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    STATOR No.  1 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    ROTOR No.   2 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    STATOR No.  2 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    ROTOR No.   3 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    STATOR No.  3 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
```

} type of machine („MIX” also for radial)



*This also fixes your design radius: If you choose “T” the duty coefficients you introduce are implemented at the tip!

You can also work with the real specific work

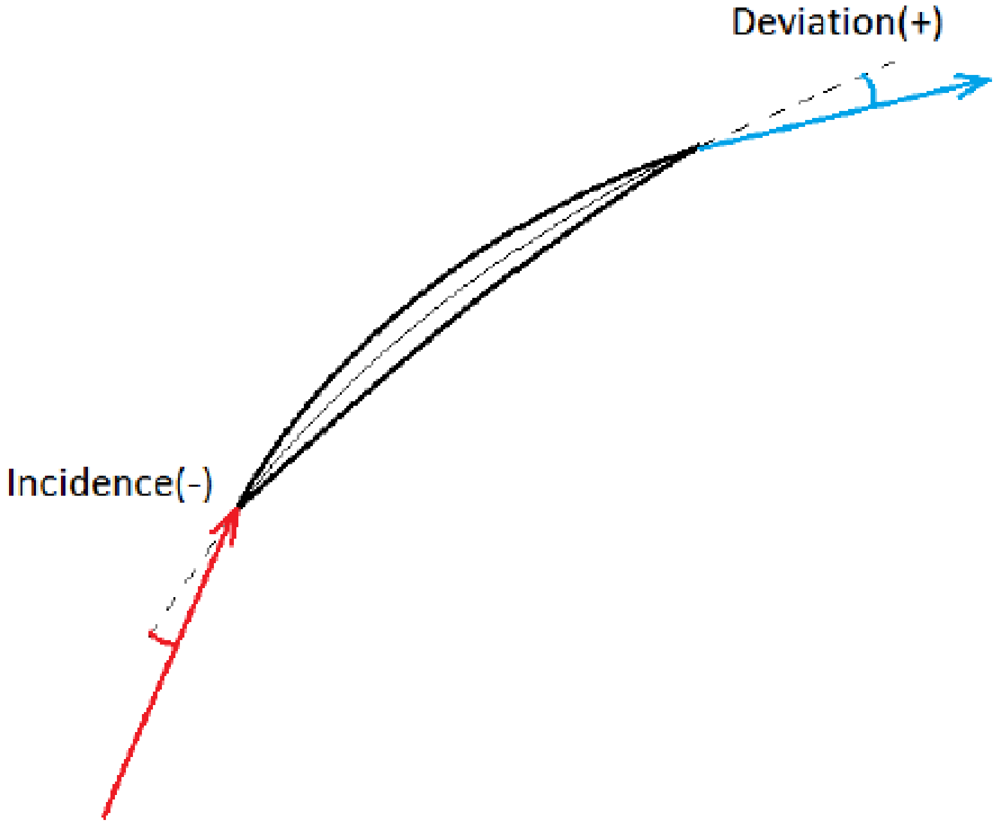
```
INPUT "FRAC_TWIST", THE FRACTION OF FREE-VORTEX TWIST  THAT YOU WANT TO USE ON THIS STAGE.
FRAC_TWIST = 1.0  GIVES FULL FREE-VORTEX TWIST.
FRAC_TWIST = 0.0  GIVES NO TWIST, SO THE BLADE ANGLES ARE THE SAME AT ALL SPANWISE POSITIONS.
VALUES OF FRAC_TWIST GREATER THAN 1.0 OR LESS THAN 0.0 CAN ALSO BE USED.
THE CURRENT VALUE IS  1.
```

Dispacement body (boundary layer)

If n, properties can be specified For each blade row separately

TUDelft

Meangen - WARNING



Stagen

- Usage: double-click on stagen-18.1.exe
  - Input: **stagen.dat** (output from Meangen).
  - Output: `stage_new.dat` (and `stage_old.dat`)
  - The *intype* file has to be set up to *N* for the use of stage_new.dat.
  - `Stagen.out` contains geometrical information

Stagen: Fine Tunning

- Meangen assumes a lot of usual parameters for the design of the machine. They can be adjusted by recompiling the code or directly in **stagen.dat**

```
287.5000      1.4000 GAS CONSTANT, GAMMA
      37          37       IM, KM
  1.2500     20.0000  FPRAT,   FPMAX
  1.2500     20.0000  FRRAT,   FRMAX
       0               IFDEFAULTS
       2           3       NOWS, N SECTIONS
 1.000                 SCALING FACTOR
```

Change the size of the mesh (IM, KM)

Controls the spacing of the mesh (do not change) (FPRAT, FPMAX, FRRAT, FRMAX)

This tells Stagen.exe how to read the file, to define more sections Meangen must be modified (NOWS, N SECTIONS)

```
********************STARTING DATA FOR A NEW BLADE ROW**************************
 BLADE ROW NUMBER =        1
 BLADE ROW TYPE   =     R
  25    70    15      NPOINTS_UP, NPOINTS_ON, NPOINTS_DWN
       0.0000          0.5000       FRACTION AXIAL CHORD    RELATIVE GRID SPACING
```

Change the size of the mesh in the meridional direction (Cells before Blade, Cells on Blade, cells after blade)

The following lines define the meridional mesh spacing (Do not change)

```
        12                  NUMBER OF BLADES IN ROW.
```

Number of Blades in the row (This comes from Meangen applying the Zweifel coefficient, do not change it randomly)

```
    0     0     1     1     1     1    0.00000    3342.00    TIP GAPS, WALL ROTNS and RPMHUB
```

Tip gaps and another features can be added! (Not required for the Project, check out documentation if curious)

TUDelft

Stagen: Fine Tunning

Defines the section in the meridional direction

```
40      0.5000    0.2500   -34.8240   28.2674          XCUP, XCDWN, BETUP, BETDWN
41    BLANK LINE
42      9                NUMBER OF POINTS ON THE STREAM SURFACE.
43     -0.300001   -0.157304   -0.016128    0.312621    0.349550    0.387025    0.663526    0.819200    0.975000
44      0.269549    0.268659    0.275405    0.338137    0.340547    0.341161    0.352513    0.353264    0.351998
45     -0.016128    0.312621    0.275405    0.338137  LEADING AND TRAILING EDGE COORDINATES
46      1.0000    0.0000     0.0000     0.0000     0.000 FCENTROID, FTANG, FLEAN, FSWEEP, FAXIAL
```

Line 40: Caution, only change after reading documentation

- XCUP: Number of upstream *local chord* length that the computational domain is extended.
- XCDWN: The same, but downstream of the Blade row
- BETUP, BETDWN: Upstream and downstream angles of the grid, the same as the blade. Do **not** change

Line 42:

- Must match the number of points in the following two lines

Line 43-44:

- Define the Surface where the Blade section is generated (Next slide)

Line 45: Do NOT change

- LE and TE coordinates, must be points in the previous list

Line 46: Check out documentation

- Multall suports more complex Blade stacking! (Not required for the project)

Stagen: Fine Tunning

Defines the section in the meridional direction

```
40      0.5000    0.2500  -34.8240   28.2674          XCUP, XCDWN, BETUP, BETDWN
41    BLANK LINE
42      9                NUMBER OF POINTS ON THE STREAM SURFACE.
43     -0.300001   -0.157304   -0.016128    0.312621    0.349550    0.387025    0.663526    0.819200    0.975000
44      0.269549    0.268659    0.275405    0.338137    0.340547    0.341161    0.352513    0.353264    0.351998
45     -0.016128    0.312621    0.275405    0.338137  LEADING AND TRAILING EDGE COORDINATES
46      1.0000    0.0000     0.0000     0.0000     0.000 FCENTROID, FTANG, FLEAN, FSWEEP, FAXIAL
```

Line 43: x coordinate

- The first and last points must be outside the computational domain.
- Multall generates This taking into account the Blade LE and TE Q angles specified.
- Multall generates:
  - 2 Points before the first row LE
  - First row LE
  - First Row TE
  - Mixing plane position
  - Second Row LE
  - Second Row TE
  - 2 Points after the second row

$$r(x) = \sum_{i=i}^{4} R_i L_{4,i}(x)$$

$$L_{4,i} = \prod_{\substack{j=1 \\ j \neq i}}^{4} \frac{(x - X_j)}{(X_i - X_j)}$$

Line 43: r coordinate

- Multall generates the gas path taking into account the (expected) changes in fluid properties as well as the (user supplied) blockage factor.
- Points bwtween the control ones are obtained by using 3 order polynomical interpolation.
- Multall guessed values are good and massflow should be matched by changing $\dot{m}$ and blockage in meangen.in, this is an (optional) opportunity to change the shape, **not** intentended as the main workflow line

TU Delft

Multall

Settings related to turbulence model, fluid model, CFL number and other simulation parameters may be changed by modifying stagen_new.dat

- Cp and gamma based on flow properties.
- CFL number for CFD simulations.
- NSTEPS_MAX determines the maximum number of iterations.
- CONLIM determines the convergence of the simulation (default value of 0.001%).

TU Delft

Multall-Convergence

- Maximum iteration, convergence limit in `stage_new.dat`:

```
    NSTEPS_MAX, CONLIM
          9000   0.001000
```

%, This is equivalent to 0.00001 in CFX

- This file also contains KM and IM, DO NOT change them here, the mesh has been already generated.
- To stop a running simulation prematurely (and still save the results):
  - Open „stopit" file
  - Change 0 to 1
- To generate intermediate results (They do not have to represent physical data, it is a numerical intermediate state):

```
5   TIME STEPS FOR PRINTOUT
     9000        9000        9000        9000        9000
```

TU Delft

Multall - run

- To run the solver in cmd and save the results to file:

  `>> multall-open-20.9.exe < stage_new.dat > results.txt` (This is one line!)

  – It Will writte in results.txt while running: Useful to control the convergence of your run.

  – Once it is finish, results.txt contains data as efficiency and mass flow of the machine

Plotting programs

- They have been lost to time. You are very free to check out the documentation, find them, and link them back to the executable.
- Transition towards ParaView: A lot of plotting can be done in PostPy (Thge most interesting metrics. If something more is required contact email, or implement it yourself!)

How to **post-process** Multall-open software package

Postprocessing – PostPy/ParaView

- Refer to PostPy documentation
- **ParaView/Tecplot** compatible files

Design **Examples**

(Credit to the correspondant author)

Design example 1
Design of first stage of HPC for NASA E3 engine

TU Delft

Design specifications

Specification	Value
Inlet total pressure (Pa)	276000
Inlet total temperature (K)	340
Overall pressure ratio	14
Mass flow rate (kg/s)	88.23
Rotational speed (rpm)	13177
1st stage inlet corrected tip speed (m/s)	379.5
IGV	No

Steps

The first step is to modify the inputs in the meangen.in file:

1. Write the parameters that are already known from the design specifications (e.g. inlet total temperature and pressure).
2. Select the number of stages considering constant work per stage. *Hint: Consider reference values for maximum stage pressure ratio (around 1.6) and maximum flow deflection to avoid separation.*
3. Select load coefficient, flow coefficient and degree of reaction based on design criteria.
4. Compute design point radius based on continuity equation.
5. Select axial chords based on selected solidity.
6. Provide estimates for row and stage gaps, and of blockage factors due to boundary layer.
7. Provide an estimated of the stage isentropic efficiency based on empirical data (e.g. Smith chart) or calculations (e.g. loss models).
8. Provide estimates of deviation and incidence angles based on models (e.g. Traupel, Greitzer, Ainley & Mathieson…).
9. Provide taper if desired (QO angles).
10. First estimate of blade max thickness and location (can be changed in Stagen).

Steps

```
C                         TURBO_TYP,"C" FOR A COMPRESSOR,"T" FOR A TURBINE
AXI                       FLO_TYP FOR AXIAL OR MIXED FLOW MACHINE
   287.000     1.400      GAS PROPERTOES, RGAS, GAMMA
   2.76   340.000      POIN,  TOIN
   1                     NUMBER OF STAGES IN THE MACHINE
M                         CHOICE OF DESIGN POINT RADIUS, HUB, MID or TIP
   13177.000              ROTATION SPEED, RPM
   88.23              MASS FLOW RATE, FLOWIN.
A                         INTYPE, TO CHOOSE THE METHOD OF DEFINING THE VELOCITY TRIANGLES
  0.82295  0.5  0.3541     REACTION, FLOW COEFF., LOADING COEFF.
A                         RADTYPE, TO CHOOSE THE DESIGN POINT RADIUS
       0.2551378           THE DESIGN POINT RADIUS
       0.0309   0.0481 BLADE AXIAL CHORDS IN METRES.
       0.2500       0.500 ROW GAP  AND STAGE GAP (fractions)
   0.00000   0.77000      BLOCKAGE FACTORS, FBLOCK_LE,  FBLOCK_TE
       0.9124              GUESS OF THE STAGE ISENTROPIC EFFICIENCY
   6.369614   9.24129          ESTIMATE OF THE FIRST AND SECOND ROW DEVIATION ANGLES
   0.3545  0.00           FIRST AND SECOND ROW INCIDENCE ANGLES
   0.2188                 BLADE TWIST OPTION, FRAC_TWIST (1 is free vortex, 0 is without twist)
n                         BLADE ROTATION OPTION , Y or N
  90  90         QO ANGLES AT LE  AND TE OF ROW 1
  90  90         QO ANGLES AT LE  AND TE OF ROW 2
n                         DO YOU WANT TO CHANGE THE ANGLES FOR THIS STAGE ? "Y" or "N"
y                         IFSAME_ALL, SET = "Y" TO REPEAT THE LAST STAGE INPUT TYPE AND VELOCITY TRIANGLES, SET = "C" TO CHANGE INPUT TYPE.
Y                         IS OUTPUT REQUESTED FOR ALL BLADE ROWS ?
N    STATOR No.  1 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
  0.0500  0.500          MAX THICKNESS AND ITS LOCATION FOR STATOR  1 SECTION No.  1
  0.0500  0.500          MAX THICKNESS AND ITS LOCATION FOR STATOR  1 SECTION No.  2
  0.0500  0.500          MAX THICKNESS AND ITS LOCATION FOR STATOR  1 SECTION No.  3
Y    ROTOR No.   1 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    STATOR No.  2 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    ROTOR No.   2 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    STATOR No.  3 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    ROTOR No.   3 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
```

Steps

Run meangen-17.4.exe, selecting input from file (write F in cmd).
Outputs:

- meangen.out: slight modifications from meangen.in (e.g. number of decimals), check if the values are roughly the same).
- stagen.dat: inputs for Stagen.

Modify stagen.dat (make changes on local blade profile characteristics based on fluid-dynamic understanding of turbomachinery behaviour, loss models, and the indications provided in the tutorial by Denton in the appendix of this ppt):

1. Change IM, KM to a value lower than the limit defined in commall.
2. Change NPOINTS to perform grid convergence study (the higher the number, the higher the accuracy of CFD results, but the higher the computational time).
   See the tutorial by Denton in the appendix for more information.
3. Number of blades can also be modified if the solidity estimate does not lead to satisfactory results.
4. Change detailed profile characteristics (see next slides).

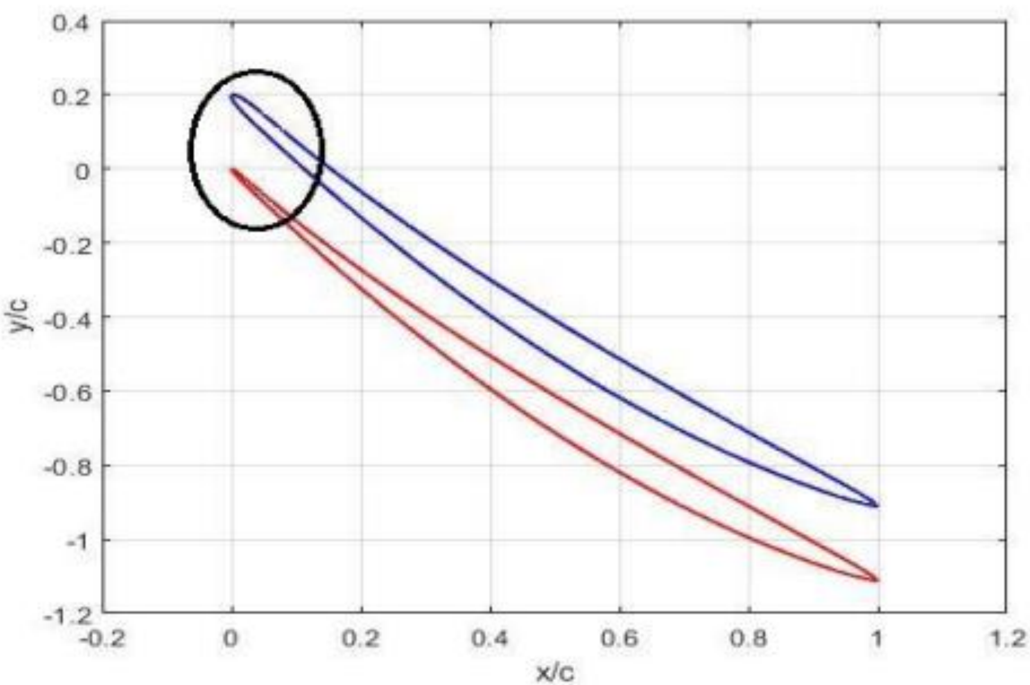
Steps

```
********************** ROW NUMBER 1 ***********************************
**********STARTING NEW BLADE SECTION, SECTION NUMBER 2 ***************
*************************BLANK LINE****************************************
   1                     INTYPE- TYPE OF BLADE GEOMETRY INPUT
   6  200    4           NPIN, NXPTS, NSMOOTH
     0.0000    -63.0805 BLADE CENTRE LINE ANGLES
     0.1139    -60.6981 BLADE CENTRE LINE ANGLES
     0.2903    -57.9017 BLADE CENTRE LINE ANGLES
     0.5018    -54.5904 BLADE CENTRE LINE ANGLES
     0.7399    -50.6373 BLADE CENTRE LINE ANGLES
     1.0000    -45.8865 BLADE CENTRE LINE ANGLES
   0.0200    0.0100    0.0750    0.4000    0.0200    0.0100    2.0000     BLADE PROFILE SPECIFICATION
   1.0000    0.0000    1.0000                        FCHORD, FPERP, FTKSCALE
   0.0000    0.5000    0.5000                        ROTN,XROT,YROT
   0.5000    0.2500  -63.0805  -45.8865              XCUP, XCDWN, BETUP, BETDWN
 BLANK LINE
   9                NUMBER OF POINTS ON THE STREAM SURFACE.
  -0.030900   -0.015520    0.000325    0.030668    0.034610    0.037529    0.054879    0.105994
   0.110775
   0.255138    0.255138    0.255138    0.255138    0.255138    0.255138    0.255138    0.255138
   0.255138
   0.000325    0.030668    0.255138    0.255138  LEADING AND TRAILING EDGE COORDINATES
   1.0000    0.0000    0.0000    0.0000    0.000 FCENTROID, FTANG, FLEAN, FSWEEP, FAXIAL
   1.0000    0.0000                              FSCALE, FCONST
```

Steps

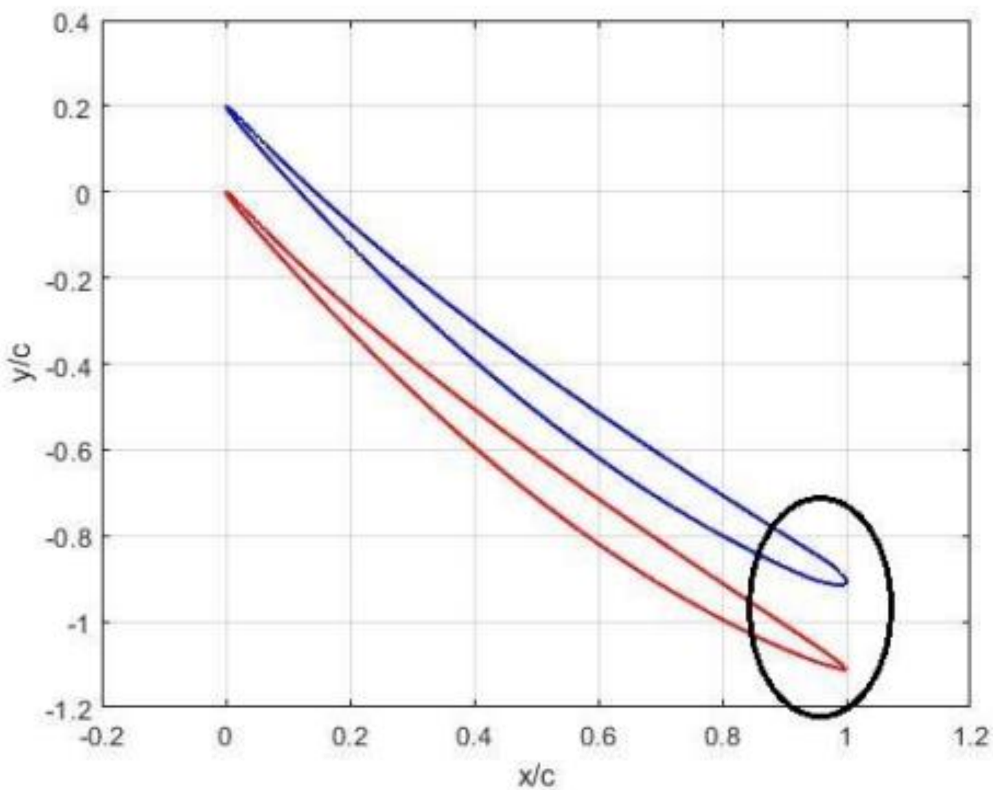
```
*********************** ROW NUMBER 1 ************************************
**********STARTING NEW BLADE SECTION, SECTION NUMBER 1 **************
*********************
   1
   6  200    4
     0.0000    -58.60
     0.1139    -55.49
     0.2903    -51.80
     0.5018    -47.4035 BLADE CENTRE LINE
     0.7399    -42.1160 BLADE CENTRE LINE
     1.0000    -35.7737 BLADE CENTRE LINE
   0.0100    0.0100     0.070     0.6000
   1.0000    0.0000     1.0000
   0.0000    0.5000     0.5000
   0.5000    0.2500   -58.6002   -35.7737
 BLANK LINE
   9                NUMBER OF POINTS ON TH
 -0.044000   -0.022003   -0.000342    0.
  0.216200
  0.167948    0.167913    0.169293    0.
  0.181694
 -0.000342    0.044138    0.169293    0.
  1.0000    0.0000     0.0000     0.0000
  1.0000    0.0000
```

$$\frac{\Delta s}{R} = -\ln\left\{1-\frac{t_{LE}}{b\cos\beta_1}\left[1.28(M_{1R}-1)+0.96(M_{1R}-1)^2\right]\right\}$$



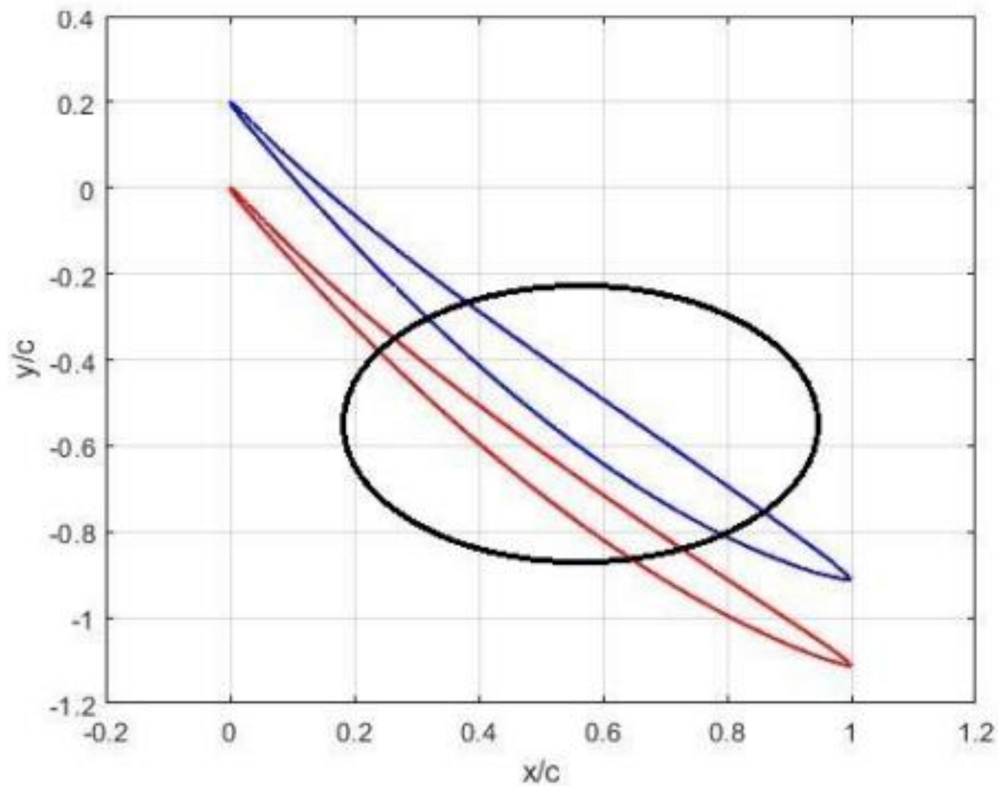
Steps

```
*********************** ROW NUMBER 1 ************************************
**********STARTING NEW BLADE SECTION, SECTION NUMBER 1 **************
*************************BLANK LINE****************************************
  1                       INTYPE- TYPE OF BLADE GEOMETRY INPUT
  6  200    4             NPIN, NXPTS, NSMOC
    0.0000     -58.6002 BLADE CENTRE LINE
    0.1139     -55.4948 BLADE CENTRE LINE
    0.2903     -51.8088 BLADE CENTRE LINE
    0.5018     -47.4035 BLADE CENTRE LINE
    0.7399     -42.1160 BLADE CENTRE LINE
    1.0000     -35.7737 BLADE CENTRE LINE
  0.0100    0.0100    0.070     0.6000
  1.0000    0.0000    1.0000
  0.0000    0.5000    0.5000
  0.5000    0.2500  -58.6002  -35.7737
 BLANK LINE
  9                  NUMBER OF POINTS ON THE
 -0.044000   -0.022003   -0.000342    0.(
  0.216200
  0.167948    0.167913    0.169293    0.1
  0.181694
 -0.000342    0.044138    0.169293    0.1
  1.0000    0.0000    0.0000    0.0000
  1.0000    0.0000
```



Steps

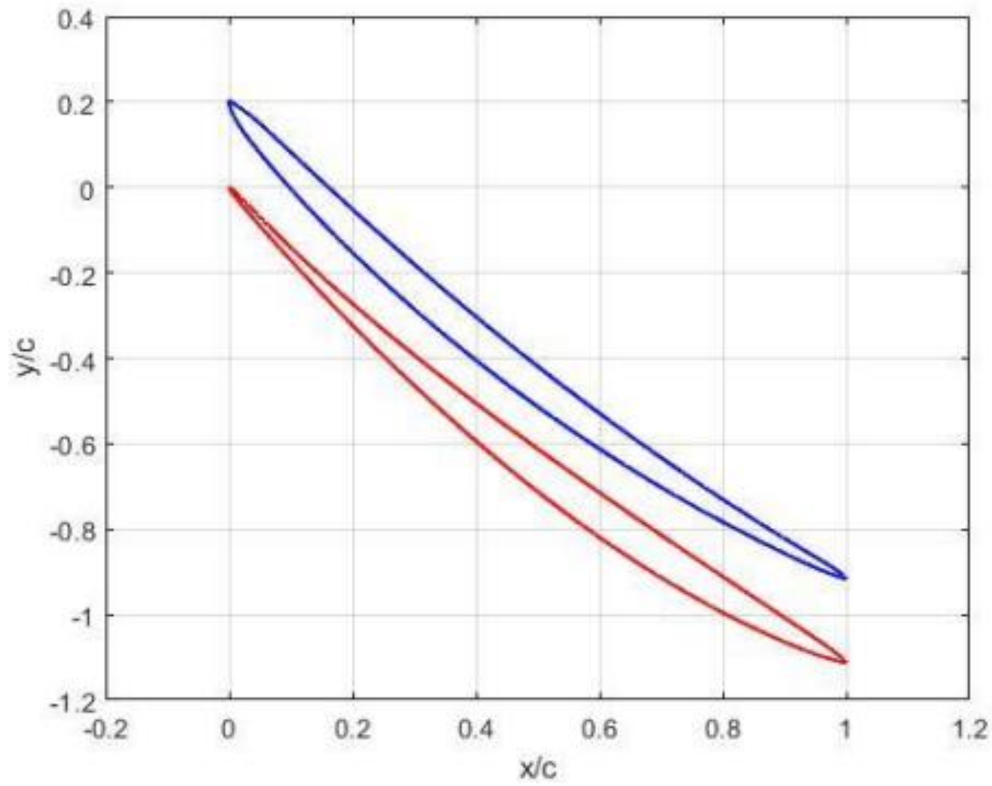
```
*********************** ROW NUMBER 1 ************************************
*********STARTING NEW BLADE SECTION, SECTION NUMBER 1 **************
*************************BLANK LINE**************************************
   1                     INTYPE- TYPE OF BLADE GEOMETRY INPUT
   6  200    4           NPIN, NXPTS, NSMOC
     0.0000     -58.6002 BLADE CENTRE LINE
     0.1139     -55.4948 BLADE CENTRE LINE
     0.2903     -51.8088 BLADE CENTRE LINE
     0.5018     -47.4035 BLADE CENTRE LINE
     0.7399     -42.1160 BLADE CENTRE LINE
     1.0000     -35.7737 BLADE CENTRE LINE
   0.0100     0.0100     0.070     0.6000
   1.0000     0.0000     1.0000
   0.0000     0.5000     0.5000
   0.5000     0.2500   -58.6002   -35.7737
  BLANK LINE
   9                NUMBER OF POINTS ON THE
  -0.044000    -0.022003    -0.000342     0.0
   0.216200
   0.167948     0.167913     0.169293     0.1
   0.181694
  -0.000342     0.044138     0.169293     0.1
   1.0000     0.0000     0.0000     0.0000
   1.0000     0.0000
```



TU Delft

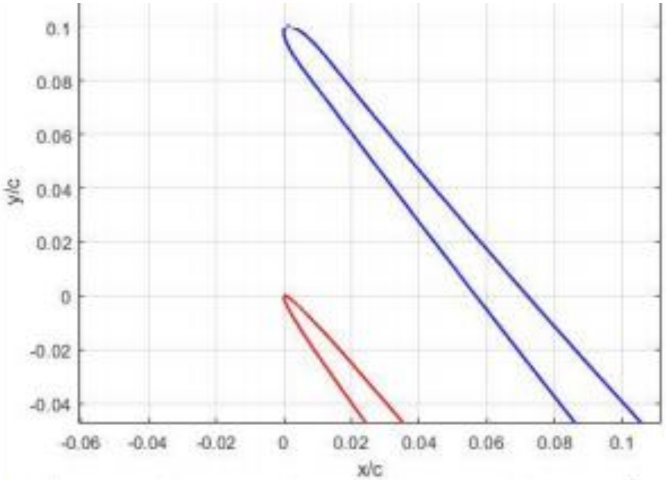
Steps

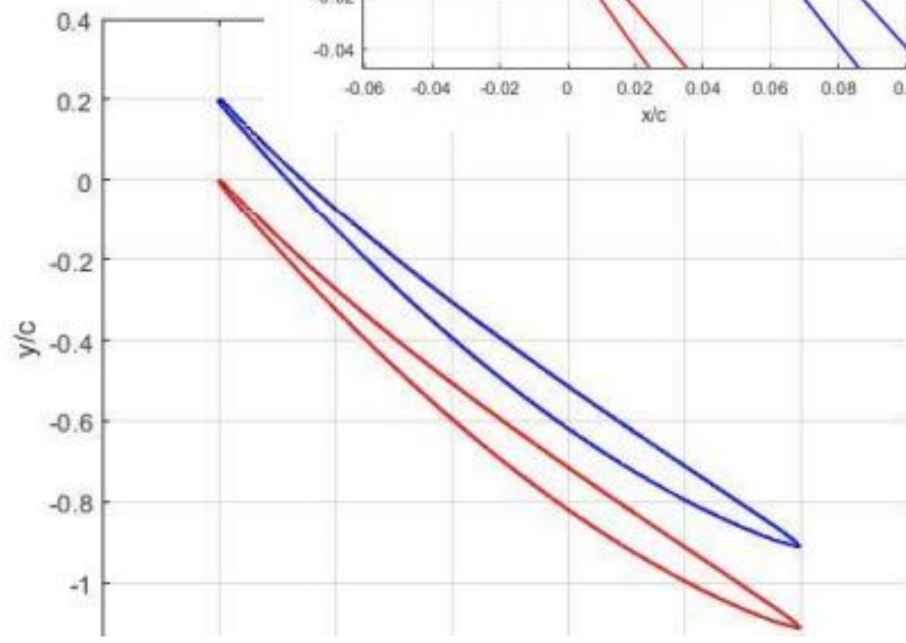
```
********************** ROW NUMBER 1 ***********************************
*********STARTING NEW BLADE SECTION, SECTION NUMBER 1 *************
************************BLANK LINE****************************************
   1                    INTYPE- TYPE OF BLADE GEOMETRY INPUT
   6  200    4          NPIN, NXPTS, NSMOC
     0.0000     -58.6002 BLADE CENTRE LINE
     0.1139     -55.4948 BLADE CENTRE LINE
     0.2903     -51.8088 BLADE CENTRE LINE
     0.5018     -47.4035 BLADE CENTRE LINE
     0.7399     -42.1160 BLADE CENTRE LINE
     1.0000     -35.7737 BLADE CENTRE LINE
   0.0100    0.0100    0.070    0.6000
   1.0000    0.0000    1.0000
   0.0000    0.5000    0.5000
   0.5000    0.2500  -58.6002  -35.7737
 BLANK LINE
   9                NUMBER OF POINTS ON THE
  -0.044000   -0.022003   -0.000342    0.0
   0.216200
   0.167948    0.167913    0.169293    0.1
   0.181694
  -0.000342    0.044138    0.169293    0.1
   1.0000    0.0000    0.0000    0.0000
   1.0000    0.0000
```



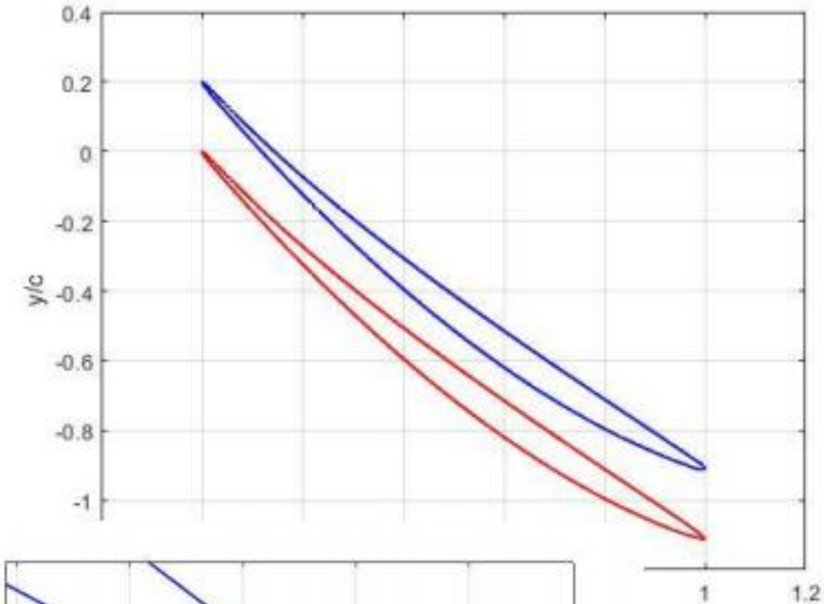
Steps

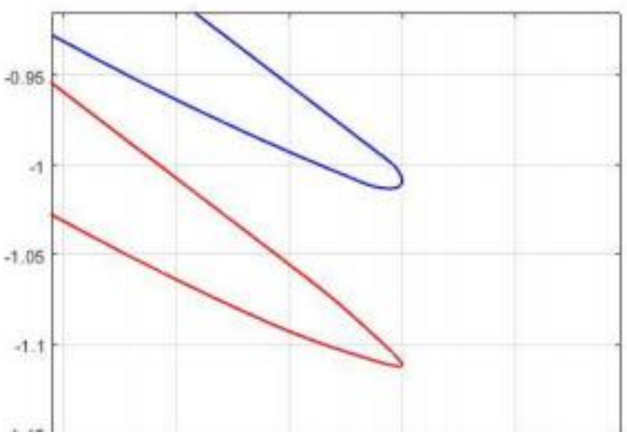
```
********************** ROW NUMBER 1 ***************************
*********STARTING NEW BLADE SECTION, SECTION NUMBER 1 ************
***********************BLANK LINE******************************
   1                    INTYPE- TYPE OF BLADE GEOMETRY INPUT
   6  200    4          NPIN, NXPTS, NSMOOTH
     0.0000    -58.6002 BLADE CENTRE LINE ANGLES
     0.1139    -55.4948 BLADE CENTRE LINE ANGLES
     0.2903    -51.8088 BLADE CENTRE LINE ANGLES
     0.5018    -47.4035 BLADE CENTRE LINE ANGLES
     0.7399    -42.1160 BLADE CENTRE LINE ANGLES
     1.0000    -35.7737 BLADE CENTRE LINE ANGLES
   0.0100    0.0100    0.070    0.6000    0.0500
   1.0000    0.0000    1.0000                      F
   0.0000    0.5000    0.5000                      R
   0.5000    0.2500  -58.6002  -35.7737            X
 BLANK LINE
   9                NUMBER OF POINTS ON THE STREAM
 -0.044000   -0.022003   -0.000342    0.044138
   0.216200
   0.167948    0.167913    0.169293    0.180201
   0.181694
 -0.000342    0.044138    0.169293    0.180201   L
   1.0000    0.0000    0.0000    0.0000    0.000 F
   1.0000    0.0000                              F
```





Steps






```
:*************************
MBER 1 **************
:****************************
:OMETRY INPUT

   0.0500    2.0000      BLADE PROFILE SPECIFICATION
  FCHORD, FPERP, FTKSCALE
  ROTN,XROT,YROT
  XCUP, XCDWN, BETUP, BETDWN

TS ON THE STREAM SURFACE.
42    0.044138    0.049501    0.054999    0.135603    0.175900

93    0.180201    0.180654    0.180712    0.182013    0.181979

93    0.180201  LEADING AND TRAILING EDGE COORDINATES
0.0000    0.000 FCENTROID, FTANG, FLEAN, FSWEEP, FAXIAL
                FSCALE, FCONST
```

Steps

Run stagen-18.1.exe. Write the name of your file in the cmd.
Outputs:

- stagen.out: blade profile coordinates.
- blade_profiles.tec: blade profiles in format readable by Tecplot.
- stage_new.dat: inputs for Multall.
- stage_old.dat: inputs for old version of Multall.

Change parameters of CFD simulation if necessary:

1. Cp and gamma have been defined in meangen.in, check if correct.
2. CFL number to ensure convergence of CFD simulations.
3. Maximum Mach number is defined as MACHLIM.
4. Convergence parameters as defined as the maximum number of steps (NSTEPS_MAX) and the convergence parameter (CONLIM), which are given standard values of 9000 and 0.1%.
5. IM and KM can again be changed in this file.

TU Delft

Steps

Run Multall by the following command in cmd (after cd-ing to the folder):

```
multall-open-20.9.exe < stage_new.dat > results.txt
```

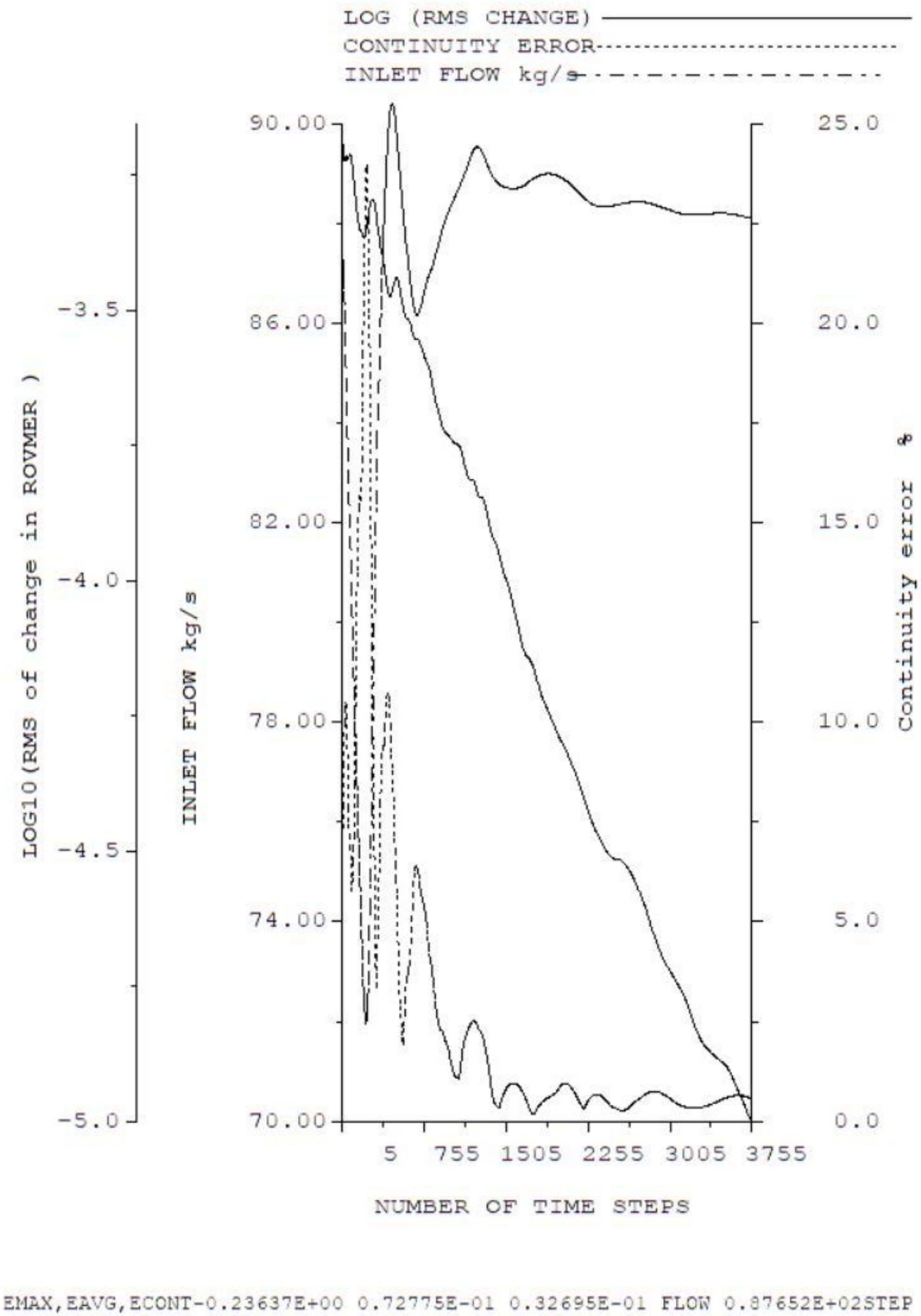
Wait until the simulation has converged (can take several minutes).

Check convergence with *histage*.

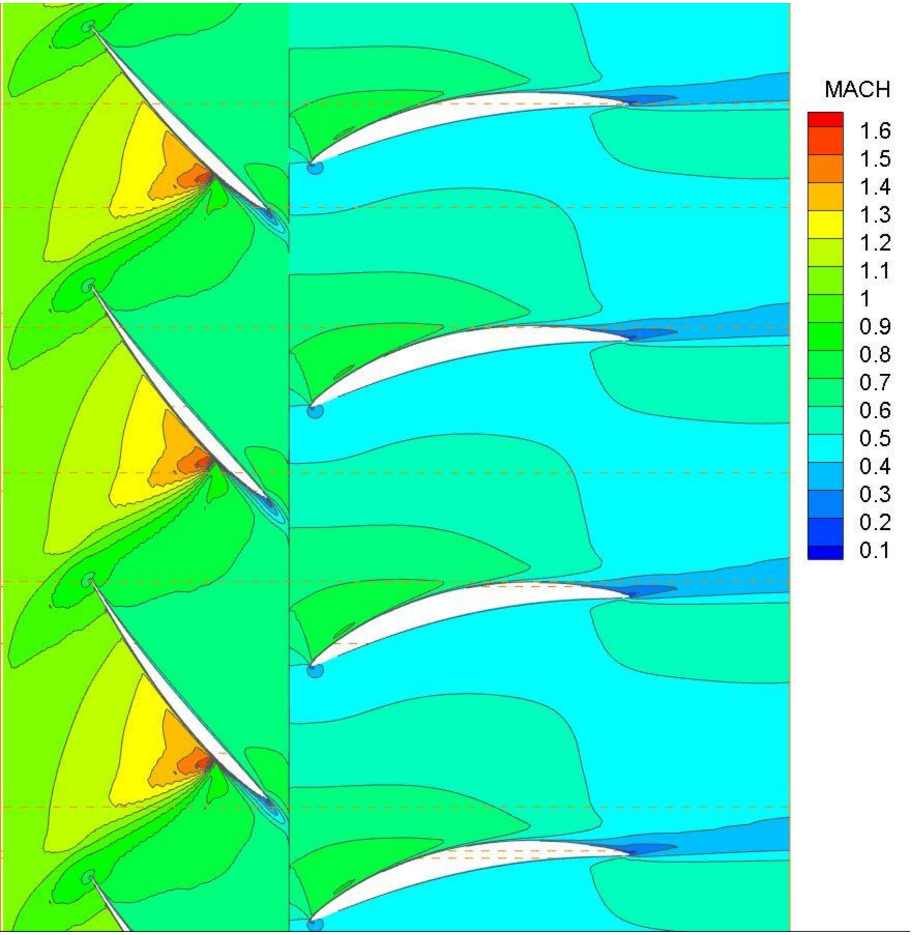
For postprocessing:

1. Convert results using *convert-to-tecplot.exe* using 1-2 blade passages. This will make a Tecplot input file from the flow_out and grid_out files.
2. Open the tecplot-input.dat file in Tecplot.
3. Obtain the distribution of different properties in Details, behind the Contour box. A certain flow slice can be selected.
4. Equations can be defined to obtain the desired parameters and compute losses.

Steps


LOG (RMS CHANGE)
CONTINUITY ERROR
INLET FLOW kg/s
LOG10(RMS of change in ROVMER)
-3.5
-4.0
-4.5
-5.0
INLET FLOW kg/s
90.00
86.00
82.00
78.00
74.00
70.00
25.0
20.0
15.0
10.0
5.0
0.0
Continuity error %
5
755
1505
2255
3005
3755
NUMBER OF TIME STEPS
EMAX,EAVG,ECONT-0.23637E+00 0.72775E-01 0.32695E-01 FLOW 0.87652E+02STEP

Steps



Mach number distribution

Total pressure distribution along radius (shockwave formation seen close to tip)

TU Delft

Steps

- Comparison of results:

Parameter	Unit	Meanline	CFD
Total-to-total efficiency	%	91.2	88.8
Mass flow rate	kg/m^3	88.23	88.13
Pressure ratio	-	1.6	1.59

- Check if target efficiency, required mass flow rate and pressure ratio are achieved.

Design example 2
3-stage axial turbine

Example: 3-stage axial turbine

- This example shows the steps from the end of a preliminary design until CFD calculation and post-processing and evaluating the results
- The following preliminary design results are given:

	Unit	Value
Inlet total pressure	bar	2
Inlet total temperature	K	500
Blade twist		Free vortex
Hub radius (const.)	m	0.4
Target total-to-total efficiency	%	90
Mass flow rate	kg/m^3	25
Rotation speed	RPM	3000
Degree of reaction	-	0.25
Flow coefficient	-	0.5
Work coefficient	-	2

- Same flow coefficient, load coefficient and degree of reaction is used for all stages.

TU Delft

Example: 3-stage axial turbine

- Blade chords are selected based on the calculated blade span and by selecting a proper aspect ratio.
- Row gaps are selected
- Incidence and deviation angles, blade thickness ratios and blockage factors are estimated
- From the preliminary calculations and design choices the Meangen input file* is constructed →

TU Delft

3-stage axial turbine - Meangen

Meangen input file (name the file `meangen.in`):*

```
T                          TURBO_TYP,"C" FOR A COMPRESSOR,"T" FOR A TURBINE
AXI                        FLO_TYP FOR AXIAL OR MIXED FLOW MACHINE
   287.500     1.400       GAS PROPERTOES, RGAS, GAMMA
     2.000    500.000      POIN,  TOIN
     3                     NUMBER OF STAGES IN THE MACHINE
H                          CHOICE OF DESIGN POINT RADIUS, HUB, MID or TIP
    3000.000               ROTATION SPEED, RPM
      25.000               MASS FLOW RATE, FLOWIN.
A                          INTYPE, TO CHOOSE THE METHOD OF DEFINING THE VELOCITY TRIANGLES
  0.250  0.500  2.000      REACTION, FLOW COEFF., LOADING COEFF.
A                          RADTYPE, TO CHOOSE THE DESIGN POINT RADIUS
       0.400               THE DESIGN POINT RADIUS
       0.050       0.040   BLADE AXIAL CHORDS IN METRES.
       0.250       0.500   ROW GAP  AND STAGE GAP (fractions)
   0.00000   0.02000       BLOCKAGE FACTORS, FBLOCK_LE,  FBLOCK_TE
       0.900               GUESS OF THE STAGE ISENTROPIC EFFICIENCY
   1.000   1.000           ESTIMATE OF THE FIRST AND SECOND ROW DEVIATION ANGLES
  -2.000  -2.000           FIRST AND SECOND ROW INCIDENCE ANGLES
   1.00000                 BLADE TWIST OPTION, FRAC_TWIST (1 is free vortex, 0 is without twist)
n                          BLADE ROTATION OPTION , Y or N
  92.000  88.000           QO ANGLES AT LE  AND TE OF ROW 1
  88.000  92.000           QO ANGLES AT LE  AND TE OF ROW 2
n                          DO YOU WANT TO CHANGE THE ANGLES FOR THIS STAGE ? "Y" or "N"
y                          IFSAME_ALL, SET = "Y" TO REPEAT THE LAST STAGE INPUT TYPE AND VELOCITY TRIANGLES, SET = "C" TO CHANGE
INPUT TYPE.
   0.00000   0.02000       BLOCKAGE FACTORS, FBLOCK_LE,  FBLOCK_TE
n                          DO YOU WANT TO CHANGE THE ANGLES FOR THIS STAGE ? "Y" or "N"
y                          IFSAME_ALL, SET = "Y" TO REPEAT THE LAST STAGE INPUT TYPE AND VELOCITY TRIANGLES, SET = "C" TO CHANGE
INPUT TYPE.
   0.00000   0.02000       BLOCKAGE FACTORS, FBLOCK_LE,  FBLOCK_TE
n                          DO YOU WANT TO CHANGE THE ANGLES FOR THIS STAGE ? "Y" or "N"
Y                          IS OUTPUT REQUESTED FOR ALL BLADE ROWS ?
N    STATOR No.  1 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
  0.3000  0.4500           MAX THICKNESS AND ITS LOCATION FOR STATOR  1 SECTION No.  1
  0.3000  0.4500           MAX THICKNESS AND ITS LOCATION FOR STATOR  1 SECTION No.  2
  0.3000  0.4500           MAX THICKNESS AND ITS LOCATION FOR STATOR  1 SECTION No.  3
Y    ROTOR No.   1 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    STATOR No.  2 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    ROTOR No.   2 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    STATOR No.  3 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
Y    ROTOR No.   3 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
```

TU Delft

3-stage axial turbine

- Before starting the calculations make sure to have all of the following files in the same folder:

convert2tecplot.exe	26/11/2017 16:43	Application	389 KB
globplot.exe	20/02/2017 21:31	Application	465 KB
histage.exe	20/02/2017 23:18	Application	465 KB
ihg_lib.dll	14/01/2003 21:11	Application extens...	268 KB
ihg_plot.exe	13/01/2003 11:50	Application	88 KB
intype	25/11/2017 12:20	File	1 KB
Meangen17.4.1.exe	12/12/2017 11:37	Application	573 KB
Multall17.5.exe	27/11/2017 19:27	Application	1,167 KB
plotall-17.1.exe	22/02/2017 16:47	Application	626 KB
Stagen17.2.2.exe	21/01/2018 12:40	Application	495 KB

TU Delft

3-stage axial turbine

- Run *Meangen* in the command line with the input file (`meangen.in`).

  `>>Meangen17.4.1.exe`

- Press „F” to input the data* from file
- This generates the input file for Stagen (next slide)

TUDelft

3-stage axial turbine

- *Meangen* ouputs 3 new files:
  - `meandesign.out` – row geometry & thermodynamics
  - `meangen.out` – echo of the Meangen input file
  - `stagen.dat` – Stagen input file
- meandesign.out includes thermodynamic properties, geometry and the data for velocity triangles for each blade row:

```
 STAGE No, ROW No, No. BLADES 1 1 33
 STAGE No, ROW No, No. BLADES 1 2 71

 *****************************************************
  CONDITIONS FOR THE FIRST BLADE ROW OF THE STAGE.
  THIS IS A TURBINE STATOR
 *****************************************************
 FIRST BLADE INLET AND EXIT ANGLES  -26.565073
74.054665
  FIRST BLADE AXIAL VELOCITY      62.8318
  FIRST BLADE INLET MACH NUMBER   0.15697631
  FIRST BLADE EXIT MACH NUMBER    0.52361447
  FIRST BLADE EXIT DENSITY        1.201726
  FIRST BLADE EXIT PRESSURE       1.6376798
  FIRST BLADE INLET STAGN PRESS   2.0000002
  FIRST BLADE EXIT STAGN PRESS    1.9741219
  FIRST BLADE REL INLET STAG PRES 2.0000002
  FIRST BLADE EXIT TEMPERATURE    474.00806
  FIRST BLADE EXIT STAGN TEMP     500.
  FIRST BLADE TIP RADIUS =        0.5021015
  FIRST BLADE INLET SPAN =        0.107626915
  FIRST BLADE AXIAL CHORD=        0.05
  FIRST BLADE ASPECT RATIO =      2.1525383

…
```

TU Delft

3-stage axial turbine - Meshing

- In `stagen.dat` the mesh size can be changed*:

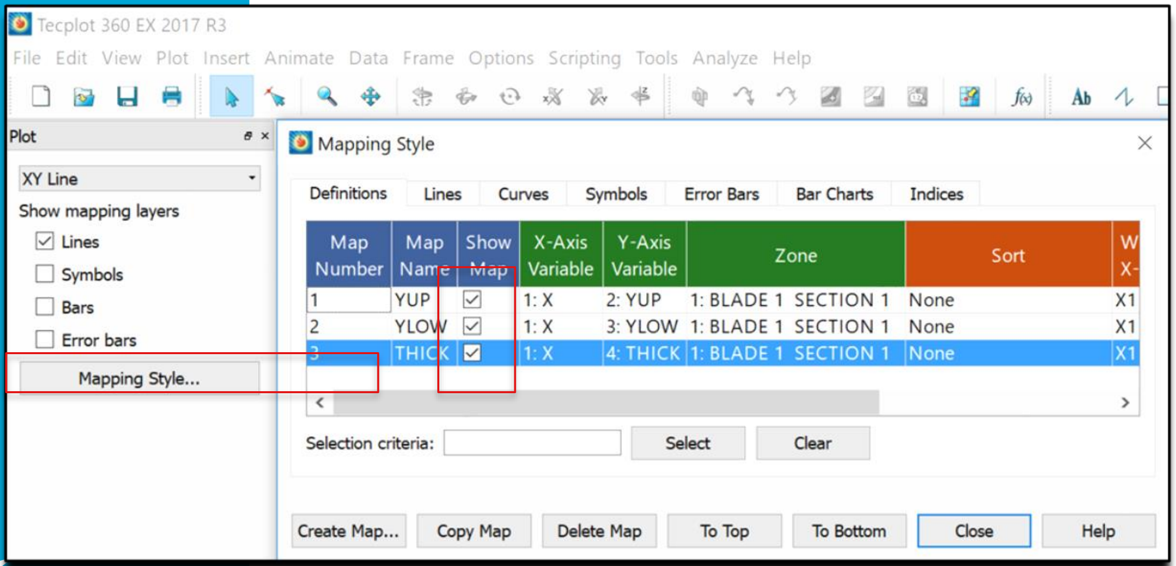
```
287.5000      1.4000 GAS CONSTANT, GAMMA
       37         37      IM, KM
     1.2500     20.0000  FPRAT,   FPMAX
     1.2500     20.0000  FRRAT,   FRMAX
        0                 IFDEFAULTS
        6          3      NOWS, N SECTIONS
    1.000                 SCALING FACTOR
```

- Run Stagen. This generates the mesh

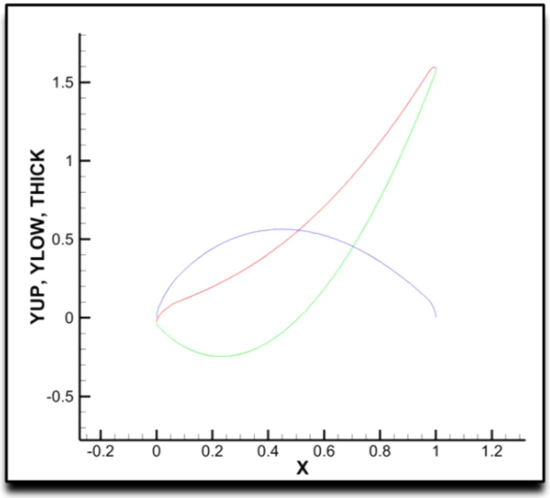
  `>>Stagen17.2.2.exe`
- Answer with yes (y) for the question about the input file name
- *Stagen* will produce the following files:
  - `stage_new.dat` – Multall input file in new format. Use this in the following steps
  - `stage_old.dat` – Multall input file in old format. Not needed
  - `blade_profiles.tec` – Blade shapes, can be opened in Tecplot
  - `stagen.out` – Blade coordinates

3-stage axial turbine – blade profiles

- Open Tecplot* and load `blade_profiles.tec`
- Click on mapping style and show all data sets

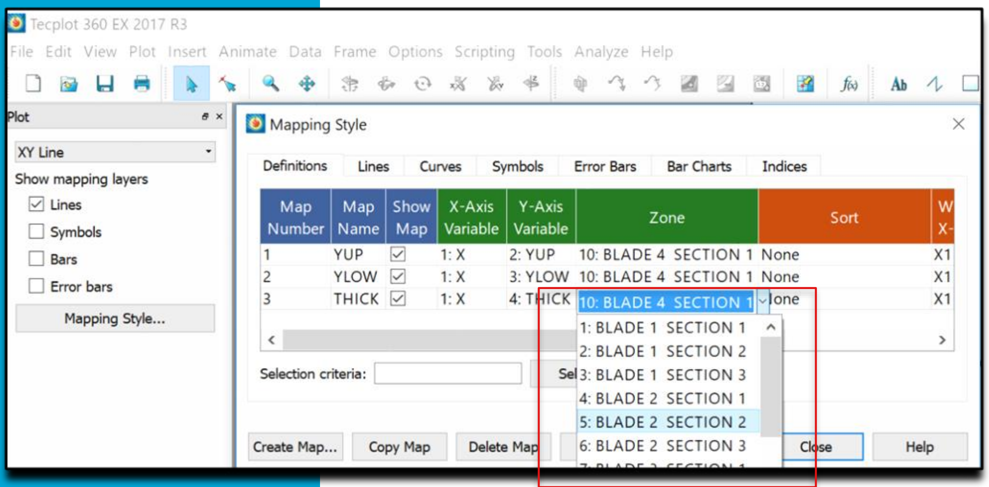


Blade shape at the 1st (hub) section of the first blade row:

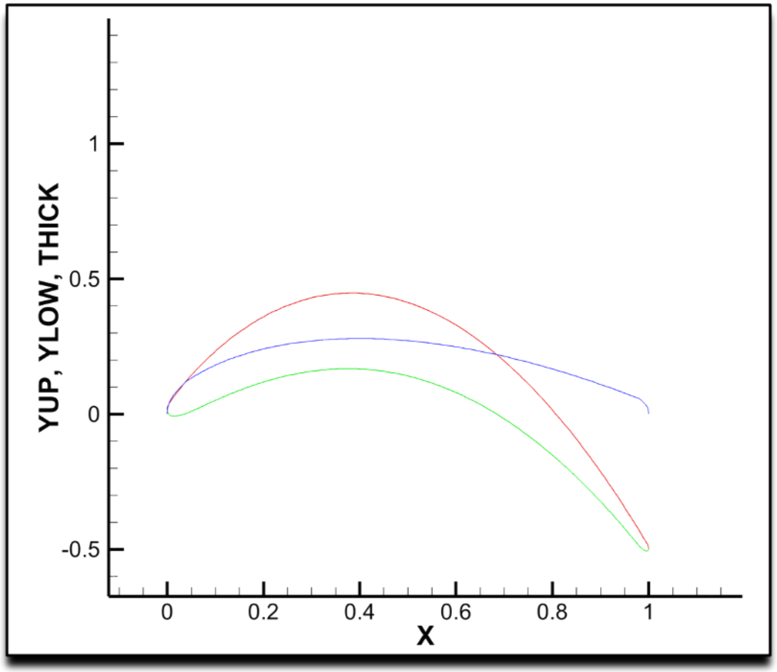


3-stage axial turbine – blade profiles

- Select zone in all 3 data sets to examine a different blade



Blade section in row 4:



3-stage axial turbine - CFD

- The *Stagen* output file includes the stopping criteria (maximum number of iterations and the convergence limit)

```
CFL,     DAMP,     MACHLIM,     F_PDOWN
 0.400000 10.000000  2.000000   0.000000
 IF_RESTART
        0
 NSTEPS_MAX, CONLIM
     9000   0.001000
```

- Run *Multall*

```
>>Multall17.5.exe < stage_new.dat
```

- The program will not work without specifying the input file name!
- Wait* until the computation converges or reaches the maximum number of iterations

3-stage axial turbine - CFD

- The run can be stopped with changing the value in the stopit file from 0 to 1. This will save the results to `flow_out` and `grid_out`.
- The calculation can be restarted form a previous run with `IF_RESTART=1`. For this the `flow_out` and `grid_out` file is needed

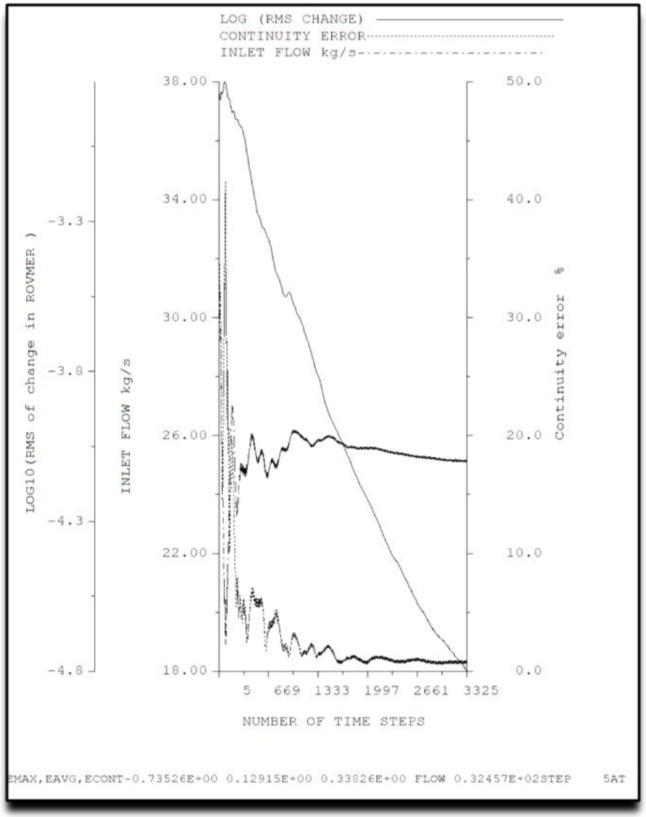
```
CFL,     DAMP,     MACHLIM,     F_PDOWN
 0.400000 10.000000  2.000000   0.000000
 IF_RESTART
          1
 NSTEPS_MAX, CONLIM
     9000   0.001000
```

- Starting from a previous run can make convergence much faster (in case of small modifications) or preliminary results can be saved by stopping and continuing the simulation

TUDelft

3-stage axial turbine - CFD

- Check the convergence of the results with *histage*.
- Check if the mass flow was constant and the imbalance acceptably low*



3 stage axial turbine - Multall

- If efficiency and mass flow is lower than required... Geometry needs to be refined
- Change the geometry (`stagen.dat`):
  - Number of blades in row
  - Blade profile specification

```
0.0400    0.0400    0.3000    0.4500    0.0200    0.0100    2.0000     BLADE PROFILE SPECIFICATION
```

TE thickness ratio   Max. t/c   Max. t/c location

  - What is the effect of these changes?

Postprocessing

- Convert your results with:

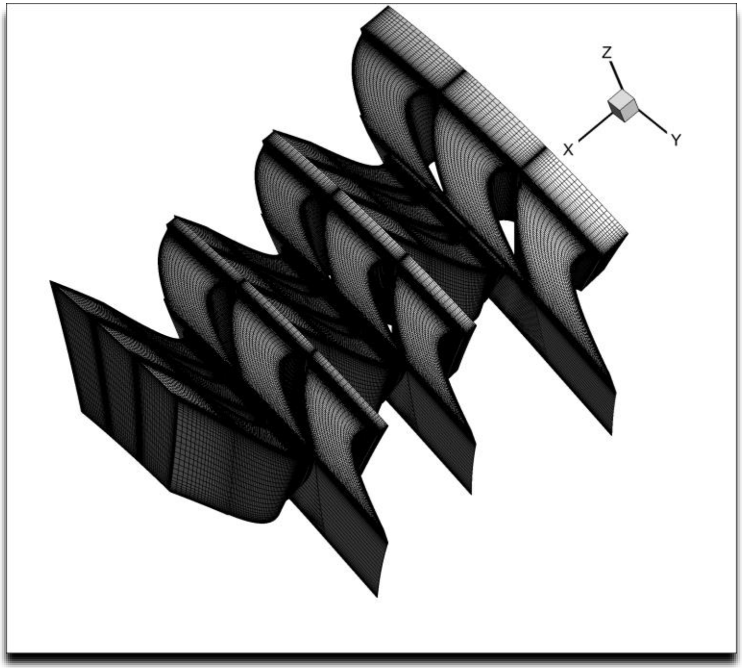
```
>> convert-to-tecplot.exe
```

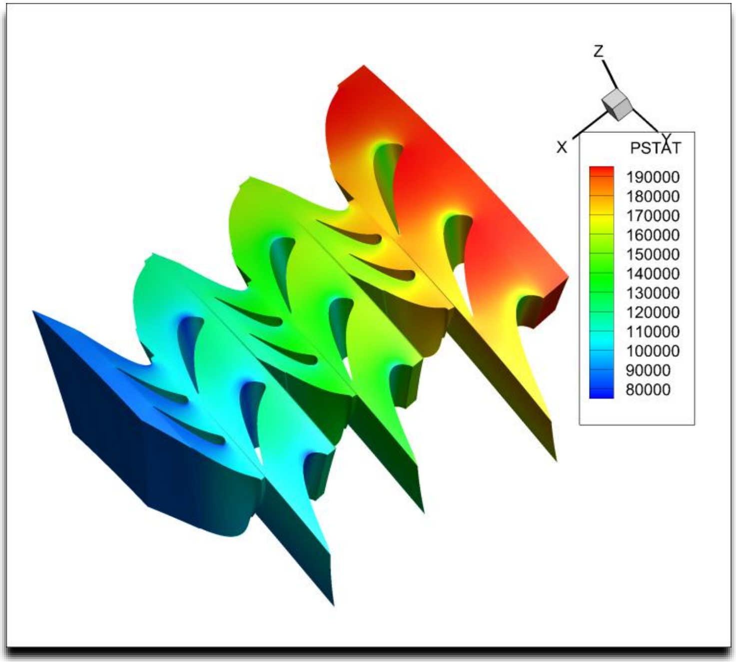
- This will make a *Tecplot* input file from the `flow_out` and `grid_out` files
- Select 2-3 blade passages. Selecting more blade passages can result in very large files (several gigabytes)
- Open the `tecplot-input.dat` file in Tecplot

TU Delft

3-stage axial turbine - Multall

- Postprocess the results to check the geometry and mesh
- Pressure distribution:

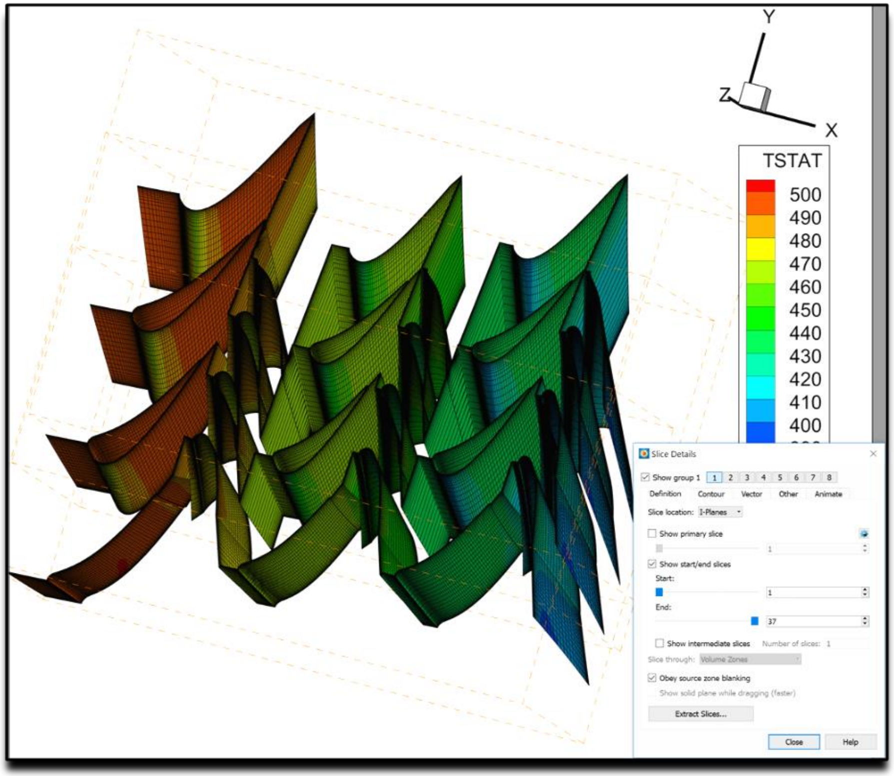




TU Delft

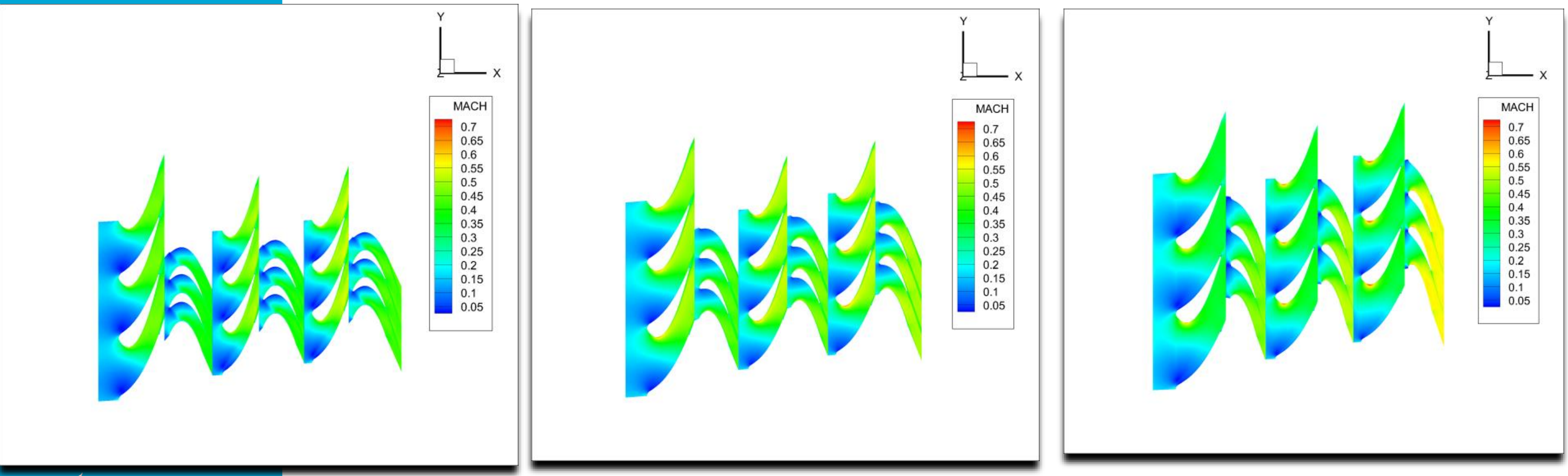
3-stage axial turbine - Blades

- To show the blade surface:
  - Plot → Slices
  - I-Planes
  - Show start/end



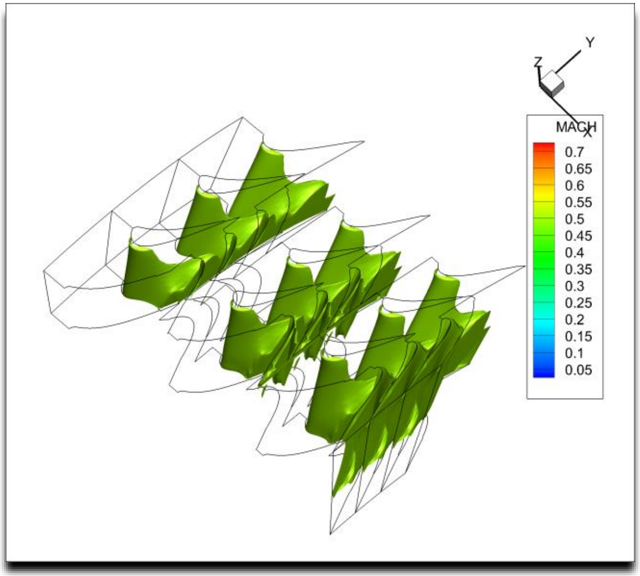
3-stage axial turbine - Multall

- Examine the flowfield in the root, meanline and tip section.
- Plot→Slices. K-Planes
- Mach number (hub, meanline, tip):



3-stage axial turbine - Multall

- In case of large Mach numbers change the blade shape (thickness and camber)
- Mach number (M=0.5 isosurface):



- Check for supersonic flow. Modify blade shape (thickness and camber) in case of large Mach numbers

3-stage axial turbine

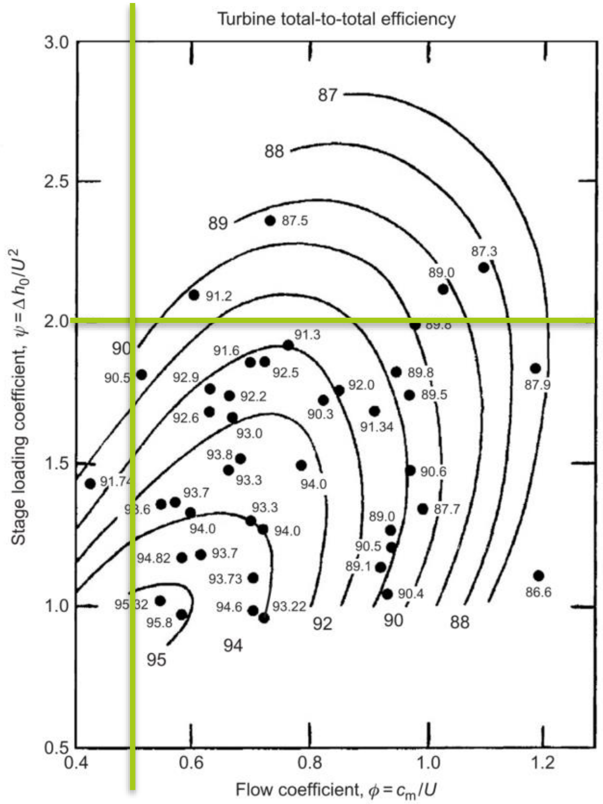
- Comparison of results:

	Unit	Meanline	CFD
Total-to-total efficiency	%	90	92.8
Mass flow rate	kg/m^3	25	25.1
Pressure ratio (t-t)	-		2.25
Power output	kW		2453

- Check if target efficiency, required mass flow rate and pressure ratio was achieved

3-stage axial turbine - Multall

- How the calculated efficiency compares with the Smith chart?



3-stage axial turbine - Questions

- Can you explain possible differences between preliminary and CFD results?
- What is the effect of changing the degree of reaction?
- What is the effect of changing the geometry (blade aspect ratio, solidity, etc)?
- What is the effect of using untwisted blades instead of free vortex?
- Run a calculation with a denser mesh (50×50 cells in the meridional section). Is there any difference in the results?

TU Delft

APPENDIX

Cooling for first stage of HPT

Cooling

Cooling flows can be added at any point on the blade and endwall surfaces. The flow is added through a series of "patches" whose I,J,K boundaries are specified in the input data. If the "mixing plane" falls within a region where coolant is being added then two separate patches, one upstream and one downstream of the "mixing plane" must be used. The coolant mass flow, stagnation temperature,

stagnation pressure, ejection Mach number and flow directions must be specified for each patch. Note that the exit relative Mach number of the coolant flow is specified and is not calculated from the local static and stagnation pressures. If it is required to model individual cooling holes then each patch may

be one grid cell in size, but this requires a great deal of input data and it is more usual to specify a single patch to cover multiple cooling flows. The overall total-to-total efficiency is calculated and

printed out, allowing for the potential work of all the cooling flows. However, the polytropic efficiencies, which are also printed out, relate the mass averaged inlet and outlet flow conditions and

are not meaningful when cooling flows are added.

Source: Denton: *J. Denton*: MULTISTAGE TURBOMACHINERY FLOW CALCULATION PROGRAM MULTALL_OPEN

Cooling

- Cooling options from the documentations:

CARD CWL1description and CWL1 .

NCWLBLADE, NCWLWALL

NCWLBLADE	The number of cooling flow patches on the blade surfaces.
NCWLWALL	The number of cooling flow patches on the hub and casing.

Set these to zero if there is no cooling on this blade row.
****

FOR N =1, NCWLBLADE input CARDS CWL2 and CWL3

CARD CWL2description and CWL2 .

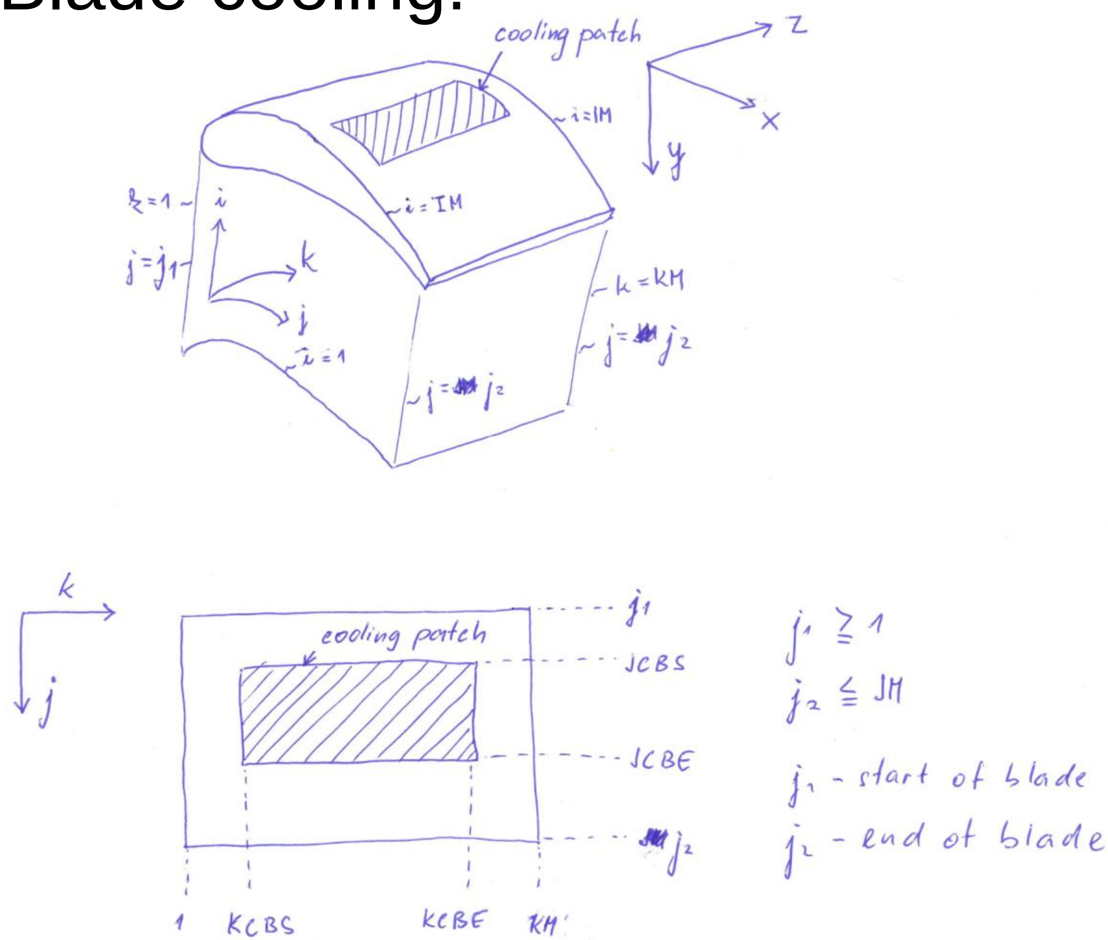
IC, JCBS, JCBE, KCBS, KCBE

IC	The "I" value of the blade surface through which coolant is being ejected. Must = 1 or IM .
JCBS	The "J" value at the start of the cooling patch.
JCBE	The "J" value at the end of the cooling patch.
KCBS	The "K" value at the start of the cooling patch.
KCBE	The "K" value at the end of the cooling patch.

TUDelft

Cooling

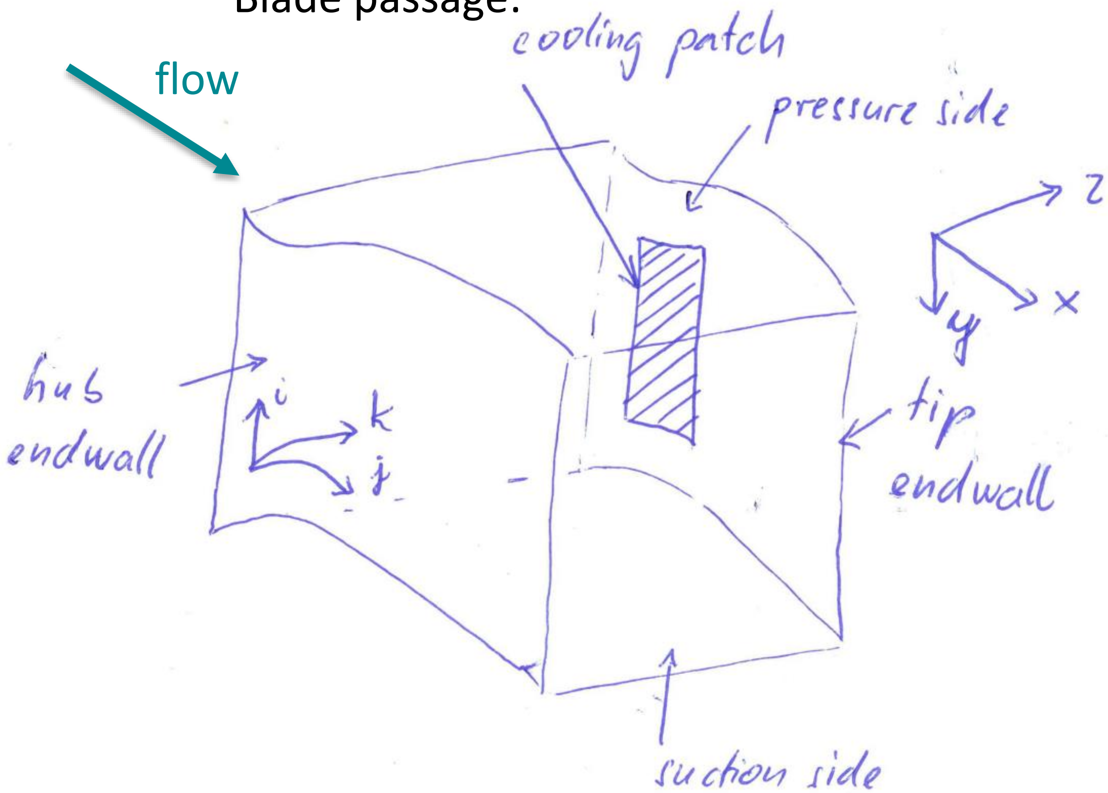
- Blade cooling:



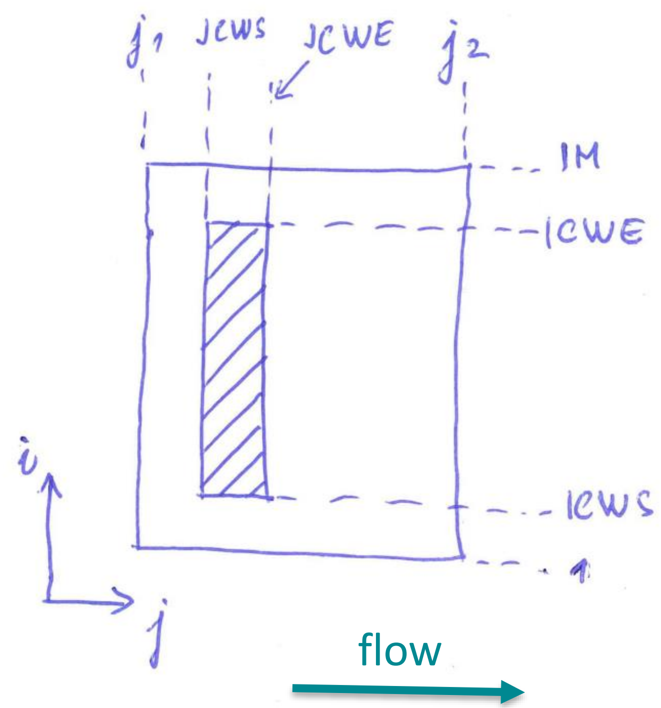
Cooling

- Endwall cooling:

Blade passage:

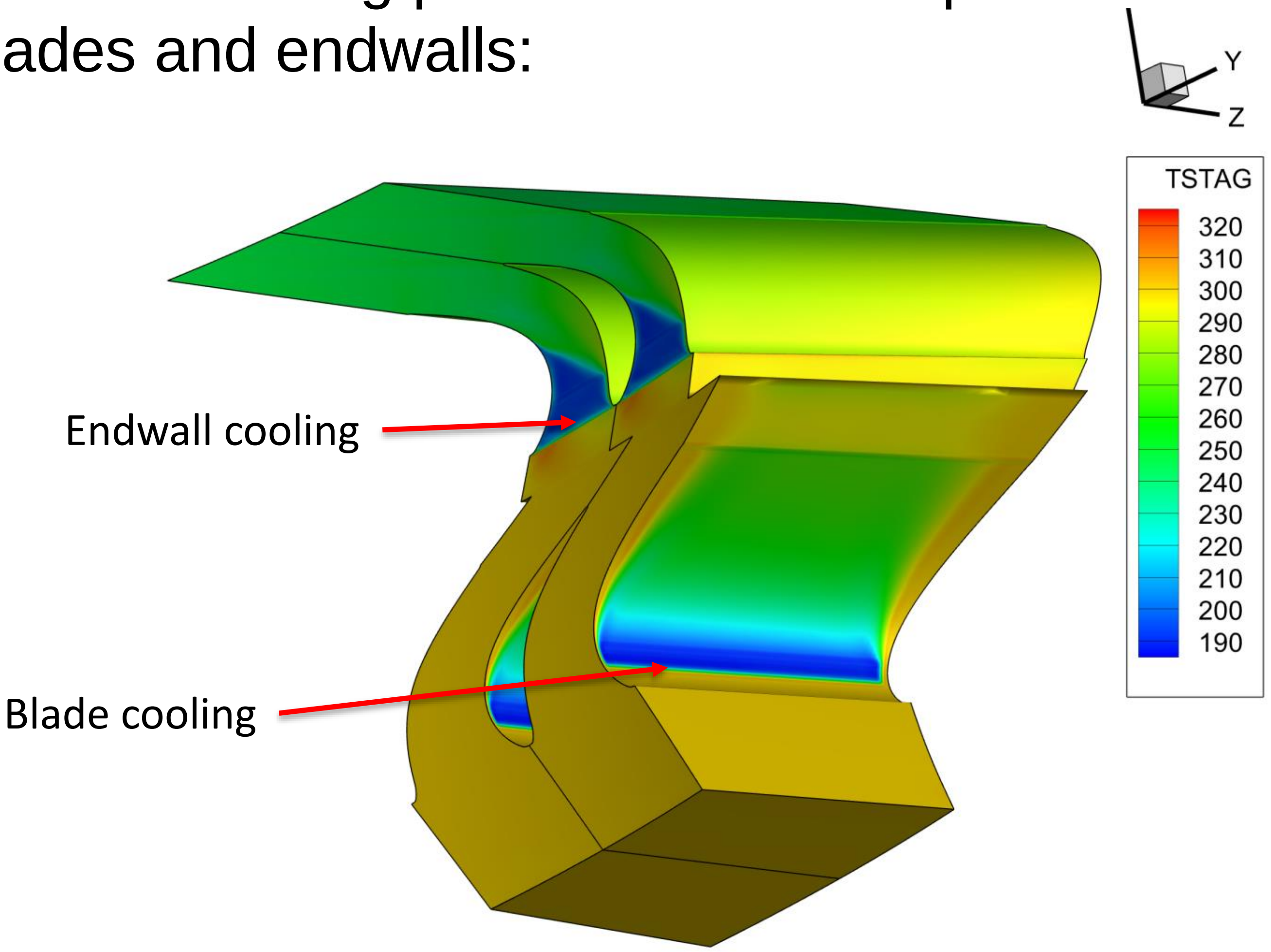


Blade surface:



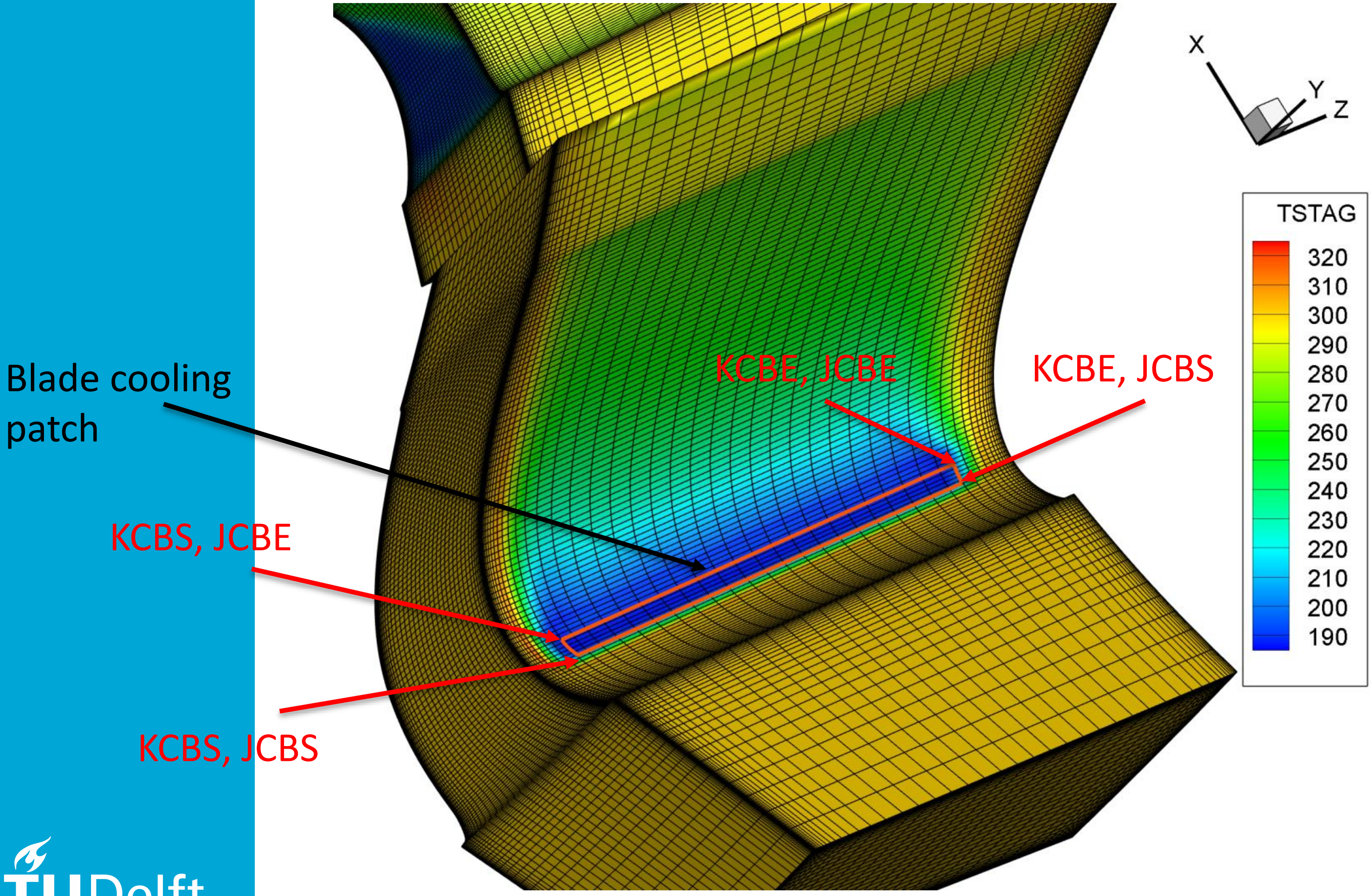
Cooling

- Effect of cooling patches on the temperature of blades and endwalls:



Blade cooling

- Coordinates of the cooling patches



Cooling – patch placement

- In `stage_new.dat` input file:

```
******************************************************************************
**********STARTING THE INPUT FOR EACH BLADE ROW*****************************
NOZZLE   ROW
     NUMBER OF BLADES IN ROW
     71
     JM         JLE        JTE
     141         26        126
```

- Indices for cooling patch should be within these limits:

$$JCBS \leq JLE, JCBE \leq JTE$$

- WARNING – If the mesh density is changed, the position of the cooling patch will also change.

Appendix

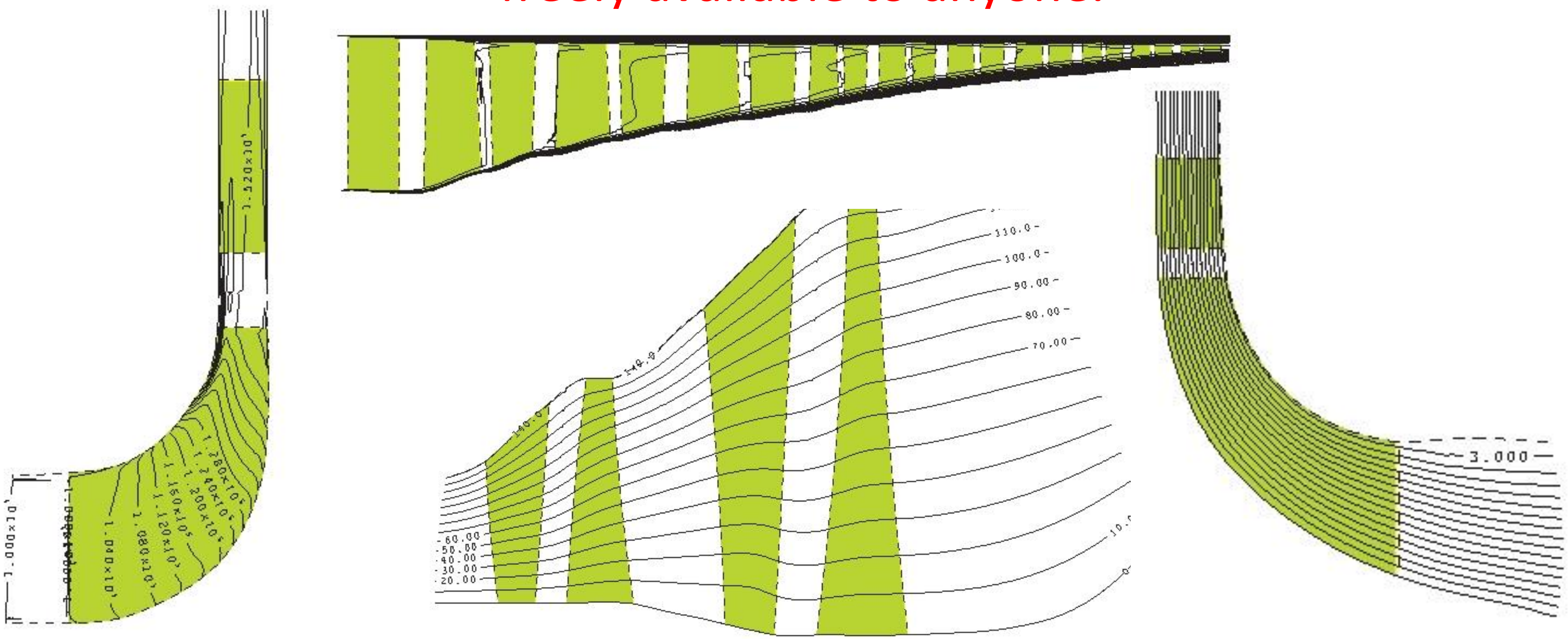
Multall tutorial by Denton

MULTALL – AN OPEN-SOURCE, CFD BASED, TURBOMACHINERY DESIGN SYSTEM

ASME GT2017-63993

John Denton
Retired from the Whittle Lab, Cambridge University, UK.

The objective of this paper is to present a turbomachinery aerodynamic design system that is relatively simple to use and is freely available to anyone.



Turbomachinery design systems are usually the property of large companies and are very complex.

The author has, over many years developed a system based on his widely-used 3D CFD code MULTALL . The system is intended to be relatively simple and easy to use. The source codes are written in FORTRAN and are freely available to any user.

It is hoped that this system will be of use to smaller companies and academics who do not have access to an “in-house” design system.

A turbomachinery design system generally consists of the following steps:

1. Specify overall parameters such as mass flow rate, mean diameter, rotational speed, inlet flow conditions, exit pressure.

2 Perform a one-dimensional mean-line calculation to obtain the annulus shape and mid-span blade angles.

3. Perform a 2D axisymmetric throughflow calculation in the inverse (design) mode to obtain the variation of flow angles along the span.

4. Repeat the throughflow calculation in the analysis mode to predict the blade losses, machine efficiency and stream surface thickness distributions.

5. Perform quasi-3D (Q3D) blade-to-blade calculations at several spanwise sections on each blade row to design the blade shapes.

6. Perform relatively coarse grid, multistage, 3D, viscous calculations for the main flow path of the whole machine to optimise the blade stacking.

7. Perform more detailed 3D calculations to include the effects of leakage flows endwall bleeds and cavities and coolant flows. These will give the final prediction of machine overall performance.

Each of these steps is likely to be repeated several times with returns to previous steps often being necessary.

TU Delft

The author has long maintained that, given a flexible geometry generator and a fast 3D solver, it is simpler and faster to omit steps 3 to 5, and go straight from the mean line design to fairly coarse grid 3D calculations to optimise the blade shapes and stacking.

This is the basis if the present design system.

1. Specify overall parameters such as mass flow rate, mean diameter, rotational speed, inlet flow conditions, exit pressure.

2 Perform a one-dimensional mean-line calculation to obtain the annulus shape and mid-span blade angles.

3. Perform a 2D axisymmetric throughflow calculation in the inverse (design) mode to obtain the variation of flow angles along the span.

4. Repeat the throughflow calculation in the analysis mode to predict the blade losses, machine efficiency and stream surface thickness distributions.

5. Perform quasi-3D (Q3D) blade-to-blade calculations at several spanwise sections on each blade row to design the blade shapes.

6. Perform relatively coarse grid, multistage, 3D, viscous calculations for the main flow path of the whole machine to optimise the **blade profiles** and the blade stacking.

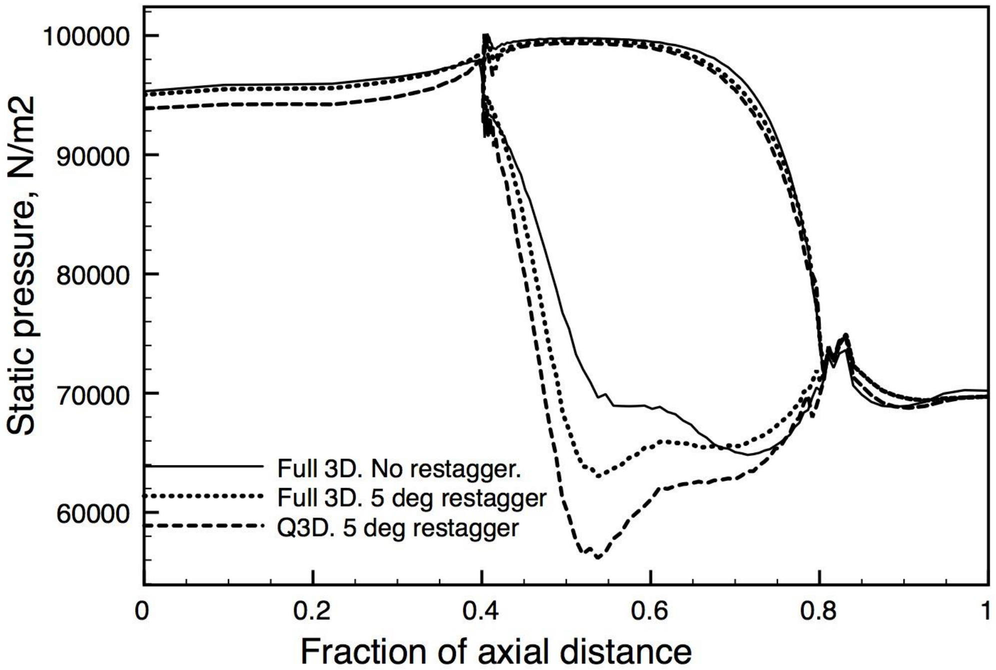
7. Perform more detailed 3D calculations to include the effects of leakage flows endwall bleeds and cavities and coolant flows. These will give the final prediction of machine overall performance.

TU Delft

The quasi-3D calculations are anyhow not accurate unless twisted stream surfaces are used, as in Wu's original suggestion.

This is virtually never done !

The figure below shows the results of a Q3D and a 3D calculation on a LP turbine blade where the mid-span section has been re-staggered by 5 deg.



Static pressure contours on the blade with re-stagger, showing 3D relief.

The Q3D calculation predicts the effect on loading to be about twice that of the 3D calculation.

THE SYSTEM CONSISTS OF 3 LINKED PROGRAMS
ALL WRITTEN IN FORTRAN

MEANGEN

Performs a 1D calculation to obtain the velocity triangles.
Sets the annulus boundaries.
Generates initial blade shapes and twists them for a free vortex design.
Writes an input file for STAGEN

STAGEN

Refines the blade shapes.
Stacks them and combines them into stages.
Writes an input file for MULTALL

MULTALL

Performs a 3D multistage calculation to predict the detailed flow pattern and overall performance.

TU Delft

MEANGEN

Takes input data either from the screen or from a file.
The input is designed to be as minimal as possible so many parameters are set by default. These can be changed by editing the program.

On the screen answer questions such as:

> Design a compressor or a turbine? Answer ‘C’ or ‘T’.
> Input the gas constant and specific heat ratio
> Input the desired mass flow rate.
> Input the rotational speed.

etc.

The screen input data is mirrored to a file and subsequent small changes to the design are most easily made by editing that file.

The initial design can be run through STAGEN and MULTALL without any further changes if required, but it will usually need to be refined by STAGEN.

MEANGEN generates data for complete stages. Single blade rows can be generated by omitting the output for one of the blade rows.

The blading parameters can defined either:

On a fixed radius and assuming repeating stage conditions. In which case only 3 parameters are needed to fix the velocity triangles. This makes it very easy to generate multiple repeating stages.

Or

On an arbitrary stream surface with changes in radius and meridional velocity. In which case 4 parameters are needed to fix the velocity triangles, which can differ from stage to stage.

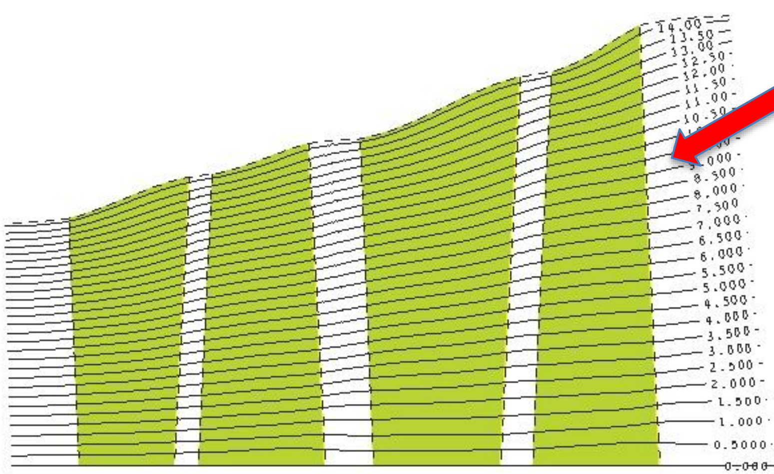
The design stream surface can be either the hub, mid-span or tip. The variation of the blade angles along the span is obtained by assuming free vortex flow.

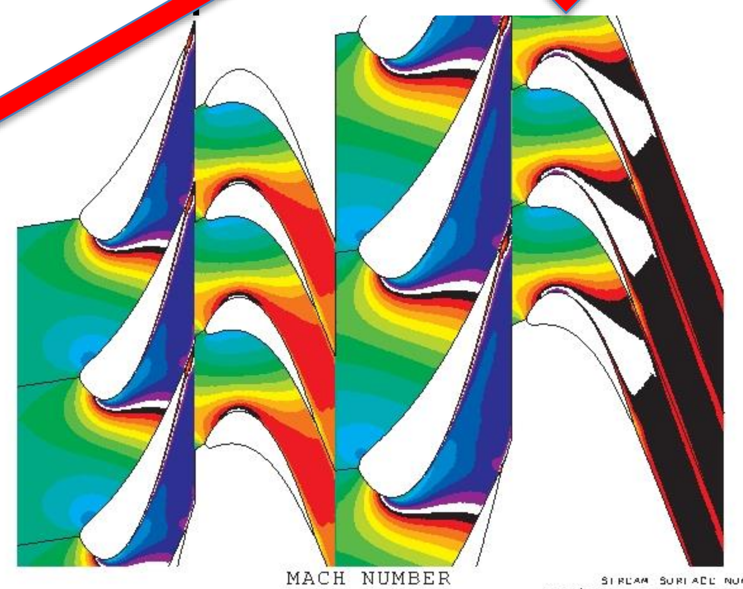
```
T                    TURBO_TYP,"C" FOR A COMPRESSOR,"T" FOR A TURBINE
AXI                            FLO_TYP FOR  AXIAL OR MIXED FLOW MACHINE
  287.150    1.400          GAS PROPERTOES, RGAS, GAMMA
    1.000  300.000          POIN,  TOIN
  2                  NUMBER OF STAGES IN THE MACHINE
H                             CHOICE OF DESIGN POINT RADIUS, HUB, MID or TIP
  3000.000                    ROTATION SPEED, RPM
   15.000                     MASS FLOW RATE, FLOWIN.
A                             INTYPE, TO CHOOSE THE METHOD OF DEFINING THE VELOCITY TRIANGLES
    0.200    0.500    2.000    REACTION, FLOW COEFF., LOADING COEFF.
A                             RADTYPE, TO CHOOSE THE DESIGN POINT RADIUS
    0.400                     THE DESIGN POINT RADIUS
    0.030    0.040          BLADE AXIAL CHORDS IN METRES.
    0.250    0.500          ROW GAP  AND STAGE GAP
    0.900                     GUESS OF THE STAGE ISENTROPIC EFFICIENCY
    1.000    1.000          ESTIMATE OF THE DEVIATION ANGLES
   -2.000   -2.000          FIRST AND SECOND ROW INCIDENCE ANGLES
   92.000   87.000          QO ANGLES AT LE  AND TE OF ROW 1
   87.000   93.000          QO ANGLES AT LE  AND TE OF ROW 2
n                    DO YOU WANT TO CHANGE THE ANGLES FOR THIS STAGE, ANSWER  "Y" or "N"
y                    IFSAME_ALL, SET = "Y" TO REPEAT THE LAST STAGE VELOCITY TRIANGLES
n                    DO YOU WANT TO CHANGE THE ANGLES FOR THIS STAGE, ANSWER  "Y" or "N"
Y                    IS OUTPUT REQUESTED FOR ALL BLADE ROWS ?
n                             STATOR No.  1 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
    0.350    0.500          MAX THICKNESS AND ITS LOCATION FOR STATOR  1 SECTION No.  1
    0.350    0.475          MAX THICKNESS AND ITS LOCATION FOR STATOR  1 SECTION No.  2
    0.350    0.450          MAX THICKNESS AND ITS LOCATION FOR STATOR  1 SECTION No.  3
n                             ROTOR No.   1 SET ANSTK = "Y" TO USE THE SAME  BLADE SECTIONS AS THE LAST STAGE
    0.300    0.400          MAX THICKNESS AND ITS LOCATION FOR ROTOR   1 SECTION No.  1
    0.250    0.400          MAX THICKNESS AND ITS LOCATION FOR ROTOR   1 SECTION No.  2
    0.200    0.400          MAX THICKNESS AND ITS LOCATION FOR ROTOR   1 SECTION No.  3
y                    STATOR No.  2 SET ANSTK= "Y" TO CHOOSE THE SAME BLADE SECTIONS AS THE LAST STAGE.
y                    ROTOR No.  2 SET ANSTK= "Y" TO CHOOSE THE SAME BLADE SECTIONS AS THE LAST STAGE.
```

This file, produced by answering questions on the screen.

(Only the numbers and characters on the left are input)

Produces this Turbine





STAGEN

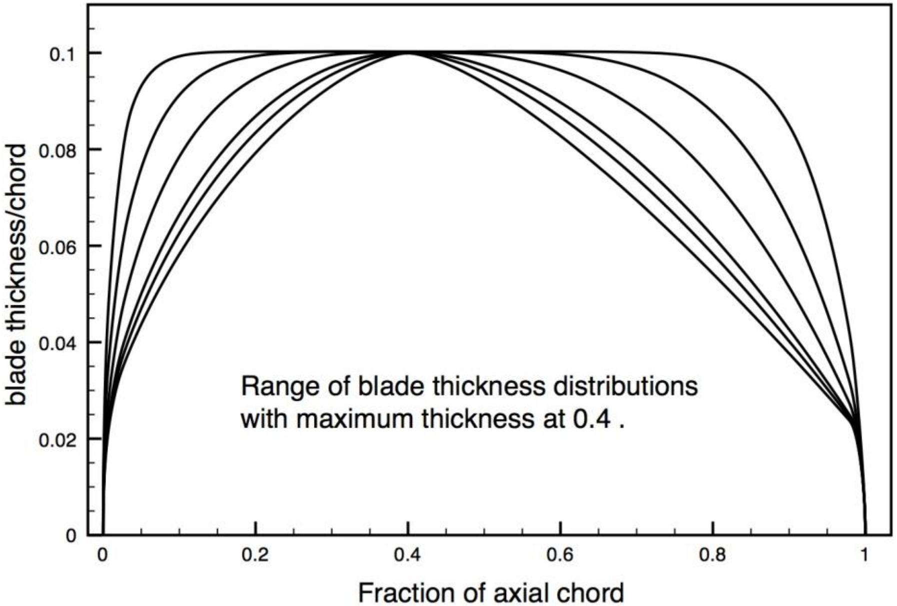
Reads in a data file produced by MEANGEN.

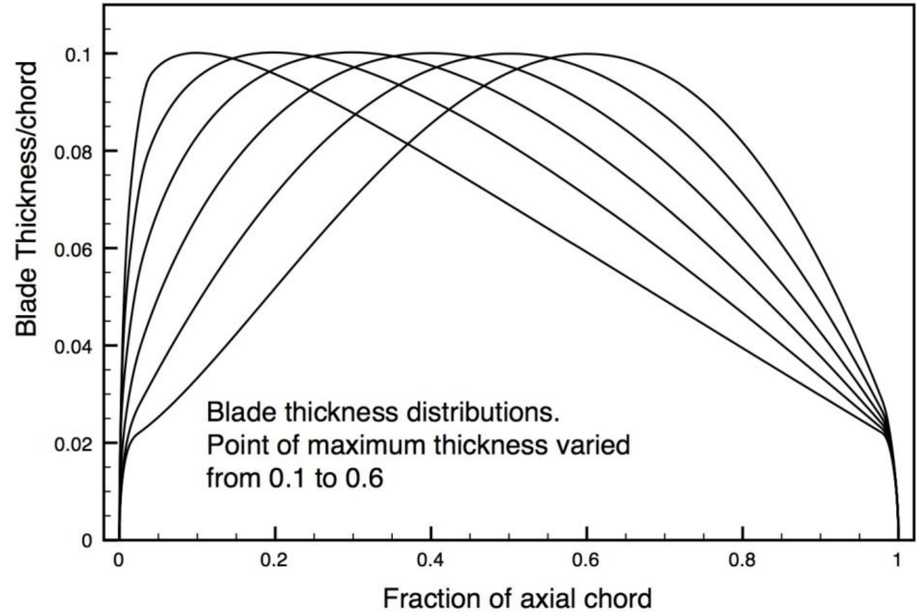
- Allows the blade sections to be changed from the initial crude guess.
- Projects the blades onto quasi-stream surfaces.
- Stacks the blades.
- Generates the grid.
- Combines the blades into stages.
- Writes an input data file for MULTALL .

As with MEANGEN many parameters are set by default and can be changed by editing the program.

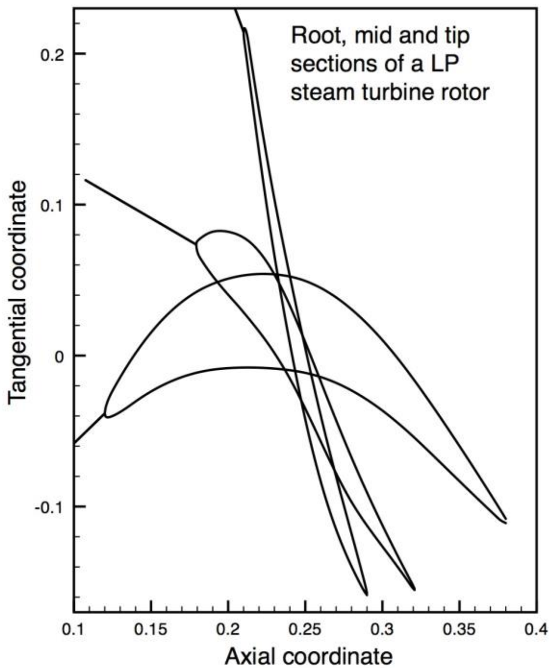
The blade profiles may be either:

- Generated from a system of equations.
- Generated from an input camber line and thickness distribution.
- Read in from an existing profile.





Blade thickness distributions from equations.



Root, mid and tip blade sections generated for an LP steam turbine last rotor.

TU Delft

MULTALL

3D multistage N-S solver developed over many years. Using on a finite volume time-marching method.

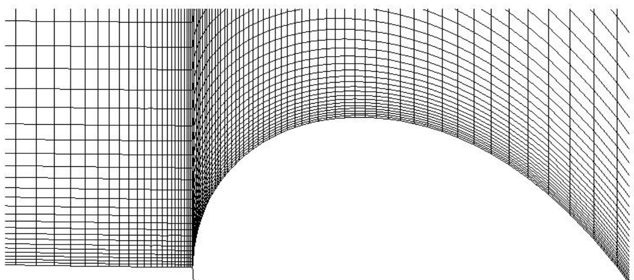
It can model most turbomachinery flow features, including:

- Mixing plane model between blade rows.
- Quasi-3D blade-to-blade calculations.
- Axisymmetric throughflow calculations.
- Three turbulence models.
- Specified boundary layer transition point, or a simple transition model.
- Surface roughness effects.
- Cooling flow addition through the blades or endwalls.
- Bleed flows from the hub or casing.
- Artificial compressibility for low Mach number and incompressible flows.
- Pinched tip model for plain tip clearances.
- Shroud leakage model for shrouded blades.
- Temperature dependent gas properties.
- Automatic matching of the grids at the mixing plane.
- Ability to refine the grids within the code.
- Automatic trailing edge cusp generation.
- Modification of the inlet boundary conditions to simulate a repeating stage.
- Ability to perform limited blade redesign within the 3D code.

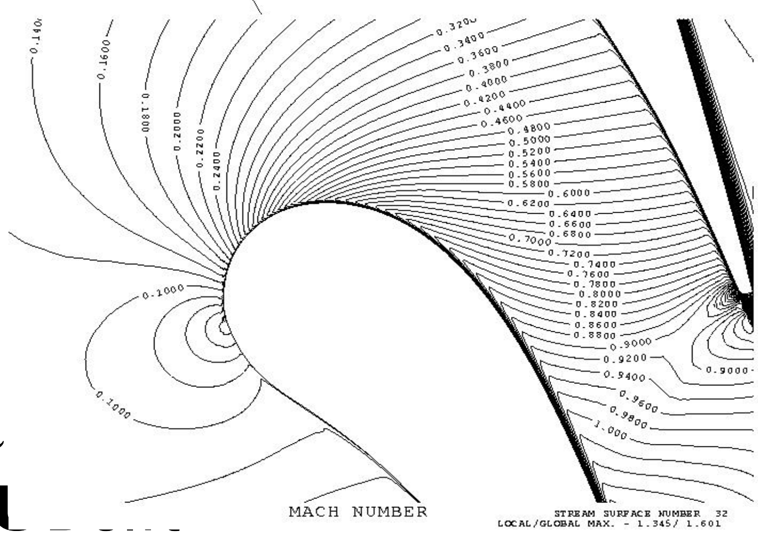
Most of these are briefly described in the paper, and in detail in the user manual, but there is no time to describe them here.

MULTALL uses a simple "H" grid, which is extremely simple to generate and is anyhow always necessary at mixing planes. However, use of an "H" mesh is much disapproved of by CFD specialists because of numerical errors on highly sheared cells.

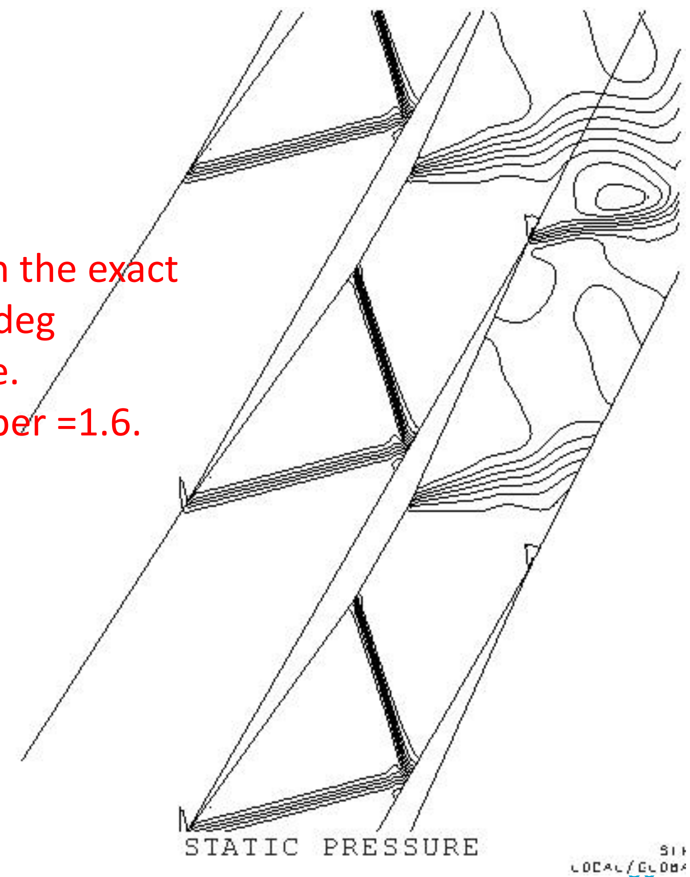
However, using cell corner storage of the variables, MULTALL works remarkably well on sheared grids, its speed and simplicity allow more cells to be used in highly sheared regions.



Turbine leading edge stagnation point

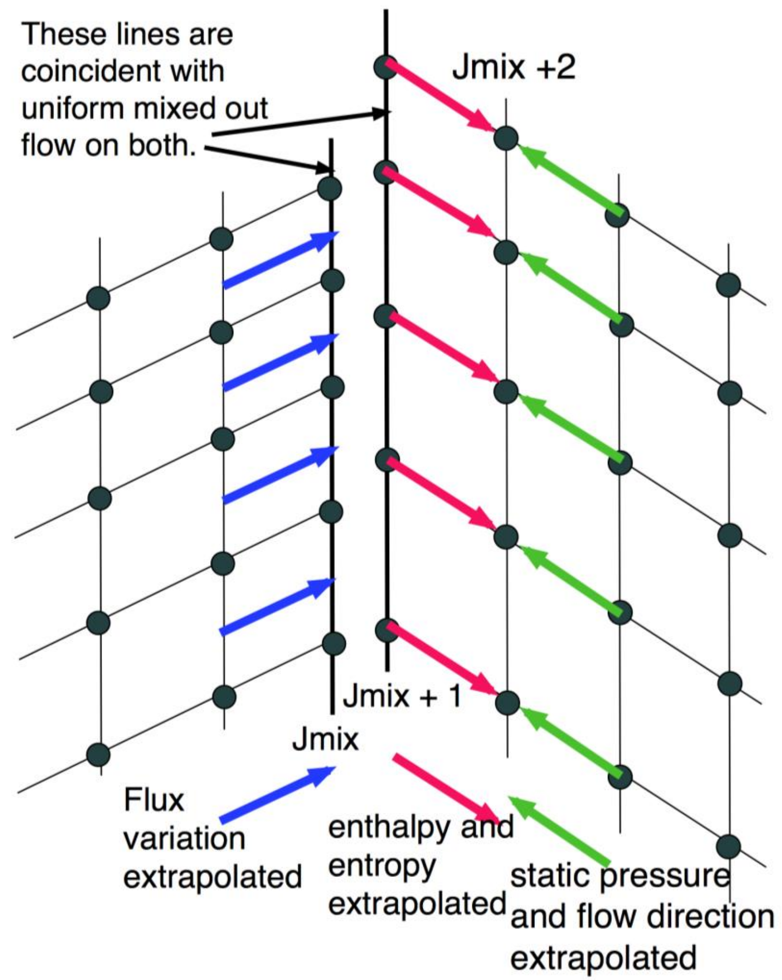


Comparison with the exact solution on a 60deg staggered wedge.
Inlet Mach number =1.6.



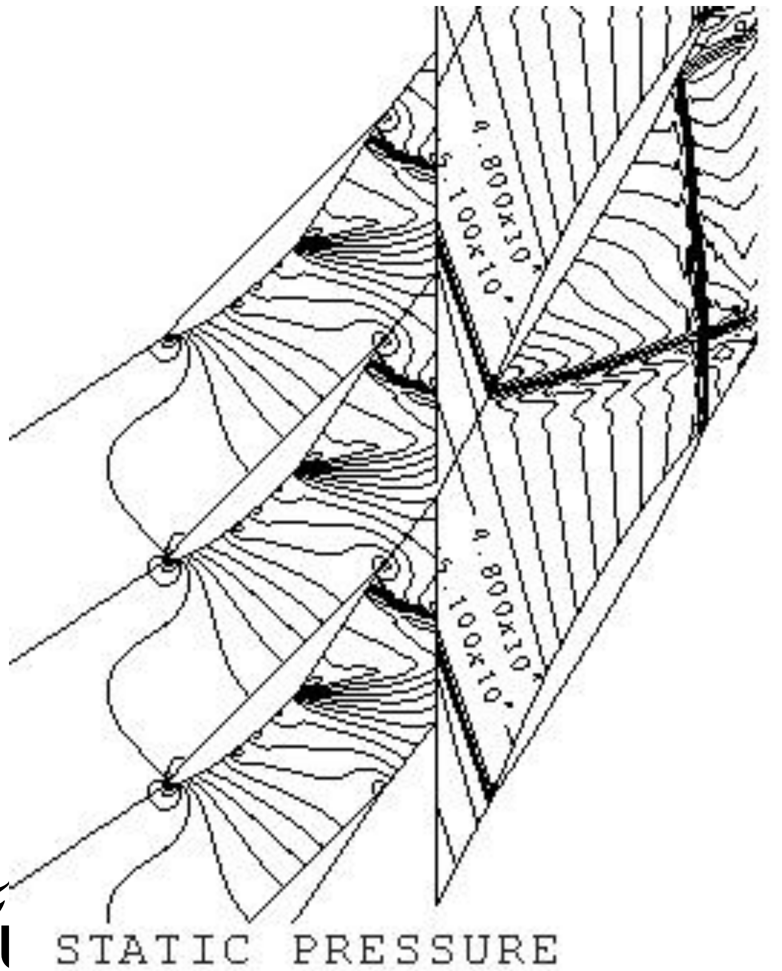
MULTALL uses mixing planes between blade rows in relative motion.

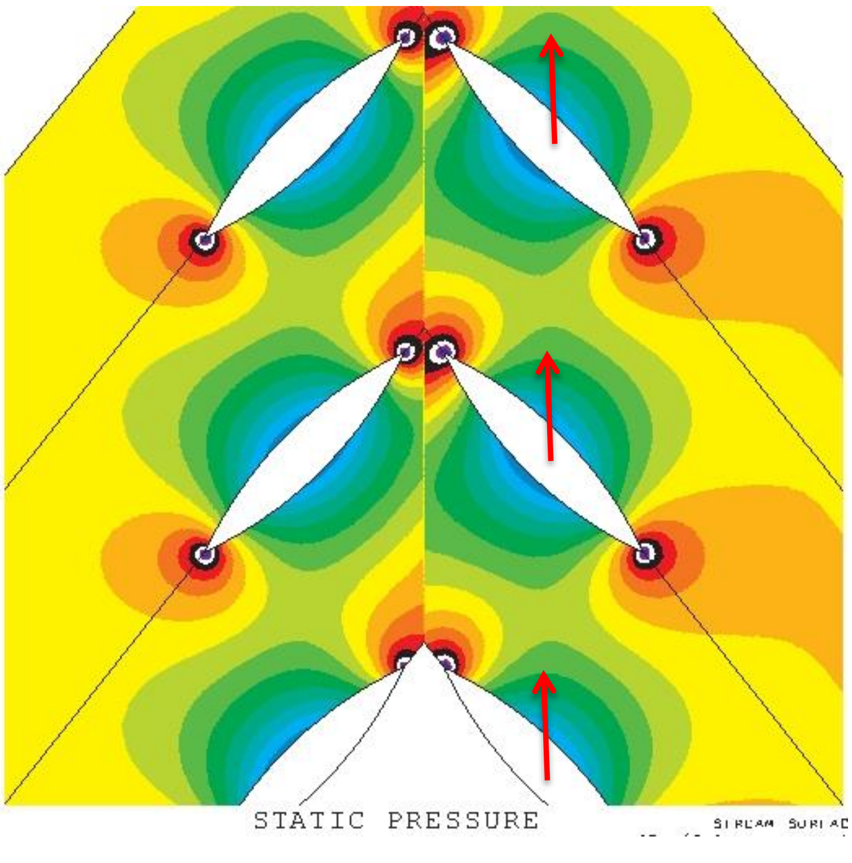
The mixing plane model has been refined over many years. There are two coincident pitchwise grid lines at the mixing plane. The flow quantities are extrapolated onto these lines and the flow between them is matched by a 1D time-marching procedure.

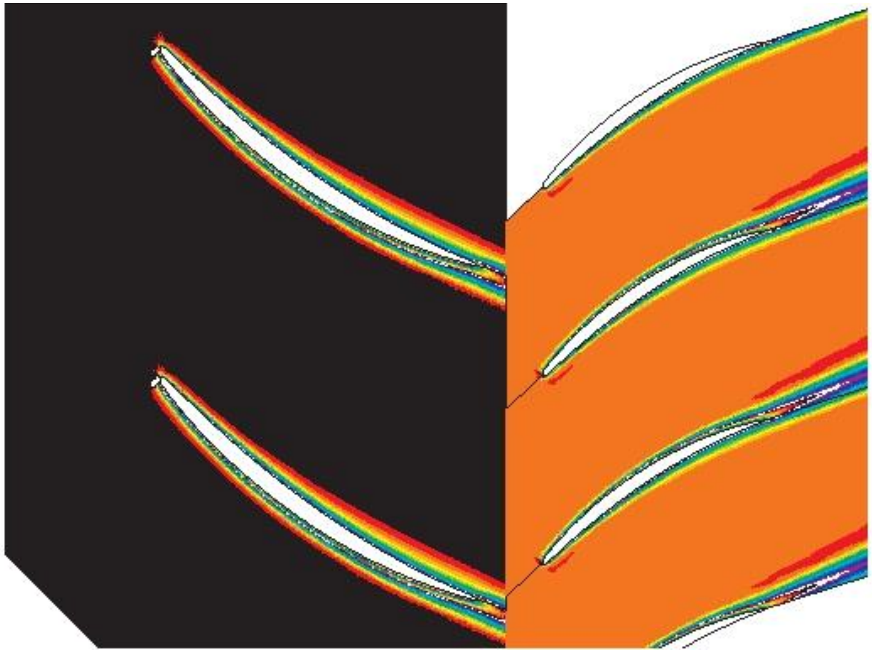


TU Delft

The mixing plane should allow pressure waves to intersect it without reflection and should mix out a potential flow without any loss. The flow downstream of it should have pitchwise uniform entropy and relative stagnation enthalpy.

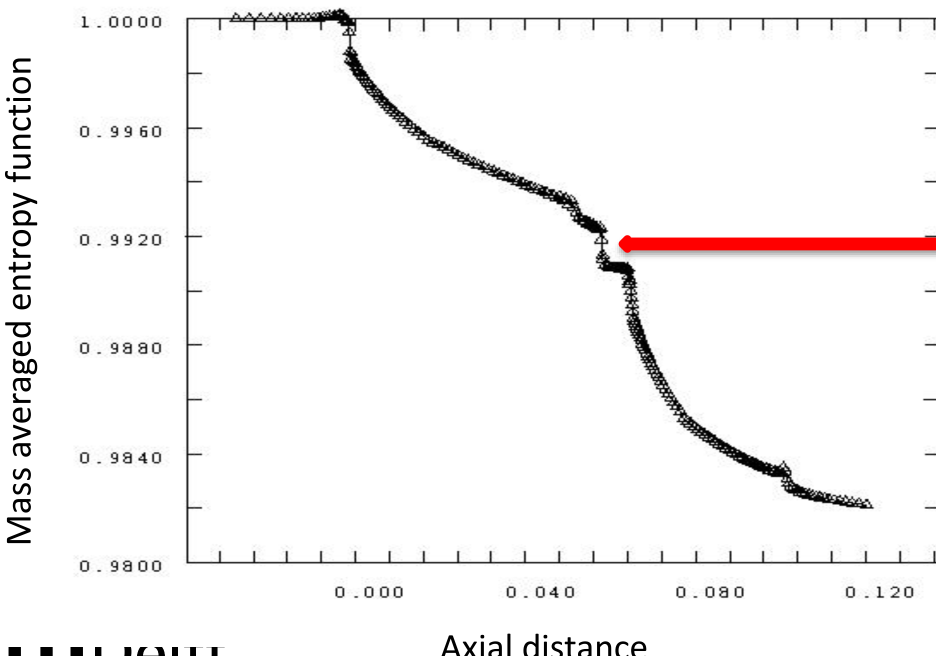






Entropy contours, the entropy is pitchwise uniform immediately downstream of the mixing plane.

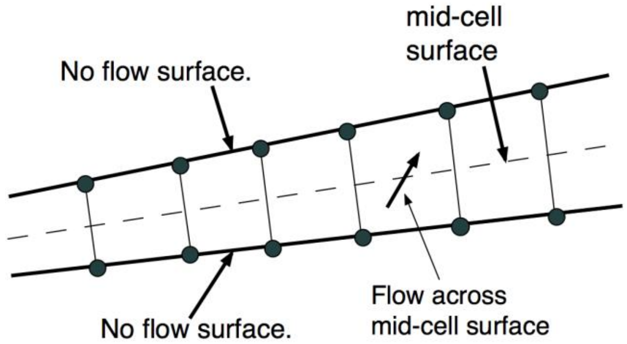
Entropy changes at a mixing plane.



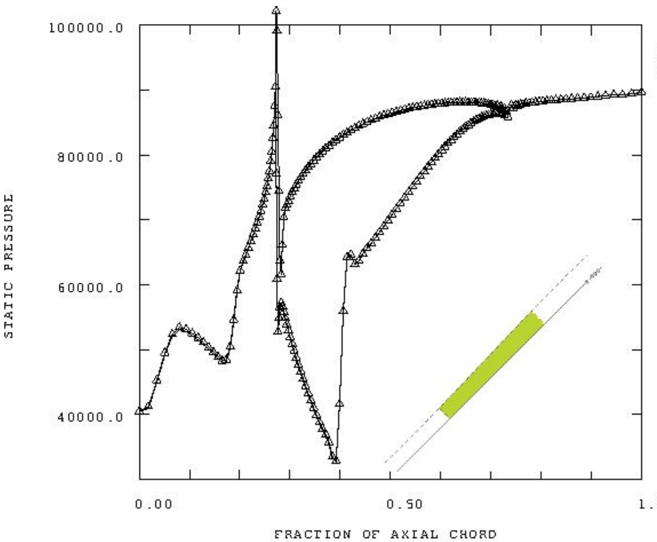
The mass averaged entropy increases at the mixing plane as required for the mixing loss.

MULTALL can be run as a quasi-3D blade-to-blade solver by setting a single cell in the spanwise direction.

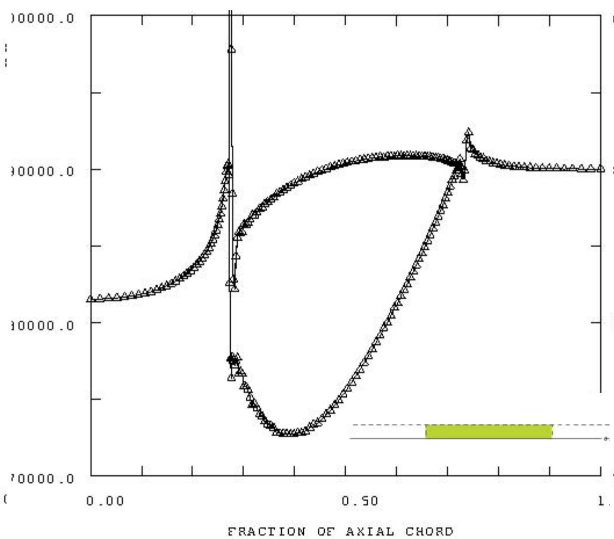
Any flow across the imaginary mid-cell line generates a pressure difference between the two end walls which keeps the flow on the stream surface.



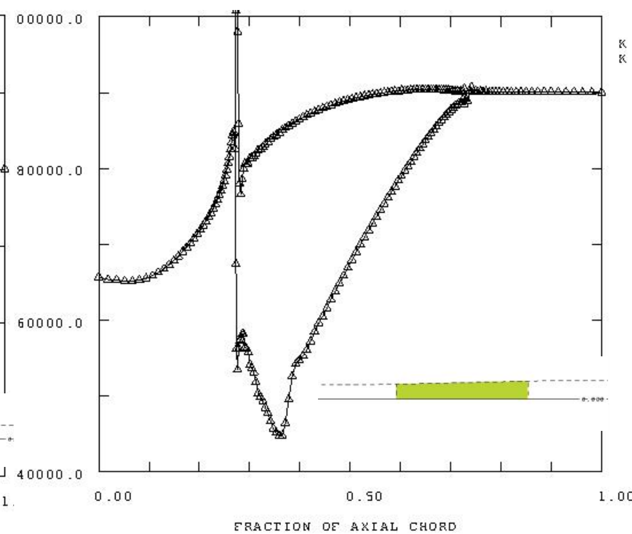
Run times are of order 5-10 seconds per blade row.



45deg flare + 20% radius change.



2D cascade. Constant stream surface thickness.



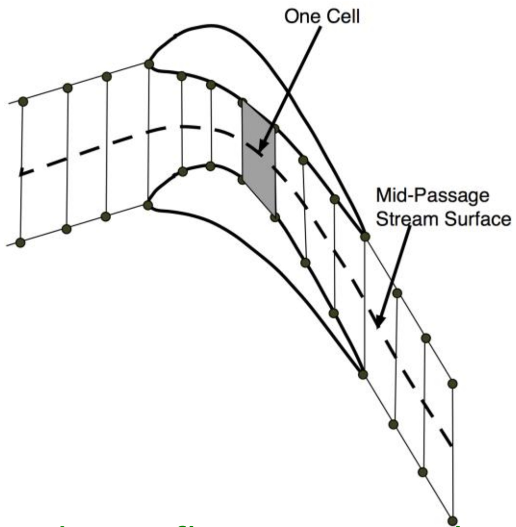
Constant radius + 25% SS divergence.

Same compressor blade row with different stream surfaces

THROUGHFLOW CALCULATIONS

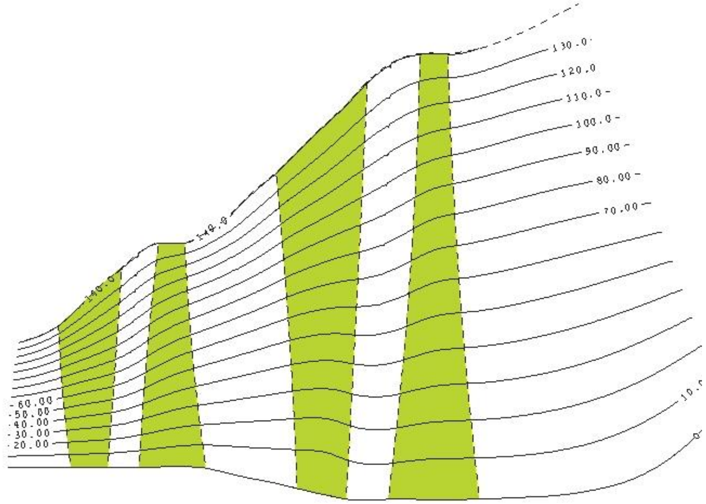
The code can be run as an axisymmetric throughflow calculation by setting a single cell in the pitchwise direction.

The full 3D geometry is input but far fewer grid points can be used.

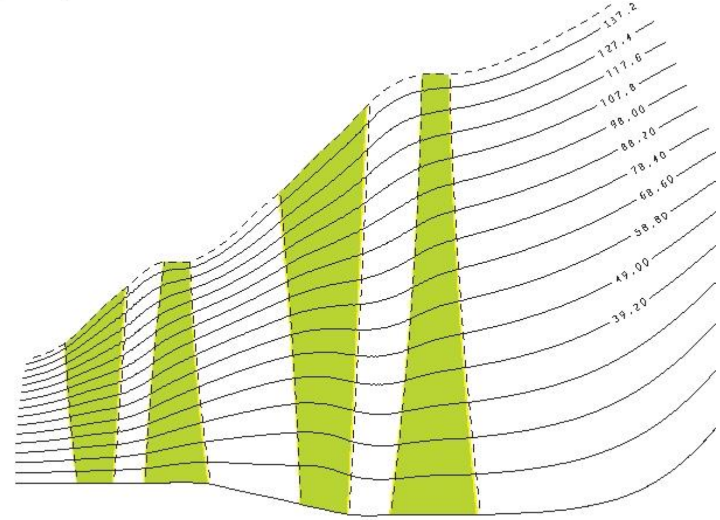


As with the B-to-B method, any flow crossing the imaginary mid-passage line is made to increase the pressure on one blade surface and decrease it on the other. This gradually builds up a blade loading which keeps the flow on the mid-passage line. Deviation and incidence to the mid-passage line are added.

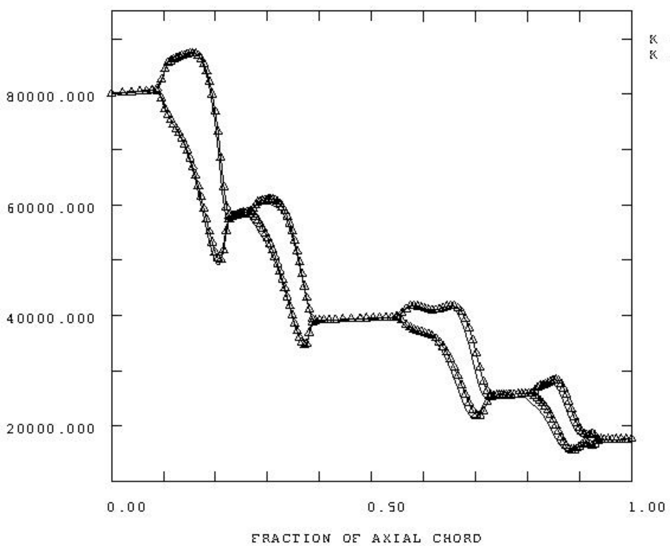
Although the blade loading calculated in this way is only a rough approximation to the true loading its overall magnitude is compatible with the momentum change.



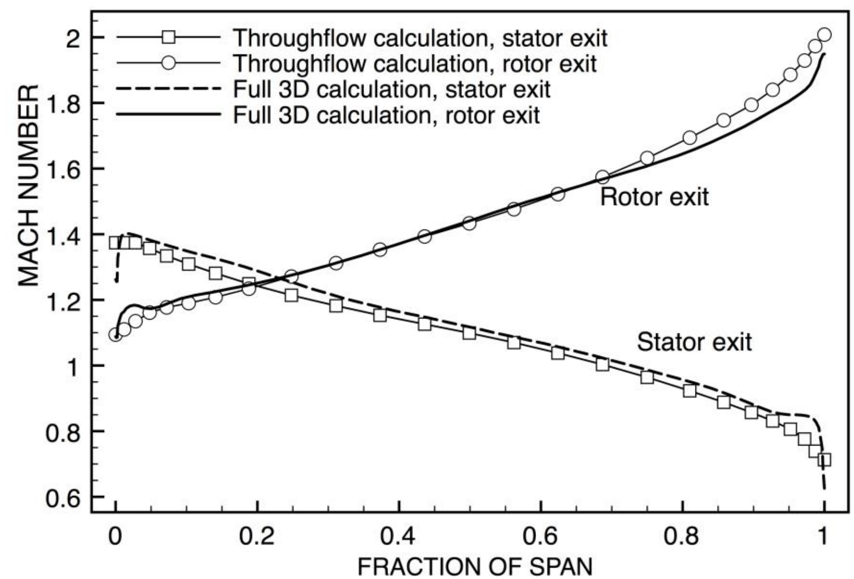
LP Steam turbine. Streamlines.
Full 3D .



LP Steam turbine
Streamlines. Throughflow.



LP steam turbine, blade surface pressure distributions from throughflow.



Comparison of throughflow and full 3D Mach numbers in LP steam turbine.

TUDelft

The throughflow method works on a wide variety of machines.

Its advantages over conventional (streamline curvature) throughflow methods are:

- It gives a crude estimate of blade loading.
- It predicts the 3D effects of blade stacking.
- The stability is not limited by the grid aspect ratio.
- It predicts viscous losses on all solid surfaces via a very simple skin friction model.
- It predicts tip leakage flows and losses.
- It predicts the growth of endwall boundary layers, but not the associated secondary flows.
- It works with a specified exit pressure rather than a specified mass flow.
- It works with choked blade rows, including predicting supersonic deviation.
- It predicts normal shock waves but not oblique shocks.
- It is very easy to change from throughflow to full 3D calculation – change 2 lines of data.

Its disadvantages are:

- It does not work in the inverse (design) mode.
- Run times are 5-10 seconds per blade row rather than a fraction of a second with streamline curvature.
- It requires a blade shape, although this can initially be a simple guess.
- It does not give realistic pressure distributions for transonic compressors, but nor does any throughflow method.
- Users are not so familiar with the method.

EXAMPLE OF USE OF THE DESIGN SYSTEM

Specify a large axial compressor with:

Mass flow rate 50kg/s
Stagnation pressure ratio = 2.0
Rotational speed = 5000 rpm
Constant tip diameter = 1m
Ambient inlet conditions 1bar, 300K .

Assuming an isentropic efficiency = 90% , a 3 stage machine with a moderate stage loading coefficient = 0.36 should satisfy the requirement.

The inlet Mach number at the tip would be near sonic so to reduce this 15° of inlet swirl is chosen. This is gradually reduced so there is no swirl at exit.

Run MEANGEN with specified inlet and outlet absolute flow angles (different for each stage), flow coefficient = 0.6, loading coefficient = 0.35.
Design on the tip stream surface.

Initially use a fairly coarse grid, 37x105x37, points per blade row. No tip clearance or hub seal leakage.

For the initial runs make changes only in MEANGEN to get good flow on the design stream surface, no attempt to optimise the blade sections at other radii.

M Produce an initial layout by running Meangen with default parameters and screen input. The solution has slightly low mass flow and pressure ratio.

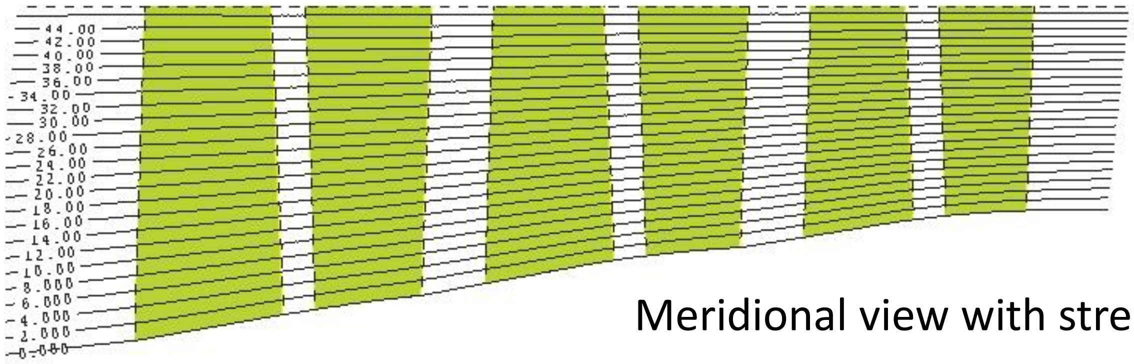
M Increase the guess of deviation angles to increase the pressure ratio. Increase the blade thickness and blade numbers. Several runs to obtain flow and pressure ratio close to the specified values.

M The blades are mid-loaded. Move the point of maximum thickness and point of maximum camber forwards to obtain more fore-loaded blades. Reduce the trailing edge thickness. Several runs to obtain the required mass flow and pressure ratio.

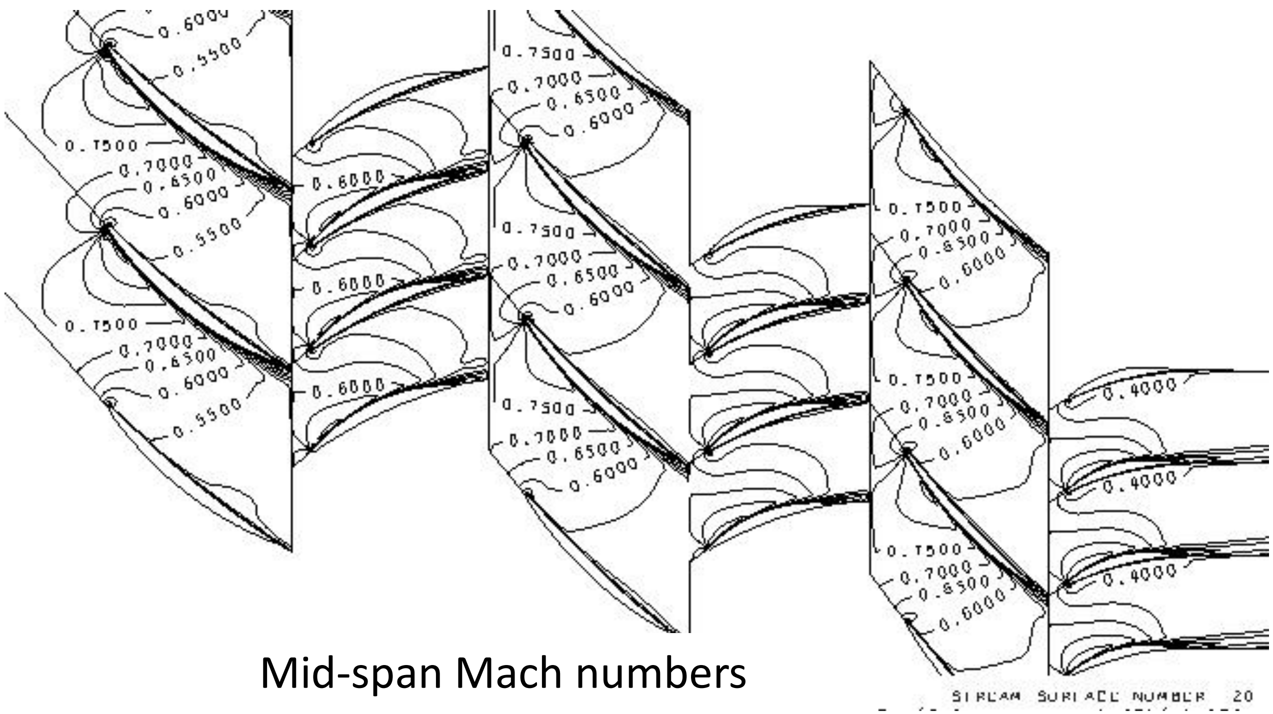
M Move the point of maximum camber slightly further forwards. The blades now have good surface pressure distributions but there are high incidences on some sections. Adjust the average incidence angles for each row. Several runs.

Total about 12 3D runs. Each run takes about 12 minutes on a single processor.

Results from the FIRST run using MEANGEN alone.



Meridional view with streamlines



Mid-span Mach numbers

Now make further changes by editing the STAGEN input file to refine the blade sections and adjust the incidences along the span

- Change to a finer 55x140x55 point grid with tip gaps on all rotors, 5 cells in each tip gap. Adjust the incidence angles along the span. Rotor 3 is most highly loaded so increase its blade numbers. Several runs.

- Add hub shroud leakages on all stators. Adjust incidences. Several runs.

- Bow all stators, with pressure surfaces leaned towards the endwalls, to reduce endwall loadings. Re-adjust the incidences. The performance is now very close to specification with a predicted isentropic efficiency of 91.5% .

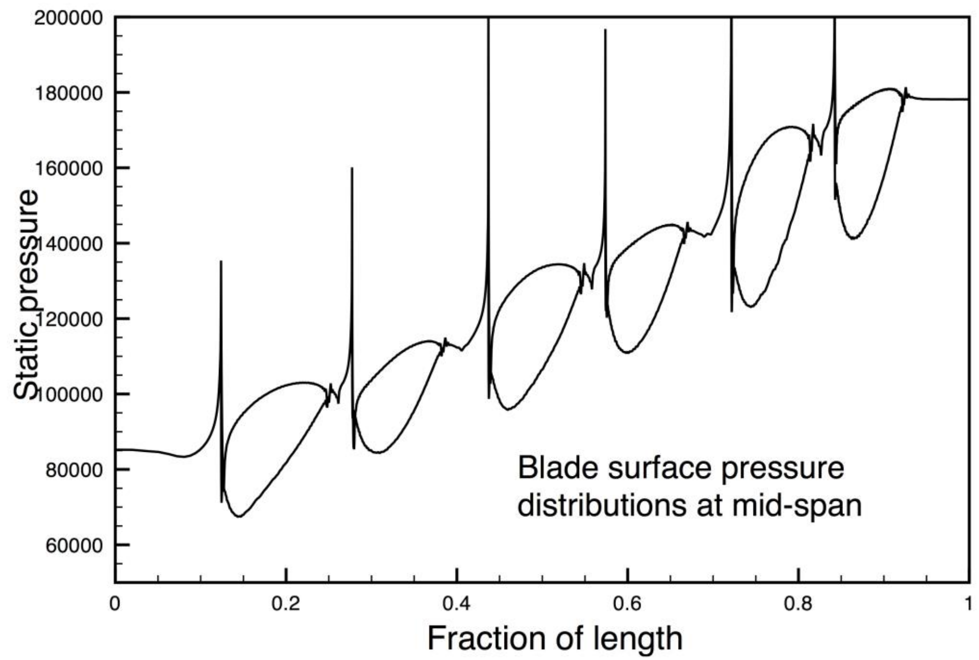
- To improve the stall margin add forwards sweep at all rotor tips by increasing the chord by 15% with a fixed trailing edge location.

- Further refine the blade incidences. Several runs.

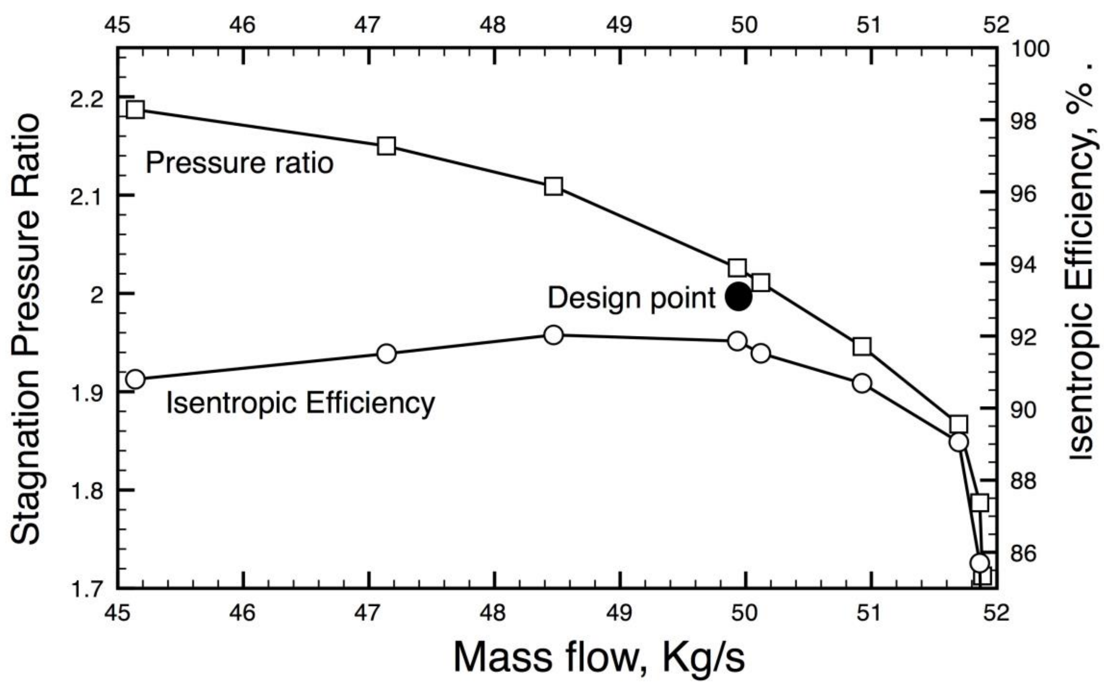
- Run a characteristic from choke to stall.

- Run a very fine grid solution, 83x299x83 points per row, at the design point to check for any grid sensitivity. The mass flow and pressure ratio each changed by about 0.1% and the efficiency was 0.4% lower.

Total about 17 3D runs, each taking about 30 minutes when starting from the previous solution.



Uniformly loaded blades



Pressure ratio and efficiency slightly exceed the design targets

In total the design used about 30 3D runs, requiring about 12 hours CPU time on a single processor of a LINUX desktop (home) computer.

However, more time than this is necessary for “thinking” about what changes to make next.

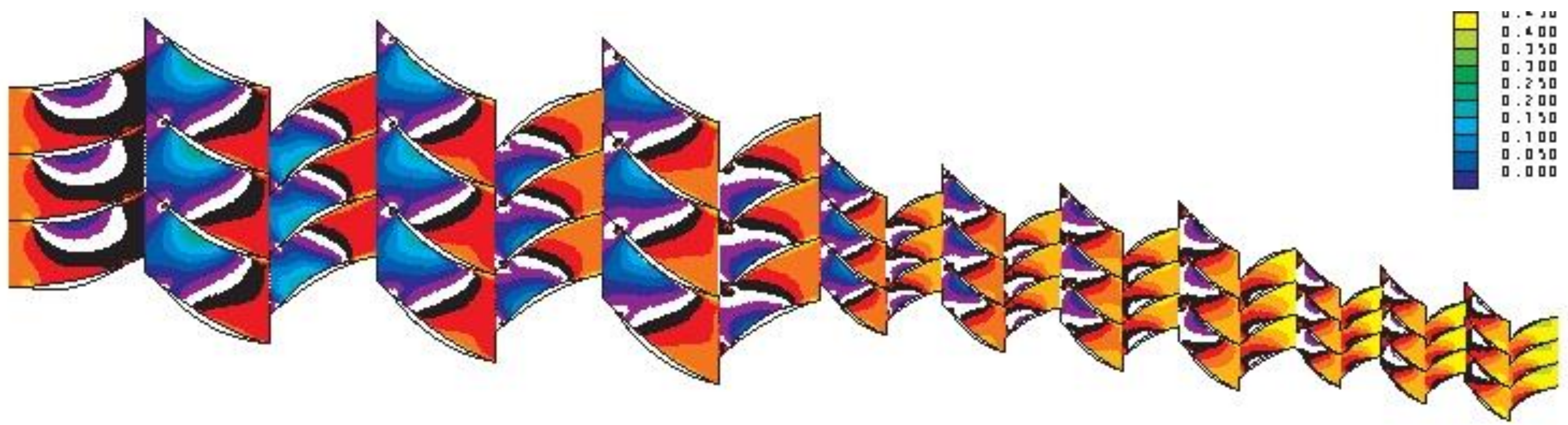
Allowing for this, the design could be completed in about 3 person days.

CONCLUSIONS

The design system is relatively simple, easy to use and fast to run.

However, as with any design process, it requires an experienced user to know what geometrical changes are needed to produce the required flow behaviour. Repeated use of the design system is a good way of acquiring such experience.

MULTALL is a useful tool in its own right. It is simpler and faster than most CFD codes and some of the techniques used in it may be of use to other CFD developers.



TU Delft

The FORTRAN source codes, user manuals and sample data sets can be downloaded as a folder named ‘multall-open’ from ‘dropbox’ using the following link.

https://www.dropbox.com/sh/8i0jyxzjb57q4j4/AABD9GQ1MUFwUm5hMWFylucva?dl=0

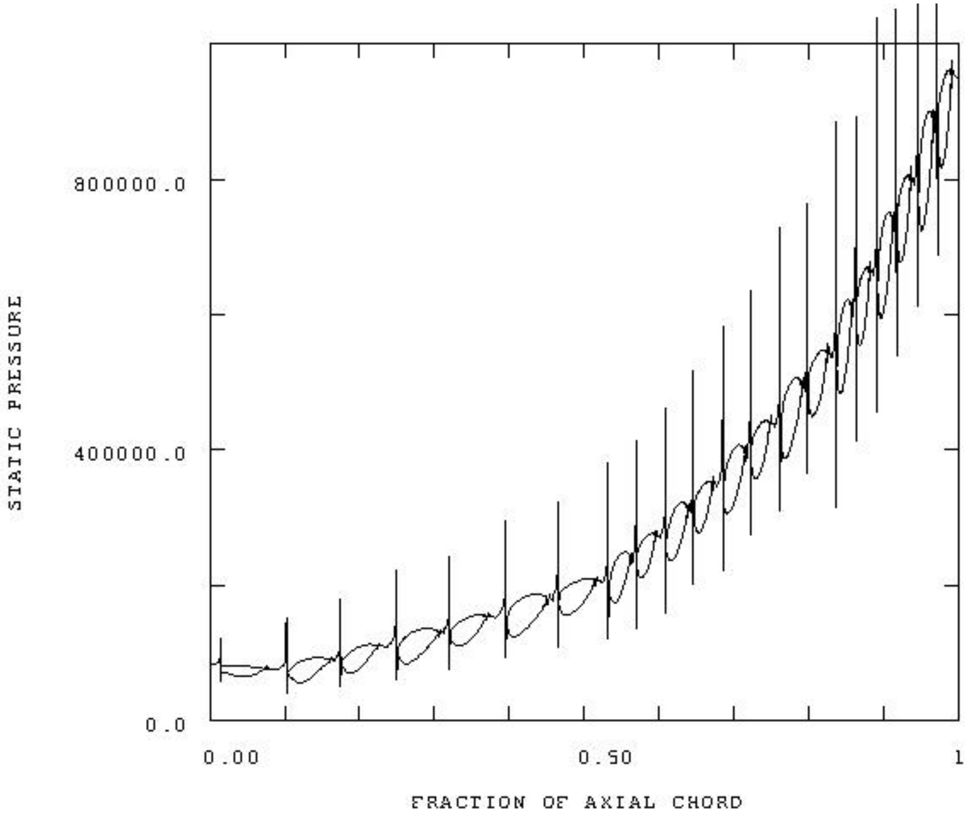
You do not need a ‘dropbox’ account to access this.

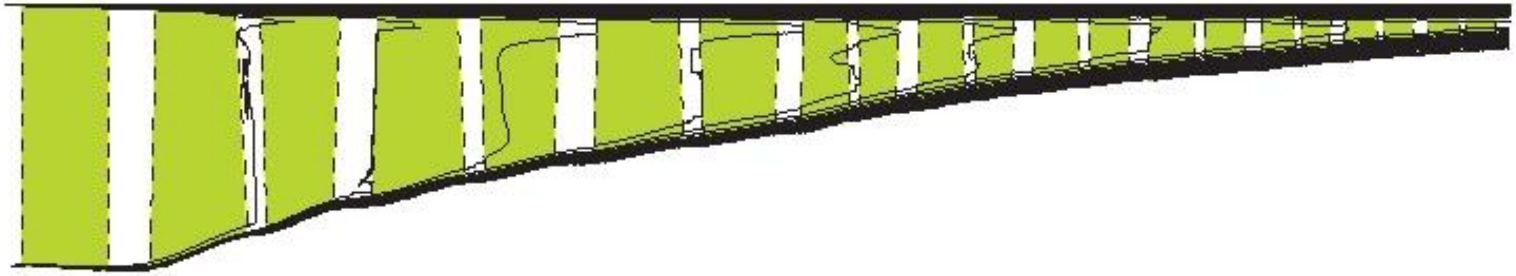
A brief description of the system and a copy of the link is also available on the web site

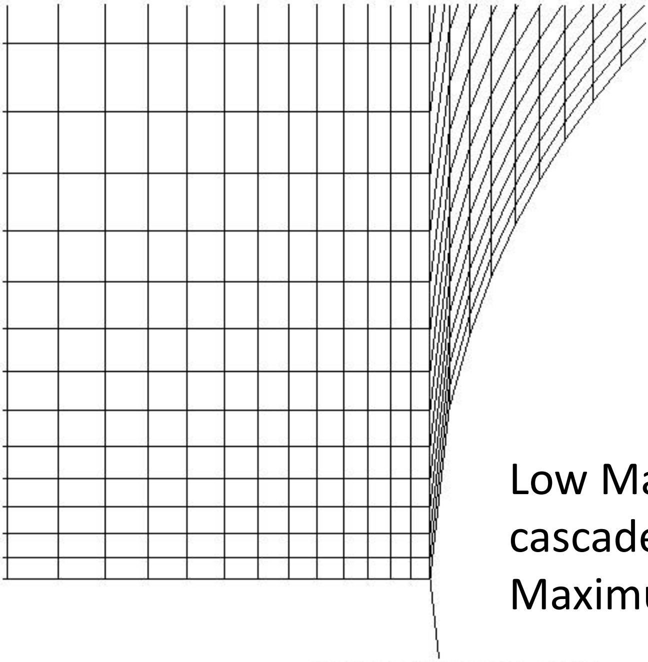
https://sites.google.com/view/multall-turbomachinery-design

Any future developments to the programs or changes to the link will be announced on this web site.

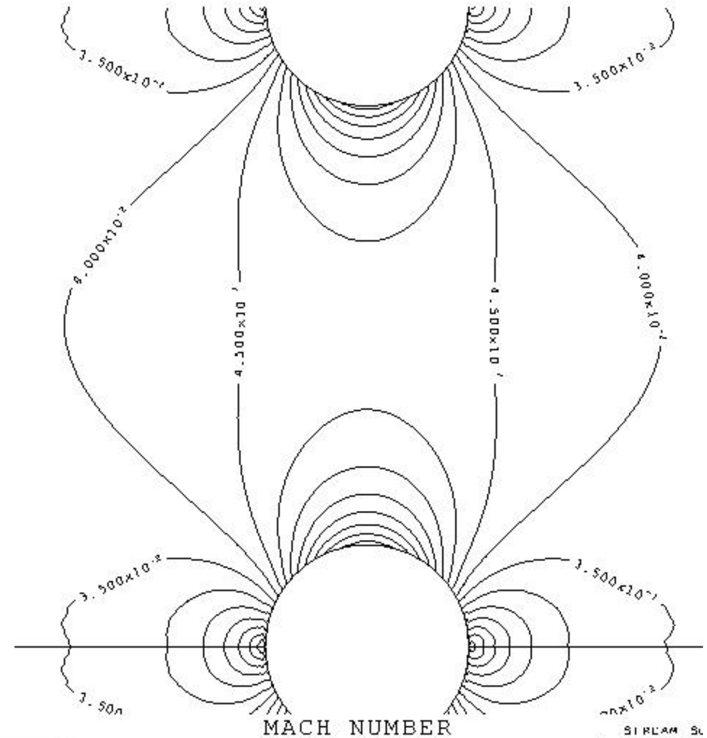
10 stage compressor with overall pressure ratio 10:1

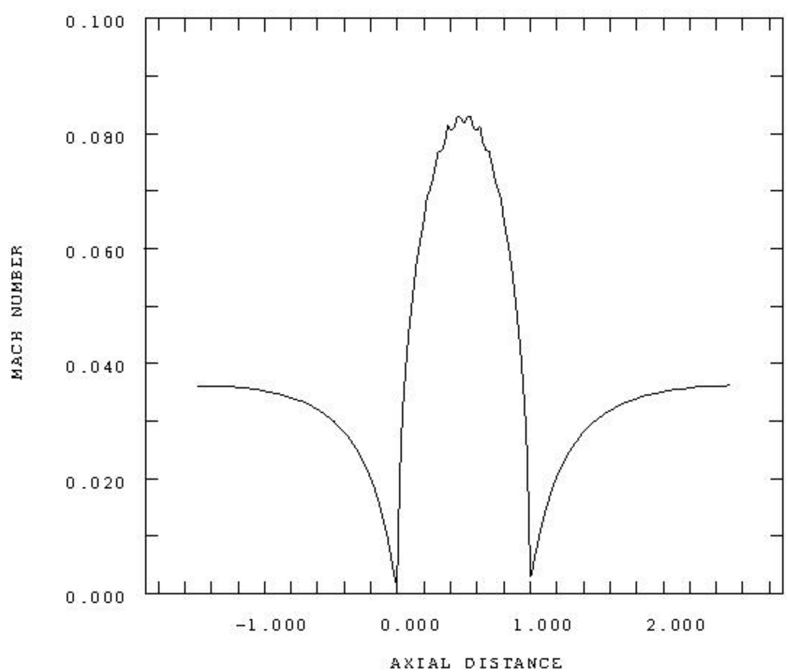


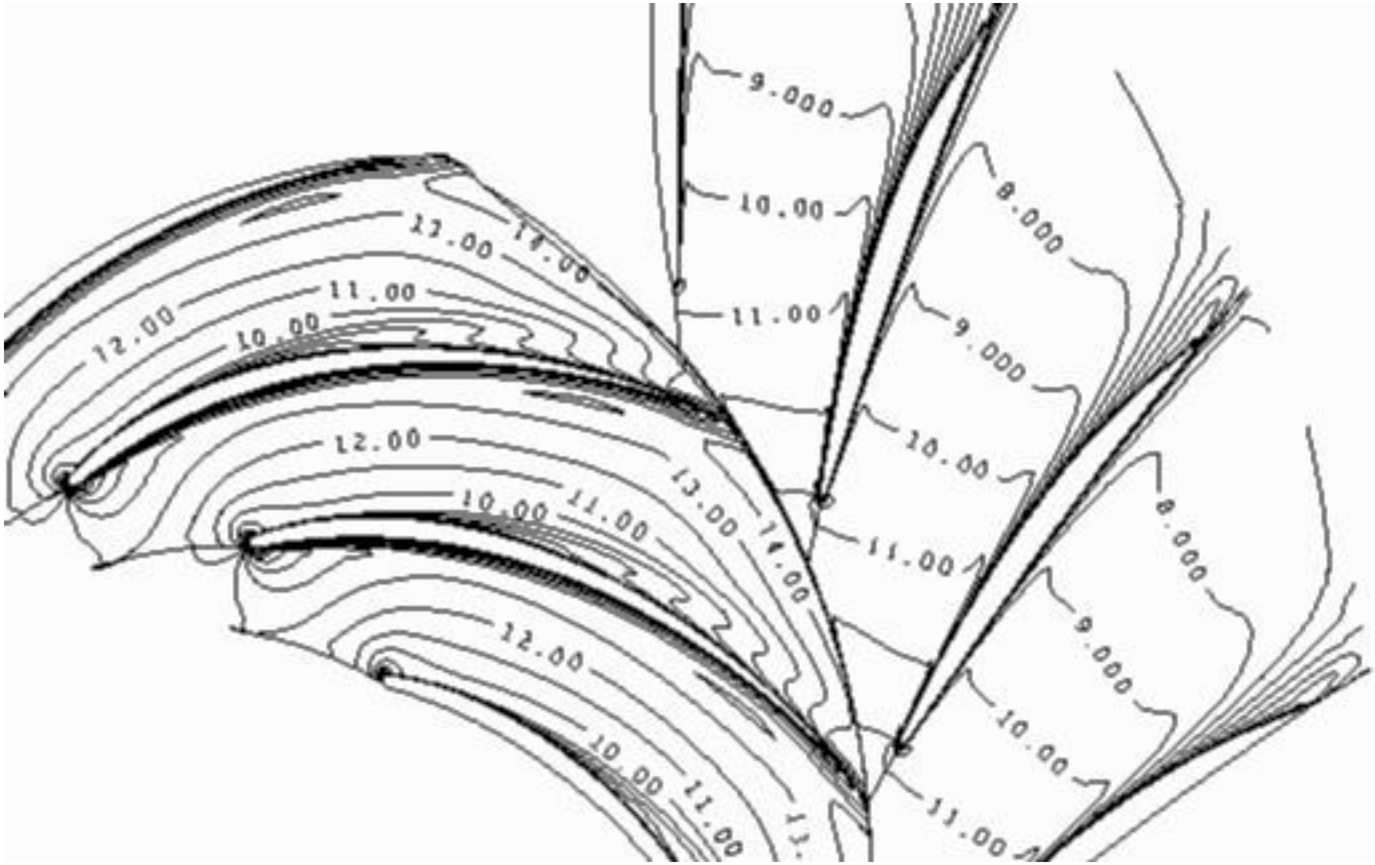




Low Mach number flow around a cascade of cylinders.
Maximum Mach number = 0.08.







Flow in a water pump. Contours of relative velocity.

TUDelft

Appendix

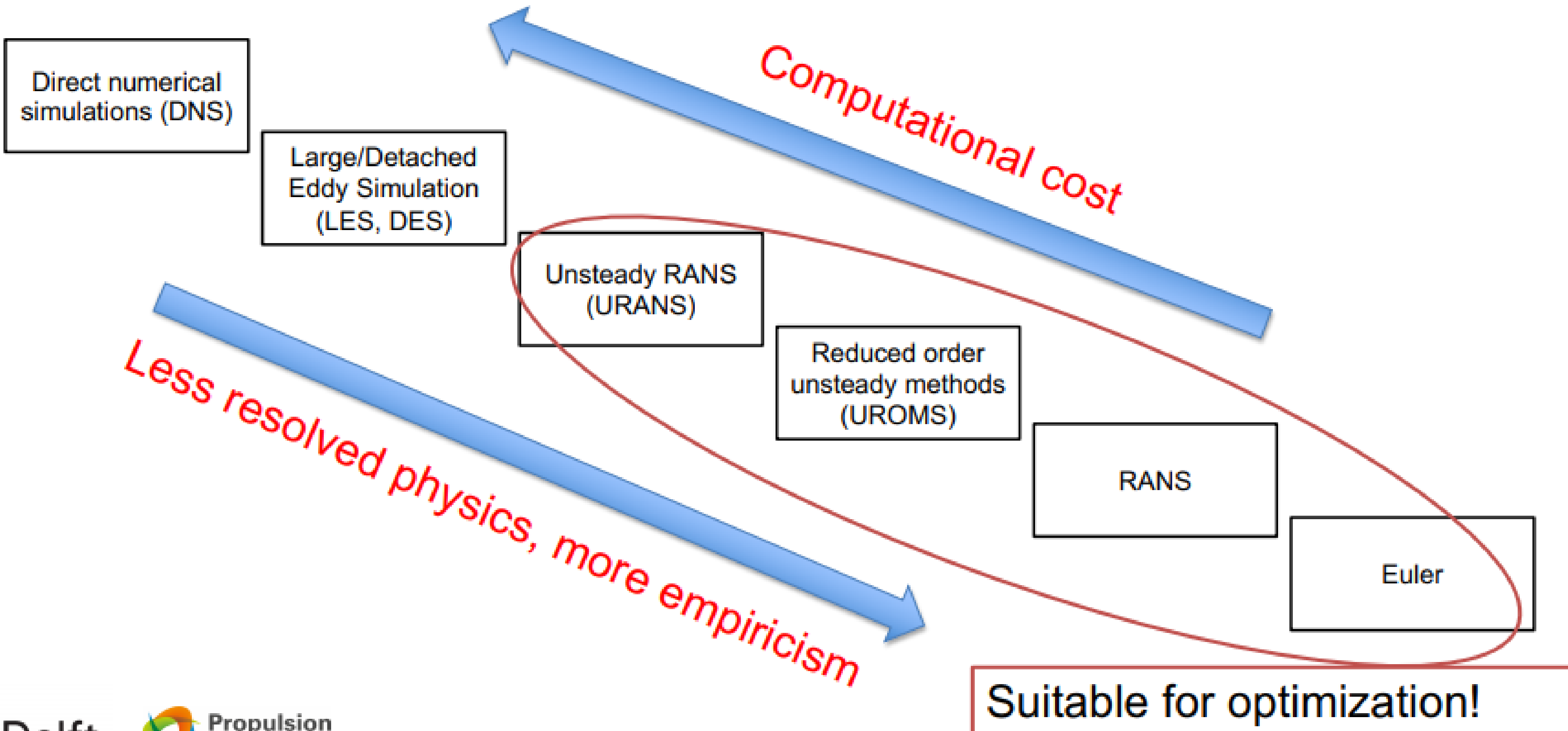
Basics on CFD for turbomachinery

Acknowledgements: A. Rubino

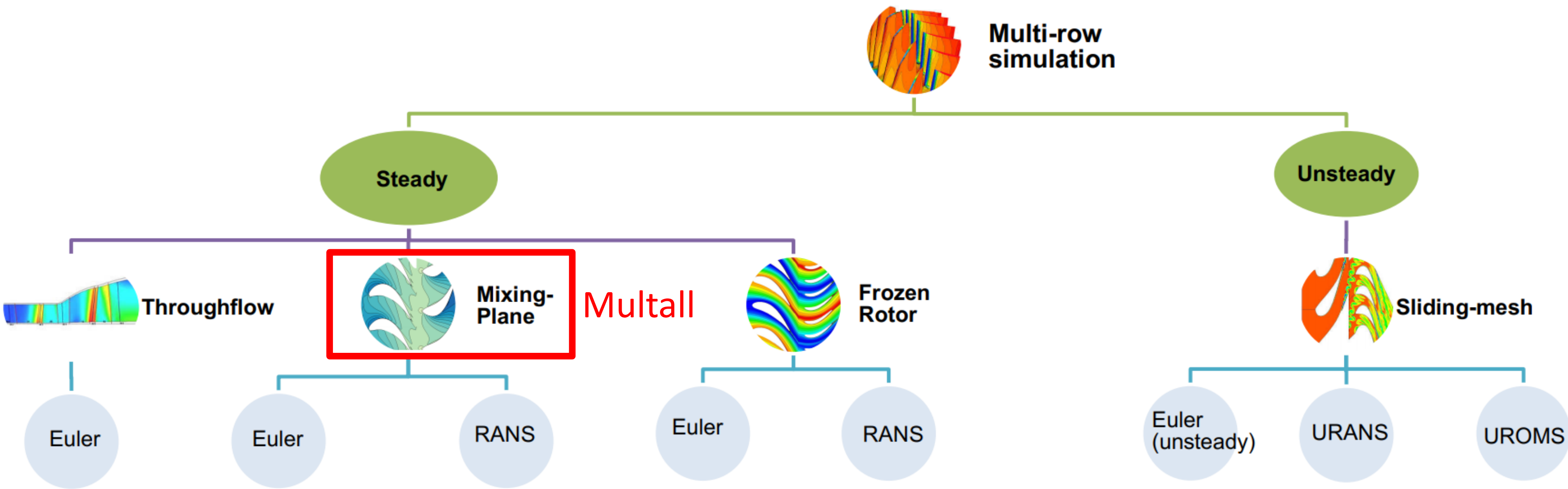
See also: http://www.cfd-online.com/Wiki/Best_practice_guidelines_for_turbomachinery_CFD

TUDelft

Hierarchy of CFD models

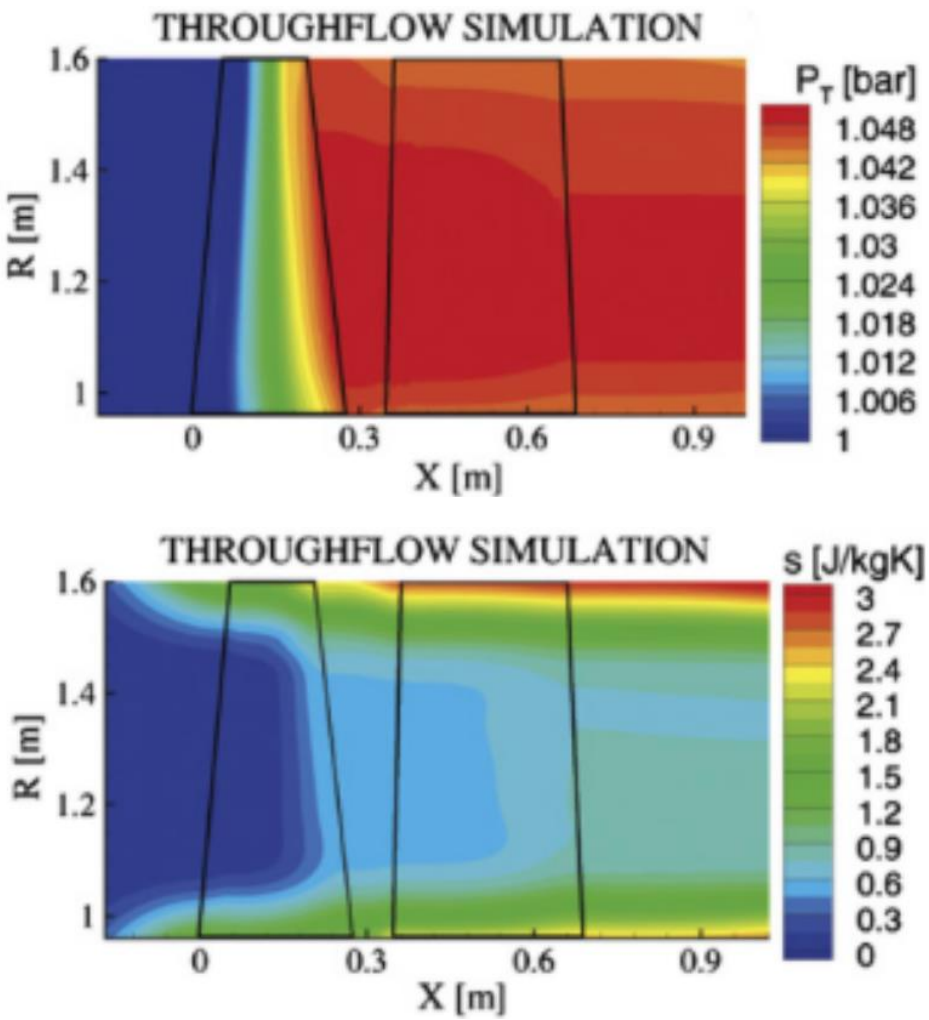


Multi-row simulation methods



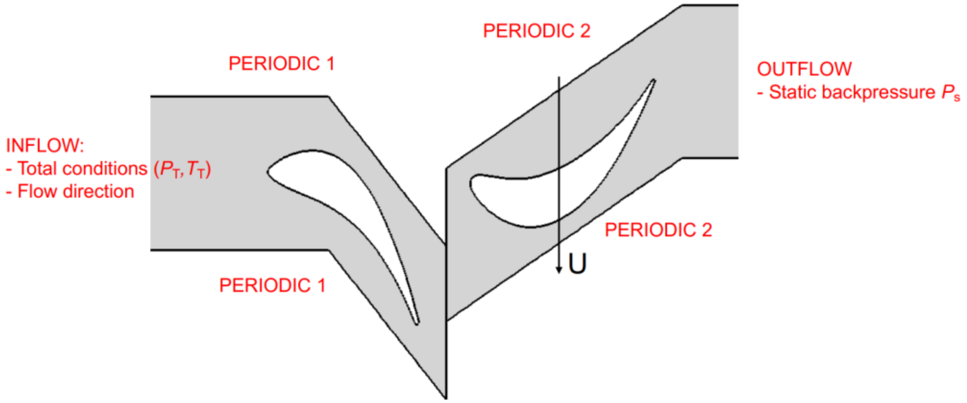
Throughflow simulations

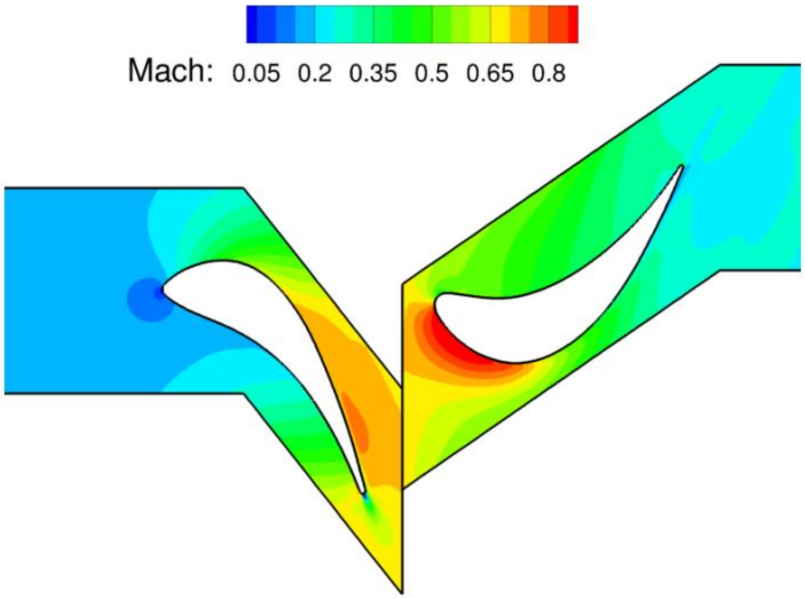
- Loss and deviation correlations.
- Accuracy determined more by correlations than by numerical method.
- Assumptions:
  - Circumferentially-averaged flow.
  - Axisymmetric.



Mixing-plane simulations

- Instantaneous mixing at mixing plane.
- Non-uniform flow after mixing plane (conservation of mass, momentum and energy).
- Low computational cost.
- Use for efficiency estimation.

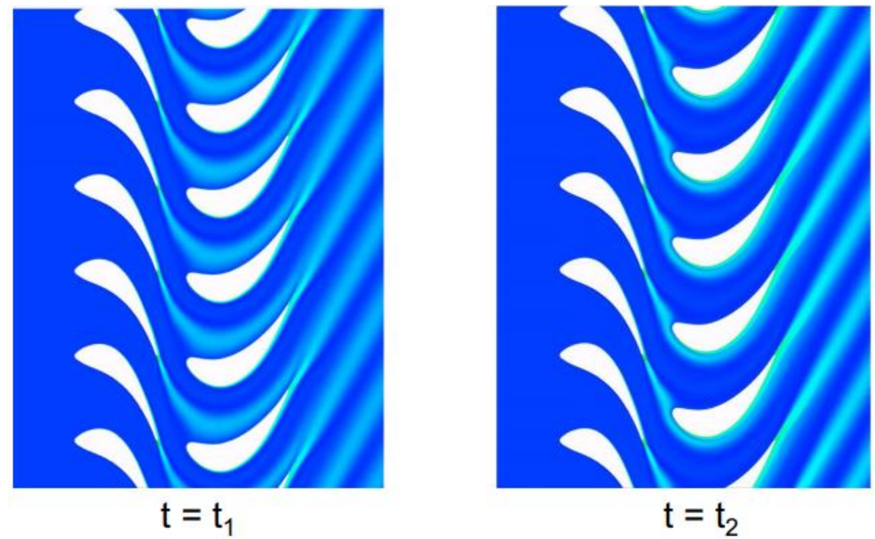


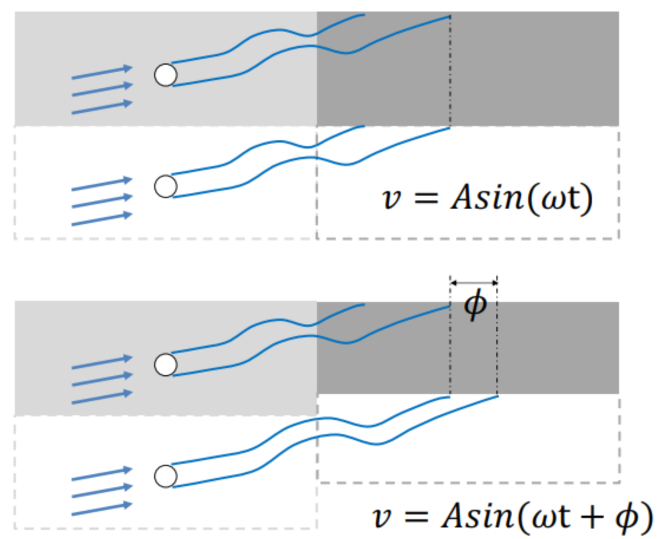


TU Delft

Frozen-rotor simulations

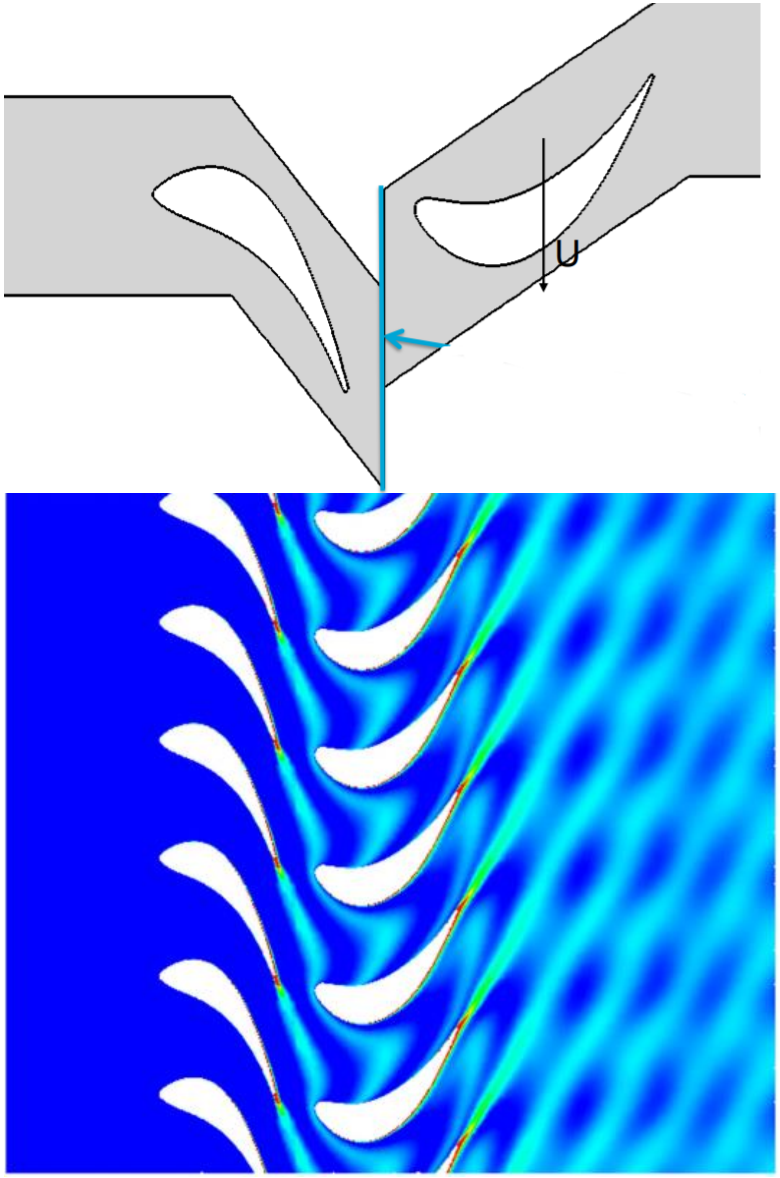
- Rotor kept frozen with respect to stator.
- Results depend on relative position rotor-stator.
- If pitch ratio between rotor-stator not integer, periodic BCs cannot be applied because of temporal lag. Use phase-lag periodic BCs.
- Use to initialize unsteady computations with sliding mesh.
- Used for interaction simulations between vaned and vaneless turbomachinery components (rotor-diffuser, impeller-volute, inlet-impeller).





Sliding-mesh URANS simulations

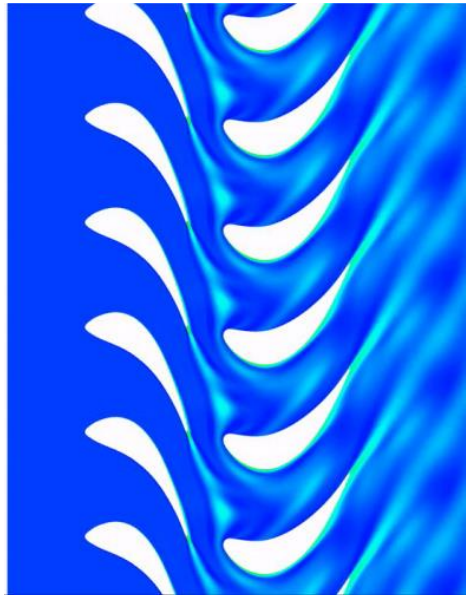
- Unsteady simulations.
- Sliding motion between stationary and rotating mesh.
- URANS as most accurate method for blade row calculations in industry.
- High cost and memory requirements (not used for design optimization).
- Initialize using a steady solution.

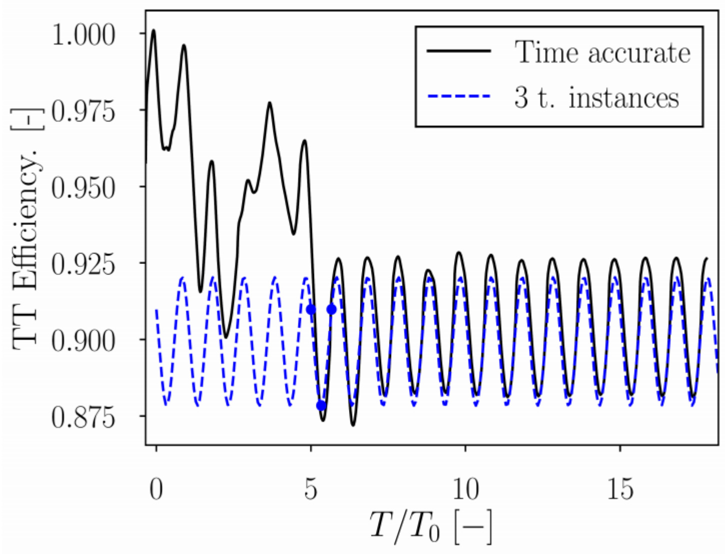


TU Delft

Reduced Order Methods (ROM): Harmonic Balance

- Unsteady simulations with a-priori known discrete set of frequencies (periodic phenomena).
- Unsteady solution only for blade passing frequency harmonics.
- Reduced computational cost, use for unsteady optimization.
- Possible numerical stability problems.
- No initial transient as with URANS.
- For >10 frequencies, computational cost can exceed URANS.
- Stator-rotor interaction (known harmonics).





TU Delft

Limitations of CFD for turbomachinery

- Difficult prediction:
  - Boundary layer transition.
  - Turbulence.
  - Endwall losses.
  - Leakage flows.
  - Leading-edge flow in compressors.
  - Trailing-edge flow in turbines.
- Modelling challenges:
  - Geometrical details.
  - Boundary conditions.
  - Freestream turbulence.
  - Endwall boundary layers.
- Check convergence, physical sense, required information.