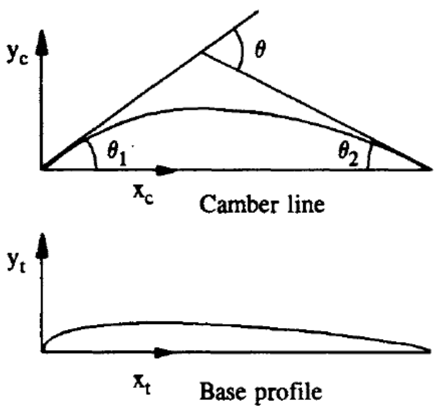
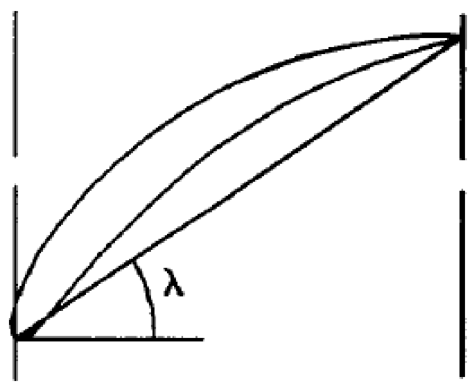
Turbomachinery design
(6 ECTS)

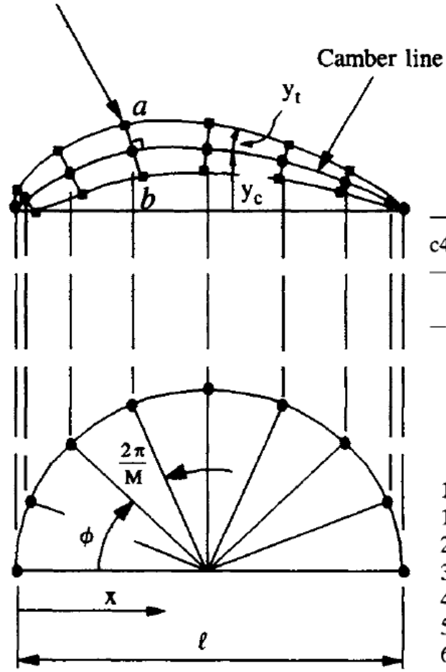
MISES LAB

Cascade profile generation



θ = camber angle
θ_1 = leading edge camber
θ_2 = trailing edge camber
ℓ = blade chord





c4		T4		NACA 0012		NACA 0015		NACA 66-010		NGTEmod	
x_t	y_t	x_t	y_t	x_t	y_t	x_t	y_t	x_t	y_t	x_t	y_t
0.00	0.00	0.00	0.00	0.00	0.000	0.00	0.000	0.0	0.0	0.00	0.000
1.25	1.65	1.25	1.17	1.25	1.894	1.25	2.367	0.5	0.759	1.25	1.375
2.50	2.27	2.50	1.54	2.50	2.615	2.50	3.268	0.75	0.913	2.50	1.910
5.00	3.08	5.00	1.99	5.00	3.555	5.00	4.443	1.25	1.141	5.00	2.680
7.50	3.62	7.50	2.37	7.50	4.200	7.50	5.250	2.5	1.516	7.50	3.195
10.00	4.02	10.00	2.74	10.00	4.683	10.00	5.853	5.0	2.087	10.00	3.600
15.00	4.55	15.00	3.40	15.00	5.345	15.00	6.682	7.5	2.536	15.00	4.180
20.00	4.83	20.00	3.95	20.00	5.737	20.00	7.172	10.0	2.917	20.00	4.550
30.00	5.00	30.00	4.72	25.00	5.941	25.00	7.427	15.0	3.53	30.00	4.950
40.00	4.89	40.00	5.00	30.00	6.002	30.00	7.502	20.0	4.001	40.00	4.820
50.00	4.57	50.00	4.67	40.00	5.803	40.00	7.254	25.0	4.363	50.00	3.980
60.00	4.05	60.00	3.70	50.00	5.294	50.00	6.617	30.0	4.636	60.00	3.250
70.00	3.37	70.00	2.51	60.00	4.563	60.00	5.704	35.0	4.832	70.00	2.450
80.00	2.54	80.00	1.42	70.00	3.664	70.00	4.580	40.0	4.953	80.00	1.740
90.00	1.60	90.00	0.85	80.00	2.623	80.00	3.279	45.0	5.0	85.00	1.500
95.00	1.06	95.00	0.72	90.00	1.448	90.00	1.810	50.0	4.971	90.00	1.270
100.00	0.00	100.00	0.00	95.00	0.807	95.00	1.008	55.0	4.865	92.50	1.170
				100.00	0.000	100.00	0.000	60.0	4.665	95.00	1.080
								65.0	4.302	97.50	0.980
								70.0	3.787	100.00	0.000
								75.0	3.176		
								80.0	2.494		
								85.0	1.773		
								90.0	1.054		
								95.0	0.408		
								100.0	0.0		

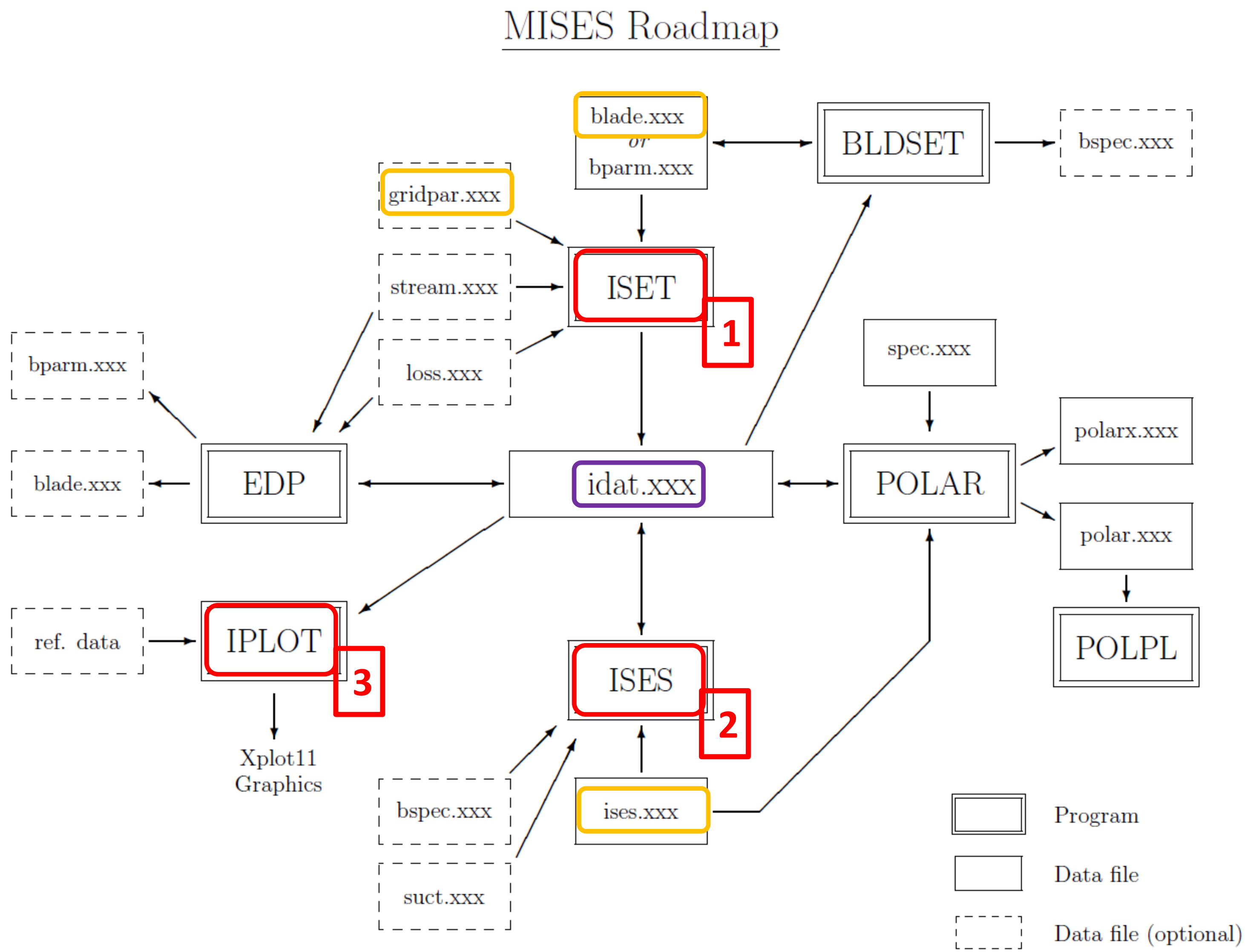
MISES description

The MISES system is a collection of programs for cascade analysis and design. This includes programs for grid generation and initialization, flow analysis, plotting and interpretation of results, and an interactive program to specify design conditions.

The block diagram for these programs is given at the end of this manual. The basic grid and flow data file for a case is the so-called *state file* named `idat.xxx`, where "`xxx`" is an extension suffix used to designate the case being run. The state file is initialized using **ISET** from the blade geometry file `blade.xxx` and the optional stream surface geometry file `stream.xxx` and the prescribed loss schedule file `loss.xxx`. The flow solver **ISES** uses the state file and a flow condition file `ises.xxx` that specifies the flow conditions and program configuration flags. The **POLAR** program performs the same calculations as **ISES** , but for a set of specified parameters. Additional design condition information can be interactively added to the state file using the **EDP** pressure edit program. The **IPLOT** program plots the flow and geometry data from the state file in an interactive plotting session.

All flow variables used by MISES are defined in the relative frame.

MISES description



MISES description

3 Streamsurface and Blade geometry definition

The blade airfoil and grid domain geometry are defined in the standard $\boxed{m' - \theta \text{ streamsurface}}$ coordinate system, shown in Figure 1. With z denoting the cylindrical axis coordinate and r the local streamsurface radius, the m' coordinate is defined by

$$m' = \int \frac{dm}{r} = \int \frac{\sqrt{dr^2 + dz^2}}{r}$$

while θ is the usual circumferential angle. The total arc length increment ds in the stream surface is given by

$$ds = \sqrt{dr^2 + dz^2 + (r\,d\theta)^2} = \sqrt{dm^2 + (r\,d\theta)^2} = r\sqrt{dm'^2 + d\theta^2} = r\,ds'.$$

The transformation from physical space to the $\boxed{m' - \theta \text{ plane is angle-preserving}}$. Hence, no transformation is required for flow angles or surface normal directions. This simplifies imposition of boundary conditions such as a specified inlet flow angle, or the normal-offsetting of the inviscid flow by the viscous displacement thickness.

For 2-D cascades, r becomes an arbitrary constant scaling length, and hence

$$m' = \frac{z}{r} \qquad \text{(2-D cascade).}$$

MISES description

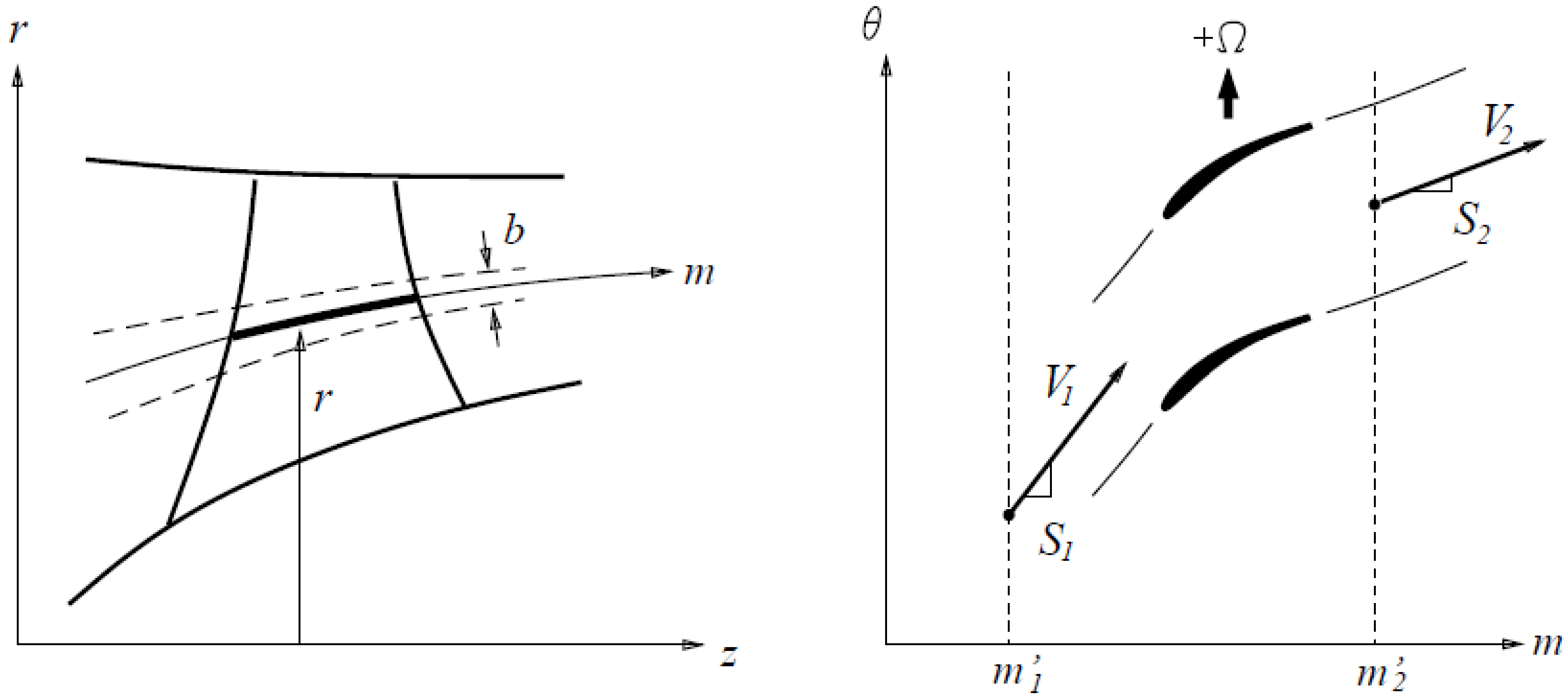


Figure 1: Streamsurface definition.

4 Input Files

`blade.xxx` *Required* unless `bparm.xxx` is used. Defines blade shape via a relatively large number of coordinate pairs.

`ises.xxx` *Required*. Specifies the flow conditions and solver control flags.

MISES description

BLADE file description

4.1 Blade coordinate file `blade.xxx`

The discrete blade airfoil geometry coordinates, however generated, are specified in the formatted file **`blade.xxx`**, which is used by the initialization program **ISET** to define the initial streamline grid. This file has the following structure.

```
NAME
SINL SOUT CHINL CHOUT PITCH
X(1) Y(1)        ←— Blade 1
X(2) Y(2)
 .    .
 .    .
X(I) Y(I)
```

`NAME`	is the name of the case, not more than 32 characters.
`SINL`	is the initial $S_1 = \tan(\beta_1)$, the tangent of the inlet flow angle relative to the axial direction. This is the default inlet flow slope at **ISET** startup, and can be changed interactively.
`SOUT`	is the initial $S_2 = \tan(\beta_2)$. With the new panel-solver grid generator in **ISET**, `SOUT` is no longer used.
`CHINL`	is the distance in m' from the blade 1 leading edge to the grid inflow plane.
`CHOUT`	is the distance in m' from the blade 1 trailing edge to the grid outflow plane.
`PITCH`	is the circumferential pitch of the cascade in radians $= 2\pi/(\text{number of blades})$

The blade coordinates `X(1),Y(1)` through to `X(I),Y(I)` are the m', θ coordinates of the blade surface, starting at the trailing edge, going round the leading edge in either direction, then going back to the trailing edge. For multiple blades the first blade definition is ended

ISES file description

4.8 Flow condition file ises.xxx

The file defines the flow conditions and boundary conditions used by the solver program **ISES** . It also configures the program into its analysis and design modes by specifying appropriate global variables and constraints.

```
GVAR(1) GVAR(2) ...  GVAR(N)
GCON(1) GCON(2) ...  GCON(N)
MINLin P1PTin SINLin XINL [ V1ATin ] <-- optional
MOUTin P2PTin SOUTin XOUT [ V2ATin ] <-- optional
MFRin HWRATin [ XSHKin MSHKin ] <-- optional
REYNin NCRIT
TRANS1 TRANS2 (TRANS1 TRANS2 for blade 2) ...
ISMOM MCRIT MUCON
BVR1in BVR2in                                      <-- optional (see below)
MOVX MOVY SCAL ROTA (MOVX MOVY ...  for blade 2)...   <-- optional (see below)
```

MISES description

```
GVAR(1) GVAR(2) ...  GVAR(N)
GCON(1) GCON(2) ...  GCON(N)
MINLin P1PTin SINLin XINL [ V1ATin ] <-- optional
MOUTin P2PTin SOUTin XOUT [ V2ATin ] <-- optional
MFRin HWRATin [ XSHKin MSHKin ] <-- optional
REYNin NCRIT
TRANS1 TRANS2 (TRANS1 TRANS2 for blade 2) ...
ISMOM MCRIT MUCON
BVR1in BVR2in
MOVX MOVY SCAL ROTA (MOVX MOVY ...  for blade 2)...
```

Global variables and constraints. Lines 1,2

The list of possible global variables is,

```
1 SINL inlet flow slope (S1)
2 SLEX grid exit slope
5 SBLE LE stagnation point (for each non-sharp LE blade)
6 PREX grid exit static pressure

7 BVR1 streamtube thickness mode 1 DOF
8 BVR2 streamtube thickness mode 2 DOF

10 REYN stagnation Reynolds number

11 PDX0 zeroth mixed inverse prescribed Pi DOF
12 PDX1 first mixed inverse prescribed Pi DOF
13 PDD0 second mixed inverse prescribed Pi DOF
14 PDD1 third mixed inverse prescribed Pi DOF
15 MINL inlet Mach number
16 MAS1 differential mass fraction DOF

20 GMODn modal geometry DOF flag n = 1, 2, ...  NGMOD

31 MOVX x-movement DOF for each blade
32 MOVY y-movement DOF for each blade
33 SCAL scaling DOF for each blade
34 ROTA rotation DOF for each blade (in degrees, CCW)

40 GPARk geometry parameter DOF flag k = 1, 2, ...  NGPAR
```

ISES file description

MISES description

```
GVAR(1) GVAR(2) ...  GVAR(N)
GCON(1) GCON(2) ...  GCON(N)
MINLin P1PTin SINLin XINL [ V1ATin ] <-- optional
MOUTin P2PTin SOUTin XOUT [ V2ATin ] <-- optional
MFRin HWRATin [ XSHKin MSHKin ] <-- optional
REYNin NCRIT
TRANS1 TRANS2 (TRANS1 TRANS2 for blade 2) ...
ISMOM MCRIT MUCON
BVR1in BVR2in
MOVX MOVY SCAL ROTA (MOVX MOVY ...  for blade 2)...
```

Global variables and constraints. Lines 1,2

The list of possible global constraints is,

```
1 Drive inlet slope S1 to SINLin
2 Drive outlet slope S2 to SOUTin

3 Set LE Kutta condition (for all non-sharp LE blades)
4 Set TE Kutta condition (for all blades)
5 Drive over/under splitter mass fraction ratio to MFRin
6 Drive inlet P0a to PSTr0 ( = 1/gamma )

7 Drive streamtube thickness mode 1 amplitude to BVR1IN
8 Drive streamtube thickness mode 2 amplitude to BVR2IN

9 Drive inlet v1/ao1 to V1ATin
10 Drive outlet v2/ao1 to V2ATin

11 Fix left endpoint of freewall segment
12 Fix right endpoint of freewall segment
13 Fix dP/ds2 at left endpoint of freewall segment
14 Fix dP/ds2 at right endpoint of freewall segment

15 Drive inlet Mach M1 to MINLin
16 Drive inlet pressure P1/Po1 to P1PTin
17 Drive outlet Mach M2 to MOUTin
18 Drive outlet pressure P2/Po1 to P2PTin

19 Drive inlet Reynolds number to REYNIN
20 Drive GMODn to GMODnin n = 1, 2, ...  NGMOD

21 Set Xshock from XSHK to XSHKIN
31 Drive MOVX to MOVXin
32 Drive MOVY to MOVYin
33 Drive SCAL to SCALin
34 Drive ROTA to ROTAin
```

ISES file description

MISES description

ISES file description

```
GVAR(1) GVAR(2) ...  GVAR(N)
GCON(1) GCON(2) ...  GCON(N)
MINLin P1PTin SINLin XINL [ V1ATin ] <-- optional
MOUTin P2PTin SOUTin XOUT [ V2ATin ] <-- optional
MFRin HWRATin [ XSHKin MSHKin ] <-- optional
REYNin NCRIT
TRANS1 TRANS2 (TRANS1 TRANS2 for blade 2) ...
ISMOM MCRIT MUCON
BVR1in BVR2in
MOVX MOVY SCAL ROTA (MOVX MOVY ...  for blade 2)...
```

Inlet, outlet conditions. Lines 3,4

MINLin $=$ inlet relative Mach number $\bar{M}_1$

P1PTin $=$ inlet static/inlet-total pressure ratio $\bar{p}_1/p_{o1}$

SINLin $=$ inlet relative flow slope $\bar{S}_1 = \tan(\beta_1) = \bar{v}_1/\bar{u}_1$

XINL $=$ inlet-condition location m'_1

V1ATin $=$ inlet relative tangential velocity ratio $\bar{v}_1/a_{o1}$

MOUTin $=$ outlet relative Mach number $\bar{M}_2$

P2PTin $=$ outlet static/inlet-total pressure ratio $\bar{p}_2/p_{o1}$

SOUTin $=$ outlet relative flow slope $\bar{S}_2 = \tan(\beta_2) = \bar{v}_2/\bar{u}_2$

XOUT $=$ outlet-condition location m'_2

V1ATin $=$ outlet relative tangential velocity ratio $\bar{v}_2/a_{o1}$

MISES description

```
GVAR(1) GVAR(2) ...  GVAR(N)
GCON(1) GCON(2) ...  GCON(N)
MINLin P1PTin SINLin XINL [ V1ATin ] <-- optional
MOUTin P2PTin SOUTin XOUT [ V2ATin ] <-- optional
MFRin HWRATin [ XSHKin MSHKin ] <-- optional
REYNin NCRIT
TRANS1 TRANS2 (TRANS1 TRANS2 for blade 2) ...
ISMOM MCRIT MUCON
BVR1in BVR2in
MOVX MOVY SCAL ROTA (MOVX MOVY ...  for blade 2)...
```

ISES file description

Viscous flow parameters. Lines 6,7

REYNin = Reynolds number
= 0.0 → inviscid calculation
(restarting a viscous case with REYNin = 0 "freezes" the boundary layers)
NCRIT = (+) critical amplification factor "n_{crit}" for e^n transition model
= (−) freestream turbulence level (τ = -NCRIT, in %) for modified Abu-Ghannam–Shaw bypass transition model
TRANS1 = side 1 surface transition trip m'/chord location
TRANS2 = side 2 surface transition trip m'/chord location

The input Reynolds number REYNin is based on the mixed-out static density, viscosity, and relative speed at m'_1, and the reference length L_{ref}.

$$\texttt{REYNin} = \frac{\bar{\rho}_1 \bar{V}_1 L_{\text{ref}}}{\bar{\mu}_1}$$

The reference length L_{ref} is the same as was used used to define the streamsurface radii R in the stream.xxx file described earlier. If a constant R=1 is specified (the default case for 2-D cascades), then L_{ref} becomes the length unit of the blade coordinates (X,Y). If (X,Y) are also defined so that the blade chord is unity, REYNin is then the usual chord-based Reynolds number.

MISES description

ISES file description

```
GVAR(1) GVAR(2) ...  GVAR(N)
GCON(1) GCON(2) ...  GCON(N)
MINLin P1PTin SINLin XINL [ V1ATin ] <-- optional
MOUTin P2PTin SOUTin XOUT [ V2ATin ] <-- optional
MFRin HWRATin [ XSHKin MSHKin ] <-- optional
REYNin NCRIT
TRANS1 TRANS2 (TRANS1 TRANS2 for blade 2) ...
ISMOM MCRIT MUCON
BVR1in BVR2in
MOVX MOVY SCAL ROTA (MOVX MOVY ...  for blade 2)...
```

Geometry movement,scaling,rotation mode amplitudes. Line 10.

MOVXin = Specified x-displacement mode MOVX
MOVYin = Specified y-displacement mode MOVY
SCALin = Specified scaling mode SCAL
ROTAin = Specified rotation mode ROTA

IPLOT description

IPLOT is the program which displays the solution in `idat.xxx` at any time whether the solution is converged or not. It is executed by the command `run xxx` and selecting the `iplot` option. Note that if the solution in `idat.xxx` is not converged, the results are physically meaningless. The top-level **IPLOT** menu is shown below.

```
 1   Blade surface plots
 2   Streamtube    plots
 3   Contour/grid  plots
 4   Wake profile  plots
 5   r,b,ln(Po) vs m'  stream surface definition plots
 6   Wheel View
 7   Dump flowfield to text file
 8   Dump BL quantities to text file

Select IPLOT option (0=Quit):
```

5.3.1 Blade surface plots

IPLOT description

The "Blade surface plots" menu brought up by the top-level option 1 allows plotting of most of the airfoil surface and wake boundary layer variables:

```
 1   Mach vs x          2     Cp vs x

 3     Hk vs x          7     Ue vs x
 4 s1 D,T vs x          8   A/Ao vs x
 5 s2 D,T vs x          9     Ct vs x
 6     Cf vs x         10 Rtheta vs x
11   Forces            13   Change blade
12   Options           14   Hardcopy toggle

15   Change x-axis coordinate type on BL plots
16   Change x-axis limits on BL plots
17   Change y-axis limits on current BL plot
18   Cursor blowup of current BL plot
19   Reset x,y-axis limits for BL plots
20   Annotation menu
21   Plot-page options
```

MISES description

IPLOT description

Three types of C_p can be displayed from the surface plots menu:

$$C_p = \frac{p - p_1}{\frac{1}{2}\rho_1 V_1^2}$$

$$\bar{C}_p = \frac{p - p_1}{p_{o1} - p_1}$$

$$C_{p_o} = \frac{p_{o1} - p}{p_{o1} - p_1}$$

The default type is C_p, but any type can be chosen using Option 12.

Two types of loss coefficients are defined from the hypothetical mixed-out state $\bar{\rho}_2$, $\bar{p}_2$ ... at m'_2, described earlier.

$$\bar{\omega} = \frac{p_{o2}^{isen} - \bar{p}_{o2}}{p_{o1} - p_1}$$

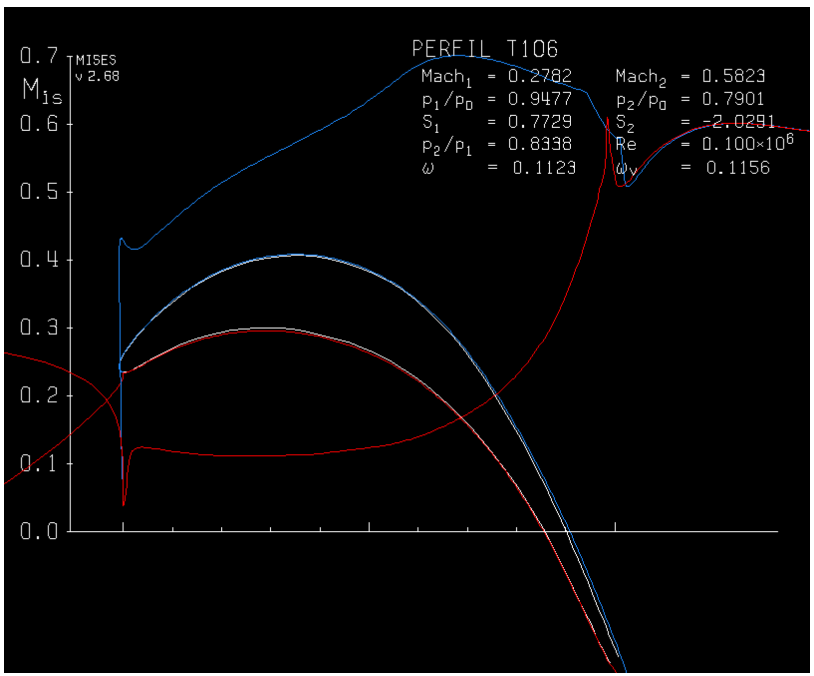
$$\zeta = \frac{p_{o2}^{isen} - \bar{p}_{o2}}{p_{o2}^{isen} - \bar{p}_2}$$

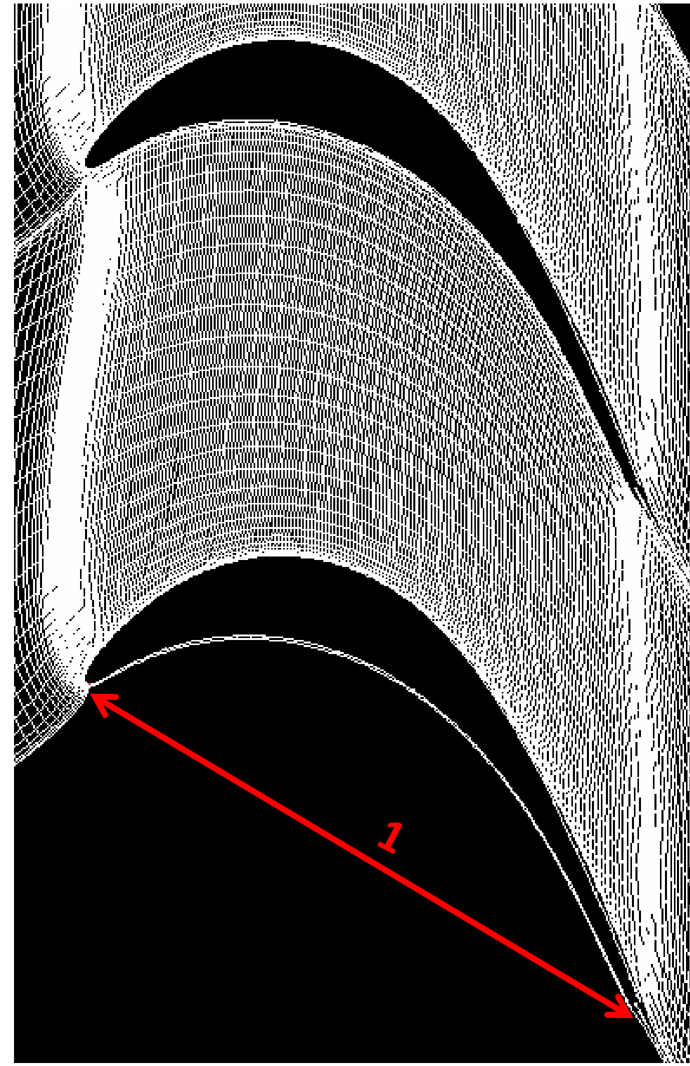
Turbine cascade [T106A]

Inlet Flow Angle

Outlet Mach Number

```
1 2 5 6 15
1 3 4 6 17
 0.3000 0.0000 0.77289 -0.90732068 | Minl P1/Po1 Sinl Xinl
 0.6000 0.790126628 -1.97966 1.46529265 | MOUT P2/Po1 Sout Xout
 0.0000 0.0000    |mfr
 100000.0  -0.8 | REYin ACRIT
 1.01  1.01 | XTR1 XTR2
 1 0.95 +1.0 |ISMOM MCRIT MUCON
 0.0 0. | Bvr1 Bvr2
 0. 0.  0. 0. 0. 0. 0. 0. 0. 0. 0.
```





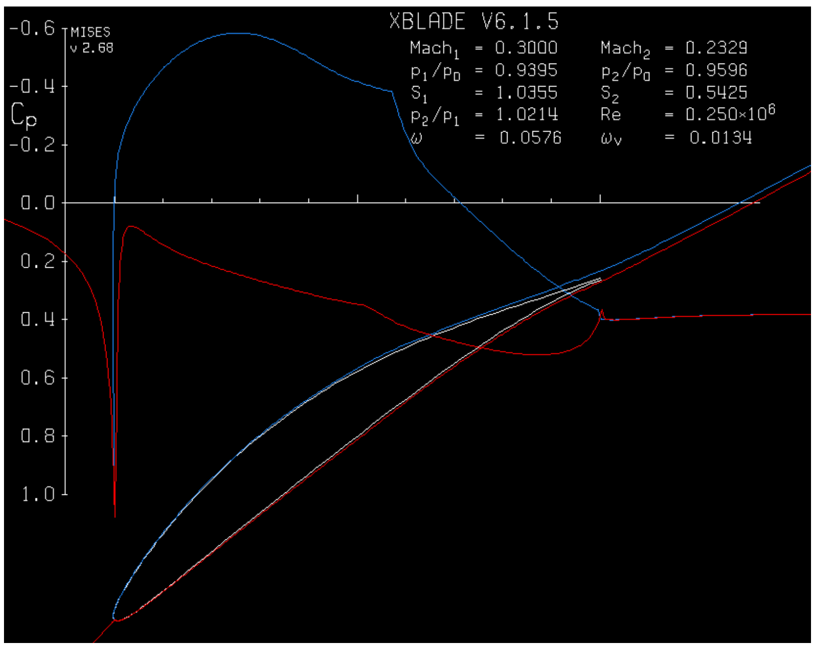
Compressor cascade [NACA65-12-10]

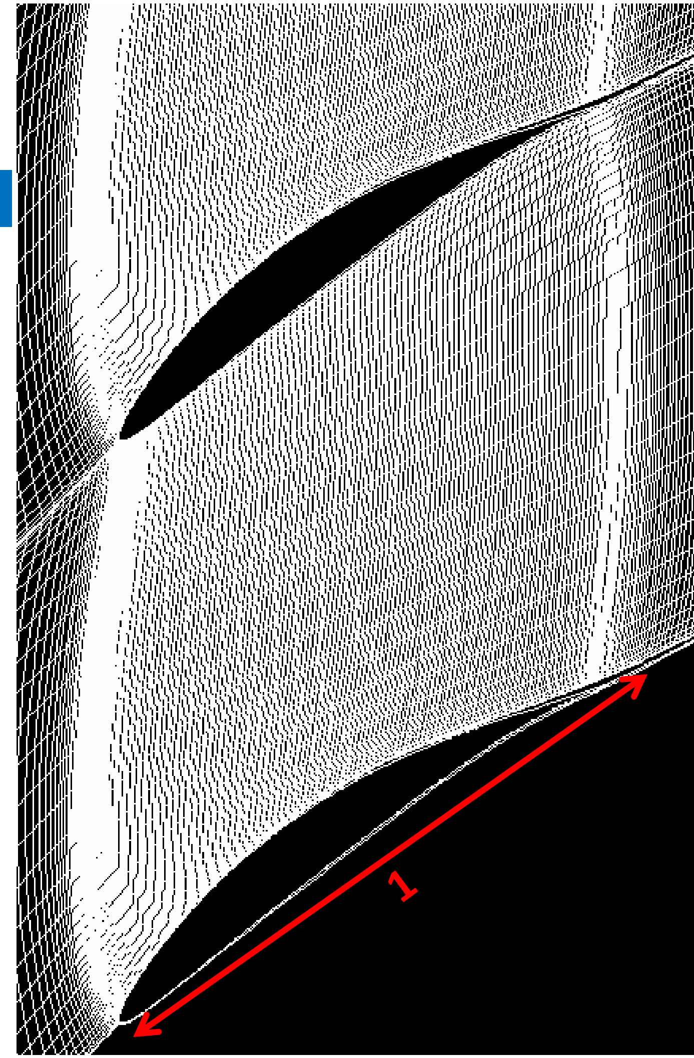
Inlet Flow Angle

Inlet Mach Number

Enables Rotation

```
1 2 5 15 34
1 3 4 15 34
 0.300000 0. 1.03553118289 -0.250000 | Minl P1/Po1 Sinl Xinl
 0.400000 0.800000 0.267949 1.250000 | MOUT P2/Po1 Sout Xout
 0.  0.000000 |mfr hwall
 250000.00 4.500 | REYin NCRIT
 1.10  1.10 | XTR1 XTR2
 1 0.95 +1.0 |ISMOM MCRIT MUCON
 0.0 0. | Bvr1 Bvr2
 0. 0. 0. 35.0
```





Exercise [All Groups]

1. Enumerate the main characteristics that describe the mid-span section of your three first airfoils obtained in Lab 1, including:
    1. Inlet / Exit flow angles
    2. Inlet / Exit flow Mach number
    3. Pitch / Chord ratio
    4. Reynolds number based on chord and inlet velocity
2. For each of the airfoils make your hypothesis / selection of
    1. Inlet / Exit metal angles
    2. Stagger angle
    3. Camber line
    4. Profile thickness distribution (see indications for each of the groups below)
3. Define the blade geometry for each of the airfoils and create the input data for MISES
4. At design conditions, run MISES and capture all significant blade surface plots (Mis, Cp, H, δ^*, θ, Cf).
5. Keeping other design conditions invariant, make a graph of total pressure loss (ζ or ω) and deviation variation with Incidence Angle (Inc) ranging from Inc=[-10 → +10]
6. Discuss the results and check out your initial estimations of deviation and loss coefficient

Exercise Group Profiles

x/c (%)	NACA (A_{10}) (línea media)	Group 2 C4	C7	Group 1 NACA65
0.000	0.000	0.00	0.00	0.000
0.5	0.250			0.772
0.75	0.350			0.932
1.25	0.353	1.65	1.51	1.169
2.50	0.930	2.27	2.04	1.574
5.00	1.580	3.08	2.72	2.177
7.500	2.120	3.62	3.18	2.647
10.00	2.585	4.02	3.54	3.040
15.00	3.365	4.55	4.05	3.666
20.00	3.980	4.83	4.43	4.143
25.00	4.475			4.503
30.00	4.860	5.00	4.86	4.760
35.00	5.150			4.924
40.00	5.355	4.89	5.00	4.996
45.00	5.475			4.963
50.00	5.515	4.57	4.86	4.812
55.00	5.475			4.530
60.00	5.355	4.05	4.42	4.146
65.00	5.150			3.682
70.00	4.860	3.37	3.73	3.156
75.00	4.475			2.584
80.00	3.980	2.54	2.78	1.987
85.00	3.365			1.385
90.00	2.585	1.60	1.65	0.810
95.00	1.580	1.06	1.09	0.306
100.00	0.000	0.00	0.00	0.000
Radio del borde de ataque		1.20	1.20	0.687
Radio del borde de salida		0.60	0.60	-

Group 3

Table 7. Base Profile T6

Station, percentage of chord	0	1·25	2·5	5·0	7·5	10·0	15	20	30
Upper and lower surface, percentage of chord	0	1·17	1·54	1·99	2·37	2·74	3·4	3·95	4·72
Station, percentage of chord	40	50	60	70	80	90	95	100	
Upper and lower surface, percentage of chord	5·00	4·67	3·70	2·51	1·42	0·85	0·72	0	

Leading edge radius = 12 per cent *t*; trailing edge radius = 6 per cent *t*; *t/l* = 10 per cent; station of maximum thickness = 40 per cent *l*.

Group 4&5

A_3K_7 Mean Line, Thickness Distribution and Coordinates for Primary Turbine-blade Series

A_3K_7 mean-line coordinates for $C_{L_0} = 1{\cdot}0$		*Thickness distribution co-ordinates $t/c = 20\%$*	
x_c	y_c	x_t	y_t
0	0	0	0
0·5	0·397	1·25	3·469
1·25	0·836	2·5	4·972
2·5	1·428	5·0	6·918
5·0	2·359	10	9·007
10	3·689	15	9·827
15	4·597	20	10·000
20	5·217	25	9·899
25	5·623	30	9·613
30	5·852	35	9·106
35	5·936	40	8·594
40	5·897	45	7·913
45	5·753	50	7·152
50	5·516	55	6·339
55	5·200	60	5·500
60	4·814	65	4·661
65	4·367	70	3·848
70	3·870	75	3·087
75	3·328	80	2·406
80	2·746	85	1·830
85	2·133	90	1·387
90	1·485	95	1·101
95	0·801	100	0
100	0		

$\frac{dy_c}{dx_{c(0.5)}} = 0{\cdot}8574$ $\frac{dy_c}{dx_{c(95)}} = -0{\cdot}1602$

Leading edge radius = 4·407% of chord
Trailing edge radius = 1·000% of chord

Camber Line: Circular Arc

Camber Line: Parabole

Guidelines and procedures for reporting

1. Lab sessions classes attendance is mandatory.
2. Work is to be done in groups of three people.
3. Each group of students need to present a report with the results of the exercise. This report will be uploaded to Aula Global web page.
4. Questions can be send via e-mail to antonio.antoranz@uc3m.es or during tutorial hours
5. General considerations for reporting:
   a) Typed document not handwritten.
   b) Short and concise paragraphs.
   c) Graphics wherever possible (tables, graphs, illustrations).
   d) Write a single compiled report per group.

Useful Linux Commands

- **mkdir** - make directories

Usage : mkdir DIRECTORY

Create the DIRECTORY(ies), if they do not already exist

- **cd** - change directory

Use cd to change directories. Type cd followed by the name of a directory to access that directory. Keep in mind that you are always in a directory and can navigate to directories hierarchically above or below.

Examples: cd DIRECTORY; cd ..; cd ~

- **mv** - change the name of a directory (or file)

Type mv followed by the current name of a directory and the new name of the directory.

Example: mv DIR NEWNAMEDIR

- **pwd** - print working directory

will show you the full path to the directory you are currently in. This is very handy to use, especially when performing some of the other commands on this page

Useful Linux Commands

- **ls** - Short listing of directory contents

Example: ls –lrt → list details about files, directories and subdirectories sorted by time modified

- **cp** - Copy files

Example: cp myfile yourfile

Copy the files "myfile" to the file "yourfile" in the current working directory. This command will create the file "yourfile" if it doesn't exist. It will normally overwrite it without warning if it exists.

- **rm** - Remove an existing file or directory (with –r option)

Removes directories and files within the directories recursively.

Example: rm –r DIRECTORY; rm FILE

- **gedit** – Text editor

Useful Linux Commands

- **How to run Mises**

1. Make a copy of Mises examples: *cp -r /opt/MisesUC3M/Lab_Session/T106 YourDir* (or *cp -r /opt/MisesUC3M/Lab_Session/NACA65 YourDir* for a Compressor Case)
2. Enter your working directory: *cd YourDir*
3. Execute command: */opt/MisesUC3M/mises2.68 CaseName* (*t106* or *naca*)
4. Mises main window:

```
   1   ISES     Euler/BL analysis
   2   IPLOT    Plotting utility
   3   EDP      Pressure-editing utility
   4   ISET     Grid generator
   5   BLDSET   Blade editor
   6   IPRINT   Print flow parameters
   7   POLAR    Euler/BL parameter-sweep analysis
   8   LINDOP   multi-point optimization driver
   9   ISEQ     point-sequence ISES execution for:   naca_..

  10   Edit  ises.naca
  11   Edit  blade.naca
  12   Copy all  *.naca  files to new  *.xxx
  13   Change current extension:   naca

  21   Print  plot.ps
  22   Toggle X-Window background color
  23   Set default number of ISES iterations

Select MISES option (0=quit):
```

5. Copy the current case (*t106* or *naca*) to a working case name using **option 12**. Provide the new case extension (e.g. *xxx*)
6. Use **option 13** to start modifying and running the working case

Useful Linux Commands

- **How to mesh a geometry**

1. From Mises main window, run **ISET** (grid generator program) with **option 4**.
2. Command sequence in ISET:

```
1+Enter+2+Enter+3+4+5+Enter+0
```

- **How to run a case**

1. From Mises main window, run **ISES** (Euler/BL analysis program) with **option 1**.
2. Command sequence in ISES:

```
Number of iterations (~30) + if converged on tolerance then 0 + if not then, enter new number of iterations + if not converged then finish
```

- **How to post process a case**

1. From Mises main window, run **IPLOT** (plotting utility) with **option 2**.
2. Command sequence in IPLOT:

```
1+i(i=1,2,3,4,5,6)+Capture or Hardcopy current plot(16)+0+0
```

3. From Mises main window, run IPRINT with option 6
4. Copy the results for exit angle and loss