# Easy 2022 Release Notes

November 2021
technet GmbH, Pestalozzistraße 8, 70563 Stuttgart, Germany

technet gmbh

## Contents

# Easy 2022 Release Notes

November 2021
technet GmbH, Pestalozzistraße 8, 70563 Stuttgart, Germany

technet gmbh

## 1 Installing

- Please note that installing Easy Release 2022 will not overwrite an older (2021, 2020, ...) existing installation of Easy. You can use Easy 2022 and an older release of Easy on the same computer.

- Easy Release 2022 requires a new password!

## 2 System Requirements

Operating systems:

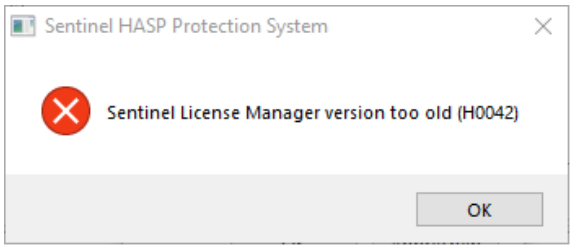
- Windows 11 (current service pack) 64-bit
- Windows 10 (current service pack) 64-bit
- Windows 8.1 (current service pack) 64-bit
- Windows 8 (current service pack) 64-bit
- Windows 7 (current service pack) 64-bit

Minimum Hardware:

- A graphic card with an OpenGL accelerator is strongly recommended.

## 3 Licensing

Easy2022 requires the latest drivers for the protection plug. If you receive the following error message when starting the program, you must install the provided Sentinel driver on your computer and, if necessary, on a license server.



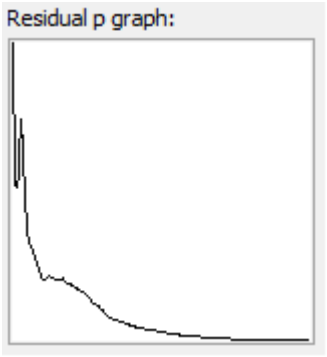

## 4 EasyShell

Settings dialog for program Repgen.

# 5 Digital wind tunnel - EasyDWT

- Log file EasyDwt.log in Easy project folder

- Residual graph for residual p

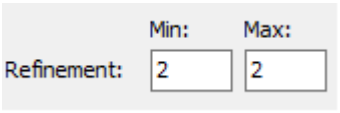



- Option: first order discretization scheme for U (more stable but less accurate calculation)

- Resume the solver: Continue the calculation with changed parameters

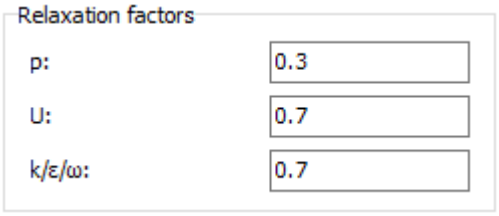
- Run-out length factor: Influences the size of the area in lee side of the buildings

- Min/max refinement: Reduces cells count on surfaces

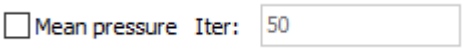



- Larger transition area between refinement steps.

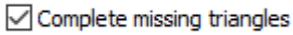
- Relaxation factors (can be lowered if residuals break out)




- Mean pressure: Create mean results from non-stable simulations



- Completion of the missing triangles is optional

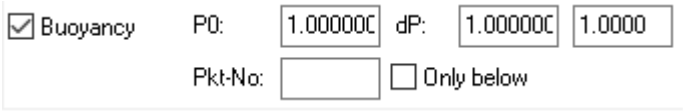
- Matching tolerance between OpenFoam an Easy mesh can be set

- Better integration in EasyShell: Modifies LOAD.INP and LOAD.STA

# 6 BeamEditor

New load type: buoyancy

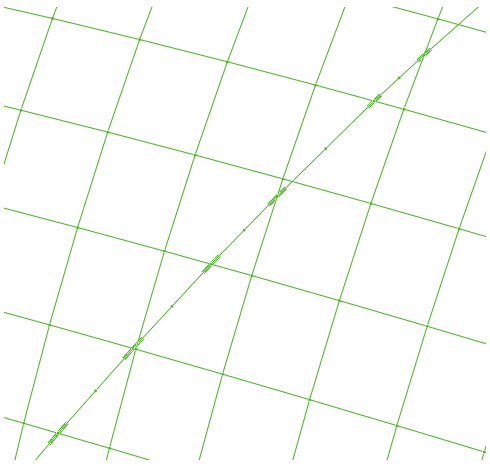



Considers the buoyancy load depending on the height.

# 7 Program CabVol

The CabVol programme is used to connect a cable or an entire cable net with a membrane. The cable net is usually only connected to the membrane with lugs, which means that all elements of a cable receive the same force in every load case. In the case of a cable net, this naturally applies to the cable elements between the nodes.

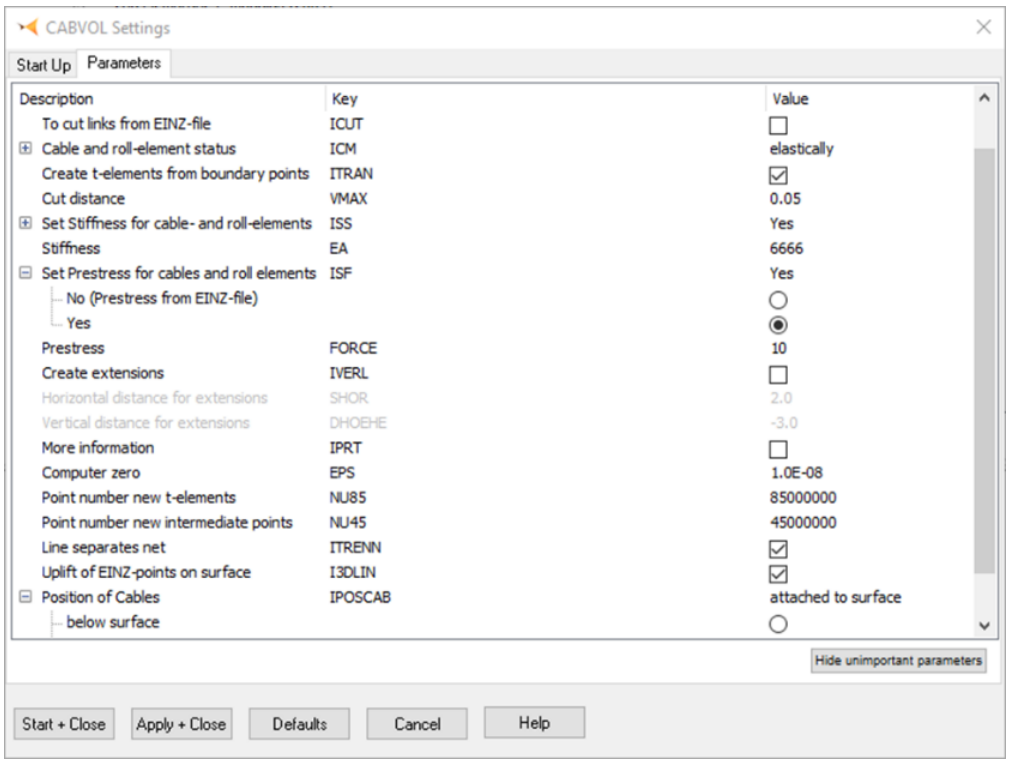
The same cable forces are realised with the help of a rol-file; in the rol-file, a cable element with 3 points and 2 cable elements always ensures the same forces in the two sections of cable and thus, in the case of continuous rol-elements, the same force results in the entire cable.



*Membrane surface with a cable and rol-elements*

New input parameters in CabVol:

| Parameter | Setting | Description |
|---|---|---|
| **ISS** | yes | All cable and rol-elements are given the stiffness EA. |
| **ISS** | no | All cable and rol- elements retain their stiffness from the EINZ-files. |
| **ISF** | yes | All cable and rol-elements get the prestress FORCE. |
| **ISF** | no | All cable and rol- elements retain their prestress from the EINZ-files. |
| **NU85** | | Point numbers for the new T-elements. |
| **NU45** | | Point numbers for the new intermediate points |
| **ITRENN** | yes | The membrane surface is made 'deformable' (in a sense, separated) at the ropes to obtain realistic deformations. The presence of an angle file is important, because the angular sizes are also affected by the 'separation'. |
| **ITRENN** | no | Without separation. |
| **I3DLIN** | yes | The plane points of the EINZ-file are placed perpendicular to the surface. |
| **I3DLIN** | no | If the points of the rope net are already on the surface it is not necessary. |
| **IPOSCAB** | Attached to surface | This method is automatically applied to the programmes Statik, StatikD, Volstan and VolstaD. Only in the Beam and Beamd programmes does this parameter have an effect. Be careful when using it: the cables can also fall down if they are placed on the wrong side of the membrane. |

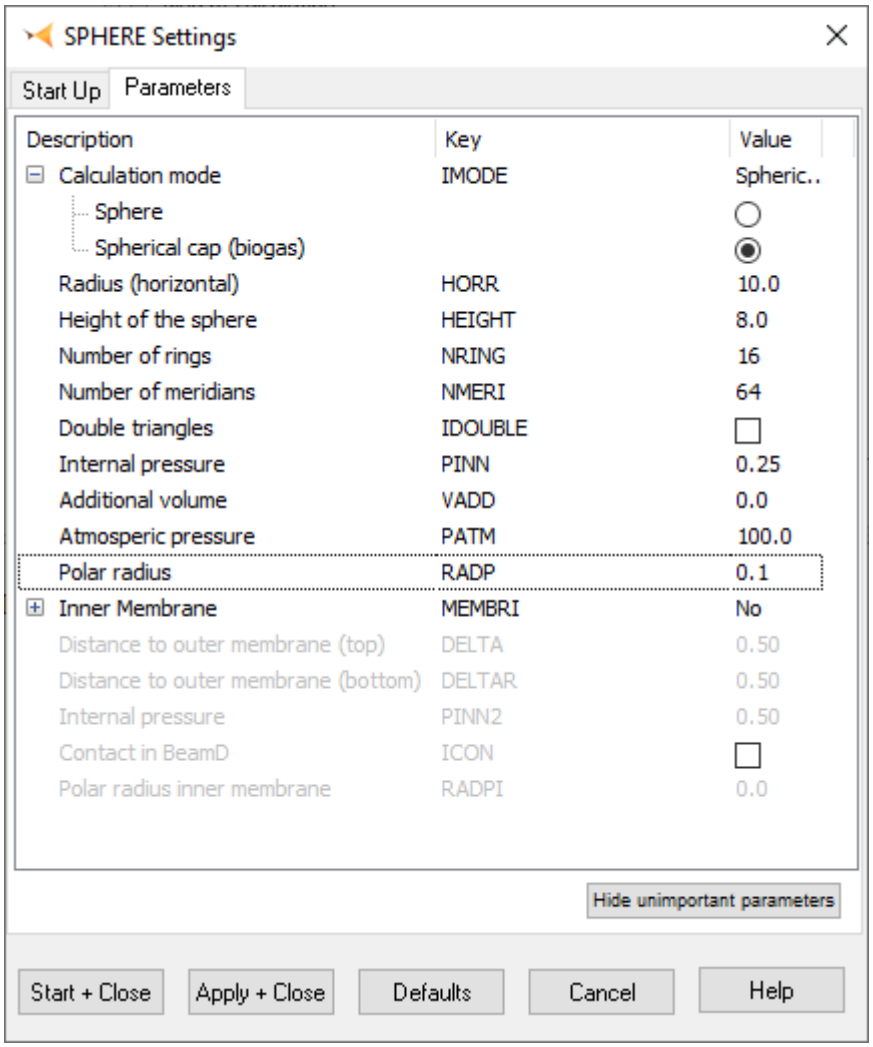


# 8 Program Sphere

The Sphere program generates spheres or spherical calottes purely parametrically. It is used especially for the generation of biogas domes.

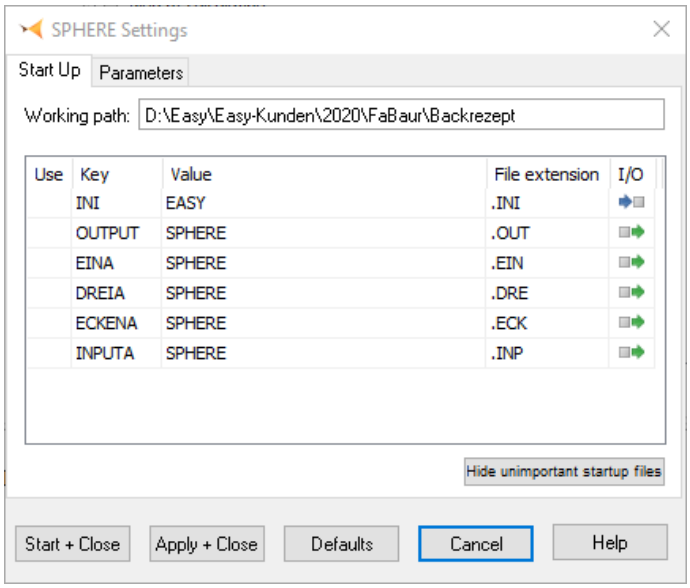
The Sphere program has the following input parameter:

| Parameter | Description |
|---|---|
| **IMODE** | Calculation mode (full sphere or spherical calotte). |
| **HORR** | Radius of the spherical calotte (generally this is not the radius of the sphere). |
| **HEIGHT** | Height of the dome. |
| **NRING** | Number of latitude circles. |
| **NMERI** | Number of longitudinal circles (meridians). |
| **IDOUBLE** | Single or double triangles. |
| **PINN** | Operating internal pressure. |
| **VADD** | Additional volume. |
| **PATM** | Atmospheric pressure. |
| **RADP** | Radius of the polar ring. |
| **MEMBRI** | With or without inner membrane. |

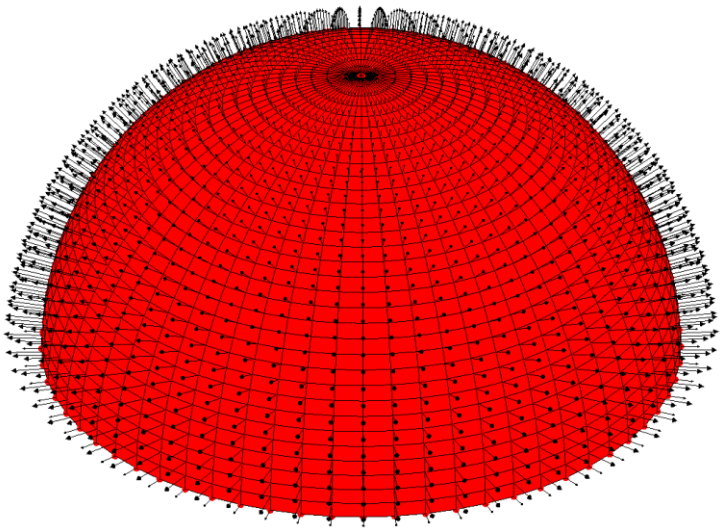



The Sphere program writes the data files, as can be seen below:

| File name | Description |
|---|---|
| **Sphere.ein** | Structure |
| **Sphere.dre** | Corresponding triangles. |
| **Sphere.eck** | corresponding polylines. |
| **Sphere.inp** | Input for the static calculation (internal pressure, volume, p*V). |
| **Sphere.las** | Internal pressure loads. |




Below you can see the result of a sphere calculation in the graphical editor GED. Above it was said that this task is to be assigned to the first method. This can now be easily checked by calculating a form finding with the external internal pressure loads. The result of this calculation corresponds exactly to the geometry Sphere.ein.



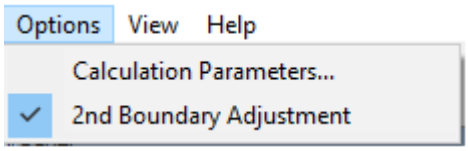
*Biogas dome with internal pressure loads*

Thus, the following control can be calculated. Start Fofin with Sphere.ein and Sphere.las. Fofin.ein and Sphere.ein are identical.

# 9 EasyTailor

- Main menu changed: “Calculation Parameter…” is now in “Options”

- „Second Boundary Adjustment“: Repeats the boundary adjustment after the compensation.
  In the case of large compensation and rotated strips, this option can be useful to obtain equal boundary lengths.

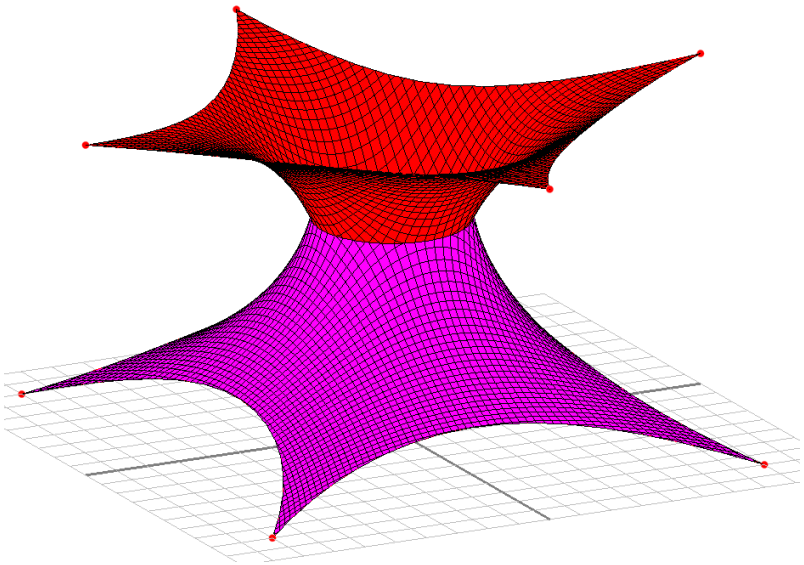



# 10 GED

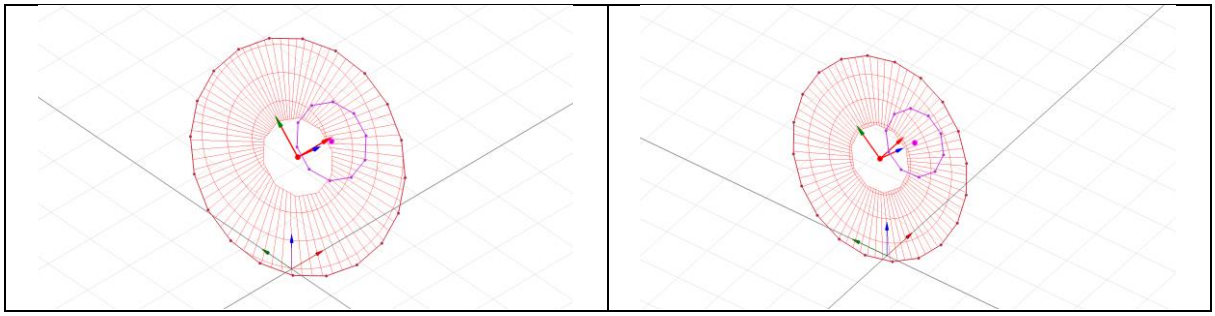
New menu item „File/Add/Angle File…”

# 11 FormEdit

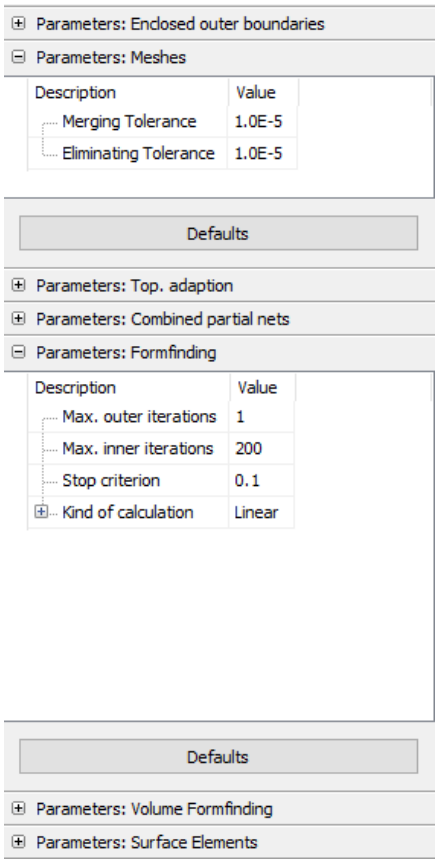
- Associated Boundaries
  A boundary can be assigned more than one time in the new version. This means that structures like the one shown in the picture are now possible. The middle ring is associated with the upper and lower boundary and is in force equilibrium.



- Mesh generation with associated boundaries
  In the previous version, the radial mesh was calculated incorrectly in the case of decentered centers of gravity. This error has been corrected (wrong on the left, correct on the right).



- Calculation Parameters
  The parameters for the calculation programs have been extended or cleaned up. In addition, a calculation is now started after changing a parameter depending on the setting "Edit/Automatic update".



# 12 Brep2Easy

Improvements in cut calculation (mesh – boundary line).