

## How to use Adsas

**Arch Dam Stress Analysis System** is a computerized version of the Trial Load method. The aim of this 'new' version of the program is only to simplify the input of the parameters (geometry, material properties, loads) and to allow a quick view the main results (deflection, principal stresses and forces on the abutments) of the analysis. The original Adsas program, developed in the 60' by USBR, remains unchanged to solve the equations.

### Geometry

Upstream and downstream facings are defined by horizontal arches whose radius (upstream or downstream) varies according to the elevation of the dam. To define such a geometry, the user must enter:

- The geometry of the crown cantilever (upstream and downstream faces)
- The position of the centers of the circles according to the elevation, called "line of centers" (upstream and downstream) which allows to know for each elevation the radius of the upstream or downstream arch.

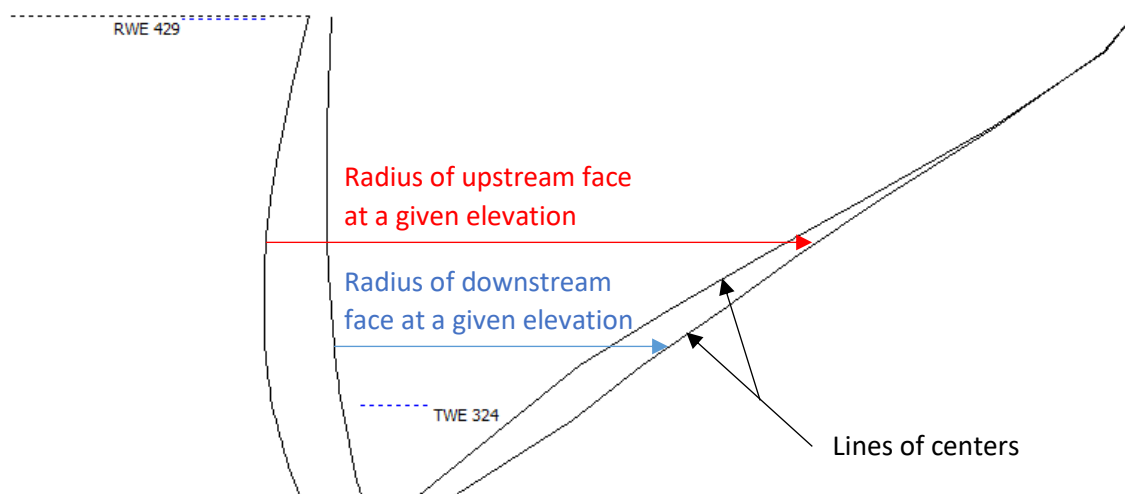
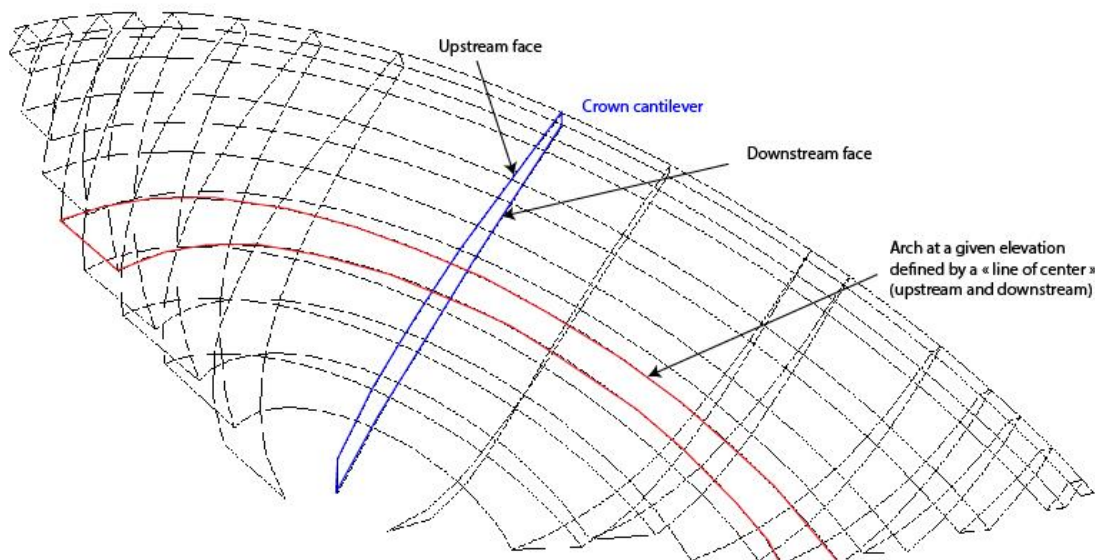


Figure 1 geometry definition

This version only allows for the moment to describe 'single centered layout': the radius remains constant on the left or right side of the reference plane and with the angle from the reference plane to the abutment.

Define the elevation of the base and the crest of the dam and the Radius

Once completed in the text boxes at the top of the form, elevation of the base and crest of dam are automatically copied in the face description and arches description tables.

Radius value is the radius of the upstream face at the crest.

The first two lines of the tables "Upstream face", "Downstream face", "Upstream line of centers" and "Downstream line of centers" are automatically created.

The screenshot shows a form with four input fields: "Nb of centers" (1), "Top of the dam" (792 m), "Base of the dam" (720 m), and "Radius" (56 m). Blue arrows point from the text "Filling in the three parameters" to the three elevation/radius fields. Below is a "Face description" section with a table. The table has four tabs: "Upstream face", "Downstream face", "Upstream line of centers", and "Downstream line of centers". The "Upstream face" tab is active, showing a table with columns: "Elevation (m)", "abscisse (m)", "slope", "Circle center abs", "Circle center elv (Circle rad (m))". The first row contains 792, 0.00, 0.000, and empty cells. The second row contains 720, 0.00, and empty cells. Below the table, the text "Automatically created" is displayed.

Figure 2 geometry menu

Adding new elevations of the downstream and upstream faces

Click on one of the cells of the line corresponding to the elevation above the one you want to add, then right click, "Insert below" and enter the new elevation. It will creates an additional row in the table.

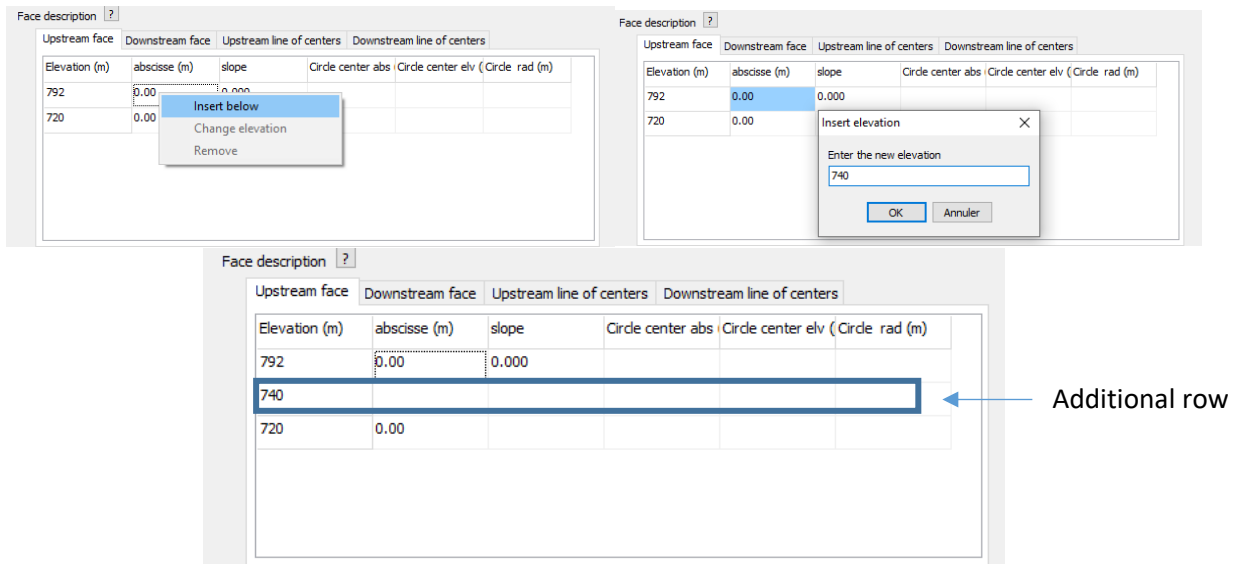


Figure 3 Figure 4 how to add new elevation

## Definition of the upstream/downstream faces of the crown cantilever and upstream/downstream line of centers

Description of the us/ds face of the crown cantilever of us/ds line of center can be done using either straight line segments or circular line segments.

- For straight line segments between elevations, the user can fill (Figure 5) :
  - o The abscissa for each elevation (0 abscissa is usually considered for the upstream face at the crest of the crown cantilever), slopes between elevations will be automatically computed once another tab is selected.
  - o The slope between the elevation considered and the elevation of line below in the table. Abscissa will be automatically computed once another tab is selected.
- Circular line segments are based on the coordinates of the center of the circle and its radius (Figure 6). Abscissa will be automatically computed once another tab is selected.

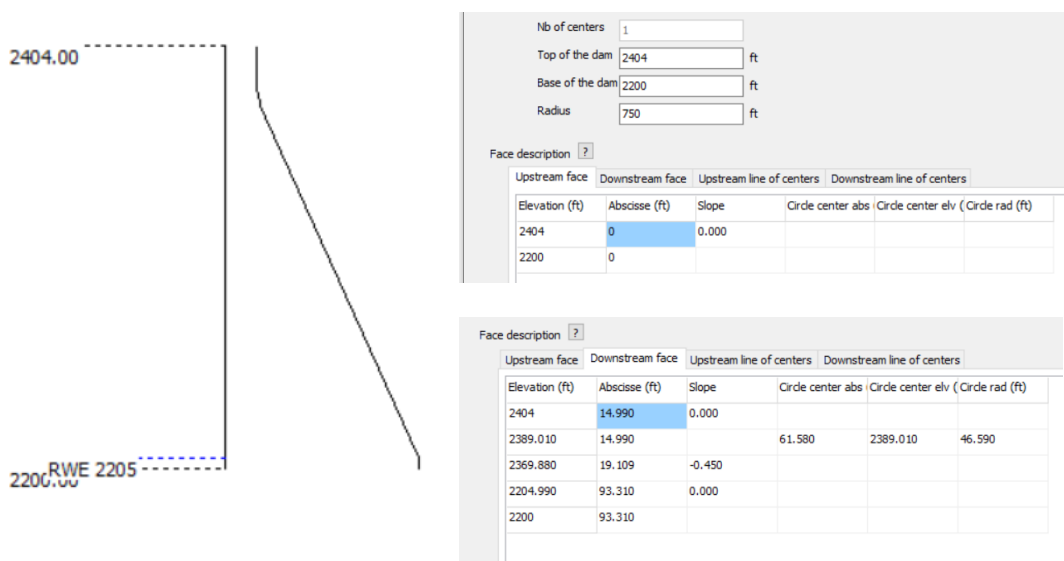


Figure 5 example of ud/ds face description with straight line segments

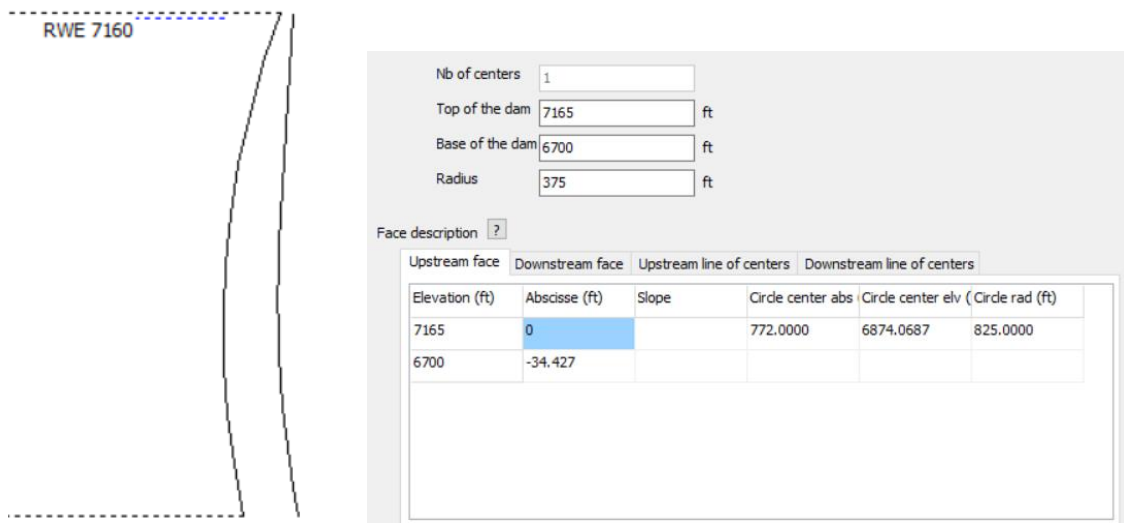


Figure 6 example of ud/ds face description with circular line segments

Description of the us/ds line of center is based on the same method.

Note that it is only possible to define a single circle for a given elevation, unlike vertical faces which can be defined by several successive circles between two elevations. French arch dams' faces are generally described by equation where radius is not constant from the reference plane to the abutment nevertheless constant radius can generally be found to roughly match the geometry at each elevation (Figure 7).

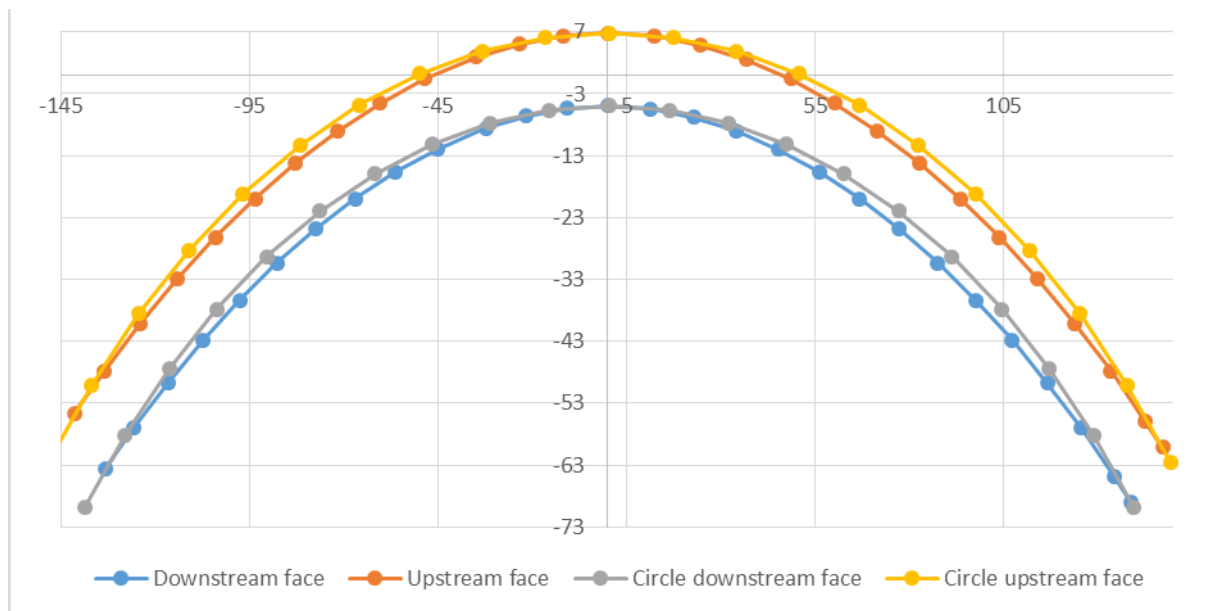


Figure 7 comparison of faces from logarithmic spiral equation and from circle equation

## Definition of the arches and cantilevers

### Elevation of arches

Selection of arch elevations for a normal arch dam begins by selecting the top and bottom arches. The crest of the dam will always be the top arch. If a spillway is located along the crest of the dam, then the next arch below the crest of the dam should be located at the spillway crest. For convenience the

lowest arch is usually the lowest topographic contour but no higher than about 15 percent of the structural height. As a first estimate of the vertical spacing of the remaining arches, you should divide the elevation between the crest and the bottom arch and space your arches the same way from crest to the toe. You can add until 9 others arches between the crest and the bottom arch.

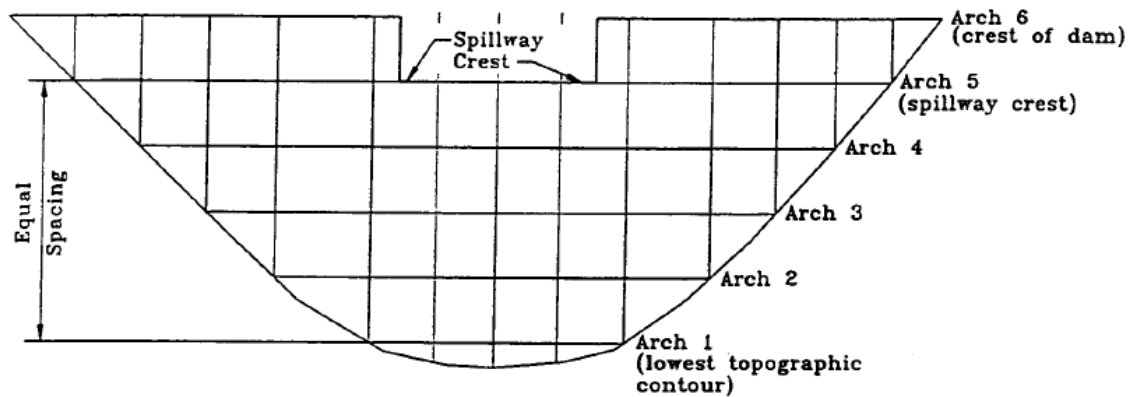


Figure 8 example of grid

## Cantilevers

The crown cantilever is always located at the maximum section (maximum dam height). With the exception of the top arch, ADSAS automatically locates additional cantilevers at the ends of each arch.

The cantilever spacing generated from the automatic placement at arch elevation will usually give noticeably unequal spaces between the cantilevers next to the crown cantilevers. This unequal spacing can be improved by adding “free cantilevers” to the layout. Free cantilevers are cantilevers added by the designer and are not based on an arch. Up to two free cantilevers can be placed on each side of the crown cantilever and are set between the crown cantilever and the abutments of the lowest arch. Their base elevation must be above the crown cantilever base elevation and below the first arch elevation.

Adding one or two free cantilevers on each side of the crown cantilever is always recommended. Analyses with no free cantilevers may produce abnormal stress distributions along the lower arch abutments.

## Arches-cantilevers grid description

In order to define the grid where balance between arch loads and cantilever loads will be achieved, user have to fill up the grid definition table presented in Figure 9. This table allows also to describe the opening of the arch at chosen elevation to define the shape of the valley.

By default, the table has only two line with top and bottom elevation. Additional elevation can be added using a right click with the mouse. The user have to fill in several inputs:

- The procedure control to define the type of cantilevers :
  - procedure ctrl = 1 : base elevation of the crown cantilever (no arch at this elevation)
  - procedure ctrl = 2 : base elevation of a free cantilever on the right side looking from downstream (no arch at this elevation)
  - procedure ctrl = 4 : base elevation of a free cantilever on the left side looking from downstream (no arch at this elevation)

- procedure ctrl = 5 : base elevation of free cantilevers on both sides (no arch at this elevation)
- procedure ctrl = 0 : an arch element will be placed at this elevation with cantilevers based on both abutments.
- The angle to abutments that can be defined from two axis:
  - From the general axis center (A on **Erreur ! Source du renvoi introuvable.**)
  - From the upstream line of center ( $\phi$  on **Erreur ! Source du renvoi introuvable.**)

Archs - Cantilevers Grid Description ?

Procedure ctrl

definition of angles to abutement

from upstream line of center (Phi)

from general axis center (A) ← Check « From general axis center »

350.00	0	31.27	32.39
335.00	0	23.23	26.4
320.00	0	19.92	19.92
300.00	5	8.48	12.94
299.20	1		

Angle to abutement RD (°)      Angle to abutement RG (°)

Figure 9 arches-cantilever grid description table

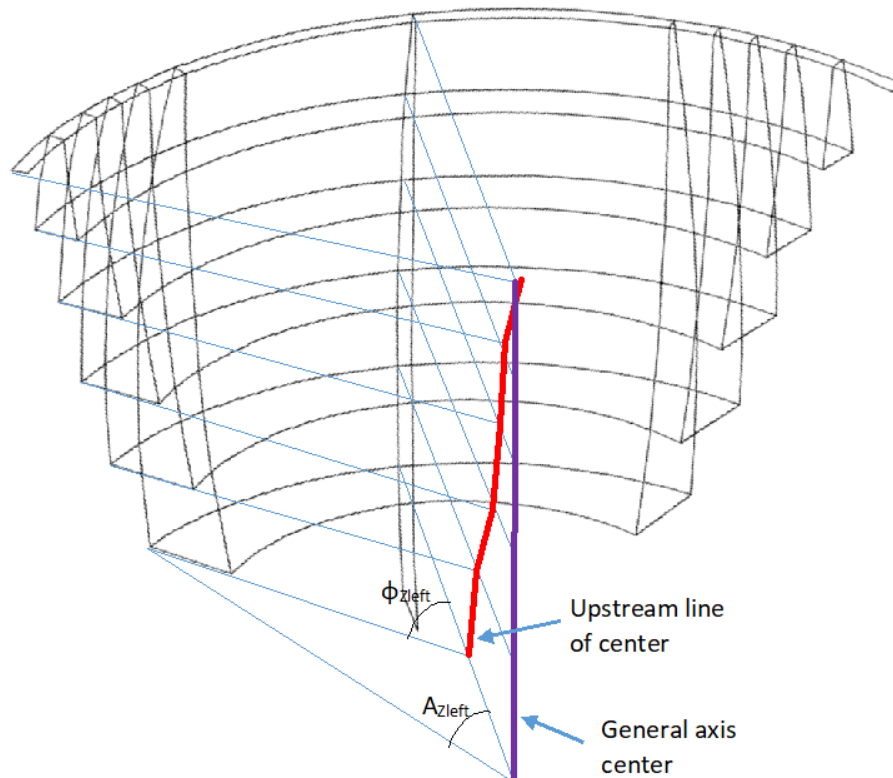


Figure 10 angles to abutment

## Concrete and foundation properties

"Material properties Menu" (Figure 11) allows to enter the properties of the concrete and the foundation. Only a uniform deformation modulus of the rock can be chosen for this early version of Adsas.

The screenshot shows a software interface for defining material properties. It is divided into two main sections: 'Concrete properties' and 'Rock foundation properties'.

**Concrete properties:**

- Modulus of elasticity of concrete: 32 GPa
- Poisson's ratio for concrete: 0.2
- Unit weight of concrete: 2400 kg/m<sup>3</sup>
- Coeff of thermal expansion: 9 E-06 /°C

**Rock foundation properties:**

- Poisson's ratio for rock: 0.2
- Deformation modulus options:
  - uniform deformation modulus of the rock: 27 GPa
  - variable deformation moduli of the rock

Figure 11 concrete and rock fondation properties

## Loads

The Loads menu allow to define the hydraulic and thermic loads.

The screenshot shows the 'Loads menu' with various parameters for defining hydraulic and thermal loads. Blue arrows point from descriptive text on the right to the corresponding input fields.

**Hydraulic loads:**

- Top of grout elevation: 430.00 m (Usually = crest of the dam)
- reservoir water elevation: 429 m (Upstream water level)
- Tailwater water elevation: 324 m (Downstream water level)

**Thermal and seismic loads:**

- Earthquake acceleration: [ ] g (Seismic load (not yet available))
- thermal load: [ ? ]

**Temperature load options:**

- Uniform: Table with columns 'Elevation (m)' and 'Uni temp(°C)'. Elevation values: 430.00, 420.00, 400.00, 380.00.
- linear: Table with columns 'Elevation (m)' and 'Lin temp(°C)'. Elevation values: 430.00, 420.00, 400.00, 380.00.

Figure 12 loads menu

The thermal loads can either be uniform, linear or both between upstream and downstream faces for each arch elevation (Figure 13).

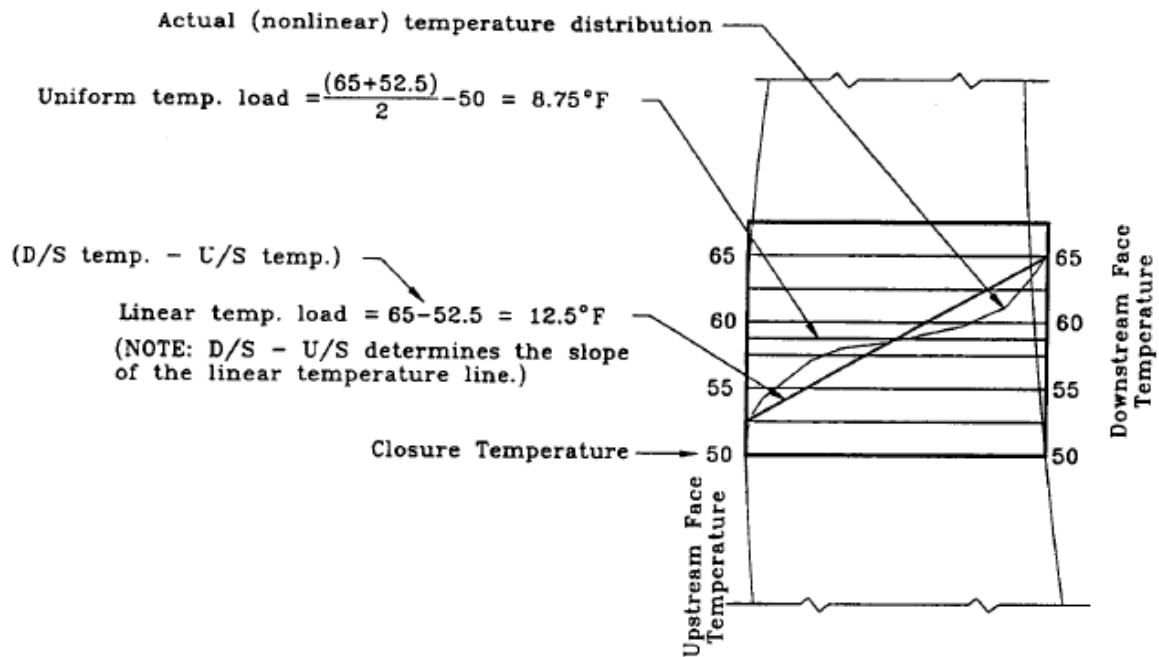
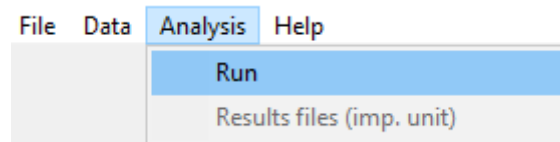


Figure 13 thermal load definition

## Run the analysis

Once every parameter have been define, you can click on "Analysis → Run": a dos window opens and launches the original executable file.



## Results

Once the calculation is complete, it is possible to view several results using the menu on the left of the application.

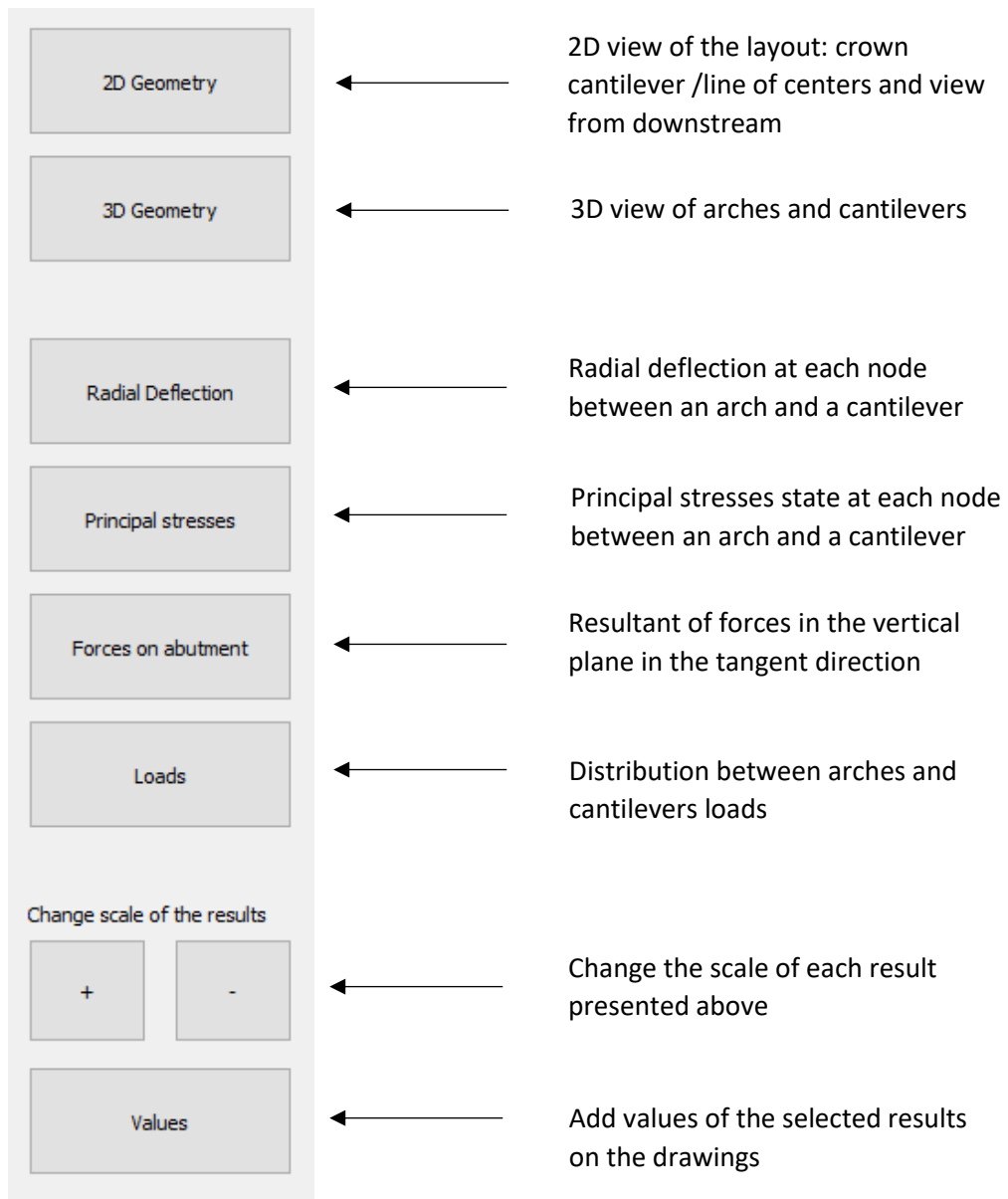


Figure 14 results menu

### 3d view

You can display the 3D view with or without the values corresponding to the thickness of the dam at the top and at the bottom of each cantilever.

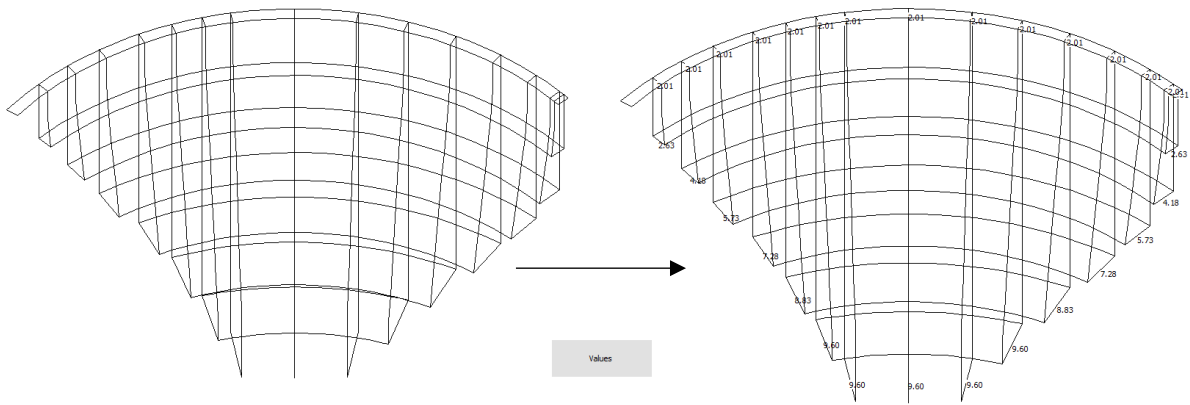


Figure 15 3D geometry menu

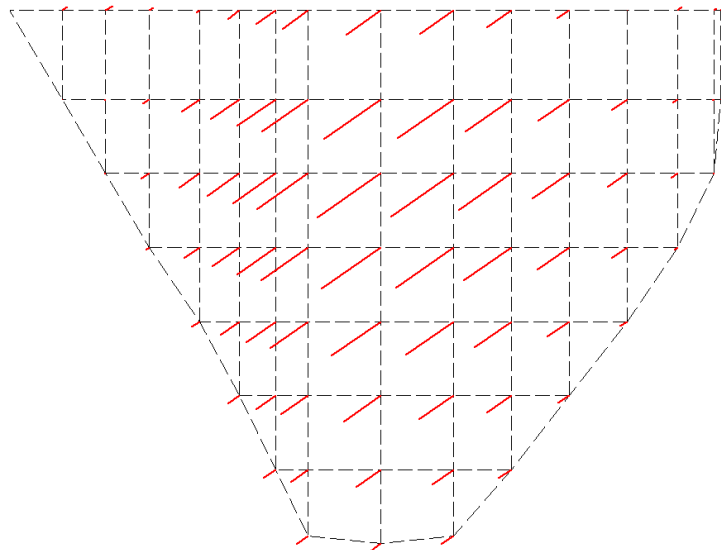
You can use the following combinations to move or change the size of the view:

- Move: Middle click
- Rotation: Ctrl + left click
- Zoom: Shift + left click
- Zoom window: Right click toward right bottom
- Initial view: Right click toward top

### Radial deflection

Radial deflection is represented by red segments at each node. If the segment is in direction of the lower left corner, it represents a displacement toward downstream. In the opposite, if the segment is in direction of the upper right corner, it represents a displacement toward upstream.

Negatives values are deflection toward downstream. The scale of the segment can be modified using the +/- buttons. Values can be printed using the 'values' button.



Developped downstream view of radial deflection (x1181)

min: -0.0093 m, max: 0.0010 m

Figure 16 radial deflection view

## Principal stresses

Principal stresses are represented by red and blue segments at each node depending on the stress at this node. Compression (negative values) are in blue and tension (positive values) are in red. The size of the segment represent the stress intensity and its angle the direction of this principal stress.

There are two representations: the first represents the principal stresses on the upstream face, and the second on the downstream face.

Values of the principal stresses can be printed using the 'values' button to obtain successively:

- The values of the first principal stress
- The values of the second principal stress

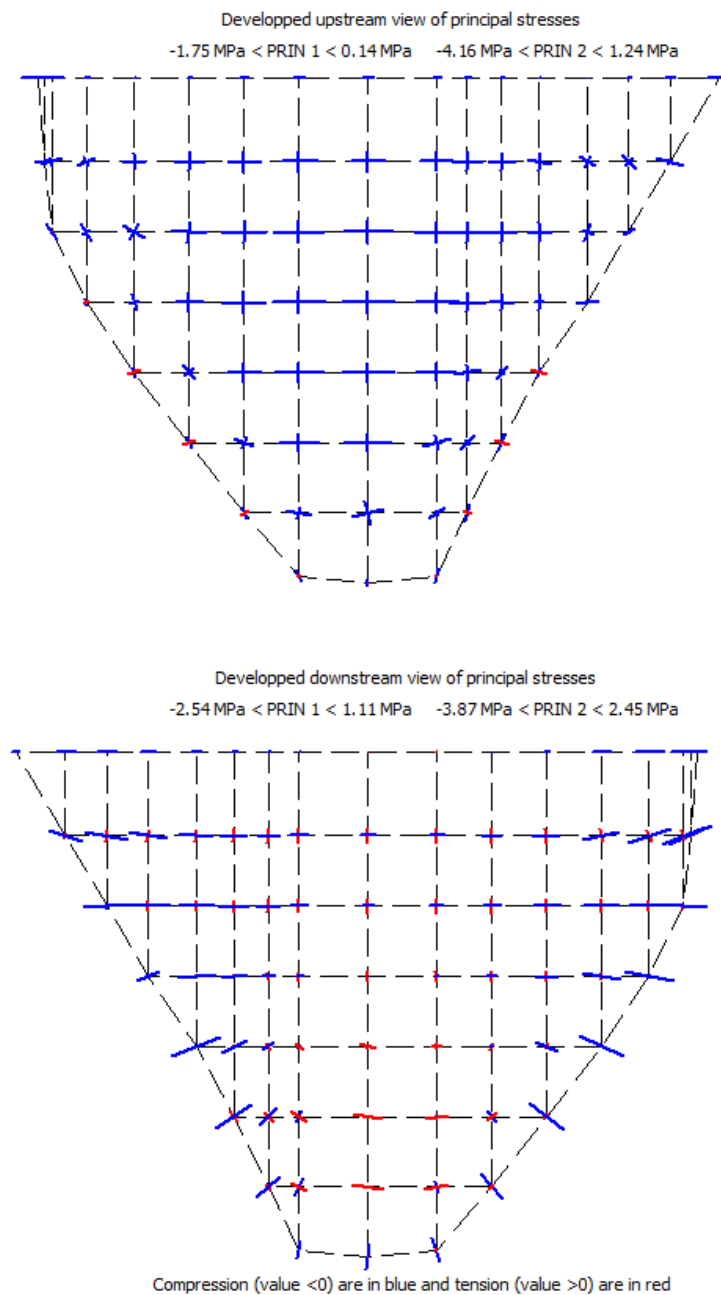


Figure 17 principal stresses on us and downstream face

## Forces on abutment

Forces on abutment are represented by red larger or smaller segments at each cantilever toe. In this early version of Adsas, only component of the resultant of forces in the vertical plane in the tangent direction of the abutment are available. By clicking on "Values" button, user can obtain successively:

- The component of the resultant forces in the vertical plane in the tangent direction of the abutment,
- The angle of the forces in the vertical plane, measured from tangential toward downward

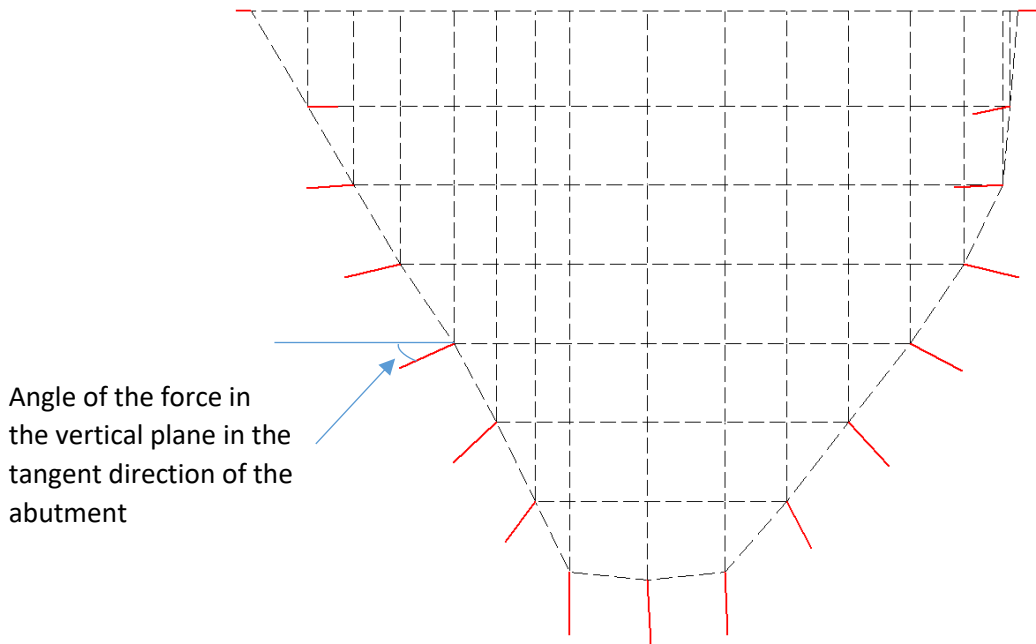


Figure 18 Forces on abutment

## Arch/Cantilever loads

The trial load method, computerized in ADSAS, allocates the loads between arches and cantilevers at the nodes. This results allows to compare arch load (red) and cantilever load (blue) for each node. That gives a glimpse of the way the structure behave and transfer the thrust at the foundation.

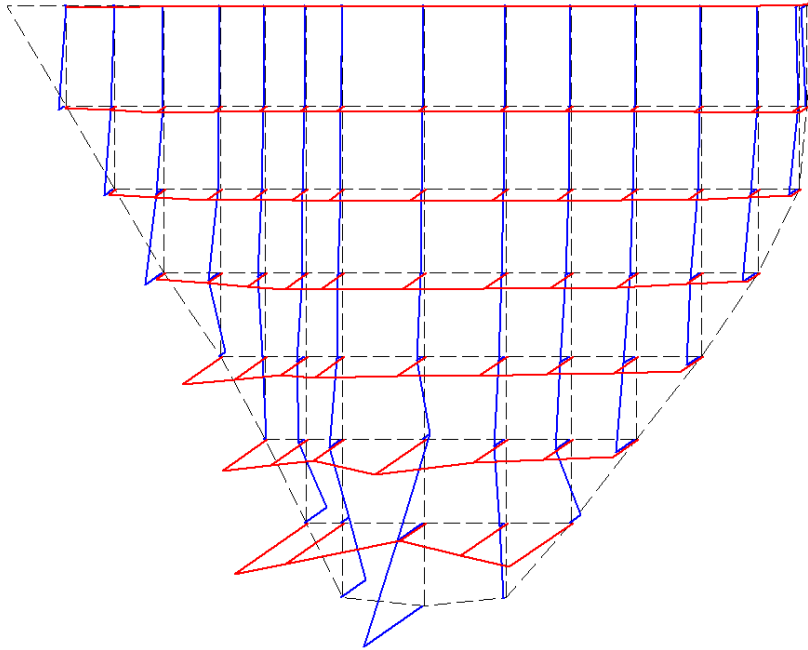


Figure 19 arches and cantilevers loads

### Full results (Imperial units only)

The original ADSAS results text file is available from the Analysis menu (Figure 20). This file are automatically generated and regroup all the assumptions and results of the analysis.

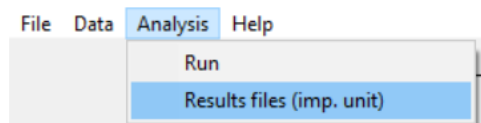
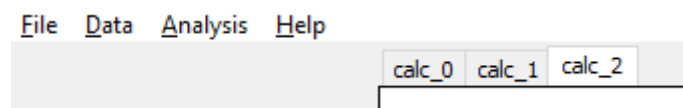


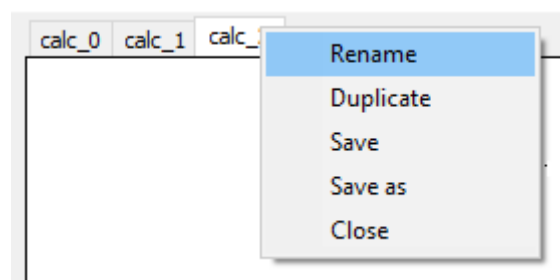
Figure 20 full results file

### Multiple analyses

It is possible to work on many projects at the same time with the use of several tabs:



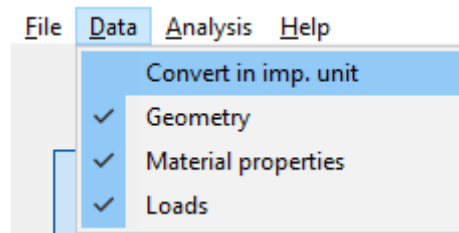
User has the possibility to Rename, Duplicate, Save or Close each tab with a right click on the tab you want to make one on these actions.



This allows you to perform several calculations in parallel to facilitate the comparison of calculation situations for example.

## Units

It is possible to switch between imperial and IS units. Imperial unit is the option by default.



Warning: You should choose your unit system at the beginning of your study and always save your project in the same unit system to prevent numerical errors.

Warning 2: If you save, close and reopen your project and want to work with the IS unit system, you have to switch again between imperial unit (by default) and IS unit. It might results with slight change of values due to the conversion which will very slightly affect the geometry (of the order of a centimeter).

## Errors – Bugs

This new version of Adsas is still under development. Several errors might occurs during the analysis due to the new graphical interface developed or due to the original Adsas solver. Try to increase gradually the complexity of the analysis. The reading of the full results file might help to understand where the problem comes from.