Topology Optimization Modeling

Ameba

User Manual

for Version 0.6.7

November 29, 2018

谢亿民设计

XIE Archi-Structure Design

目　录

- About Ameba
- User agreement
- Register
- Start a new project
  - Start a 2D optimization project
  - Start a 3D optimization project
- Function Introduction
  - Domain
    - PointsDomain
    - LinesDomain
    - CurvesDomain
    - SurfacesDomain
  - Ameba2D
    - Load2D
    - TangentLoad2D
    - NormalLoad2D
    - NondesignDomain2D
    - Support2D
  - Ameba3D
    - Load3D
    - NormalLoad3D
    - NondesignDomain3D
    - Support3D
  - Mesh
    - Mesh
  - PreProcess
    - Symmetry
    - Material
    - OptParameters
    - PreProcessing
    - Sensitivity
    - MultipleLoadCase
  - PostProcess
    - Display
    - Step
    - RenderDisplay
  - Cloud
    - Solver
    - Login

MeshTools
Rebuilding2D
Remeshing3D
AmebaSmooth
Subdivision
MeshDual
FillHoles
MeshChecker
MeshPipe
OffsetMesh
Solver Window
Logs

About Ameba

Ameba is a type of single-celled organism which is capable of changing its shape in different environments.

Based on the Bi-directional Evolutionary Structural Optimization (BESO) technique originally proposed by Professor Yi-Min (Mike) Xie, his team at XIE Technologies has developed a topology optimization software tool called Ameba. According to their design requirements,the users can apply different loading and boundary conditions to the initial design domain. During the computational process by this software, the design domain can evolve, similar to an ameba, into various shapes, and eventually reach an organic form which is also structurally efficient

The team at XIE Technologies is committed to providing an advanced, then easy-to-use topology optimization tool, than can enhance your creative designs and accelerate your product development.

User agreement

User agreement

Welcome to Ameba

1. Thank you for using the "Ameba" topology optimization plug-in and online computing services.
2. Please read the following terms carefully. If you disagree with any of the terms of the agreement, you may choose not to enter "Ameba". When you register successfully, whether you enter the "Ameba" website or use the "Ameba" computing service, or publish any content on the "Ameba" website, it means that you (ie "users") completely accept all terms under the agreement.

Privacy and Intellectual Property

1. The "Ameba" topology optimization allows the user to submit a model file for optimization calculations. The calculation results are transmitted over the Internet to the user's home location and the service itself is not responsible for storing user-related models and calculation results.
2. The user’s design and calculation of the intellectual property of the optimization result are owned by the user. We will protect the user’s personal information and design data as much as possible to prevent the user data from being leaked or infringed.

About Using the Product

1. After the success of user's registration, the "Ameba" account number and corresponding user name and password and other account information will be generated. After the user completed the application registration process, the user will be entitled to use the "Ameba" account. This right belongs only to the original applicant, and is prohibited from giving, borrowing, renting, transferring or selling. The user should save and use its user name and password carefully and reasonably. Improper user care may result in hacking or password theft. The responsibility lies with the user.
2. In the process of using the "Ameba" service, user may not upload, copy, publish, transmit, or reproduce the following:
   a) Oppose the basic principles set by the Constitution;
   b) Endangering national security, leaking state secrets, subverting state power, and undermining national unity;
   c) Damage to national honors and interests;
   d) Inciting ethnic hatred, ethnic discrimination, and undermining national unity;

e) Undermining the country’ s religious policy and promoting cults and feudal superstitions;
f) spread rumors, disrupt social order, and undermine social stability;
g) Dissemination of obscenity, pornography, gambling, violence, homicide, terror or abetment of crimes;
h) insulting or slandering others and infringing the legal rights and interests of others;
i) Information containing other contents prohibited by laws and administrative regulations.

3. When using "Ameba" to optimize computing services, the user needs to generate a temporary authorization code on the site. The use right of the authorization code's belongs only to the initial registrant and is prohibited from granting, borrowing, renting, transferring or selling.

Disclaimer

1. "Ameba" will try its best to ensure the reliability of calculation results and optimization results through various means. At the same time, please be noted that there are some unavoidable position defects in this software. When the user uses the calculation results for engineering practice, please judge the rationality of the structure and conduct the necessary design review.
2. "Ameba" assumes no responsibility for service interruptions or other defects caused by force majeure or "Ameba" failure to control, but will endeavor to reduce the resulting loss and impact to the user.

About the Agreement

In accordance with the development of the Internet and changes in relevant laws, regulations and regulatory documents, or due to business development needs, "Ameba" will amend or change the terms of service of this agreement when necessary. Users can log in to the “Ameba” website to check the latest version of the relevant agreement terms. The revised service terms will effectively replace the original service terms once they are published. If the user continues to use the services provided by "Ameba", it means that the user has accepted the modified service terms; if the user does not accept the modified service terms, it shall stop using the services provided by "Ameba".

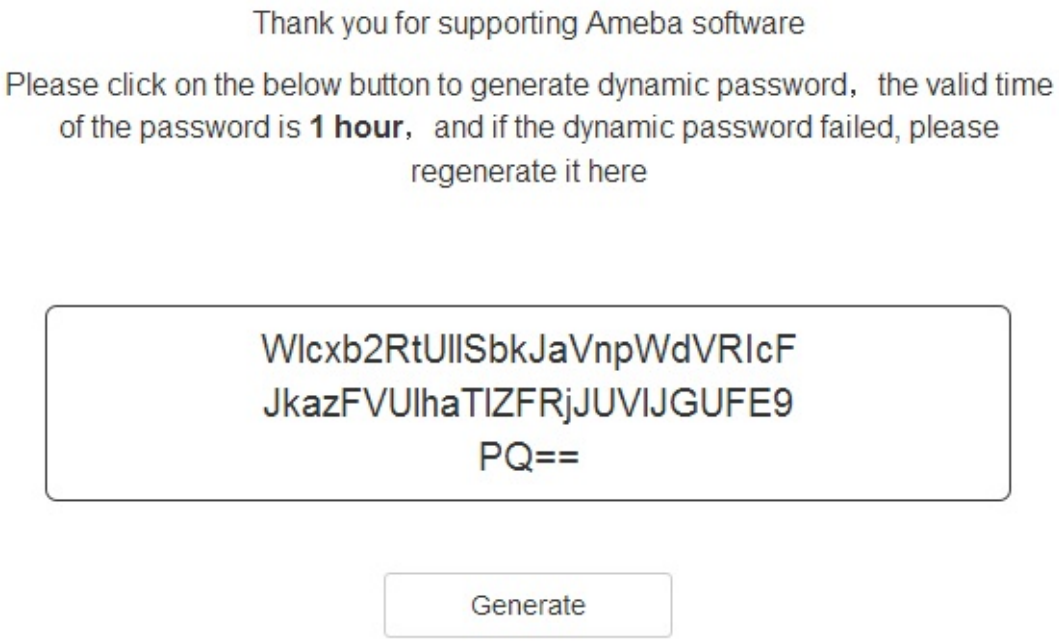
Register

Register and Login

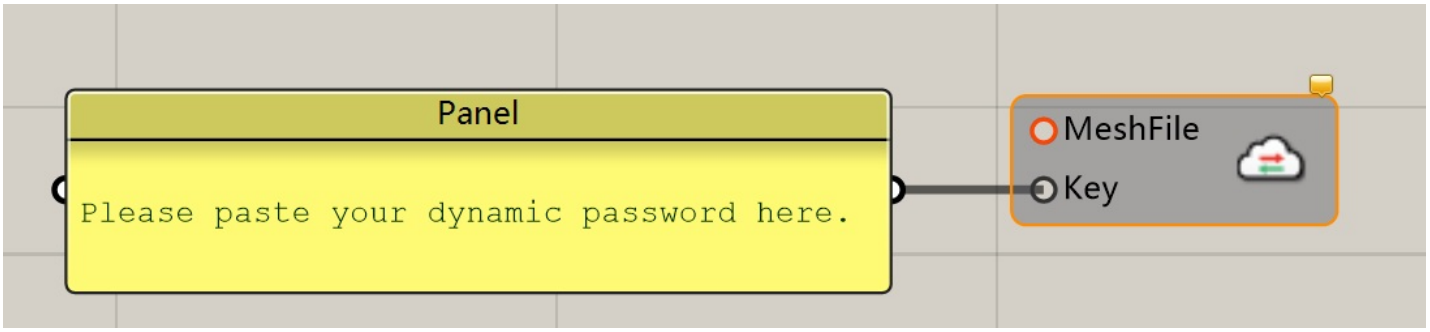
User login http://ameba.xieym.com

Click the top right corner Login | Register , click the top right corner panel after login ,

click【computing services】



Switch to dynamic password web page , click Generate button , a dynamic password will be generated in the upper box, which is used to login cloud server 。



Logout

Click the top right panel, logout。

Start a new project

Start a new project

- Start a new project
  - Preparation
  - Requirements of plug-in environment :
  - Install plug-in
  - Software operation interface
  - Work Space

Preparation

Download the Ameba.exe file and run the installation.

The unit system of Ameba is: N, mm, Mpa. Therefore, it is best to keep mm as the unit of Rhino's model.

Requirements of plug-in environment :

.NET Framework 4.6.1

Rhinoceros 6 or no less than Rhinoceros 5 SR12 (Windows) + Grasshopper(Ver>0.9)

Install plug-in

Rhino plug-in Download

Software operation interface

Work Space

Rhinoceros

Start a 2D optimization project

Start a 2D optimization project (Rhino)

- Start a 2D optimization project (Rhino)
  - Calculation instructions :
  - Attention :
  - Example :
    - 1. Define a design domain
    - 2. Generate mesh
    - 3. Define the support
    - 4. Define the load
    - 5. Define other parameters
    - 6. Computing services
  - 7. Visualize calculation results
    - 8. Post-processing of two-dimensional topology optimization

Calculation instructions :

2D calculation refers to the calculation of a plane object to approximate the simulated 3D environment. The advantage is that the calculation speed is faster.

Attention :

All 2D objects must be closed 2D objects (a flat surface or a closed curve)

Example :

2D calculations can only be performed in Rhino's Top view.

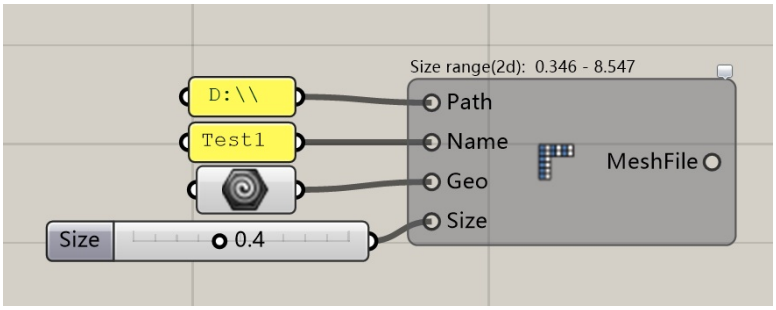
---

1. Define a design domain

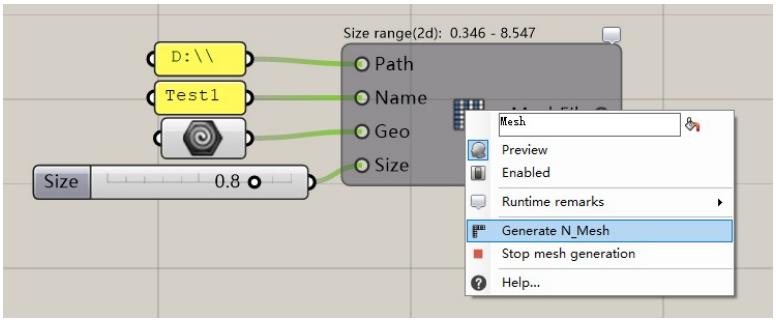
A plane represents a closed design domain.

2. Generate mesh

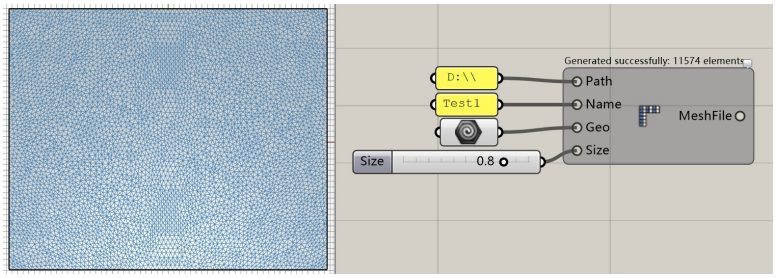
Use the Mesh component to pick up the model, then right click on the component button Generate N_Mesh.



Wait for a while, if the component is still yellow or red, you can click the small tab in the upper right corner of the component to view the error information. Usually the mesh becomes white when the step completes (Attention: Size must be set within the Size range).

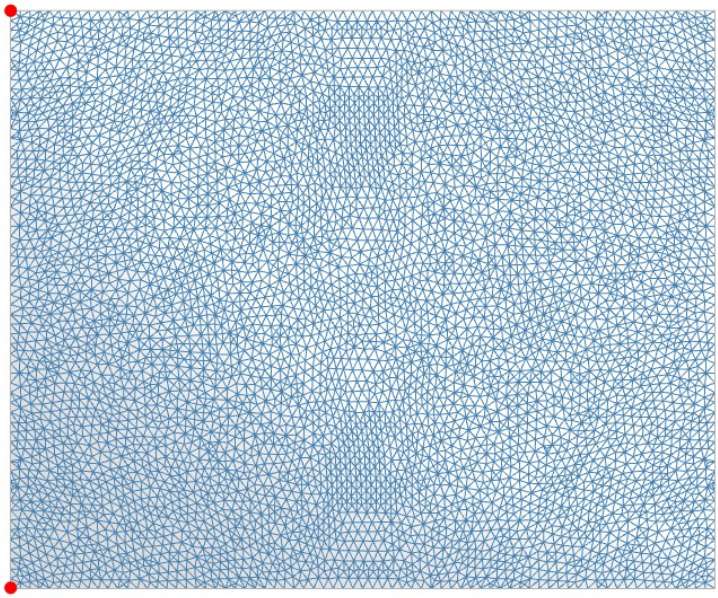


After the success of dividing the mesh, we will see that the model is covered with blue mesh edges, and the top of the component shows how many mesh cells have been generated.

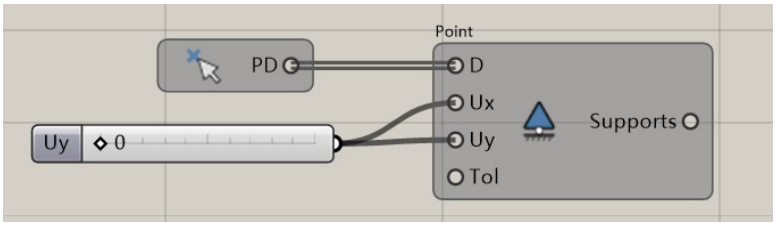


3. Define the support

The support defines the red point below.

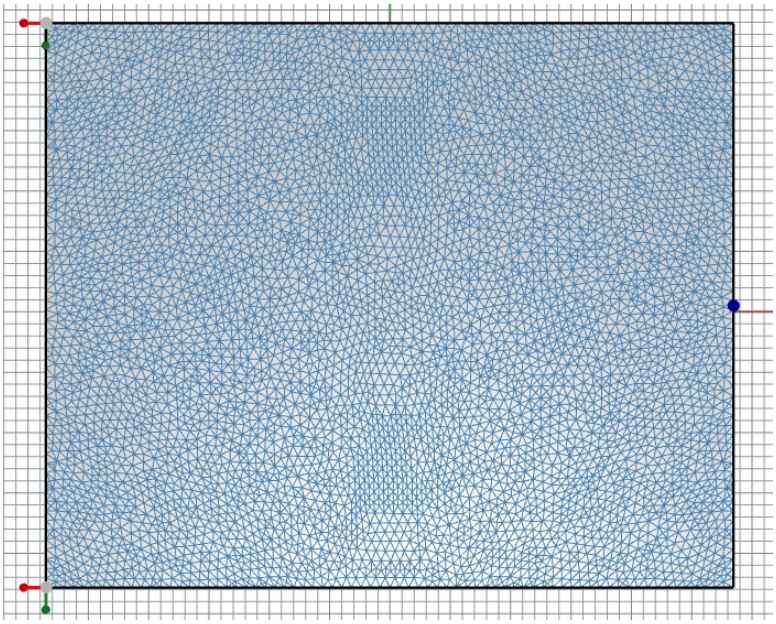


The following figure defines the fixed support in the XY direction.

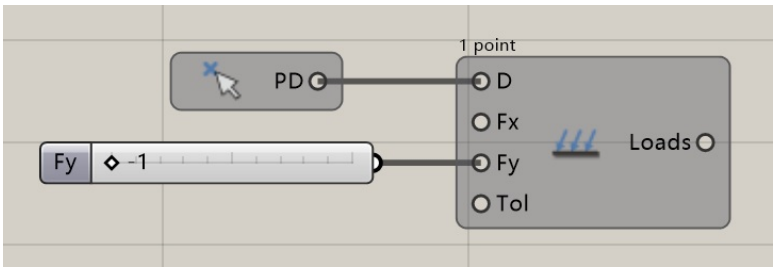


4. Define the load

Define the load at the blue point below.

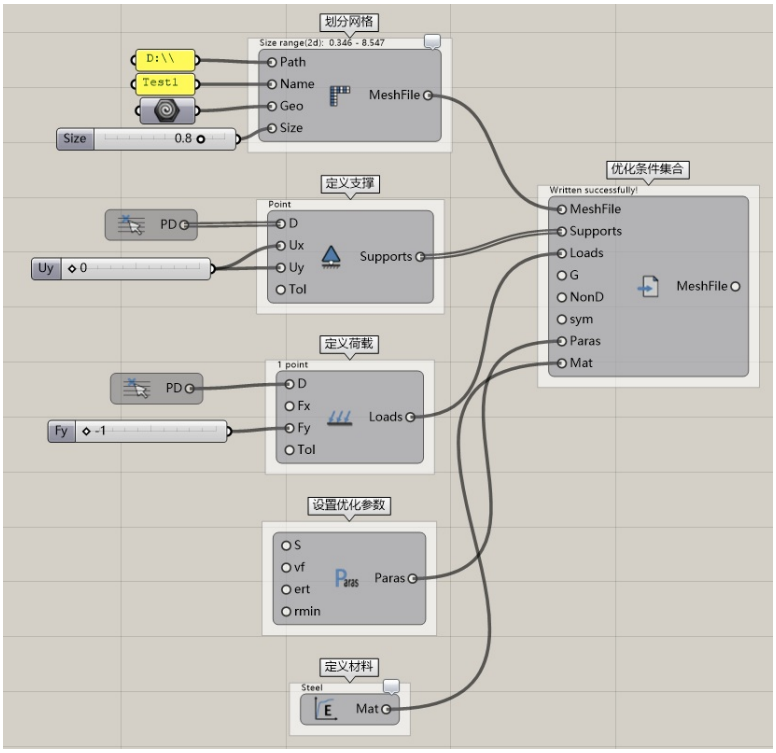


Define the point load in the Y-axis direction



5. Define other parameters

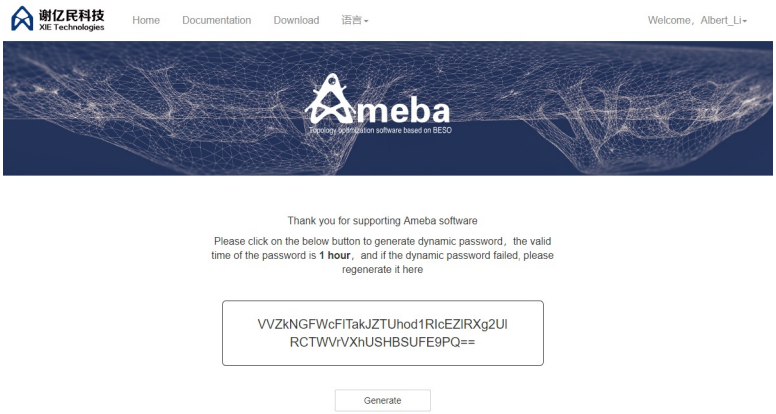
This example all stays default.

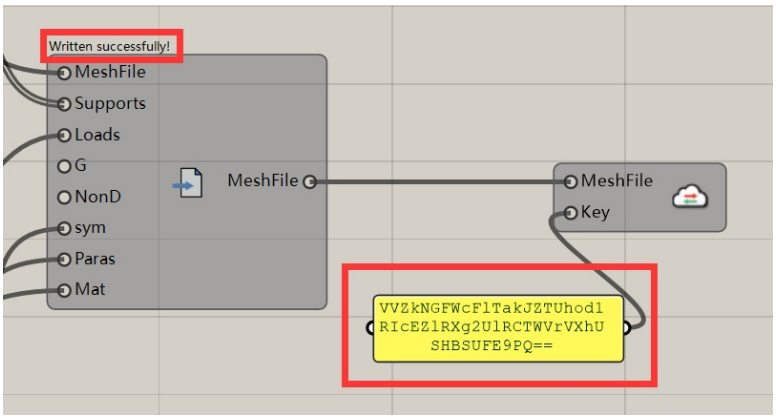


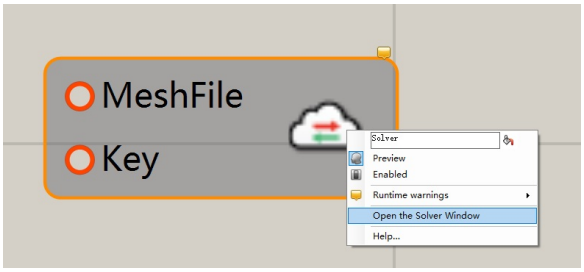
6. Computing services

First of all, you are supposed to insure PreProcessing component has revealed "Written successfully!" . Then you need to log in https://ameba.xieym.com/key_en/ to get the key and enter it into a panel component. Finally, connected them with Solve component.

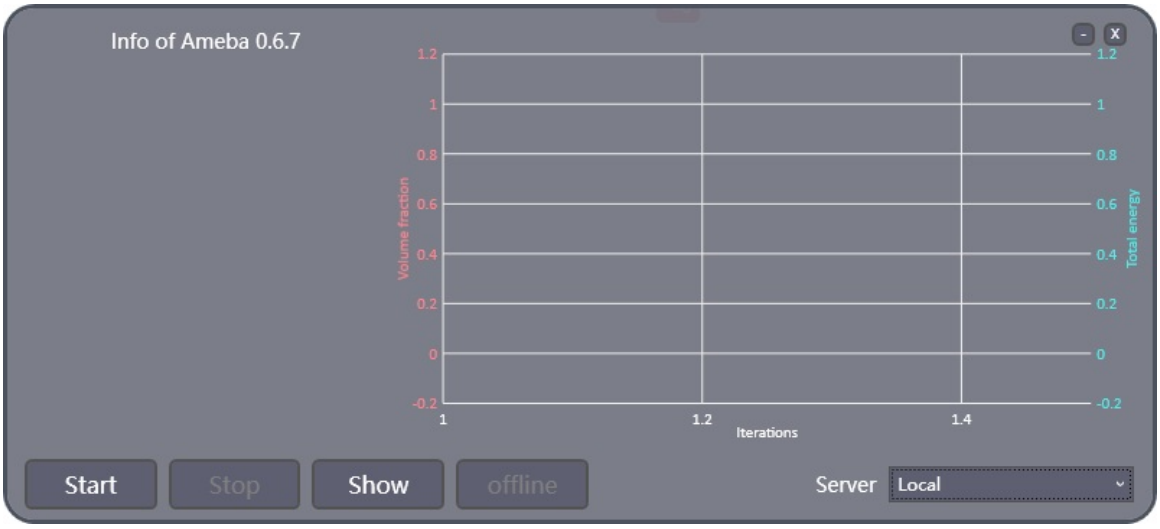
Notes: The key has just an hour of validity. But it can't be shut downwhen your project is under calculation. You need to regenerate it from Ameba's website.







Right-click Solve component and select "Open the Solve Window". You can also double-click this component. After operations, Solver Window can be opened.

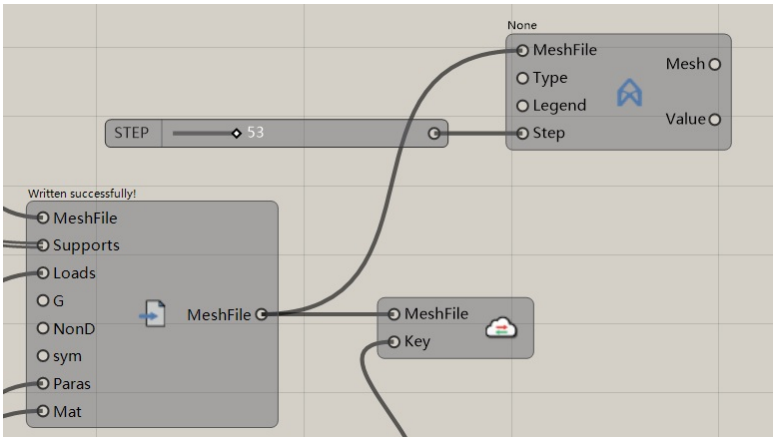


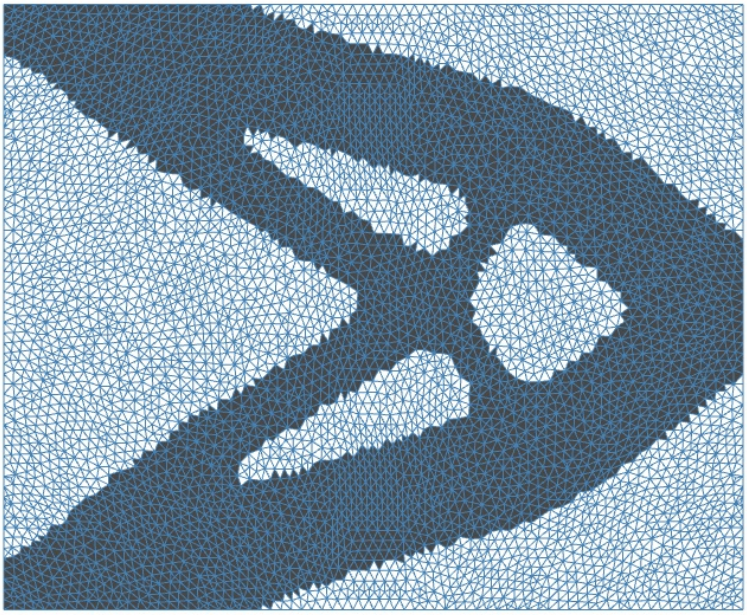
Then, you are supposed to select a server, China Shanghai or US.Virginia, and click Start button. Please wait a moment until the Solver Information Panel displays "The calculation has started. Please wait patiently." Next, you can have a cup of coffee and wait about ten minutes(depends on your mesh model, load case and internet speed). After completion, you can review the result model using Display component.

You could click the Solver Window and get more details.

7. Visualize calculation results

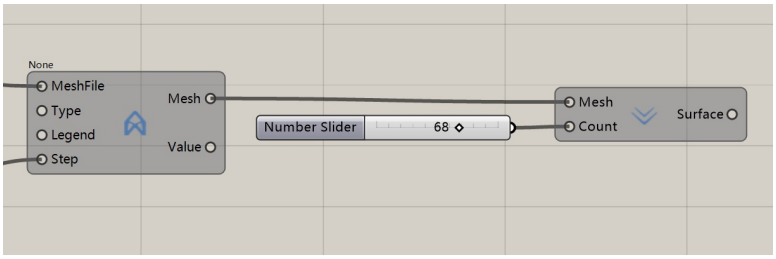
Access to the Display and Step component. In the above, the Last Result Button of the calculation window can directly adjust the Step to the number of steps in the latest calculation. After the calculation is completed, you can click the Last Result Button to view the final calculation result.

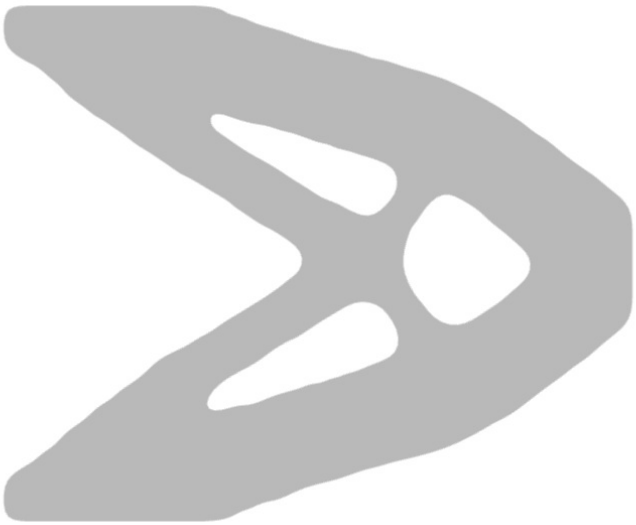




8. Post-processing of two-dimensional topology optimization

Post-process the resulting mesh model with the Rebuilding2D component to achieve higher model quality.





Start a 3D optimization project

Start a 3D optimization project

- Start a 3D optimization project
  - Calculation instructions：
  - Matters need attention：
  - Example：
    - 1. Define a design domain
    - 2. Generate mesh
    - 3. Define the support
    - 4. Define the load
    - 5. Define other parameters
    - 6. Computing services
      - You could click the Solver Window and get more details.
    - 7. Visualize calculation results
    - 8. Post-processing of three-dimensional topology optimization

Calculation instructions：

Three-dimensional calculation refers to carry out optimization calculation of a closed three-dimensional object, can be any closed three-dimensional entity.

Matters need attention：

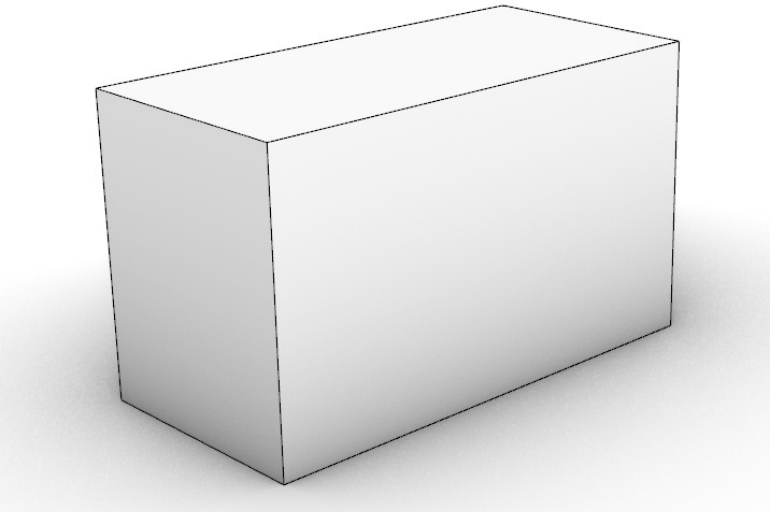
All 3D objects must be closed 3D geometry.

Example：

All 3D calculations are done in the perspective view of Rhino。

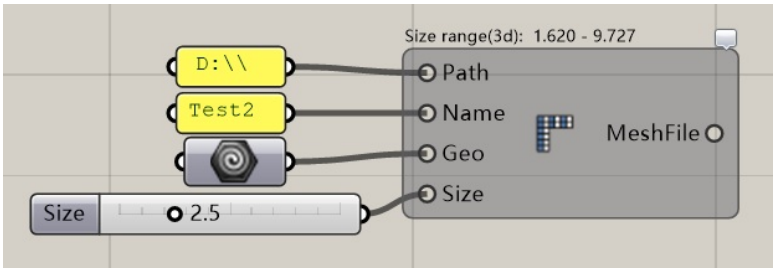
1. Define a design domain

A box represents a closed design domain.

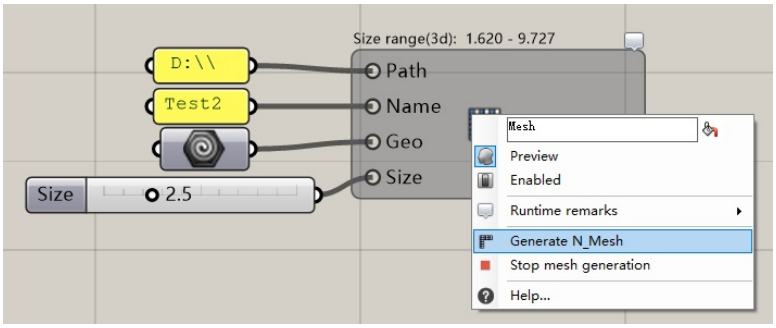


2. Generate mesh

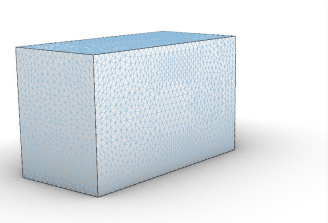
Use the Mesh component to pick up the model, then right click on the component button Generate N_Mesh.

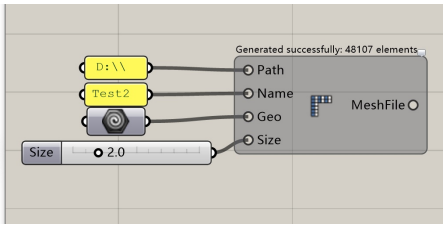


Wait for a while, if the component is still yellow or red, you can click the small tab in the upper right corner of the component to view the error information. Usually the mesh becomes white when the step completes (Attention: Size must be set within the Size range).



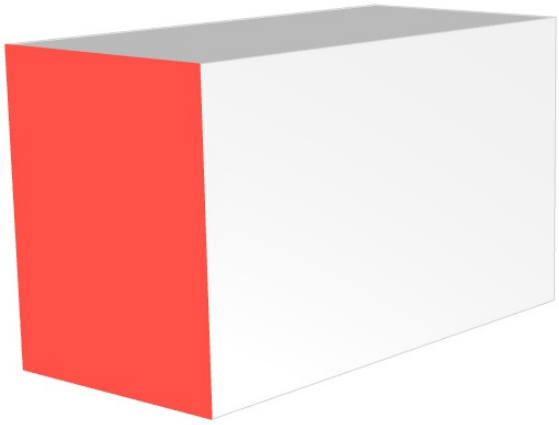
After the success of dividing the mesh, we will see that the model is covered with blue mesh edges, and the top of the component shows how many mesh cells have been generated.



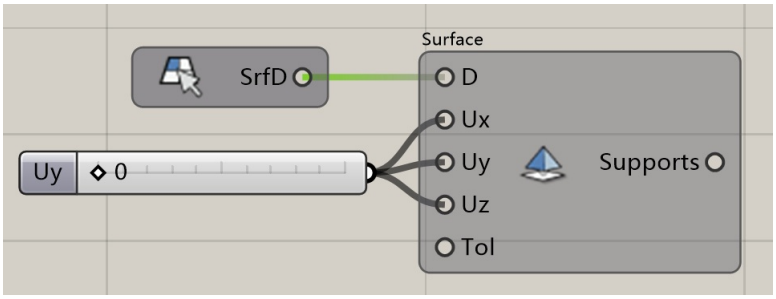


3. Define the support

The supports are defined at the red surface of the figure below.

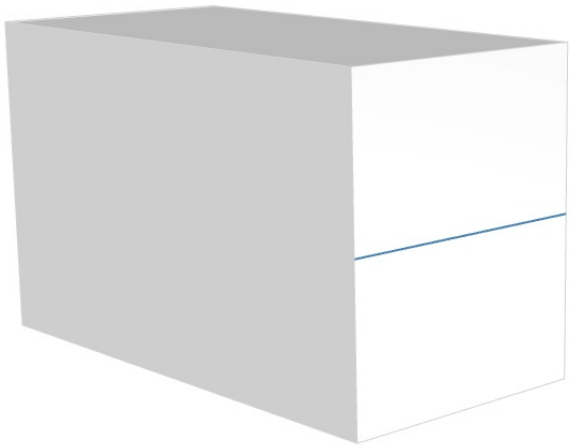


The following figure defines the fixed support in the XYZ direction.

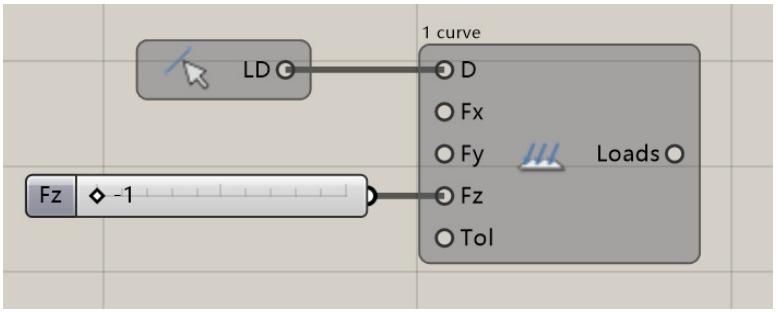


4. Define the load

Define the load at the blue point below.

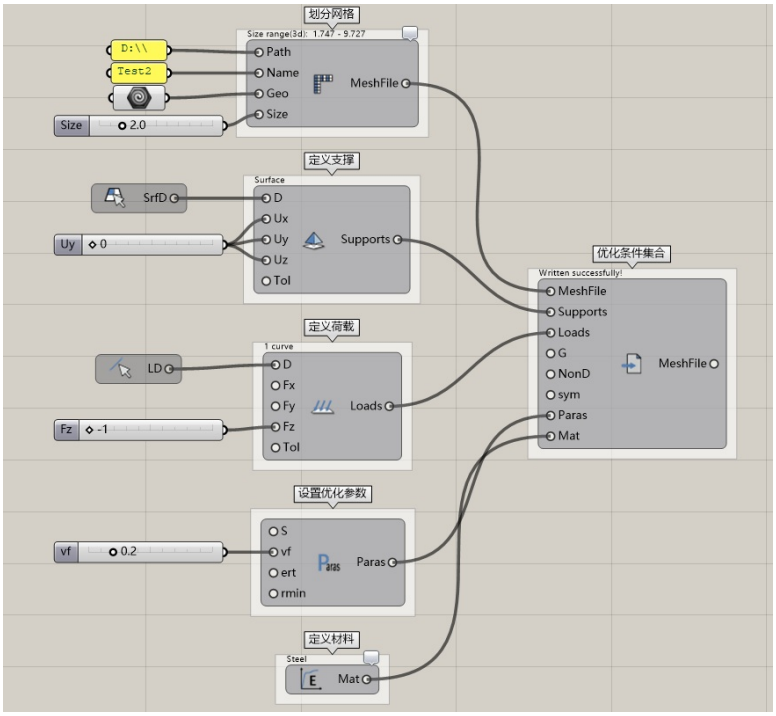


Define the point load in the Z-axis direction



5. Define other parameters

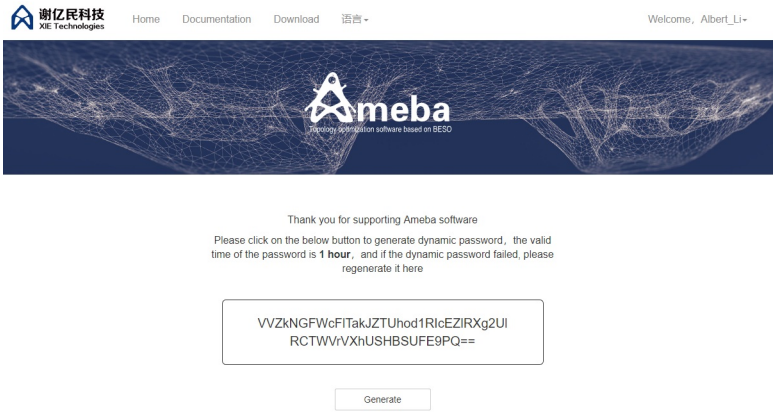
The volume fraction of this example is 20%, and all other parameters remain the default.

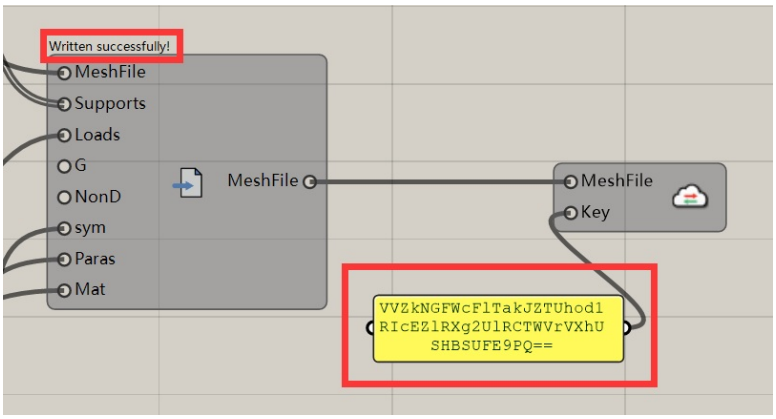


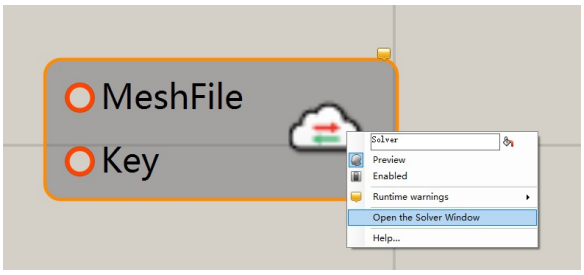
6. Computing services

First of all, you are supposed to insure PreProcessing component has revealed "Written successfully!" . Then you need to log in https://ameba.xieym.com/key_en/ to get the key and enter it into a panel component. Finally, connected them with Solve component.

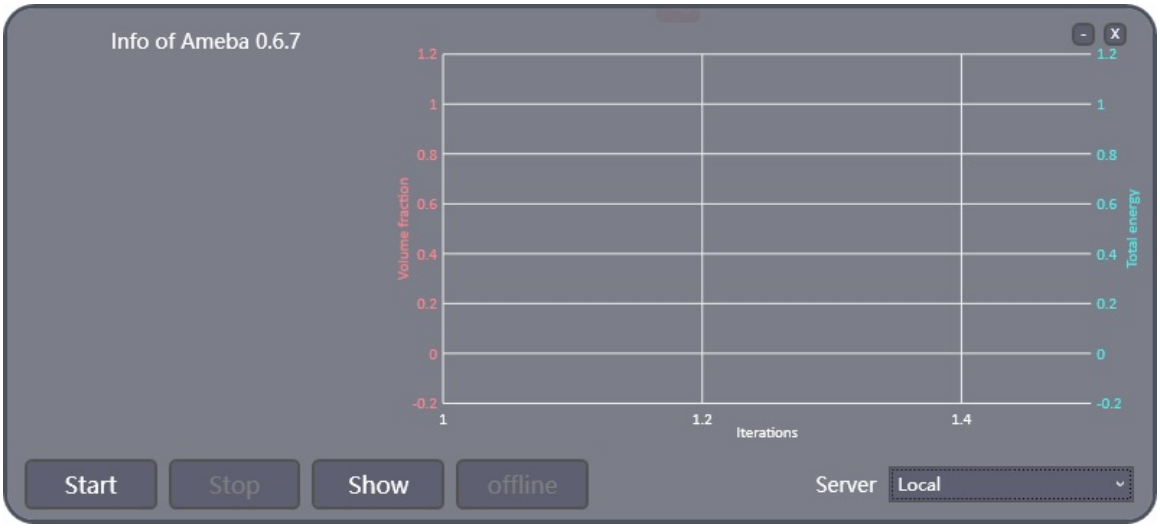
Notes: The key has just an hour of validity. But it can't be shut downwhen your project is under calculation. You need to regenerate it from Ameba's website.







Right-click Solve component and select "Open the Solve Window". You can also double-click this component. After operations, Solver Window can be opened.



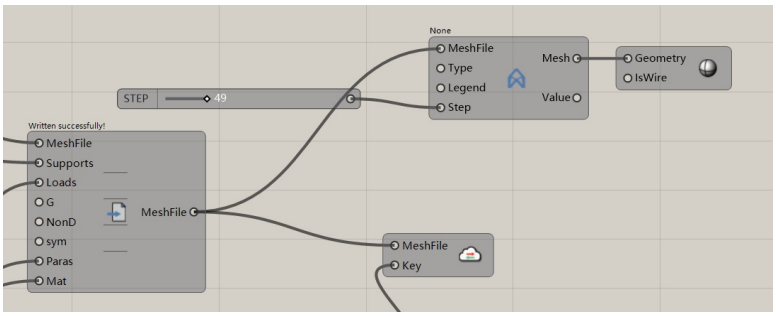
Then, you are supposed to select a server, China Shanghai or US.Virginia, and click Start button. Please wait a moment until the Solver Information Panel displays "The calculation has started. Please wait patiently." Next, you can have a cup of coffee and wait about ten minutes(depends on your mesh model, load case and internet speed). After completion, you can review the result model using Display

component.

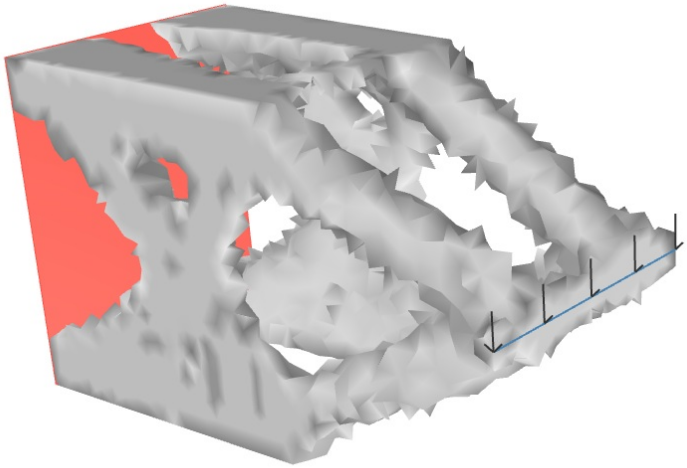
You could click the Solver Window and get more details.

7. Visualize calculation results

Access to the Display and Step component. In the above, the Last Result Button of the calculation window can directly adjust the Step to the number of steps in the latest calculation. After the calculation is completed, you can click the Last Result Button to view the final calculation result.



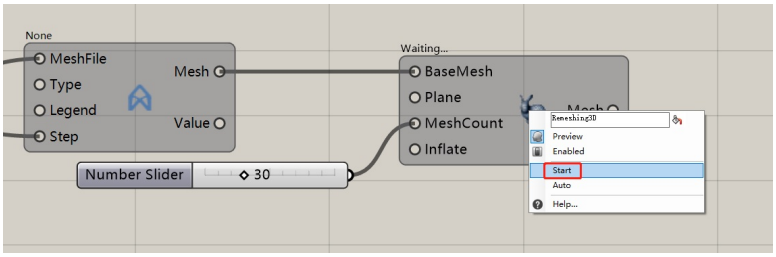
As shown in the figure below, the result model of 3D topology optimization is obtained.



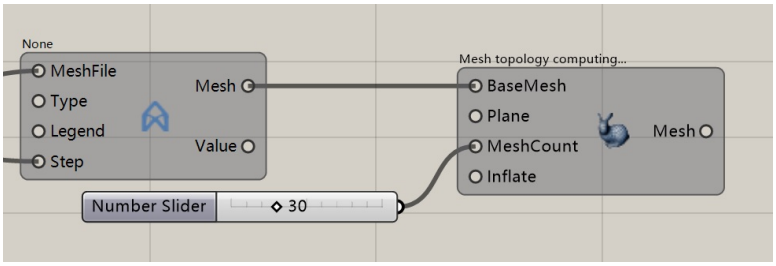
8. Post-processing of three-dimensional topology optimization

Since the calculated mesh model is rough, it is difficult for designers to continue editing. We can use Ameba's MeshTools to post-process the model.

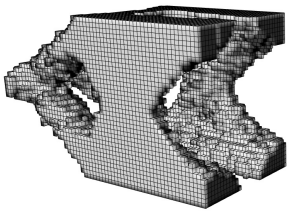
Use Remeshing3D to perform mesh reconstruction on a three-dimensional topology-optimized model. Click Start to start the calculation (Auto mode can calculate in real time, but it may cause a jam).

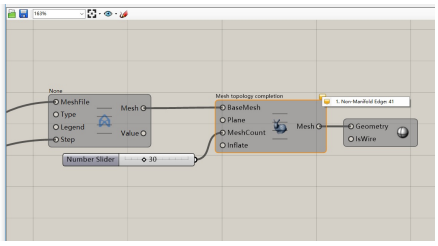


After starting the calculation, the information above will become [Mesh topology computing...]

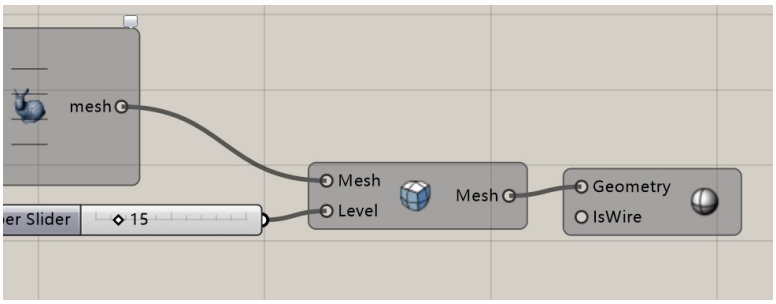


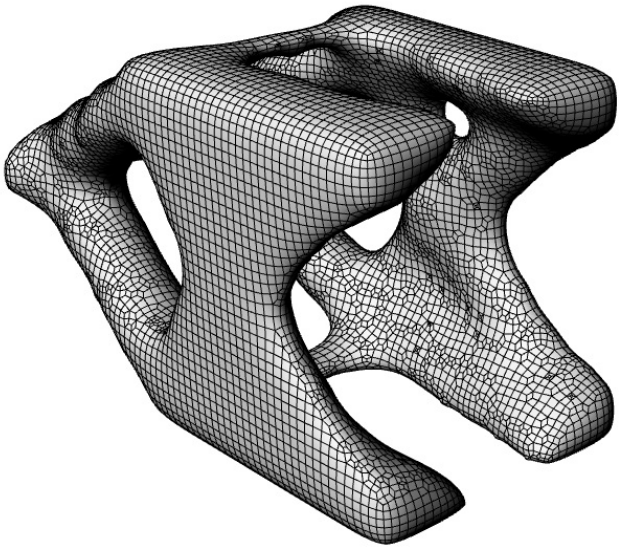
After the calculation is completed, a structured mesh model stacked by Mesh Box will be obtained. Due to the limitation of the Free version, this model may generate unsmooth edges, and the MeshCount parameter can only be input within 30.





Finally, we access the AmebaSmooth component to smooth the model to get a smooth and re-editable mesh model.





Function Introduction

Function Introduction

Domain

A group of components for getting Rhinoceros objects.

Ameba2D

A group of components used to set two-dimensional topology optimization conditions. Users can use these components to set loads, non-design areas, and supports.

Ameba3D

A group of components used to set three-dimensional topology optimization conditions. Users can use these components to set loads, non-design areas, and supports.

Mesh

Convert the user's model into a mesh model that can be used for calculations.

Preprocess

For the component group of calculating the result pre-processing, the user can use the module under the group to set the symmetry constraint, material, optimization parameters, sensitivity options, multiple working conditions, and also collect the pre-processing parameters to connect the cloud server.

Postprocess

For the component group used for post-processing of calculation results, the user can use the module under the group to query and display the calculation result, and also reconstruct the mesh model.

Cloud

A component group used to interact with the cloud server. Users can use the components under the group to perform cloud computing or log in to the server.

MeshTools

Optimized mesh model.

Domain

Domain

A group of components for getting Rhinoceros objects.

PointsDomain

Get the selected point object.

LinesDomain

Get the selected line.

CurvesDomain

Get the selected curve.

SurfacesDomain

Get the selected surface.

PointsDomain

PointsDomain

Description:

Get the selected point object.

Usage:

Right-click on the battery, then select 【Set points】, and draw points directly in Rhino.

Output:

PD: point objects.

LinesDomain

LinesDomain

Description：

Get the selected line.

Usage：

Right-click the battery, then select 【Set lines】, and draw lines directly in Rhino.

Output:

LD:lines domain.

CurvesDomain

CurvesDomain

Description:

Get the selected curve.

Usage:

Right-button battery, then select 【Select lines】, and pick up straight objects directly in Rhino.

Output:

CurveD: curves domain.

SurfacesDomain

SurfacesDomain

Description:

Get the selected surface.

Usage:

Right-click the battery, then select 【Select surface】, and pick up the plane surface directly in Rhino.

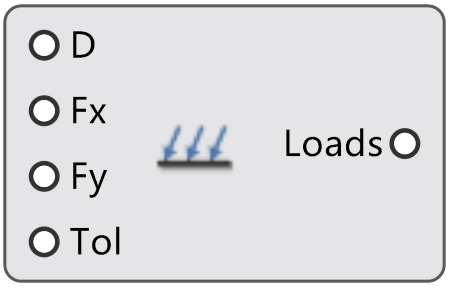
Output:

SrfD: surfaces domain.

Ameba2D

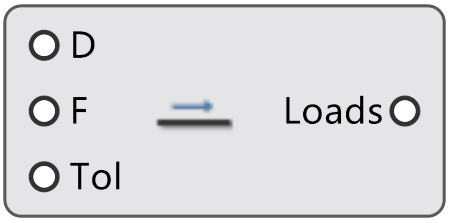
Ameba2D

A group of components used to set two-dimensional topology optimization conditions. Users can use these components to set loads, non-design areas, and supports.



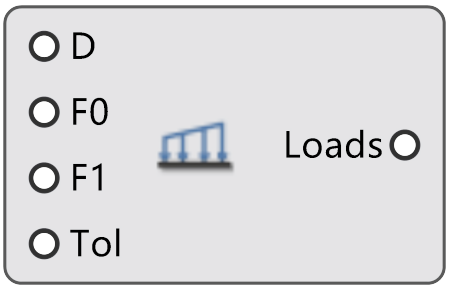
Load2D

Apply a two-dimensional load along a certain direction.



TangentLoad2D

Define the two-dimensional shear load on the surface of the object.

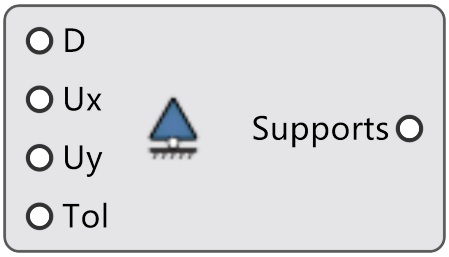


[NormalLoad2D =230x]](NormalLoad2D.md)(NormalLoad2D.md)

Apply a two-dimensional load along the normal to the surface of the object.

NonDesignDomain2D

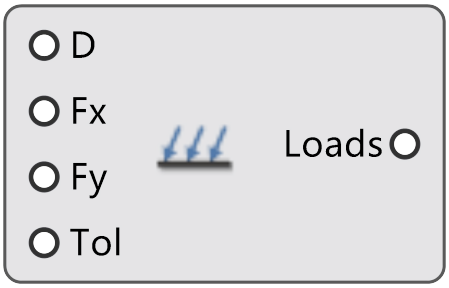
Define a two-dimensional non-design area.



Support2D

Define two-dimensional support.

Load2D



Load2D

Description：

In the two-dimensional analysis, a general load is defined and the load can only be applied to the surface of the object.

(This component must be used after generating the mesh.)

Usage：

Right-click on ShowSelElement to display the cell to which the load is applied and the cell center point to represent the cell.

Input：

D: Enter the area of a load. You can input points, lines or surfaces.

Fx: Enter the load in the X-axis direction.

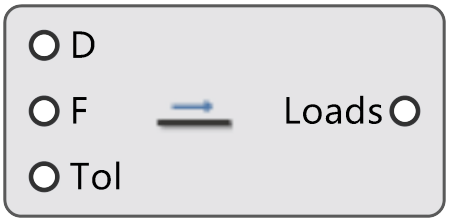
Fy: Enter the load in the Y-axis direction.

Tol: The tolerance for area determination. This parameter will be invalid when you input surfaces.

Output：

Loads: Output a load.

TangentLoad2D



TangentLoad2D

Description：

In the two-dimensional analysis, a shear load is defined and the load can only be applied to the surface of the object.

(This component must be used after generating the mesh.)

Usage：

Right-click on ShowSelElement to display the cell to which the load is applied and the cell center point to represent the cell.

Input：

D: Enter the area of a load. You can input circles or line.

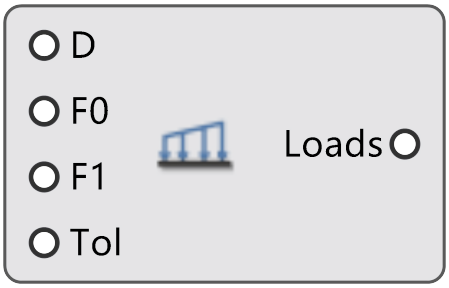
F: The size of the line load.

Tol: The tolerance for area determination. This parameter will be invalid when you input surfaces.

Output：

Loads: Output a load.

NormalLoad2D



NormalLoad2D

Description：

Define a normal uniform load along a curve in 2D problem.

(This component must be used after generating the mesh.)

Usage：

Right-click on ShowSelElement to display the cell to which the load is applied and the cell center point to represent the cell.

Input：

D: Enter the area of a load. You can input circles or lines.

F0: Load size of left end.

F1: Load size of right end.

Tol: The tolerance for area determination. This parameter will be invalid when you input surfaces.

Output：

Loads: Output a load.

NondesignDomain2D

NondesignDomain2D

Description：

Design a non-desgin domain in 2D problem.

(This component must be used after generating the mesh.)

Usage：

Right-click on ShowSelElement to display cells that are not designed, and use cell center points to represent cells.

Input：

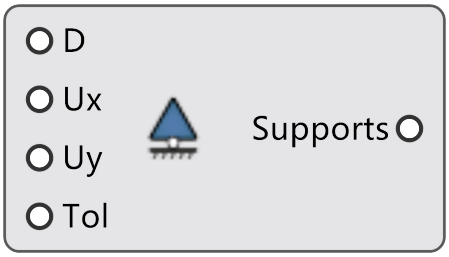
D: Reference line of non-desgin domain. You can input lines or surfaces.

Tol: The tolerance for area determination. This parameter will be invalid when you input surfaces.

Output：

NonD: Non-desgin domain.

Support2D



Support2D

Description：

Define a support in 2D problem.

(This component must be used after generating the mesh.)

Input：

D: Enter a support domain. You can input points, lines or surfaces.

Ux: The displacement in the X-axis direction. If you don't input any vaule, there will be no constrains in this direction. If you input 0, there will be constrained in this direction. If you input a non-zero value, this value will mean displacement in this direction.

Uy: The displacement in the Y-axis direction. If you don't input any vaule, there will be no constrains in this direction. If you input 0, there will be constrained in this direction. If you input a non-zero value, this value will mean displacement in this direction.

Tol: The tolerance for area determination. This parameter will be invalid when you input surfaces.

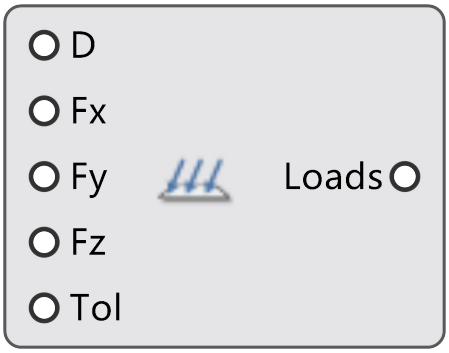
Output：

Supports: Output a support.

Ameba3D

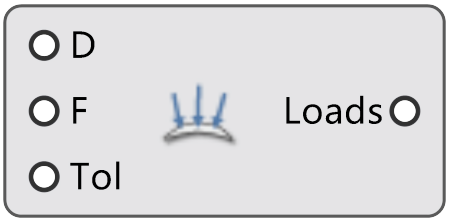
Ameba3D

A group of components used to set three-dimensional topology optimization conditions. Users can use these components to set loads, non-design areas, and supports.



Load3D

Apply a three-dimensional load along a certain direction.

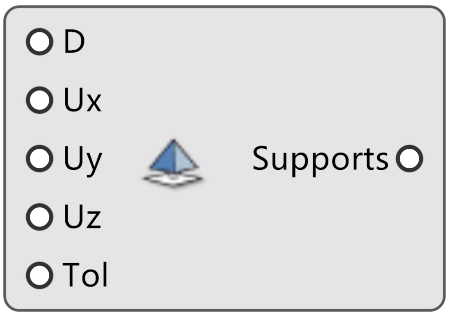


NormalLoad3D

Apply a three-dimensional load along the normal to the surface of the object.

NonDesignDomain3D

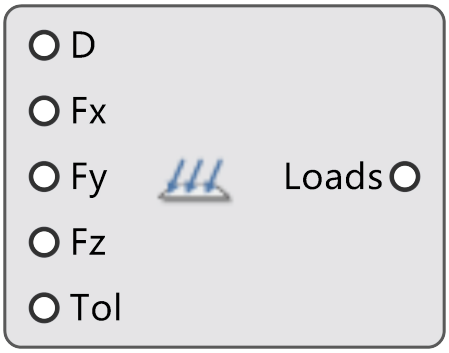
Define three-dimensional non-design areas.



Support3D

Define three-dimensional supports.

Load3D



Load3D

Description：

Define a genaral load in 3D problem.

(This component must be used after generating the mesh.)

Usage：

Right-click on ShowSelElement to display the cell to which the load is applied and the cell center point to represent the cell.

Input：

D: Enter the area of a load. You can input points, lines, surfaces or closed soilds.

Fx: Enter the load in the X-axis direction.

Fy: Enter the load in the Y-axis direction.

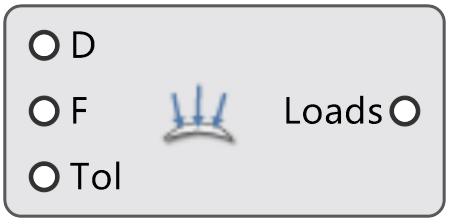
Fz: Enter the load in the Z-axis direction.

Tol: The tolerance for area determination. This parameter will be invalid when you input closed soilds.

Output：

Loads: Output a load.

NormalLoad3D



NormalLoad3D

Description：

In the three-dimensional analysis, define a load that is perpendicular to the surface and the load can only be applied to the surface of the object.

(This component must be used after generating the mesh.)

Usage：

Right-click on ShowSelElement to display the cell to which the load is applied and the cell center point to represent the cell.

Input：

D: Enter the area of a load. You have to input surfaces.

F: Load size.

Tol: The tolerance for area determination. This parameter will be invalid when you input closed soilds.

Output：

Loads: Output a load.

NondesignDomain3D

NonDesignDomain3D

Description：

In the two-dimensional analysis, define a non-designed area (only valid in two-dimensional analysis). (This component must be used after generating the mesh.)

Usage：

Right-click on ShowSelElement to display cells that are not designed, and use cell center points to represent cells.

Input：

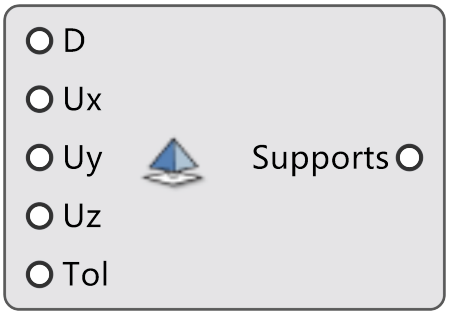
D: Enter the area of a load. You can input lines, surfaces or closed soilds.

Tol: The tolerance for area determination. This parameter will be invalid when you input closed soilds.

Output：

NonD: Output non-design area.

Support3D



Support3D

Description：

In the three-dimensional analysis, define a support (only valid in three-dimensional analysis).

(This component must be used after generating the mesh.)

Input：

D: Enter the area of a load. You can input points, lines, surfaces or closed soilds.

Ux: The displacement in the X-axis direction. If you don't input any vaule, there will be no constrains in this direction. If you input 0, there will be constrained in this direction. If you input a non-zero value, this value will mean displacement in this direction.

Uy: The displacement in the Y-axis direction. If you don't input any vaule, there will be no constrains in this direction. If you input 0, there will be constrained in this direction. If you input a non-zero value, this value will mean displacement in this direction.

Uz: The displacement in the Z-axis direction. If you don't input any vaule, there will be no constrains in this direction. If you input 0, there will be constrained in this direction. If you input a non-zero value, this value will mean displacement in this direction.

Tol: The tolerance for area determination. This parameter will be invalid when you input closed soilds.

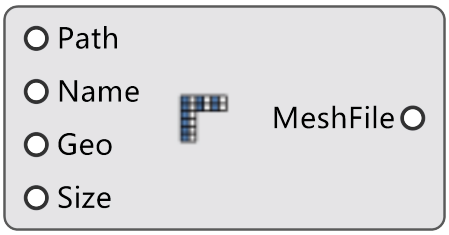
Output：

Supports: Output a support.

Mesh

Mesh

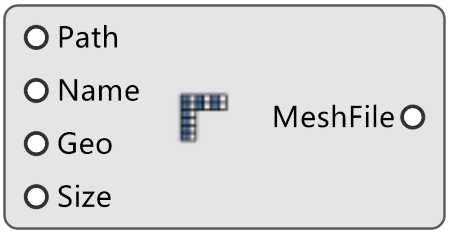
Convert the user's model into a mesh model that can be used for calculations.



Mesh模块

The objects to be analyzed are meshed for cloud operations.

Mesh



Mesh

Description：

(If it is a two-dimensional calculation, the plane must be XY plane, the Z value of the coordinates of all the vertices of the plane must be 0)

Usage：

Set all the inputs, then right click on the battery and select Generate N_Mesh to mesh.

Input：

Path: Define a directory to store project files.

Name: Define a project name.

Geo: Define an analysis area.

Size: Divide the size of the grid.

Attention：

After settingPath, Name, Geo, Size range will appear, and Size can only be set to within the Size range below the component.

Output：

MeshFile: Connect Display, PreProcessing and Solver for data transmission.

PreProcess

Preprocess

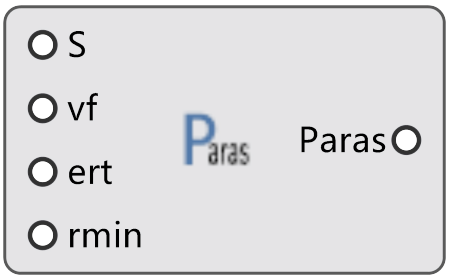
For the component group of calculating the result pre-processing, the user can use the module under the group to set the symmetry constraint, material, optimization parameters, sensitivity options, multiple working conditions, and also collect the pre-processing parameters to connect the cloud server.

Symmetry

Symmetric constraints.

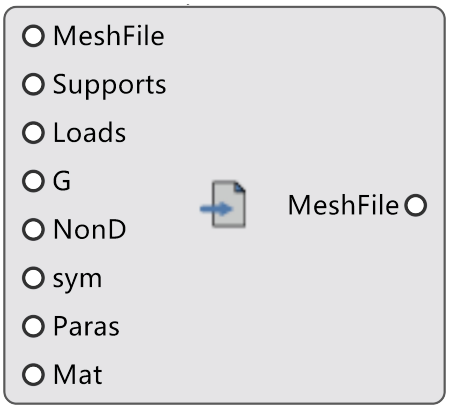
Material

Material settings.



OptParameters

Optimize parameter settings.



PreProcessing

Pre-processing parameter set.

Sensitivity

Sensitivity option.

MultipleLoadCase

Multiple settings.

Symmetry

Symmetry

Description：

If a symmetric constraint is imposed, define a line (two-dimensional) or a plane (three-dimensional) as the axis of symmetry or face.

(This component must be used after generating the mesh.)

Input：

Sym: Enter the axis of symmetry (two-dimensional) or the plane of symmetry (three-dimensional).

Tol: Tolerance value.

Output：

Syms: Symmetric constraint.

Material

Material

Description：

Define the material of the object.

Attention：

E and u inputs are valid only when set to User.

Usage：

Right-click on the battery and select some of the preset materials.

Input：

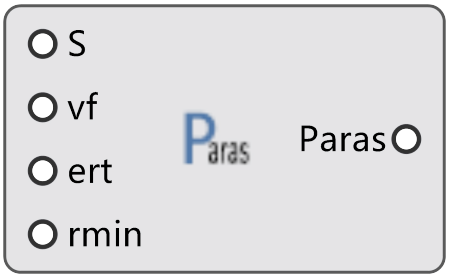
E: modulus of elasticity.

u: Poisson's ratio.

Output：

Mat: Material parameters.

OptParameters



OptParameters

Description :

Define some parameters of the BESO algorithm.

Input :

S: Sensitivity. Presets can be selected from PreProcessing.

```
In short, sensitivity is the basis for the increase and decrease of the unit.
At present, the strain energy density and the Mises stress can be selected.
When the unit volumes are all the same, the calculation results of the strain
energy density or the Mises stress are the same, but the unit sizes are
generally different. In order to consider the influence of the unit volume,
the strain energy density is selected.
```

vf: Constrained volume fraction.

```
The constrained volume fraction is the percentage of the volume of
the material area to be retained in the original design area (0-1).
This value cannot be too small. Otherwise, the material is not enough
to establish a structure. The calculated structure is not established,
and the calculation will not be convergence.
```

ert: Evolution rate.

```
Evolution rate is the volume percentage of material cut at each step.
(recommended 0.01-0.05)
```

rmin:过滤半径。（默认为3倍的网格）

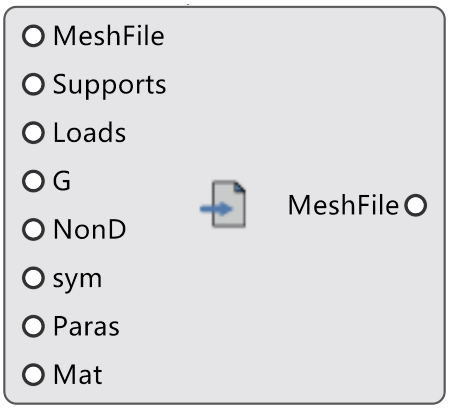
```
It can be understood that the center of the i-th unit is the center,
the radius is Rmin (the three-dimensional is the ball), and the
sensitivity of the i-th unit is determined according to the distance
```

```
between the inner unit and the i-th unit.
```

Output :

Paras: Parameter set.

PreProcessing



PreProcessing

Description：

Write all the definitions of the analysis to a file and read the file as a cloud.

输入端：

MeshFile: Transfer the data from the Mesh battery to this battery.

Supports: Connect all the supports as a List.

Loads: All loads are accessed as a List.

G: Gravity (defined by gravity, you can customize the direction and size of gravity).

NonD: Non-designed area. (If not, keep it as default).

Sym: Symmetric constraint. (If not, keep it as default).

Paras: Optimization parameter set.

Mat: Material.

Output：

MeshFile:Mesh File.

Sensitivity

Sensitivity

Description：

Some presets for the sensitivity of the BESO algorithm are given.

Output：

S: Sensitivity。

MultipleLoadCase

MultipleLoadCase

Description：

The input end of the component is a load case, and one load case can be composed of one or more loads;

Attention：

The input end of the component is a load case, and one load case can be composed of one or more loads;

输入端：

Case1: The first condition.

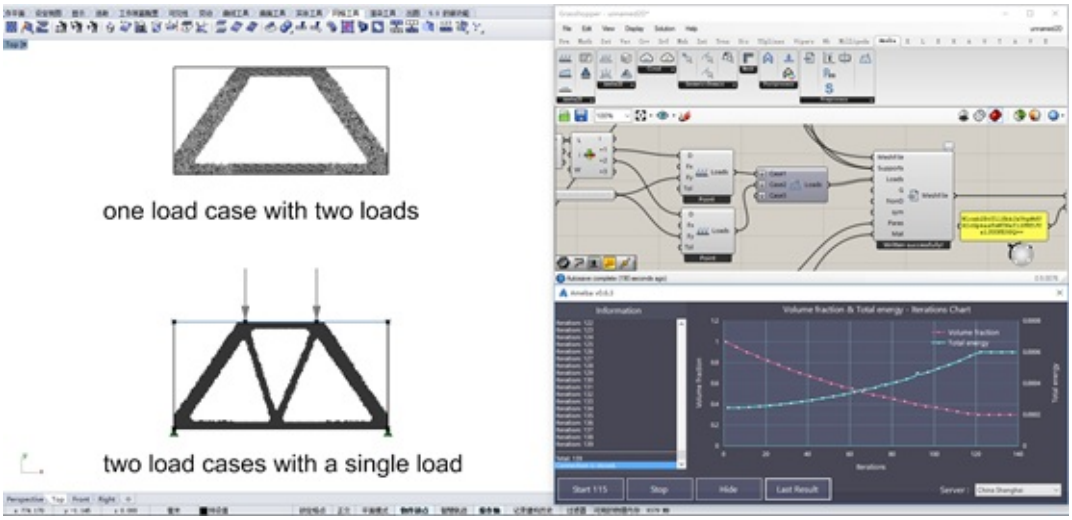
Case2: The second condition.

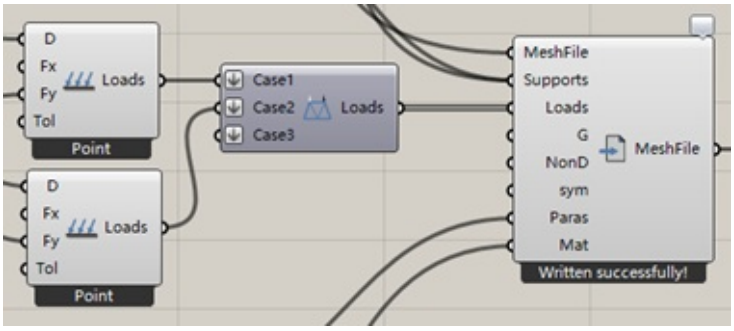
...

输出端：

Loads: Multiple working conditions collection.

示例：




D
Fx
Fy
Tol
Loads
Point
D
Fx
Fy
Tol
Loads
Point
Case1
Case2
Case3
Loads
MeshFile
Supports
Loads
G
NonD
sym
Paras
Mat
MeshFile
Written successfully!

PostProcess

Postprocess

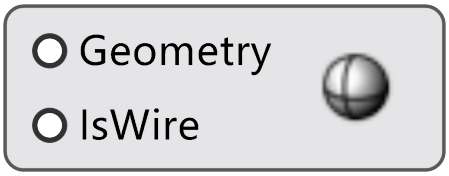
For the component group used for post-processing of calculation results, the user can use the module under the group to query and display the calculation result, and also reconstruct the mesh model.

Display

Display the result.

Step

The number of iteration steps.



RenderDisplay

Display the model.

Display

Display

Description：

Display the calculation result.

After the server completes the calculation, you will receive many files (topology optimizated result) from server. You need to connect Step component with Display component. Then, you can pull Step component's button for looking through these files. If you want to have a real-time view, you just left-click "Show" button in the calculation window.

Input：

MeshFile: Transfer the data from the Mesh component to this componenet.

Type: Display type ("None" means no cloud image, "Mises" means Mises stress, "Principal" means main stress)

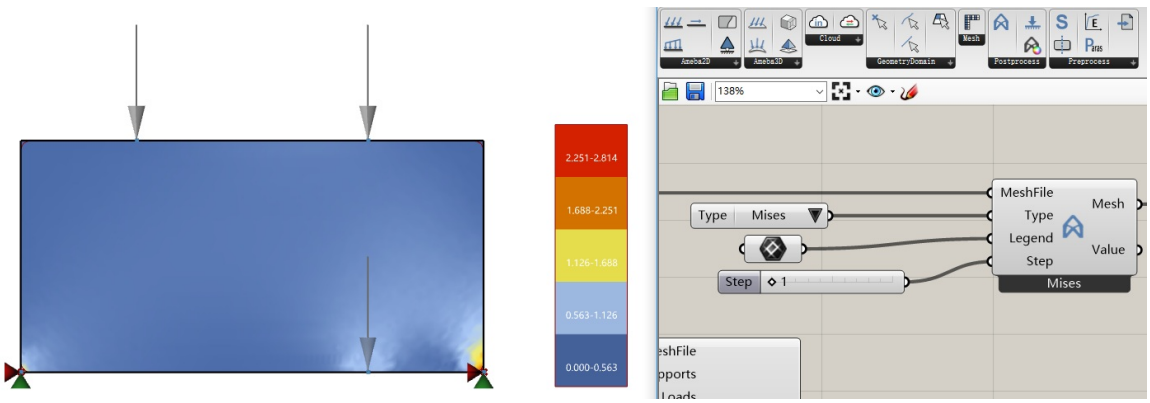
Step: Each step of the calculation (the total number of steps is different depending on the input mesh)

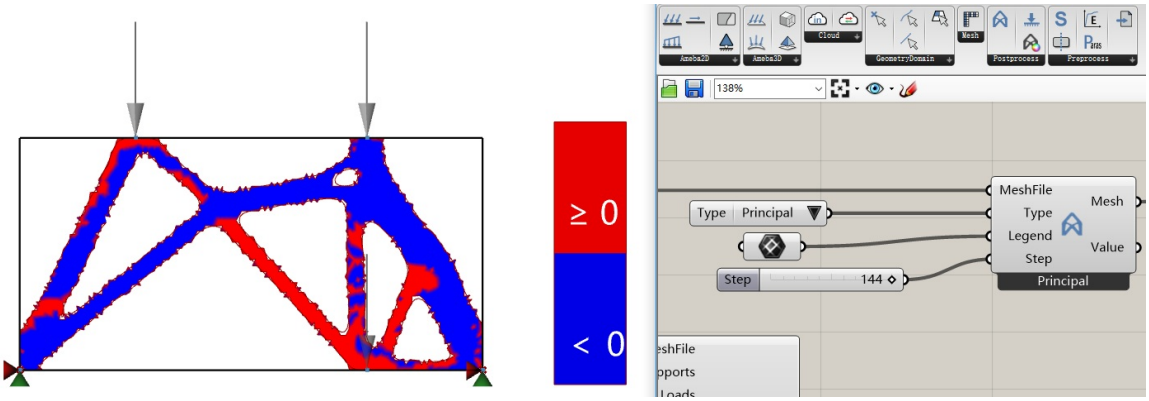
Output：

Mesh: Corresponding to the mesh of Step.

Value: The result value of the display type corresponding to the mesh vertex.

Example：




≥ 0
< 0
138%
Type
Principal
Step
144
MeshFile
Type
Legend
Step
Mesh
Value
Principal

Step

Step

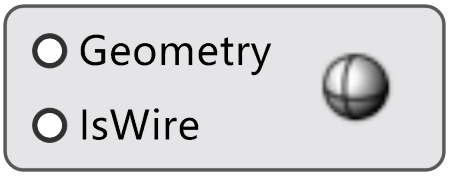
Description：

After the server completes the calculation, you will receive many files (topology optimizated result) from server. You need to connect Step component with Display component. Then, you can pull Step component's button for looking through these files. If you want to have a real-time view, you just left-click "Show" button in the calculation window after connecting Step component with Display component.

Output：

Step: The number of steps calculated.

RenderDisplay



RenderDisplay

Description :

Display the model as a white model with black edges.

Input :

Geometry: The model to be displayed.

IsWire: Whether to display mesh edges or surface boundaries.

Cloud

Cloud

A component group used to interact with the cloud server. Users can use the components under the group to perform cloud computing or log in to the server.

Solve

Deliver cloud computing and monitor cloud computing results.

Login

Log in to the cloud server.

Solver

Solver

Description：

Start cloud computing.

Usage：

The default server is "Shanghai", if it is an overseas user, you can switch to "Virginia".

Connect all the inputs, then right click on the component, select Display calculation window, open the server calculation window, press the window button Start to start the calculation, press Stop to stop the operation.

Hide：

Hide the window.

Last Result：

Automatically jump to the AmebaStep component to the latest number of files returned.

Attention：

The new version is that the calculation feedback is calculated and returned in real time, so when the calculation converges to its own needs, you can press Stop to end it, instead of waiting for all calculations to complete like the old version.

Input：

MeshFile: Transfer the data from the Mesh component to this component .

Key: dynamic password.

Login

Login

Description：

Log in to the server.

Usage：

Double-click the component to enter the Ameba login screen, and then register to log in. (This is what needs to be done before cloud computing).

Output：

Ver: The current Ameba version. (Users should update the latest version at the first time to reduce the possibility of BUG).

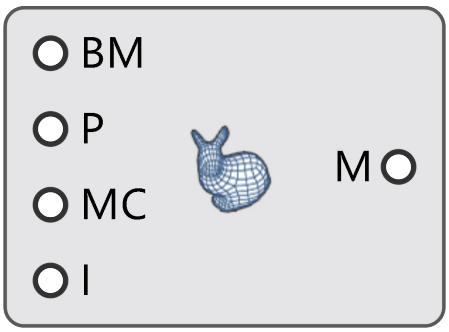
MeshTools

MeshTools

Optimized mesh model.

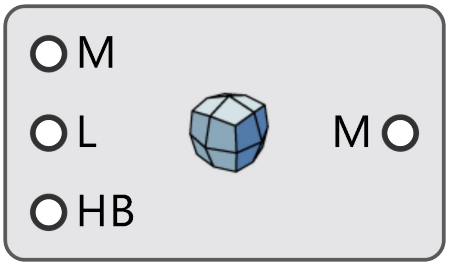
Rebuilding2D

The mesh calculated by Ameba2D is reconstructed into a Trimmed surface.



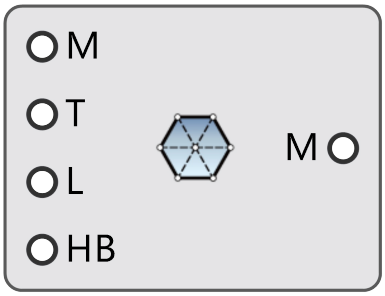
Remeshing3D

Reconstruct the mesh.



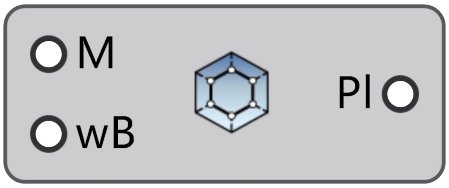
AmebaSmooth

Using the Laplacian Smoothing algorithm to optimized the mesh model.



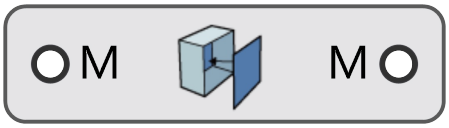
Subdivision

Subdiving the mesh using Catmull-Clark or Loop algorithms.



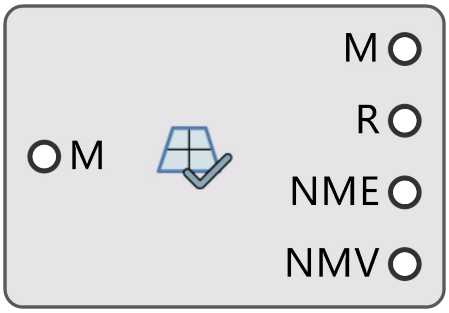
MeshDual

This component computes the dual of the mesh. Triangle meshes' dual graph are hexagons, quadrangle meshes' dual graph are still quadrangles (expressed by polylines).



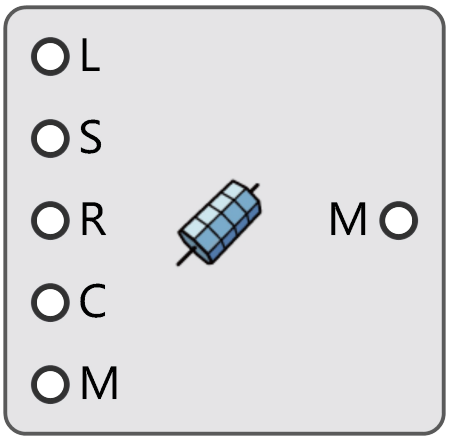
FillHoles

Create simply some mesh faces to fill the holes.



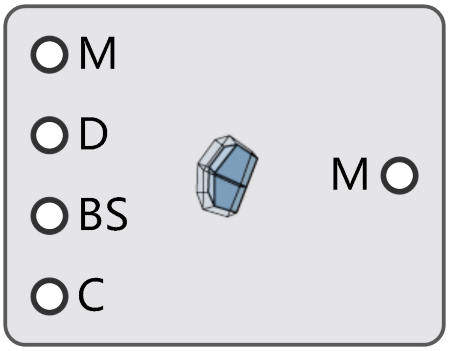
MeshChecker

Check and report informations that whether your mesh has non-manifold edges(or non-manifold vertices) or not.



MeshPipe

Create pipe meshes according lines.



OffsetMesh

Offset your mesh.

Rebuilding2D

Rebuilding2D

Description：

The mesh calculated by Ameba2D is reconstructed into a Trimmed surface.

Input：

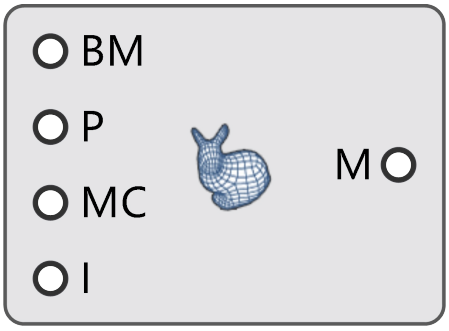
Mesh: The mesh model to be optimized.

Count: The number of subdivision points. The larger the value, the closer it is to the original mesh.

Output：

Surface: Output a Trimmed Surface.

Remeshing3D



Remeshing3D

Description：

Reconstruct the mesh.

Input：

BaseMesh: Enter a mesh. It can be the result of the calculation, or it can be the result of calculating after the smooth.

Plane: Set the direction of the calculated Box.

MeshCount: The number of grids. The larger the value, the larger the value of the generated box, but this value does not represent the total number of grid, just a relative value.

Inflate: Controls the calculated edge, the default is 0.1.

Output：

Mesh: Remeshing mesh.

Right click menu：

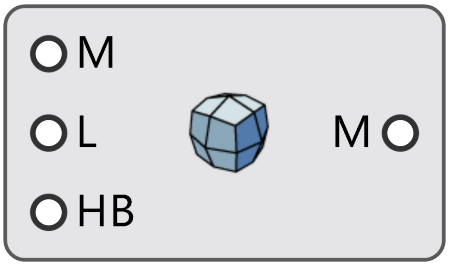
Start: Start the calculation.

Auto: Automatic calculation, but may appear to be stuck.

Attention：

1. This component is usually used with Ameba Smooth to get a smooth mesh model.
2. In the Free version of Ameba, the resulting mesh may produce unsmooth edges (which may cause the model could not be further post-processed), and the number of meshes is limited to 30 or less.
3. The complete Remeshing feature will be available in the Ameba Pro version.
4. The mesh input by this component can be a non-manifold mesh or an unconnected mesh.

AmebaSmooth



AmebaSmooth

Description：

Using the Laplacian Smoothing algorithm to process the mesh model.

Input：

Mesh: Enter a mesh.

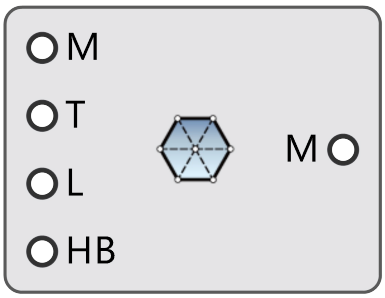
Level: The number of smoothing iterations for mesh.

HoldingBoundaries: Smooth the mesh on the premise of holding mesh's boundaries.

Output：

Mesh: The mesh after the smoothing process

Subdivision



Subdivision

Description：

Subdiving the mesh using Catmull-Clark or Loop algorithms.

Input：

Mesh: Enter a mesh.

Types: Defines how to subdivide the mesh:

- 0: Catmull-Clark Subdivision. This component will subdivide the mesh using Catmull-Clark algorithm.
- 1: Loop Subdivision. This component will subdivide the mesh using Loop algorithm.

Level: The number of subdiving iterations for each mesh faces.

HoldingBoundaries: Smooth the mesh on the premise of holding mesh's boundaries.

Output：

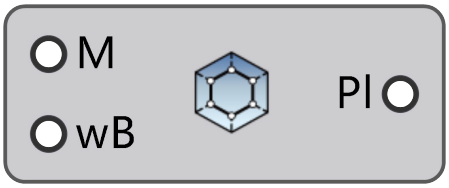
Mesh: The mesh after the subdividing process

Algorithm Details：

Catmull-Clark Subdivision Algorithm Details：https://en.wikipedia.org/wiki/Catmull–Clark_subdivision_surface

Loop Subdivision Algorithm Details：https://en.wikipedia.org/wiki/Loop_subdivision_surface

MeshDual



MeshDual

Description：

This component computes the dual graph of the mesh.

Input：

Mesh: Enter a mesh.

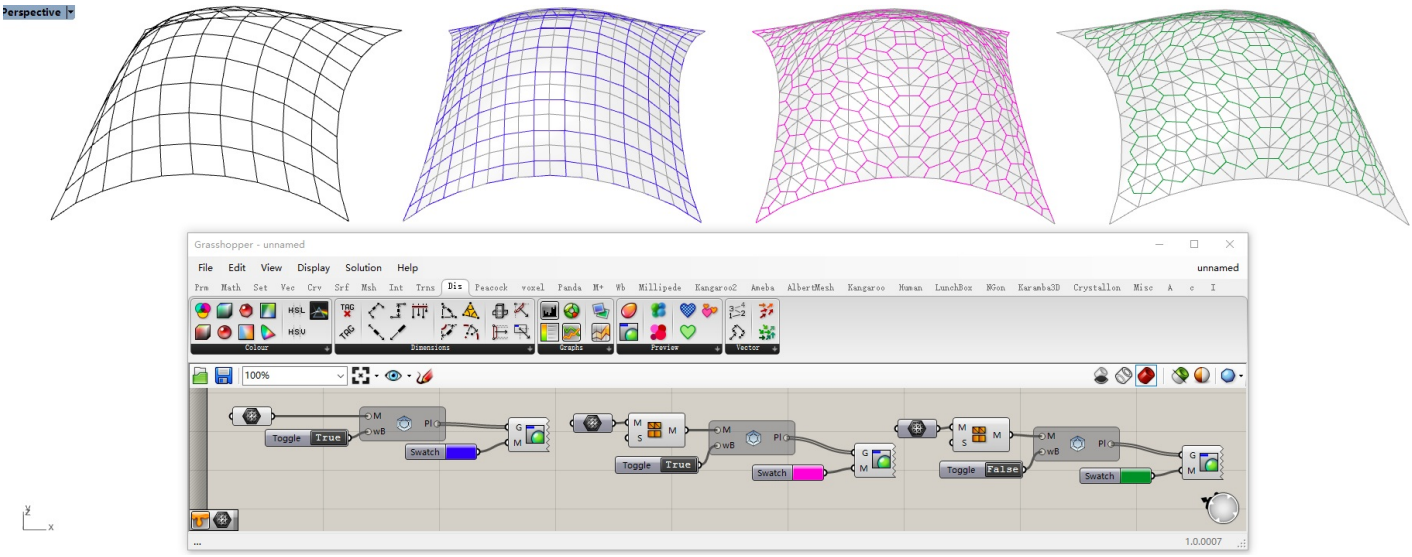
withBoundary: Whether you want to create a dual graph with boundary or not.

Output：

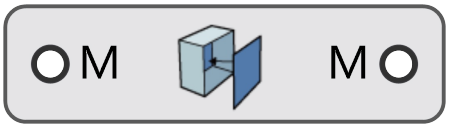
Polyline: Dual polylines.

Algorithm Details：

Sometimes, we use it for designing building's curtain walls. The edge count of a polygon depends on its center point, which is a vertex from your original mesh. For example, if that vertex's valence is 6 (means this vertex is shared by 6 triangles) , you will get a hexagon though MeshDual. So, if you have a uniform triangular mesh (Every vertex valence is close to 6), you will get many hexagon polylines via this algorithm. However, if you input a quadrilateral mesh, you will get a new set of quadrilateral polylines, which are generated by interconnecting the center points of original quadrilateral mesh faces.



FillHoles



FillHoles

Description：

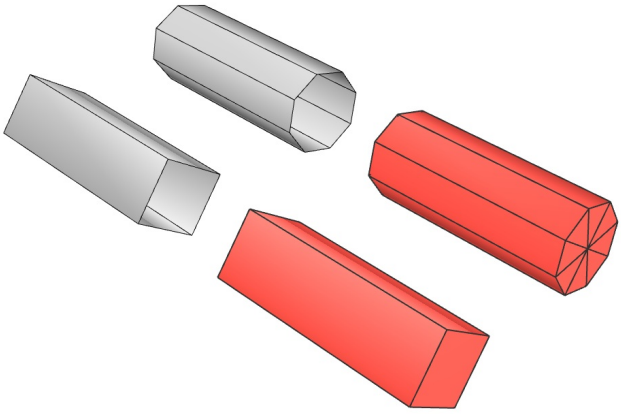
Create simply some mesh faces to fill the holes.

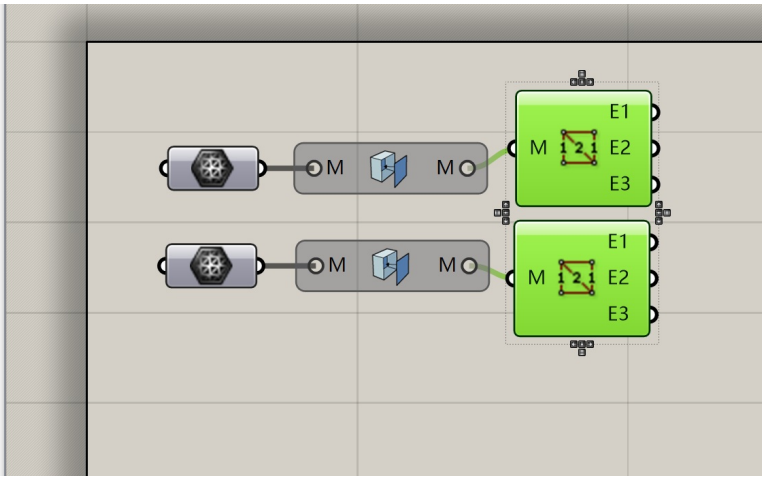
Input：

Mesh: Enter a mesh.

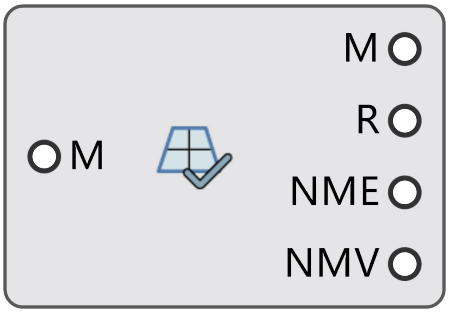
Output：

Mesh: Fill the mesh's holes.





MeshChecker



MeshChecker

Description：

Check and report informations that whether your mesh has non-manifold edges(or non-manifold vertices) or not.

Input：

Mesh: Enter a mesh.

Output：

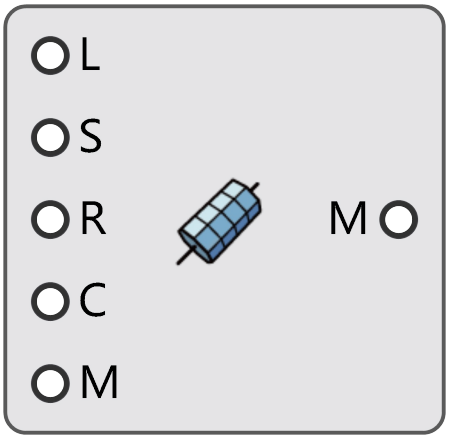
Mesh: the mesh has been welded.

CheckReport: report informations about your mesh.

Non-Manifold Edges: If your mesh has some non-manifold edges, it will output them.

Non-Manifold Vertices Indices: If your mesh has some non-manifold vertices, it will output them.

MeshPipe



MeshPipe

Description：

Create pipe meshes according lines.

Input：

Line: Enter a set of lines.

Sides: The side count of pipe's section.

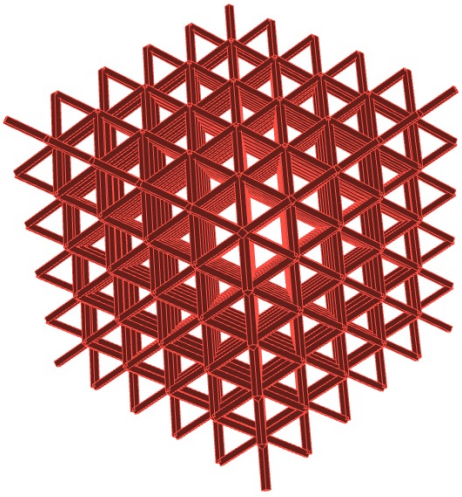
Radius: The radius of pipe's section.

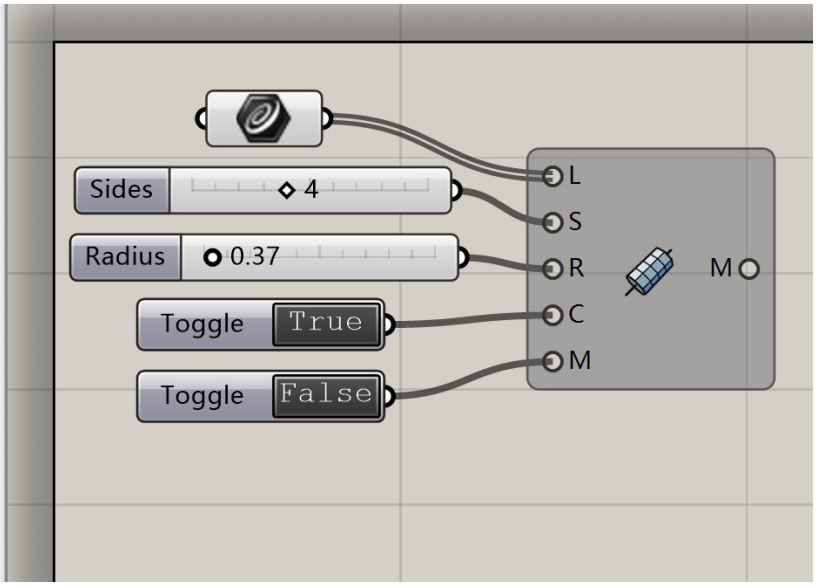
Cap: Whether to generate closed pipes.

Merge: Whether to merge these pipes.

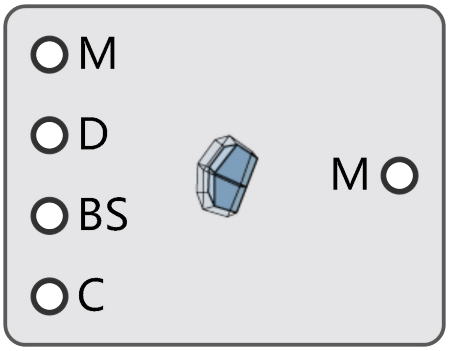
Output：

Mesh: Output a mesh.





OffsetMesh



OffsetMesh

Description：

Offset your mesh.

Input：

Mesh: Enter a mesh.

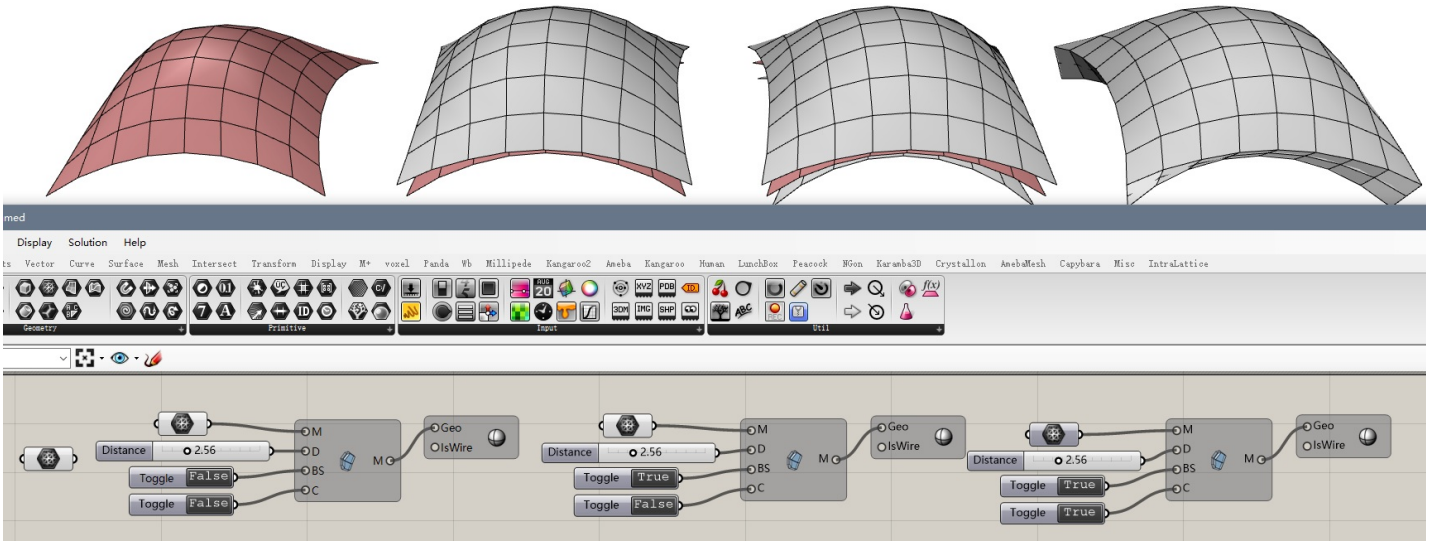
Distance: the distance you want to offset.

IsBothSides: If true, the mesh will offset on both sides.

IsClosed: If true, you can get a soild mesh.

Output：

Mesh: Output a mesh.

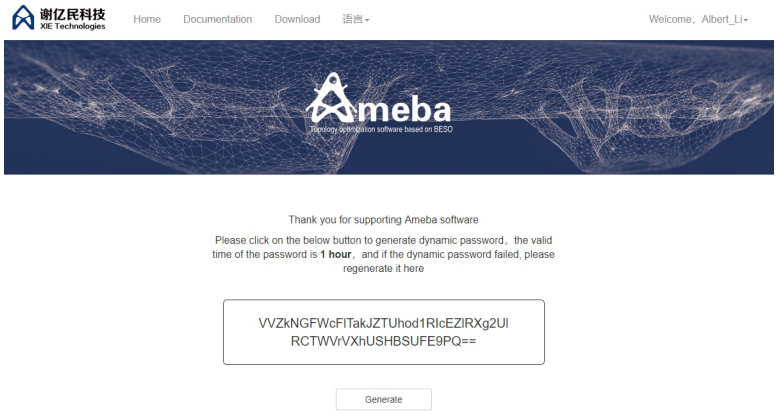


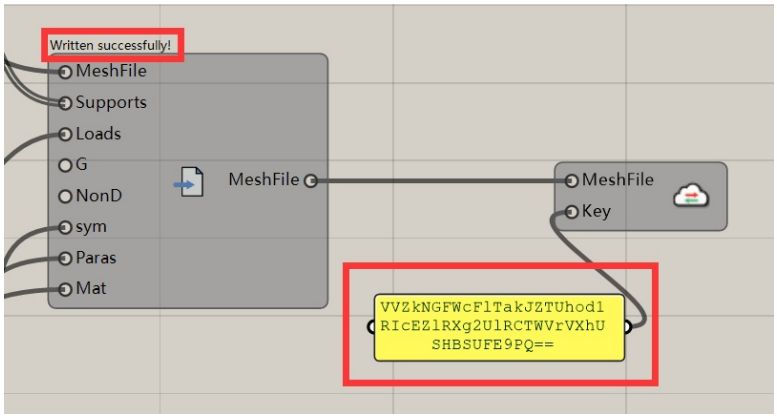
Solver Window

Preparatory Work

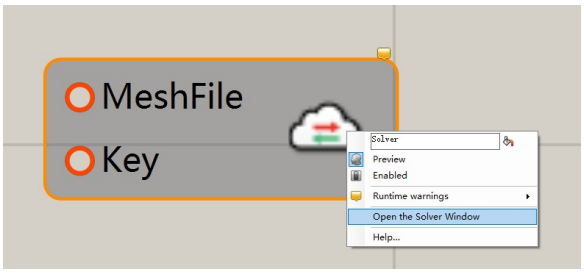
First of all, you are supposed to insure PreProcessing component has revealed "Written successfully!" . Then you need to log in https://ameba.xieym.com/key_en/ to get the key and enter it into a panel component. Finally, connected them with Solve component.

Notes: The key has just an hour of validity. But the key which is out of date can't be shut down the solve process. So you can regenerate it from Ameba's website.

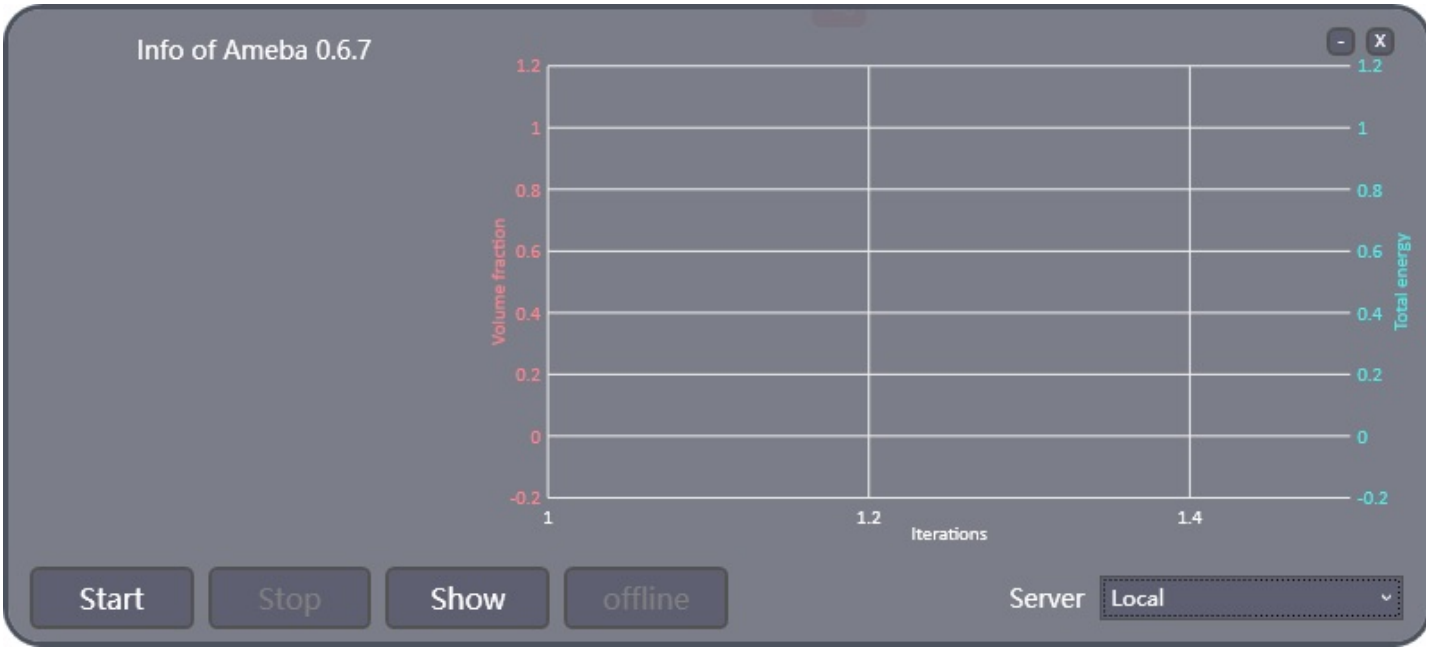




How to start it



The one way is right-click Solve component and select "Open the Solve Window". You can also double-click this component. After operations, Solver Window will be opened.

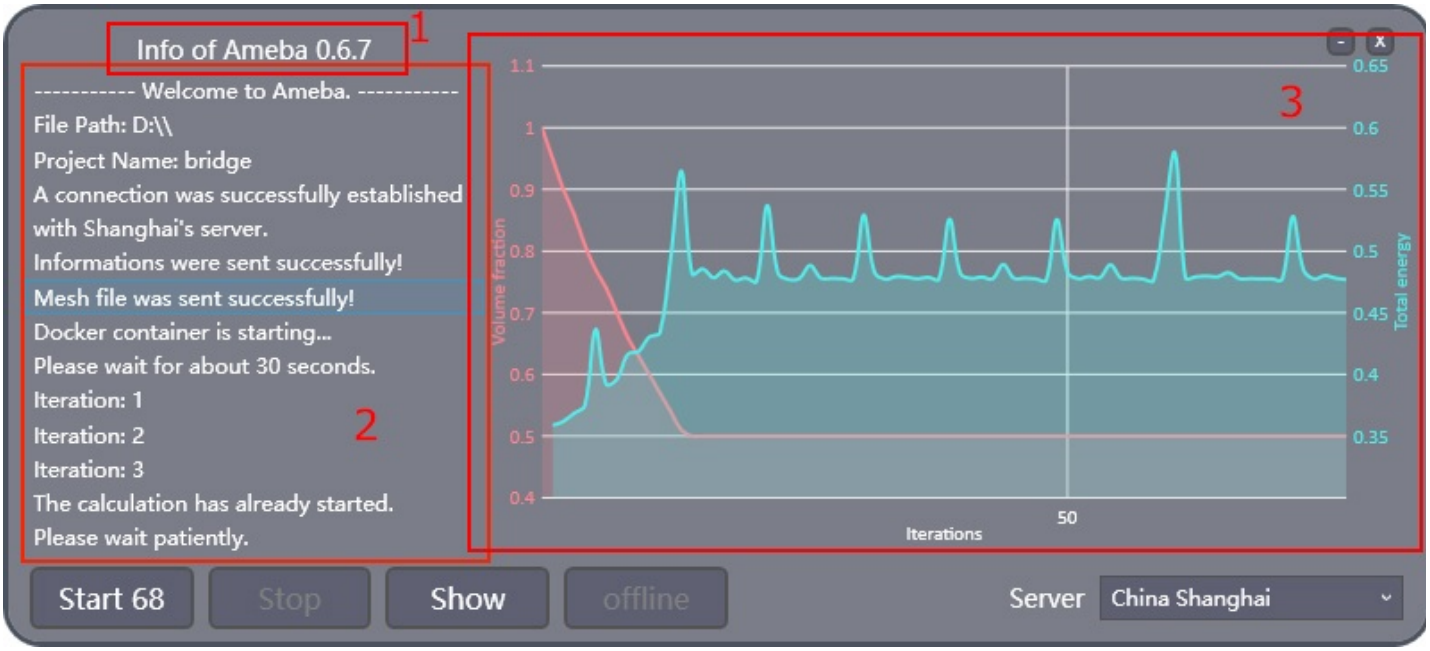


Then, you are supposed to select a server, China Shanghai or US.Virginia, and click Start button. Please wait a moment until the Solver Information Panel displays "The calculation has started. Please wait patiently." Next, you can have a cup of coffee and wait about ten minutes(depends on your mesh model, load case and internet speed). After completion, you can review the result model using Display component.

Window Introduction

The Solve Window

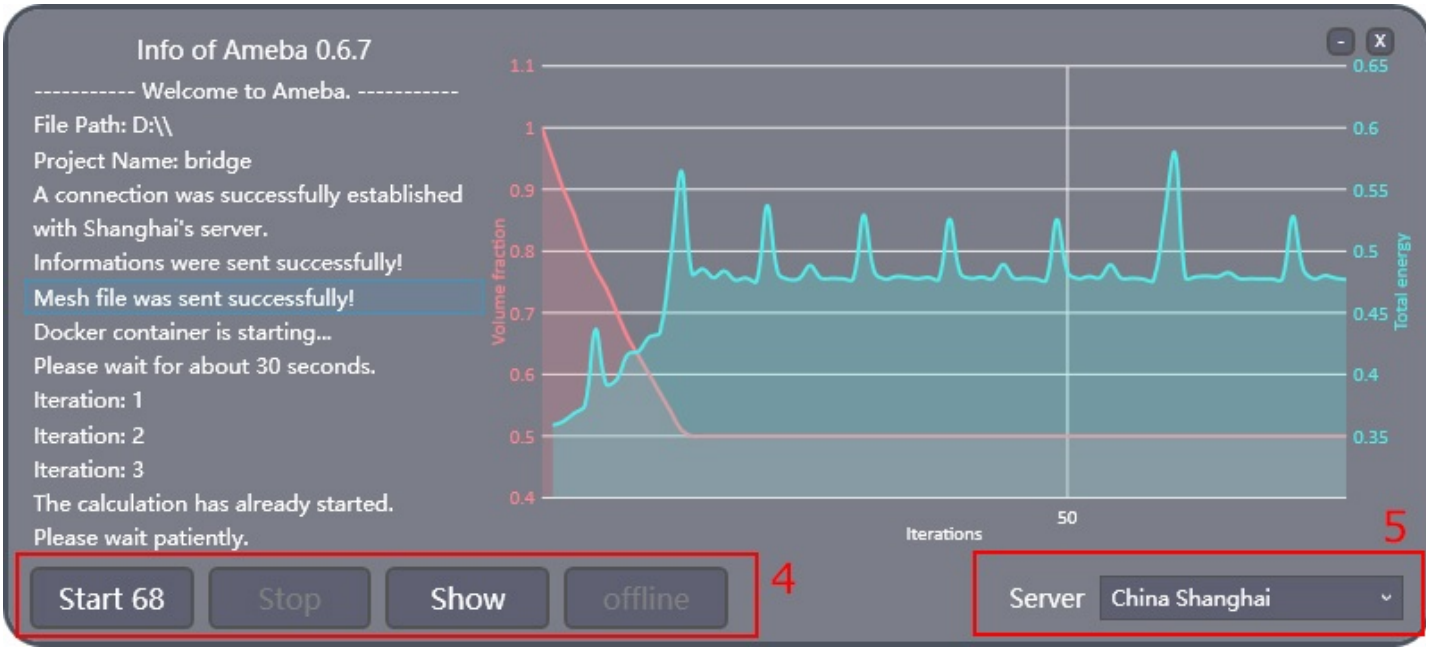
1. Title: The title of the information panel. You can get the version number from it.
2. Solver Information Panel: It will display some informations for the file transfer status, calculation process and error reporting.
3. Broken Line Graph: A broken line graph which is about the Volume fraction,Total energy and Iterations. The Volume fraction will be reduced to your appointed value according to OptParameters component. The Total energy means the gross strain energy. The more stable this parameter is, the easier calculation result converges.



4. Buttons:

- Start: Start to solving. Once the calculation is started, a counter will appear to monitor if the server connection is normal. If the counter stoped for a long time, there is a problem with the server connection.
- Stop: Shut down the solving process.
- Show: If you click this button, it will turn into "Auto" and show solving process model in real-time.
- Offline: Start to calculate offline. Even if your computer has shut down, solving process is still continue. This is a function in Pro Version.

5. Server: Before starting a solving process, you have to choose a server. At present, it just has two servers can be used. Chinese user could select "China Shanghai", and "US.Virginia" Server is recommended for overseas users. The "Local" server is for developers only.



Error Reporting and Solution

- The preparation stage of the solution will take at about 30 seconds (because of Docker's start). This phrase, "The calculation has already started.", maybe appears later than the actual calculation (depends on your network speed). It doesn't affect the calculation result.
- "mesh.xml is larger than 6MB.": The file size of your mesh is limited to 6MB in the Beta Version (The Pro Version will be released soon). If the information panel reminds you about it, you have to reduce the mesh size for decreasing your mesh count.
- "Memory Overflow.": That means your model and conditions are too complex. You need to decrease your mesh count or simplify your case of loads or supports.
- "Please check your case of loads or supports": This phrase illustrates the case of loads or supports has some mistakes such as applying the load directly to the support.
- "Can not connect to the server.": You should check your internet connection, firewalls or antivirus program and so on.
- "Your key has been overdue. Please regenerate a new key!": You are supported to log in https://ameba.xieym.com/key_en/ to get a key again.

Logs

2018.03.31 Ameba 0.6.0 released.

2018.03.31 Ameba 0.6.0 user manual(Chinese version) released

2018.04.16 Ameba 0.6.0 user manual(English version) released

2018.04.23 Ameba 0.6.1 released:

1. Solved recieving interruptions;
2. optimized the speed of recieving;
3. optimized the speed of 【Display】component.

2018.06.22 Ameba 0.6.2 released:

1. Mises stress and principal stress cloud image display function;
2. Add a US server;
3. Add surface Load and Support;
4. The calculation of the component is no longer affect main interface.

2018.07.27 Ameba 0.6.3 released:

1. Add multi-case optimization function;
2. Enable visualize and return the calculated information and results in real time;
3. Optimized the calculation speed.

2018.08.30 Ameba 0.6.4 released:

1. New component display style;
2. Add RenderDisplay, AmebaSmooth,Rebuilding2D and Remeshing3D components;
3. Calculation can support arbitrary load area;
4. Calculation can support any non-designed area;
5. Adopt more resource-saving support and load display methods.