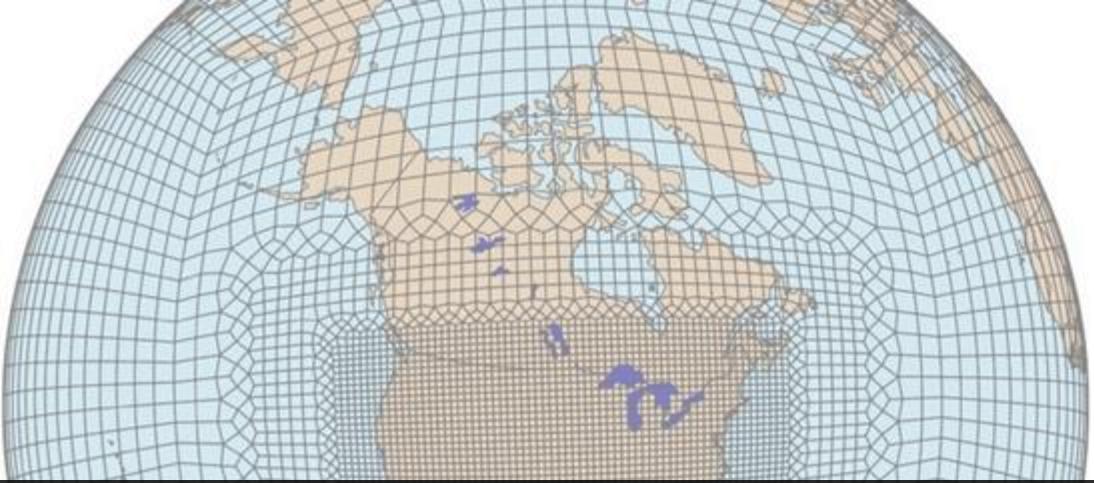
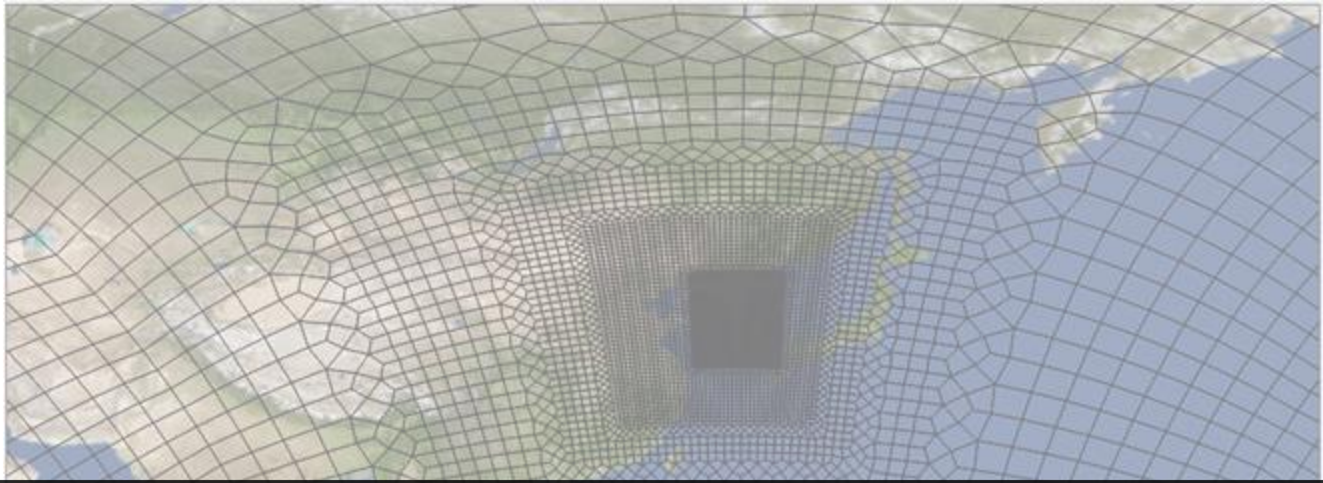




Creating and Using Variable Resolution Grids in CESM

MUSICA Tutorial at Nanjing University

Contributors:
Patrick Callaghan, Adam Herrington, Paul Ulrich, Colin Zarzycki, The CSEG team,
Julio Bacmeister, Peter Lauritzen, Duseong Jo, Louisa Emmons

September 2024

The continuing development of VR tools is supported by NSF CAM7 funding
US NSF Award ID 1755088

This material is based upon work supported by the National Center for Atmospheric Research, which is a major facility sponsored by the National Science Foundation under Cooperative Agreement No. 1852977.

VRM Tools

Repository:
https://github.com/ESMCI/Community_Mesh_Generation_Toolkit

Documents:
https://github.com/ESMCI/Community_Mesh_Generation_Toolkit/tree/master/VRM_tools/Docs
(but some parts out of date)

Wiki page with current instructions:
https://wiki.ucar.edu/display/MUSICA/Custom+Grid+in+CESM3

Outline

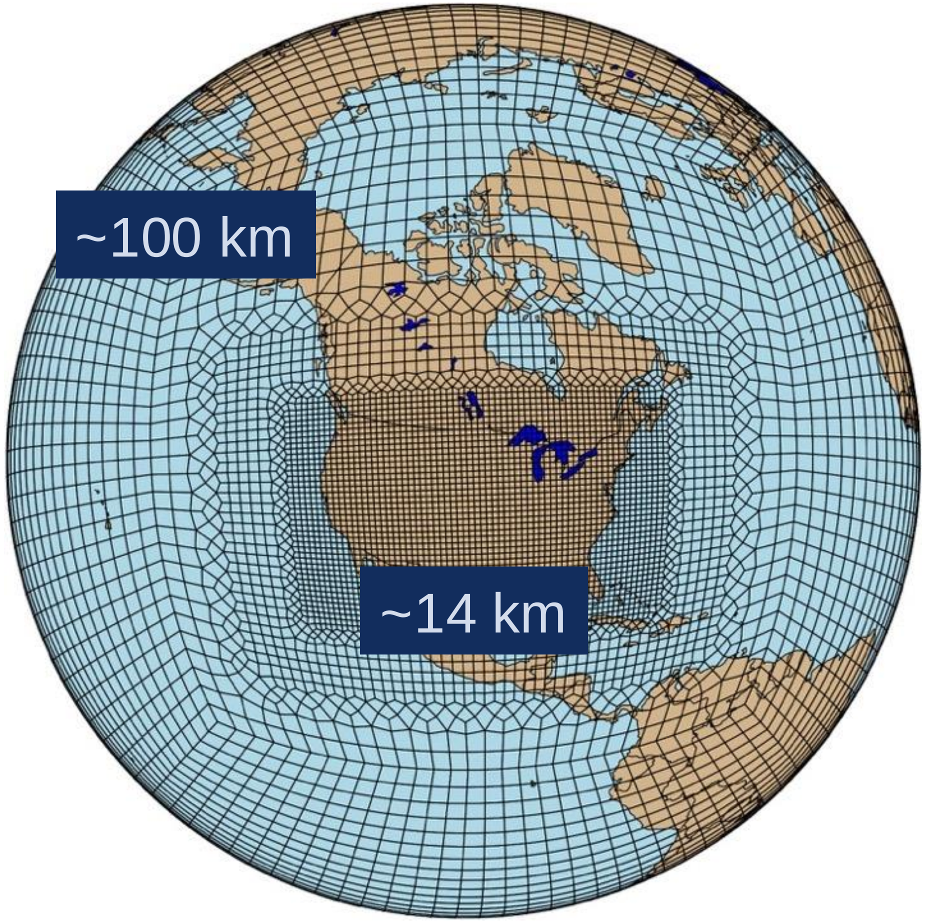
Overview of Creating a Grid

- Existing grids
- Summary of tools
- Recommendations and tips for a successful grid
- Steps needed to use new grid in MUSICAv0

Detailed instructions

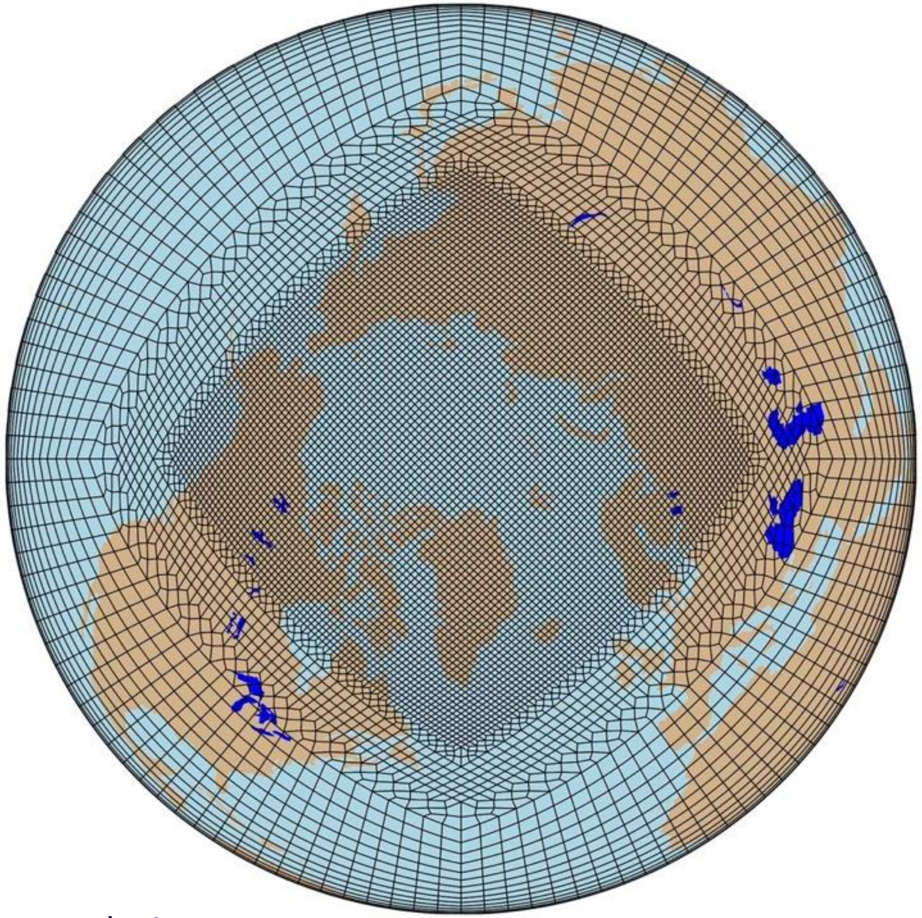
Available in CESM (v2.2 and later)

CONUS



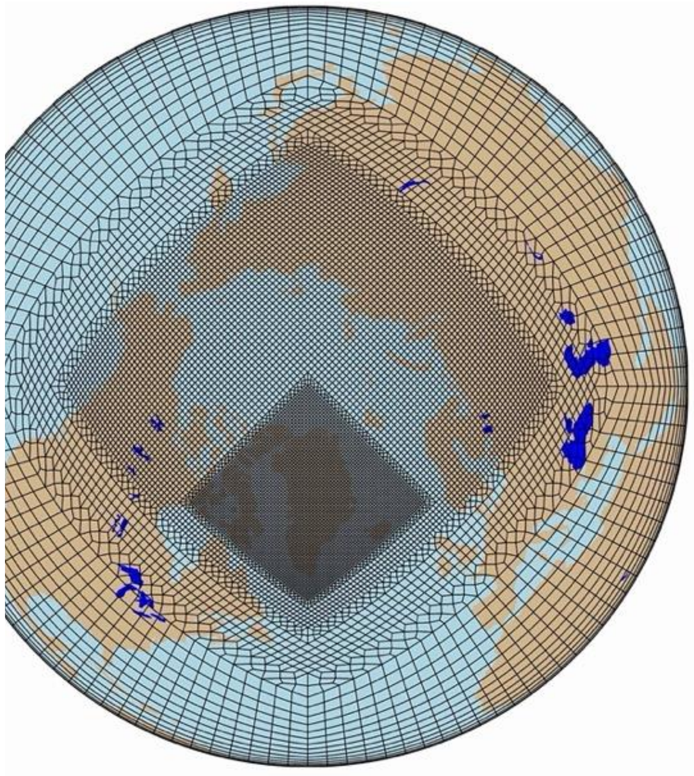
Resolution:
ne0CONUSne30x8_ne0CONUSne30x8_mt12

ARCTIC



Resolution:
ne0ARCTICne30x4_ne0ARCTICne30x4_mt12

ARCTICGRIS
(Greenland Ice Sheet)



Resolution:
ne0ARCTICGRISne30x8_ne0ARCTICGRISne30x8_mt12

NCAR
UCAR

Refined Grids Available for Many Regions

https://wiki.ucar.edu/display/MUSICA/Available+Grids

Pages / MUSICA Home

Available Grids

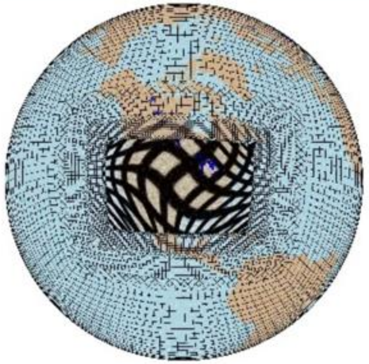
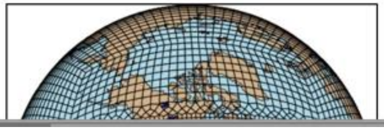
Created by Louisa Emmons, last modified on Nov 16, 2023

A number of grids have been created by various users for use in MUSICAv0 which we list here to demonstrate the diverse capability of MUSICAv0. The CONUS ne30x8 and ARCTIC grids are available resolutions in CESM2.2, but the other grids have been developed for various science applications which have not yet been published. In the future we plan to have a public repository of grids, or may provide some grids in future model results.

Protocol: Please contact the developer of the grid if you are interested it in using it and include them as co-author of any work using that grid.

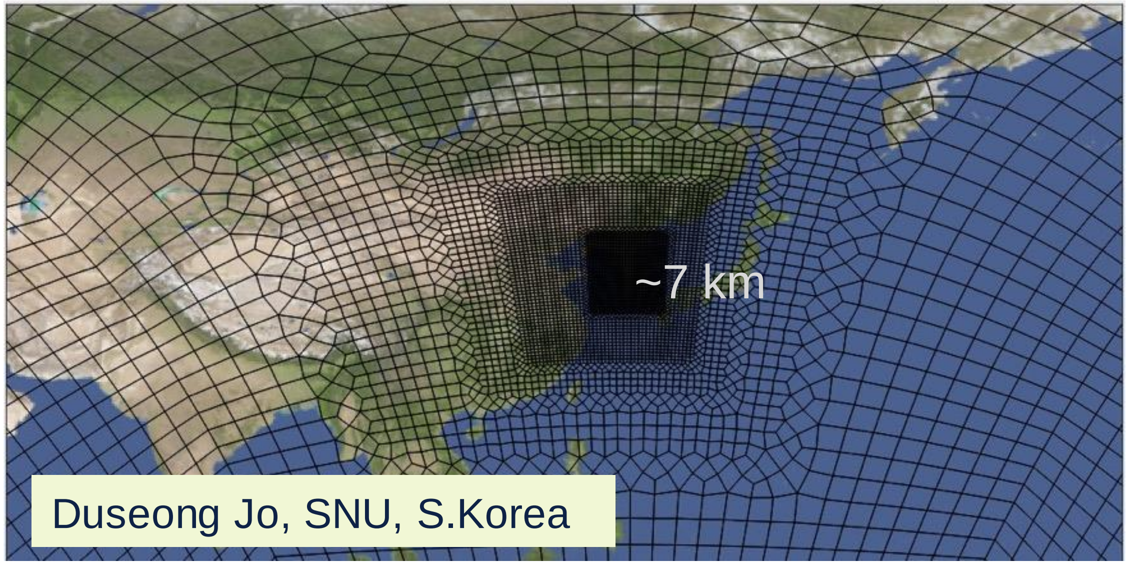
Emissions may be available for sharing for these grids - contact the grid developers. We will develop a public repository for those as well.

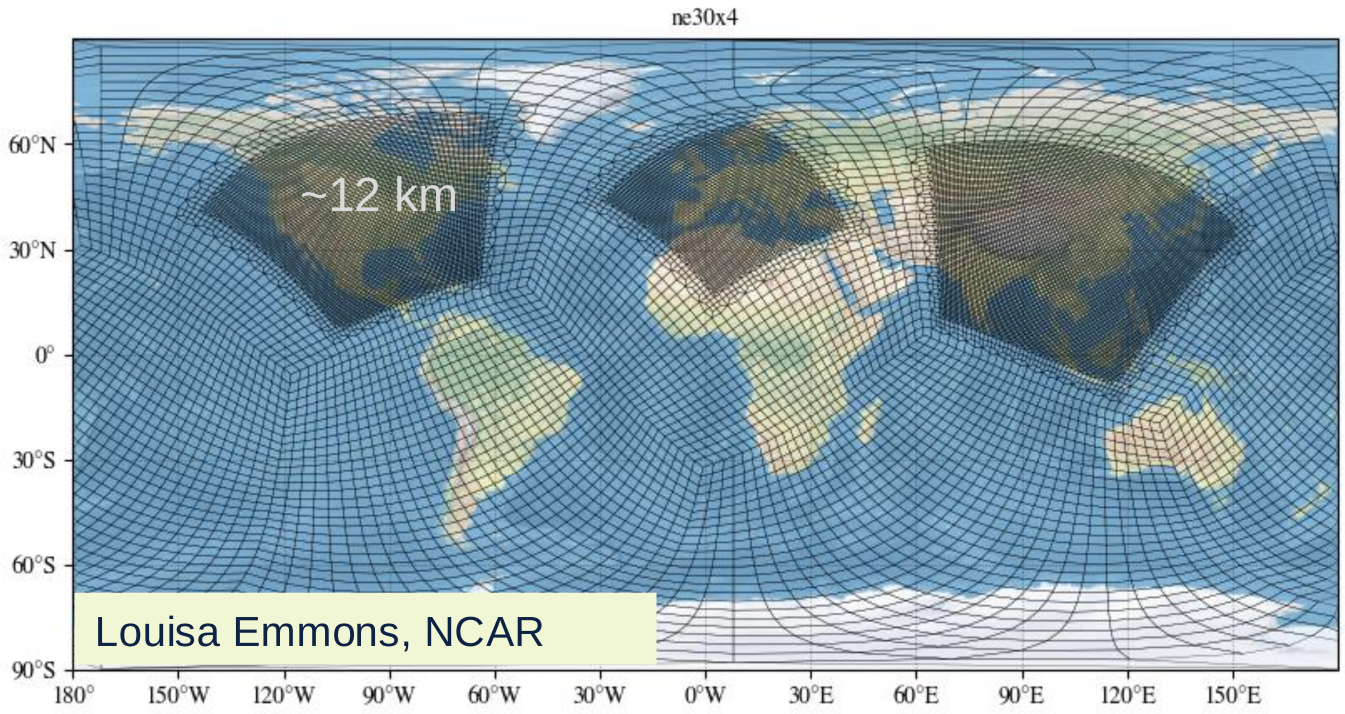
The grid resolutions (e.g., ne30x{N}) are defined at the bottom of this page.

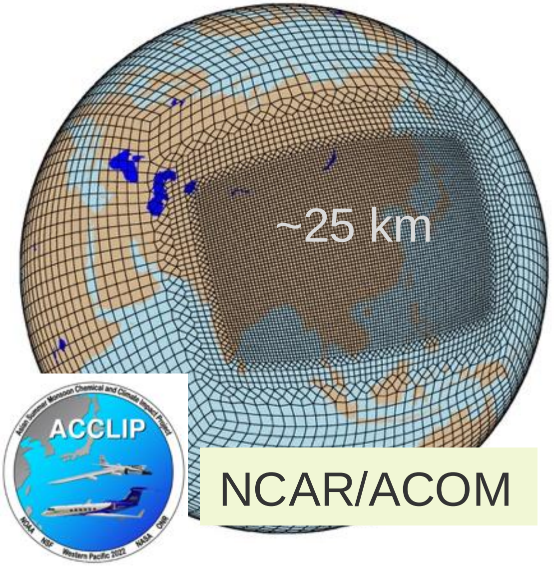
Refined region	Resolution, Repository, Contact	Image (click for full size)
CONUS 1/8 degree (14 km)	Resolution: ne0CONUSne30x8_ne0CONUSne30x8_mt12 Repository: part of CESM2.2 (CAM User's Guide) Output from community simulation - DOI: https://doi.org/10.5065/tgbj-yv18 Publications: • Schwantes, R. et al., JAMES, in press. • Tang, Wenfu, et al., JGR-Atmospheres, in review.	
CONUS 1/4 degree (28 km)	Resolution: ne0np4.CONUS.ne30x4_mt12 Repository: /glade/campaign/acom/acom-weather/MUSICA/musica_repo/ne0np4.CONUS.ne30x4 Contact: Louisa Emmons, NCAR/ACOM	

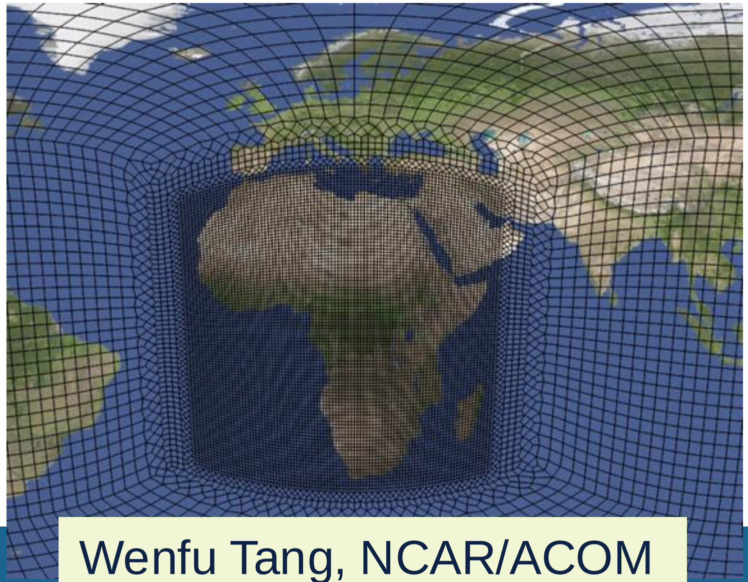
Screenshot

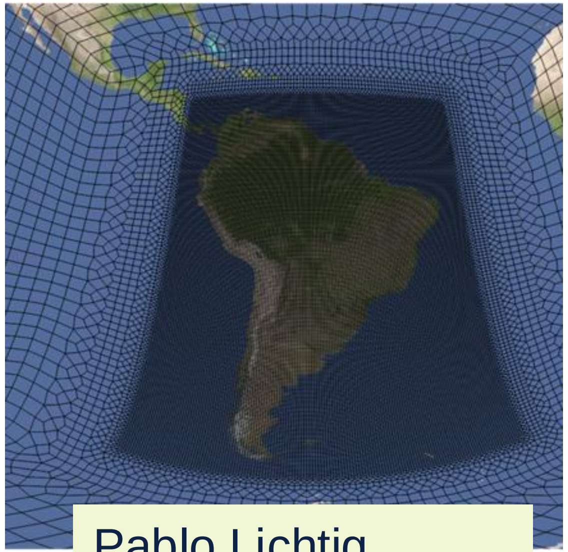
Custom Grids











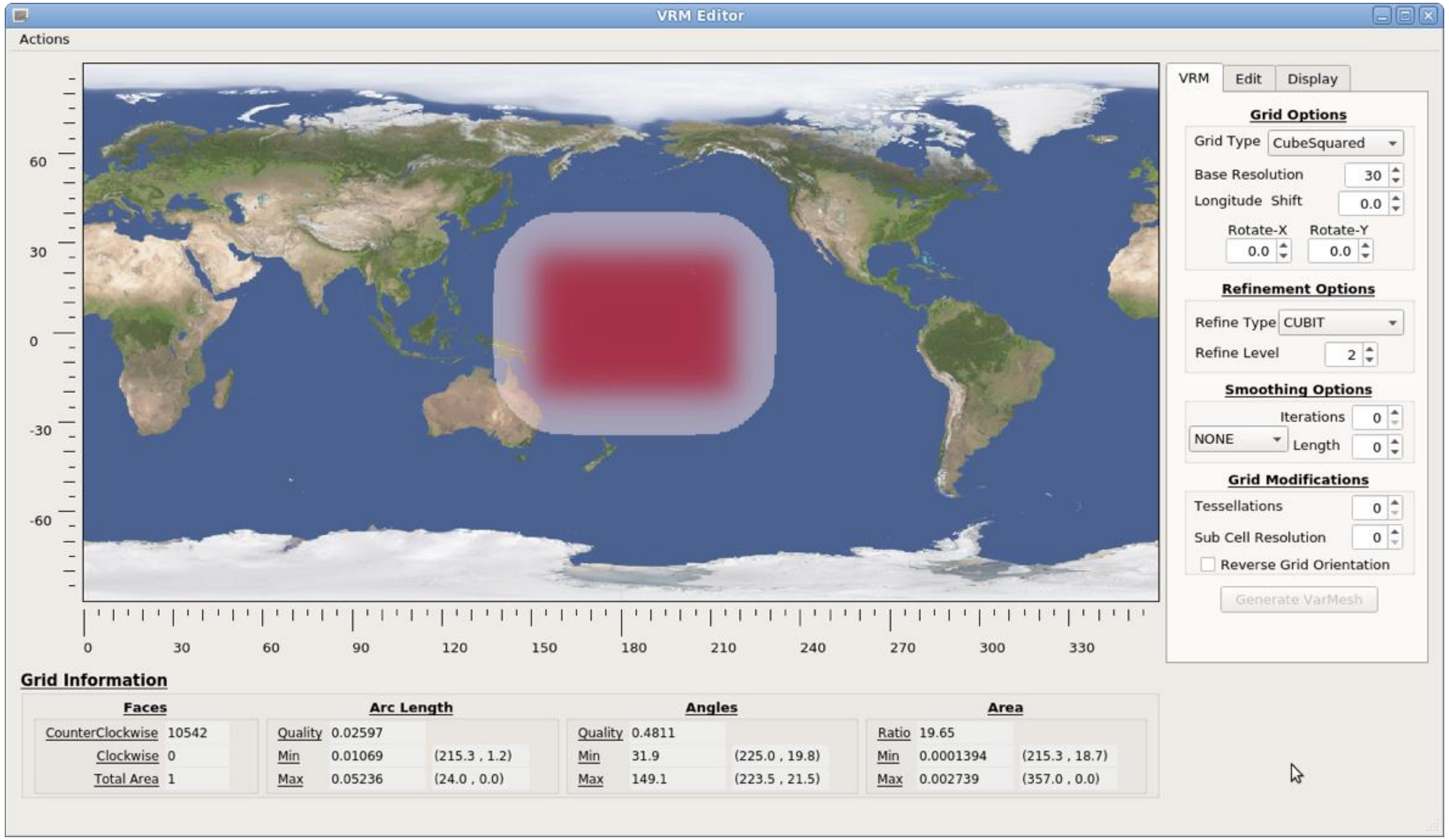
Steps to create a new grid and use in MUSICAv0

- Download source code and build for VRM_Editor, etc.
- Create a new grid
- Create a repository of mesh files and initial value datasets
- Install new grid in CESM
- Establish defaults for stable model runs
- Run CAM to spin-up surface state
- Regrid emissions data for the new grid (mass-conserving)
- Regrid MERRA/GEOS meteorology data to the new grid (if nudging)
- Run MUSICAv0

Creating a New Grid with VRM Editor

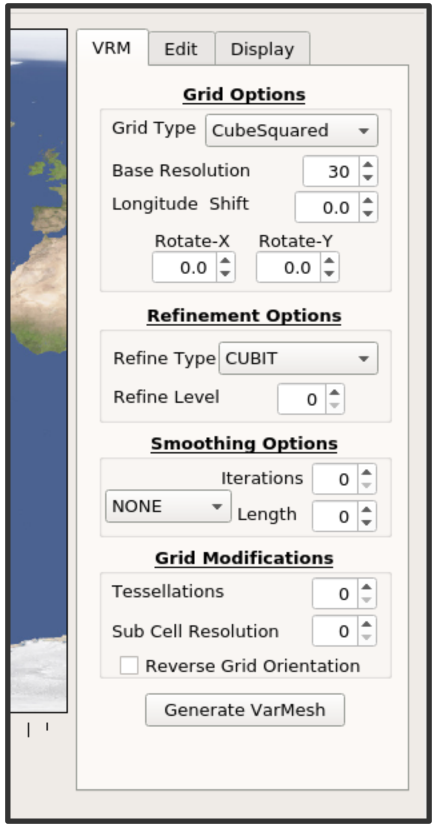
The approach is to edit a map of continuous refinement values [0.,1.] on a (Lat,Lon) grid directly. Values = 1 define the desired region of refinement, shown in red.

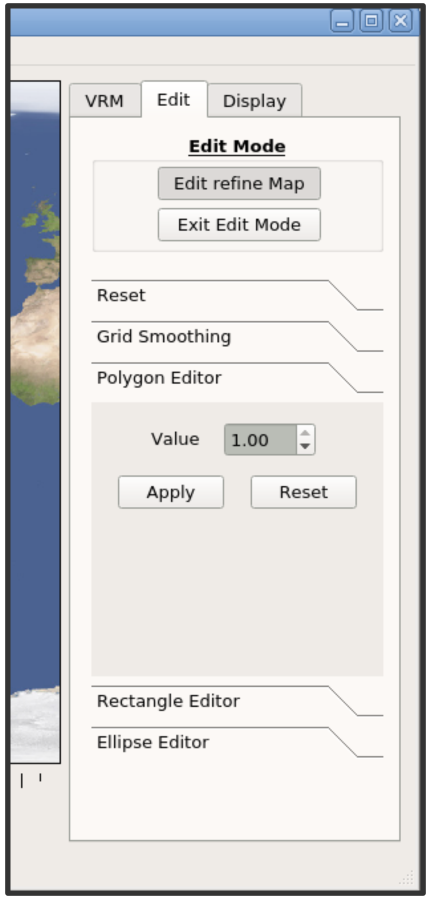
- These values are stored/read from a netCDF file with a defined (Lat,Lon) grid.
- The Actions menu at the upper left has options for reading/writing refinement maps and newly created grids.
- When a grid is created, the table at the bottom of the interface provides quality information about the grid. Including the min/max area, angle, arc length, and the locations of those extreme values.
- On the right is a set of tools for editing and creating the grid.

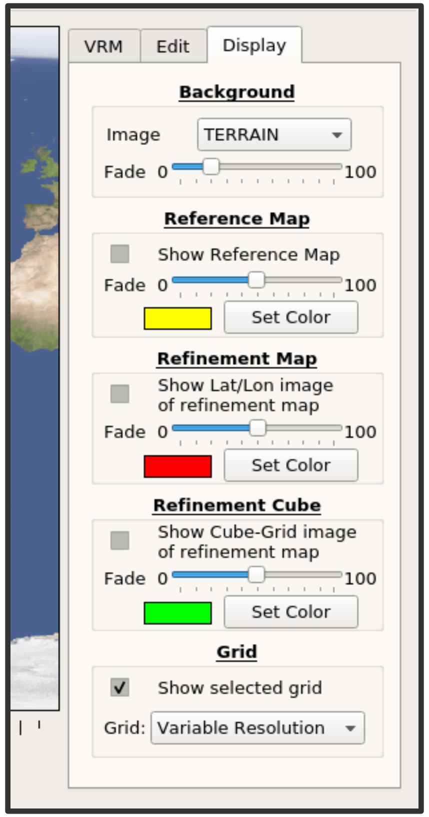


VRM Editor menus

- The editor has tools used to create and modify the refinement map.
- There is also set of options control the appearance of the displayed values in the editor.



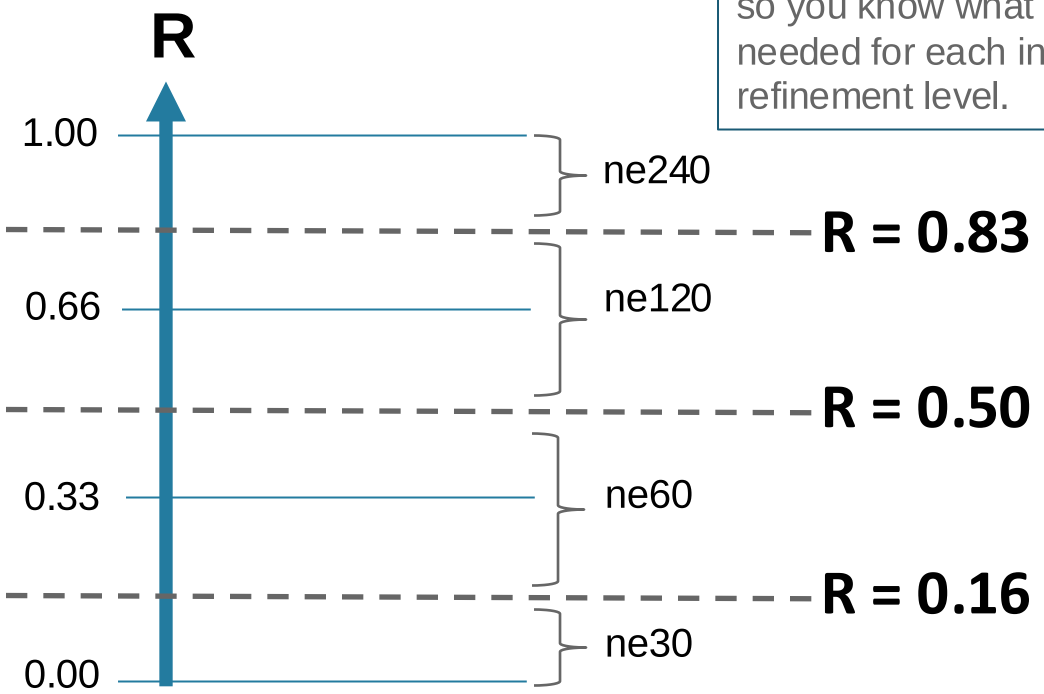




VRM Editor Refinement Levels

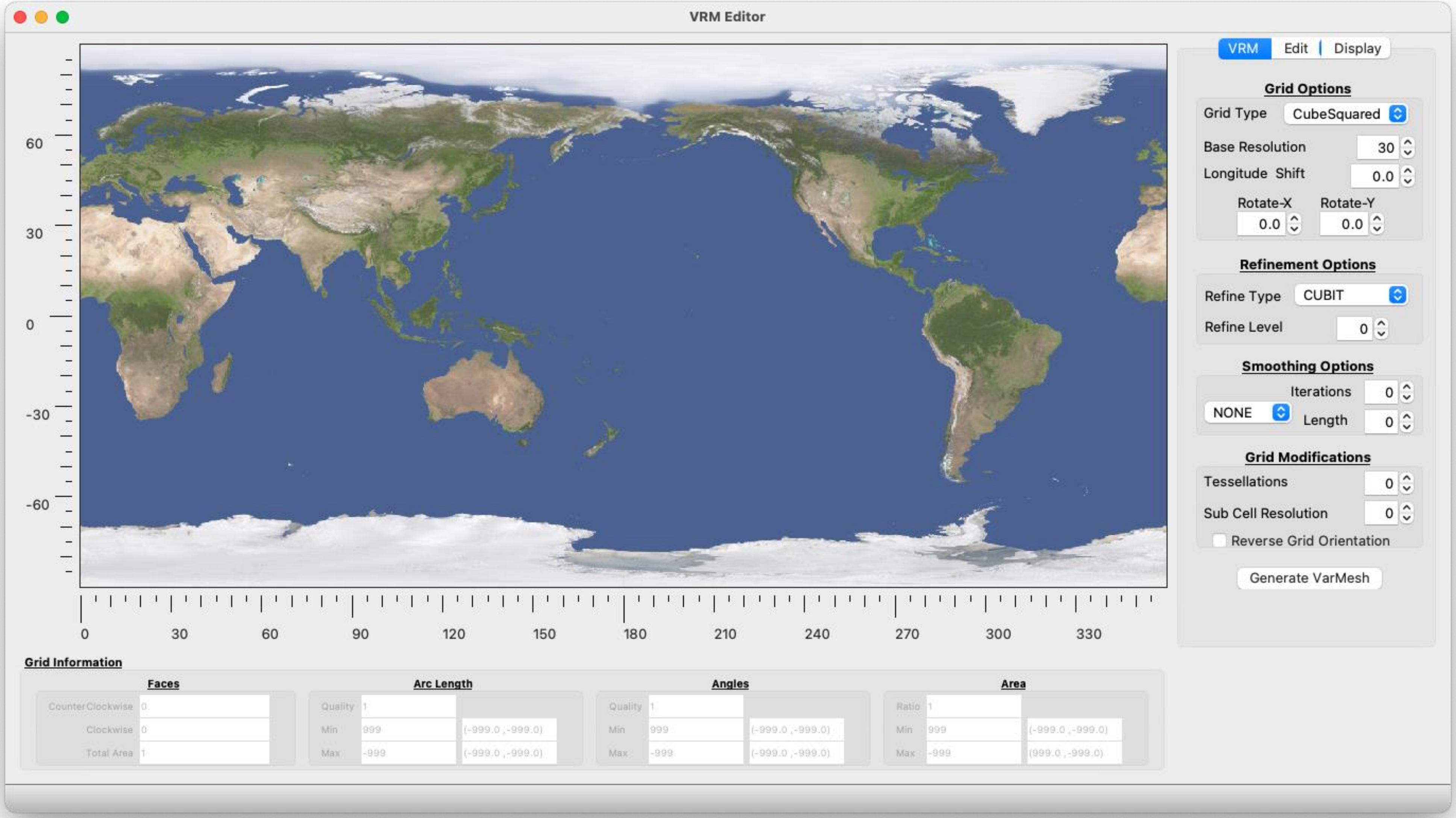
Consider a ne30 grid with a ne240 refinement.

PLAN AHEAD:
Work out these numbers for the refinement level you want so you know what values are needed for each intermediate refinement level.



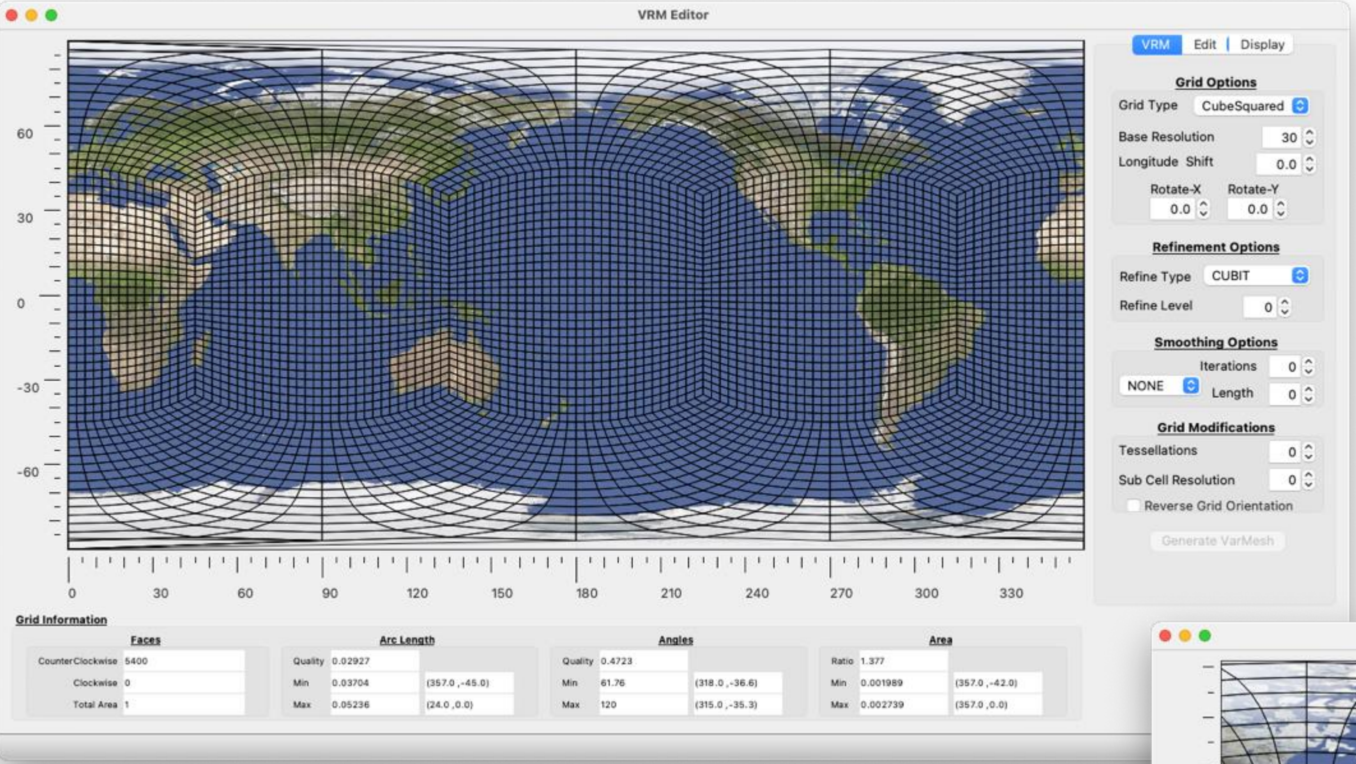
- The range of refinement values is discretized and assigned to refinement levels as shown.
- For ne240 the the refinement level is 3, so the interval in R values is (1/3). Setting values equal to 1 defines the highest resolution region.
- Nominally, to specify a ne120 transition region around the ne240 domain, a halo of R values equal to 0.66 is added. Similarly, subsequent halos can be added using the appropriate value of R
- For each element the average R value is discretized to the corresponding refinement level and this integer value is assigned to the element.
- Note: To implement transitions between resolutions, SQuadGen automatically assigns refinement level values around the perimeter of a high resolution region to the next lowest resolution.

Creating a New Grid

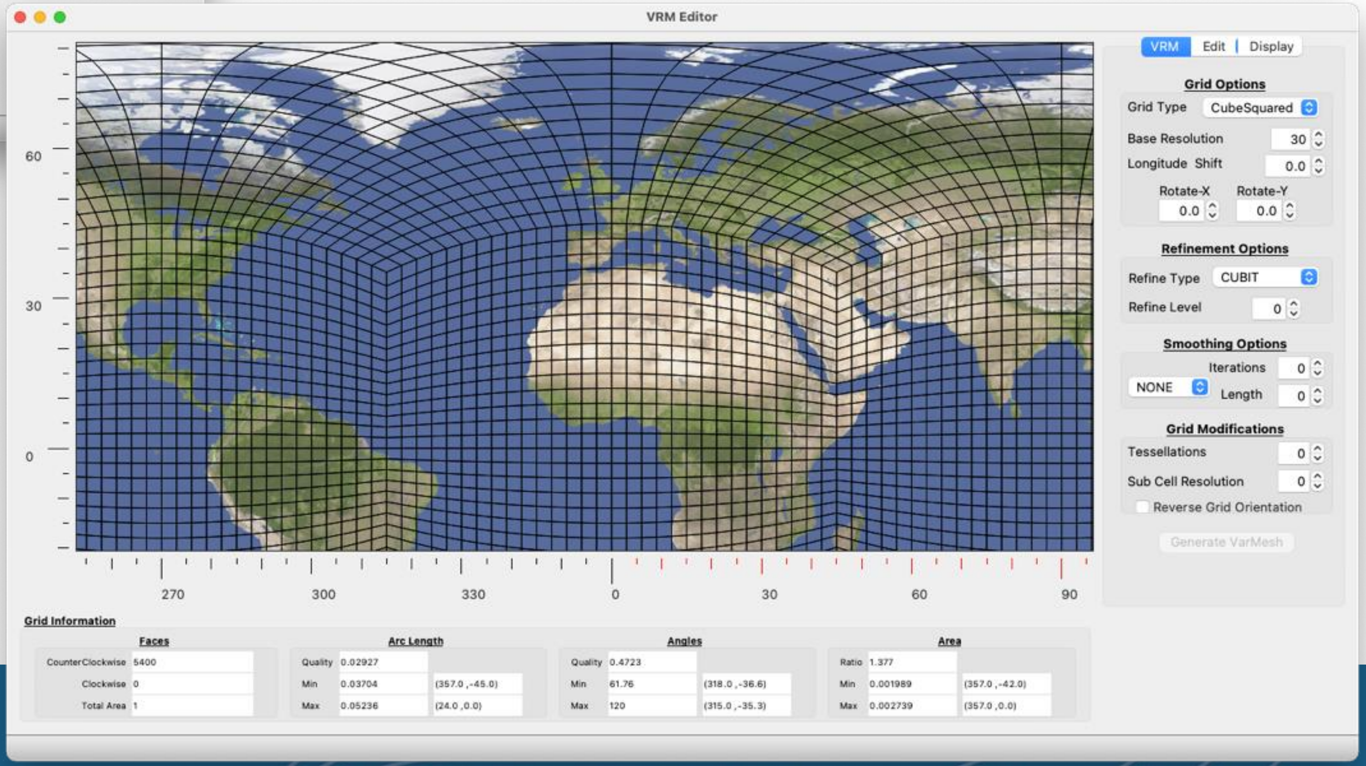


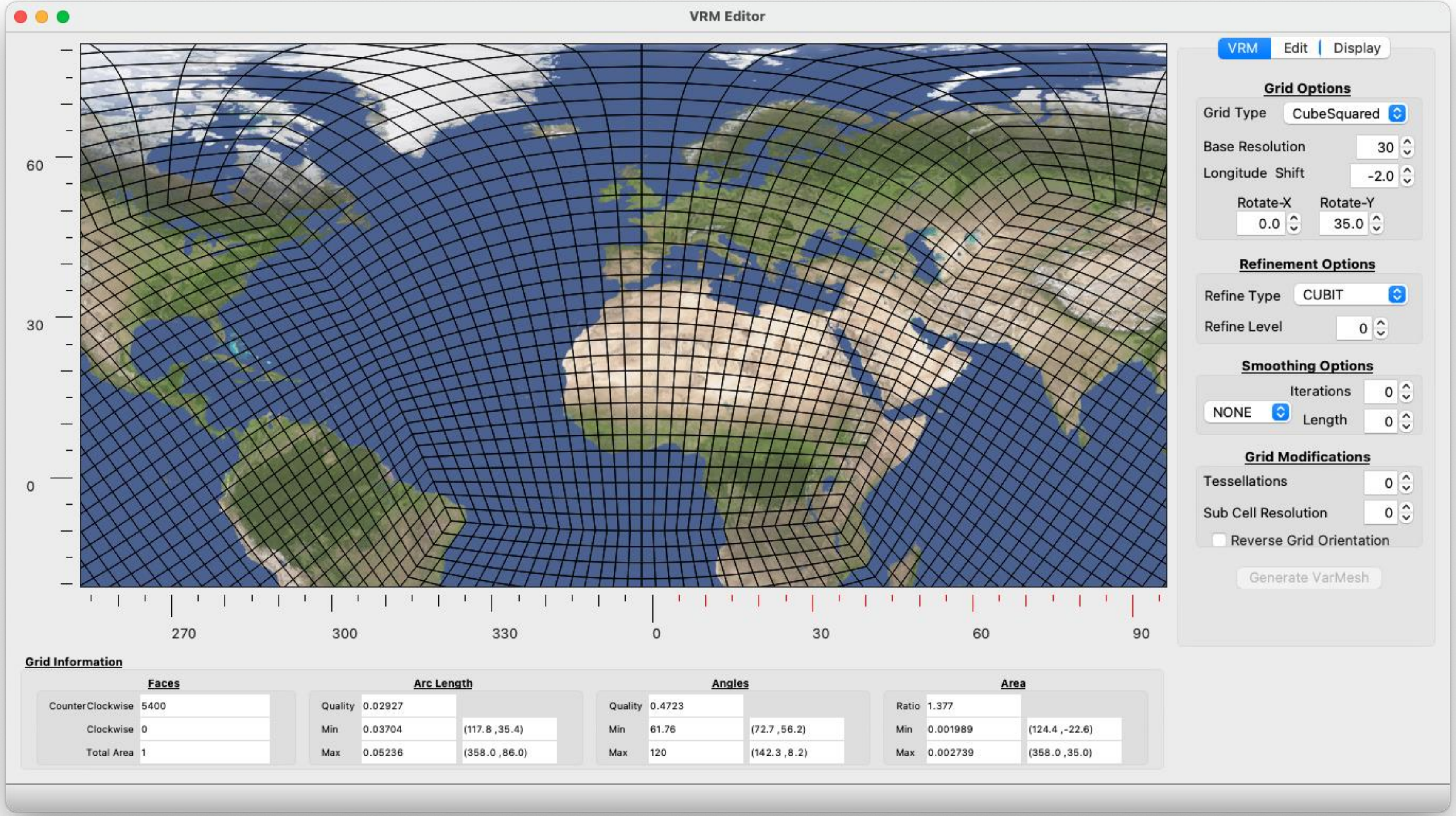
CubeSquared
Base: 30

Click
Generate Varmesh
to get base grid



- Use arrow (right/left, up/down) keys to recenter map
- Zoom in with scroll





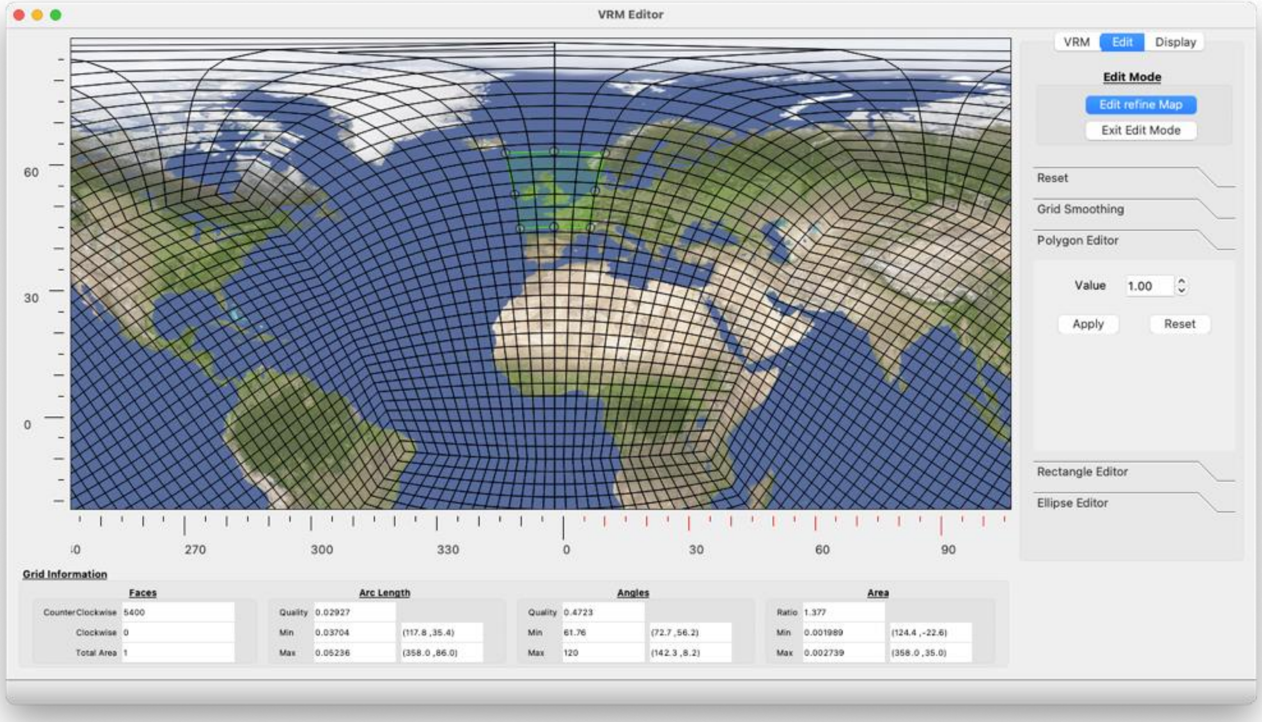
Want to center a cube face over region of interest
By changing some combination of:

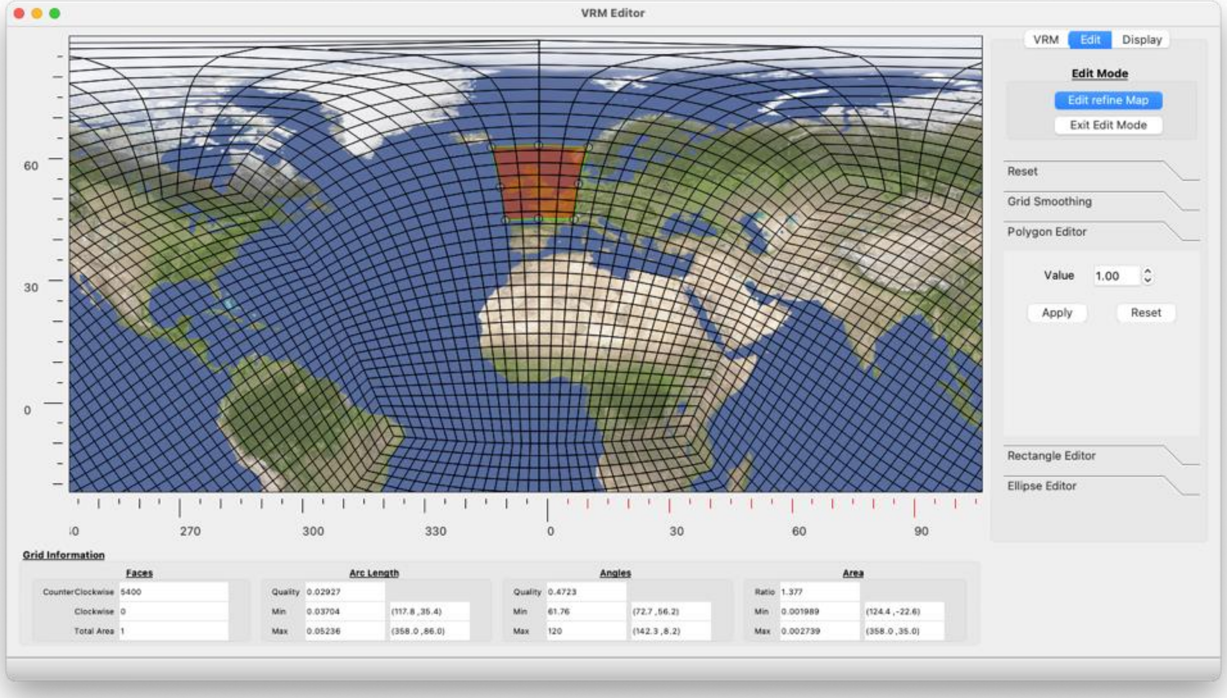
- Longitude Shift
- Rotate-X
- Rotate-Y

Click
Generate VarMesh

Go to Edit menu to create refined region

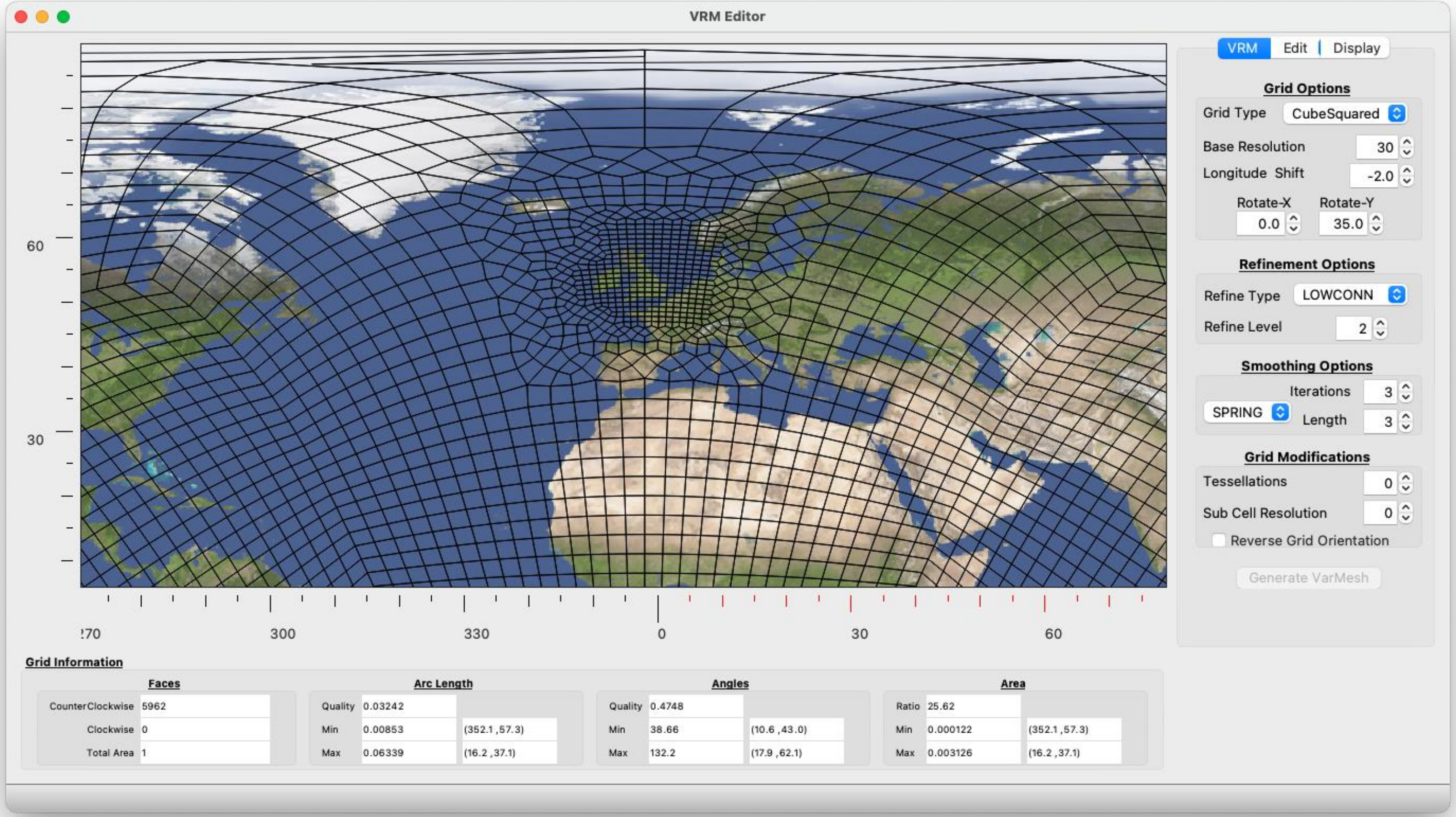
- Edit refine Map
- Polygon Editor (green box is over center of Pacific by default)
- Move 8 vertices to desired region
- Try to follow grid lines
- Click Apply to generate region
- Exit Edit Mode (Yes to save)





Go back to VRM menu

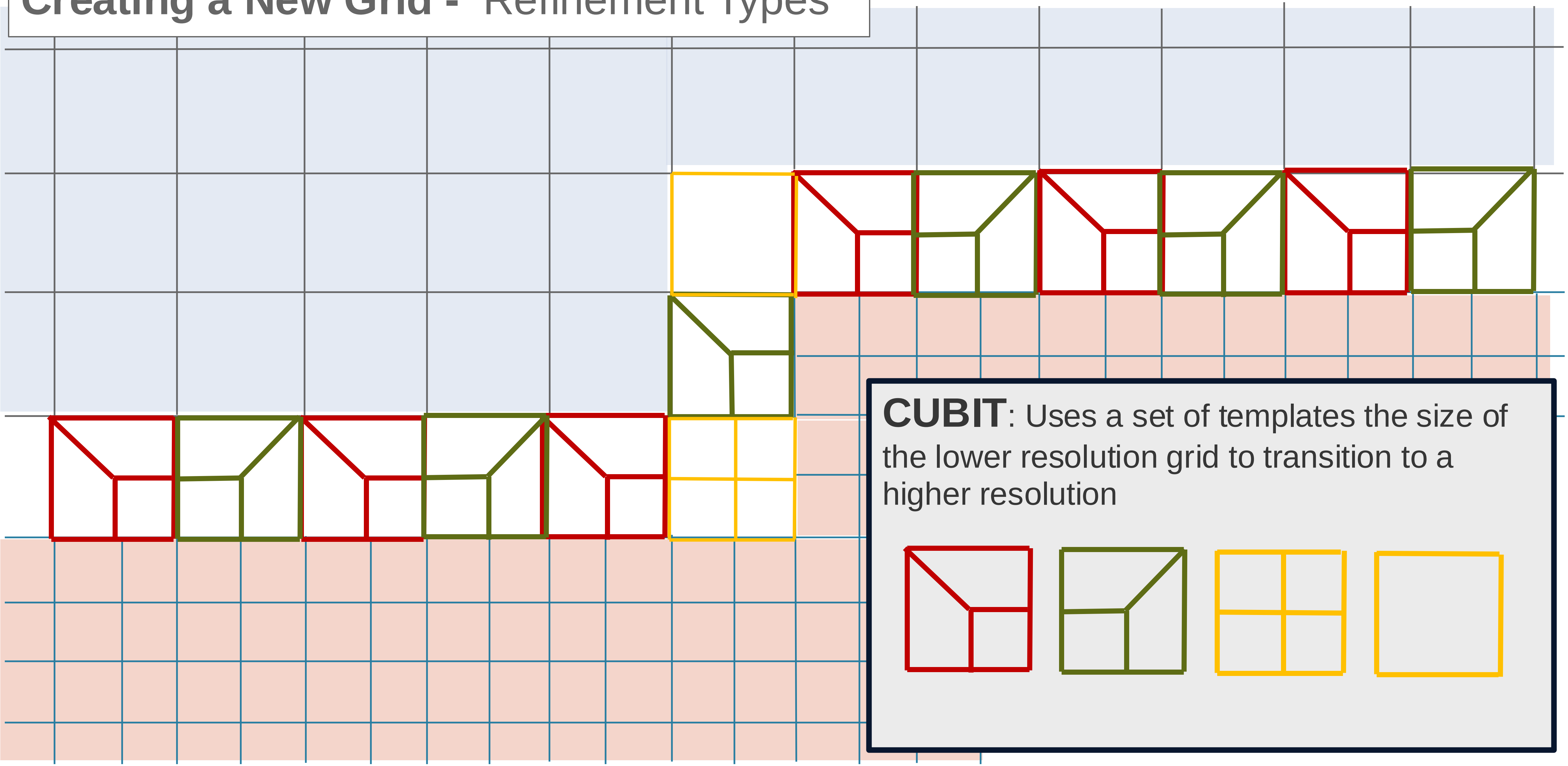
- Set Refine Type to LOWCONN (recommended, CUBIT also ok)
- Set desired refine level (1 for half, 2 for quarter, etc.)
- Set smoothing option to Spring, with Iterations: 3; Length: 3
- Click Generate VarMesh



Go back to Edit menu to adjust refinement box to remove uneven edges

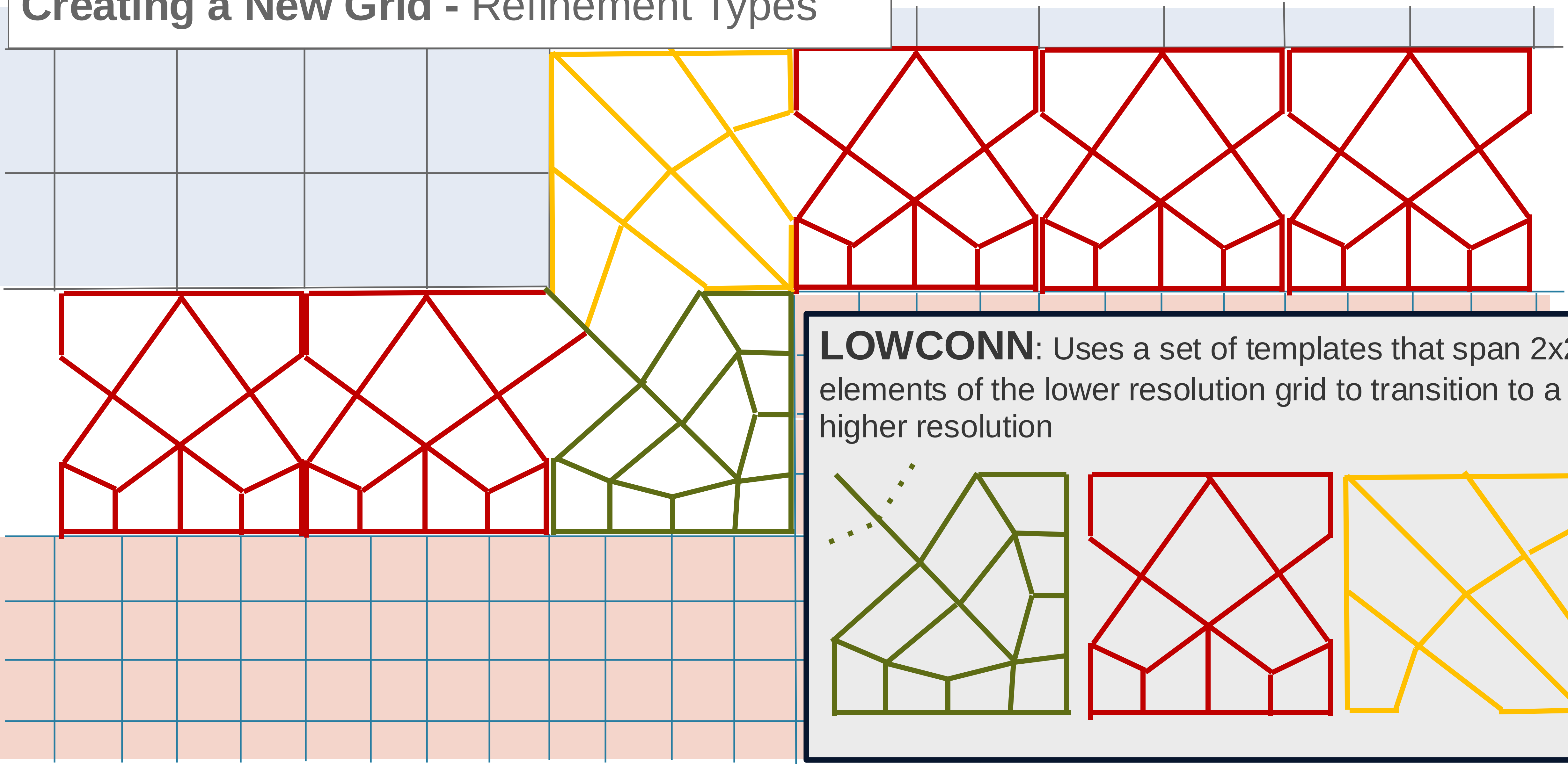
May want to add a halo of intermediate resolution

Creating a New Grid - Refinement Types

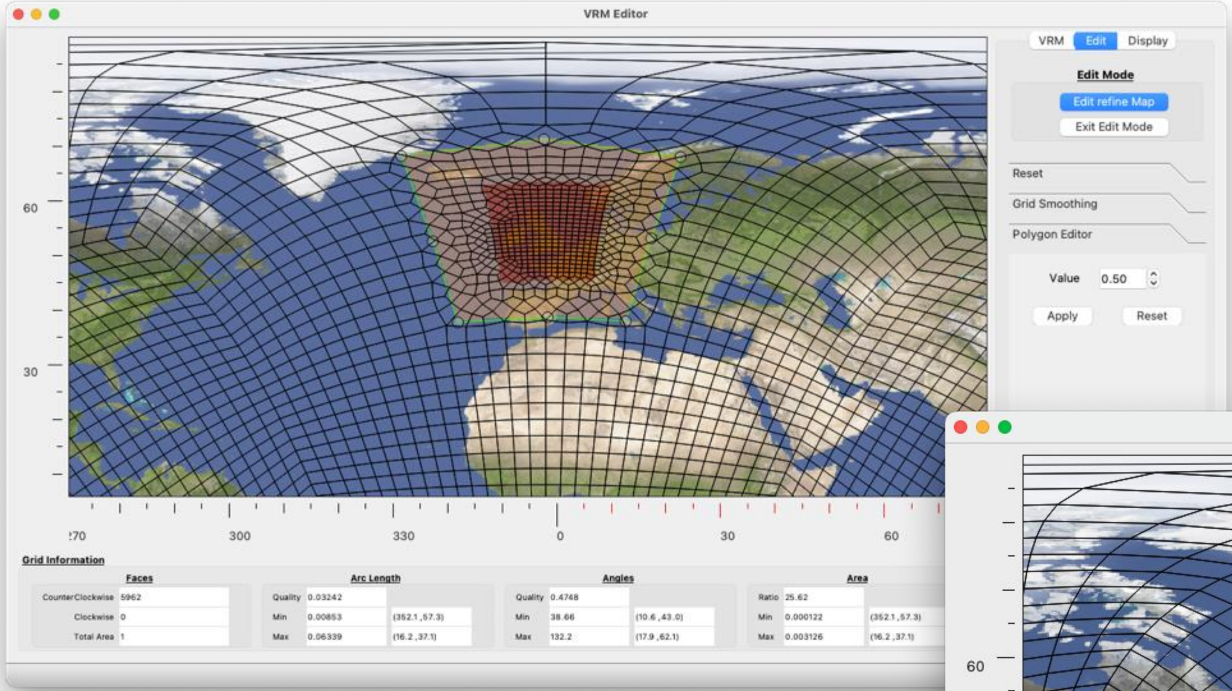


CUBIT: Uses a set of templates the size of the lower resolution grid to transition to a higher resolution

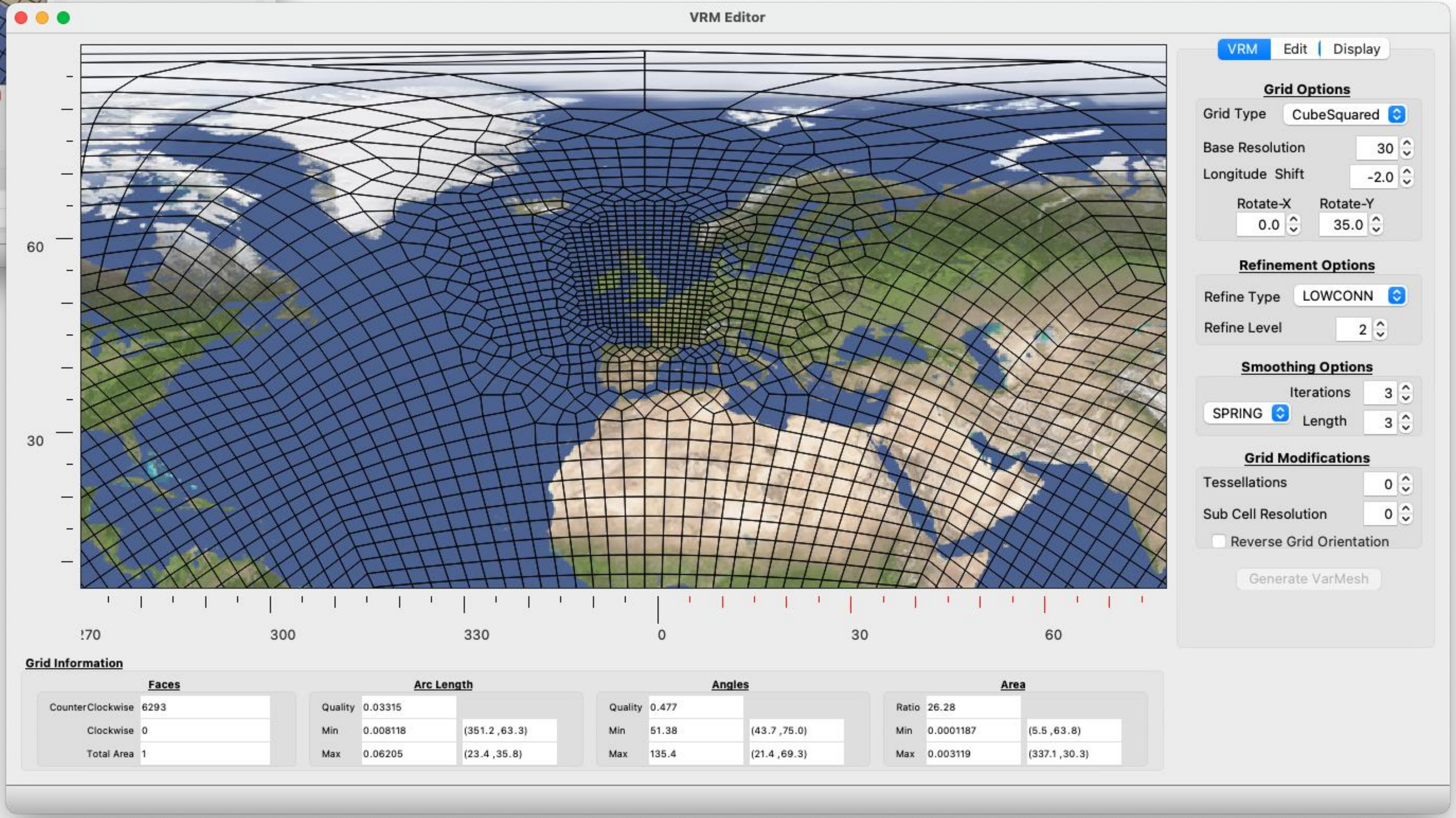
Creating a New Grid - Refinement Types



LOWCONN: Uses a set of templates that span 2x2 elements of the lower resolution grid to transition to a higher resolution



This grid has uneven edges that should be corrected before using.

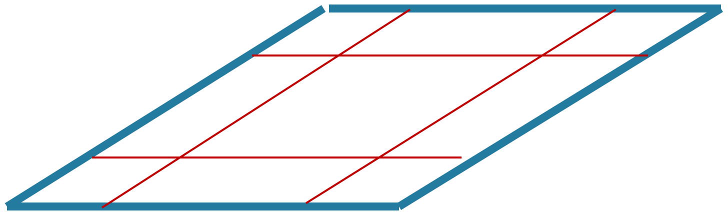


When satisfied with grid,

- Save Refinement Map {REFMAP_UK01_Lon-2-Y35_halo.nc}
- Write Exodus file {UK01_ne30x4_EXODUS.nc}

Summary of Best Practices

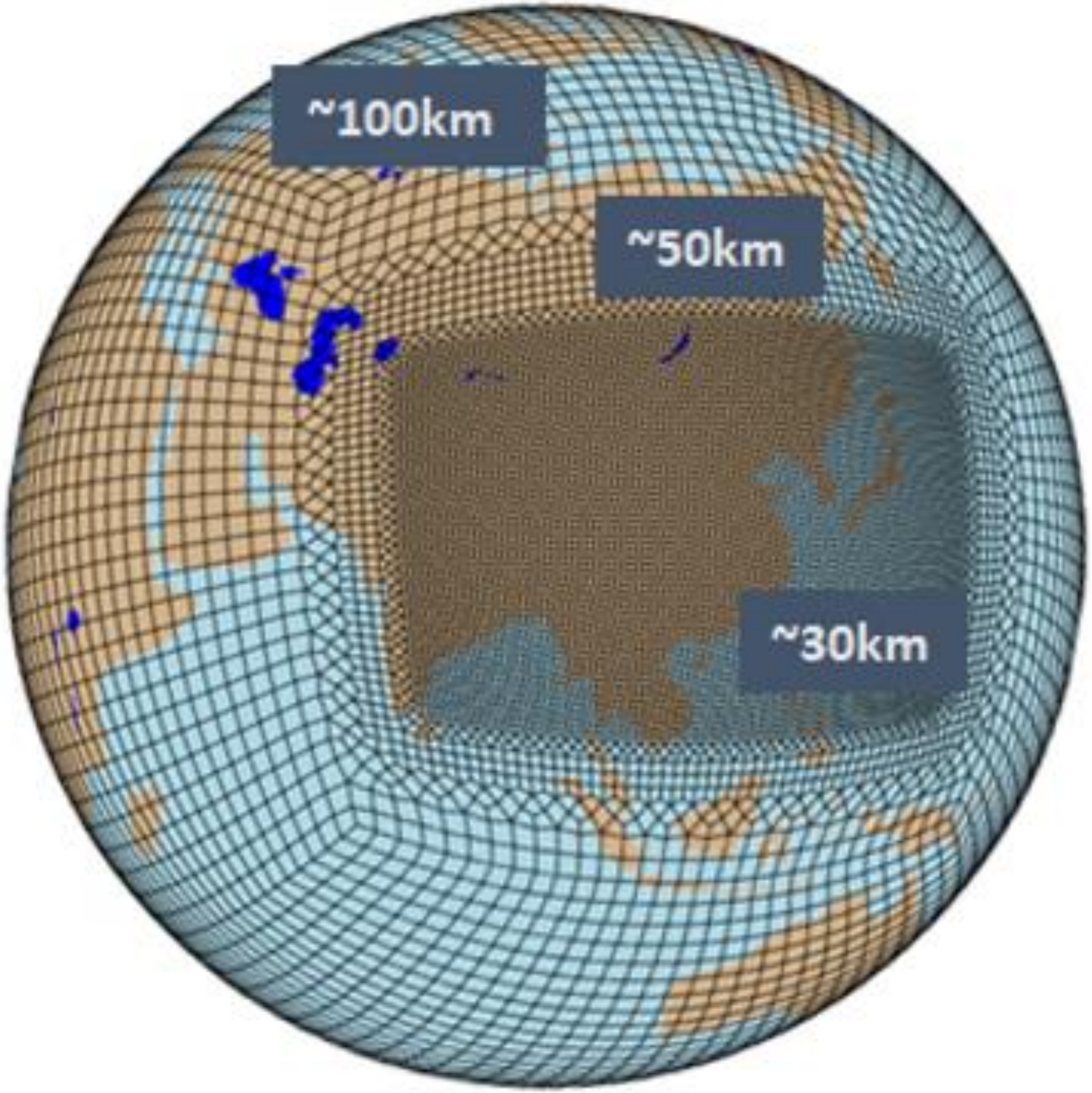
- Try to center the refinement on a cube face.
- Refinement regions whose boundaries are aligned with (rectangular) spectral element grid lines tend to behave better numerically.
- Regions that "look" good – have a uniform/symmetric appearance – tend to behave better numerically.
- In general, it is better to use LOWCONN instead of CUBIT.
- Convex regions tend to behave better numerically compared to regions that have concave edges.
- When creating a high resolution region (e.g. ne240), try to make each halo (successive resolutions) several cells wide on all sides of the domain.
- AVOID aggressive refinement at the edges of the cube faces.
- Generally, corner angles in the range (45-135) are okay, particularly with LOWCONN. Once the angles leave this range, users should identify the elements identified a the bottom of VRM_Editor and try to adjust the grid.
  [This is a FUZZY boundary!, elements that are near the limiting values may still be acceptable]



Summary of Best Practices (cont.)

- When making a small refined region you might want to start with ne60 grid (instead of ne30).
- In the polygon editor of VRM Editor, the number of vertices can not be increased more than 8. When making a large refined region try using multiple polygons in VRM Editor to get edges that follow the SE grid lines.
- Sometime there are only minor irregularities along a refinement edge. To smooth edges in VRM Editor, try adjust Lon shift and Rotate-X, Rotate-Y by small amounts (0.1 deg), this sometimes resolves the problem.
- You may want to save refinement map frequently, because the VRM editor sometimes crashes. When there is a crash, you can restart your work from the saving point.
- If you are having trouble with a grid, e.g. it needs large sub-cycling settings, then there is likely a problem with your grid. In the long run, you are better off iterating with variations/adjustments to your grid to get better performance.

Some Variable Resolution Examples

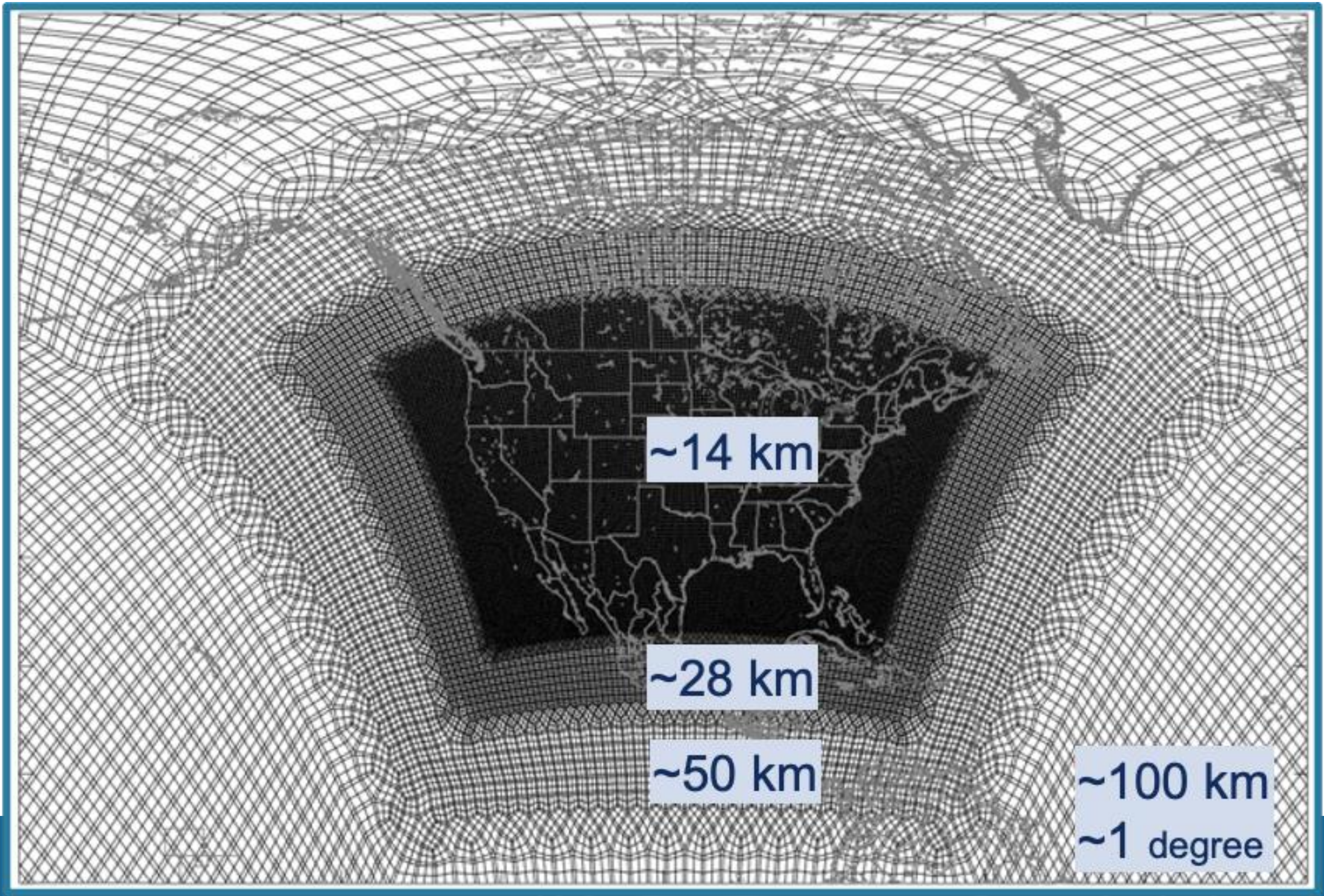


ne30x8 CONUS grid

Refined region follows base grid lines

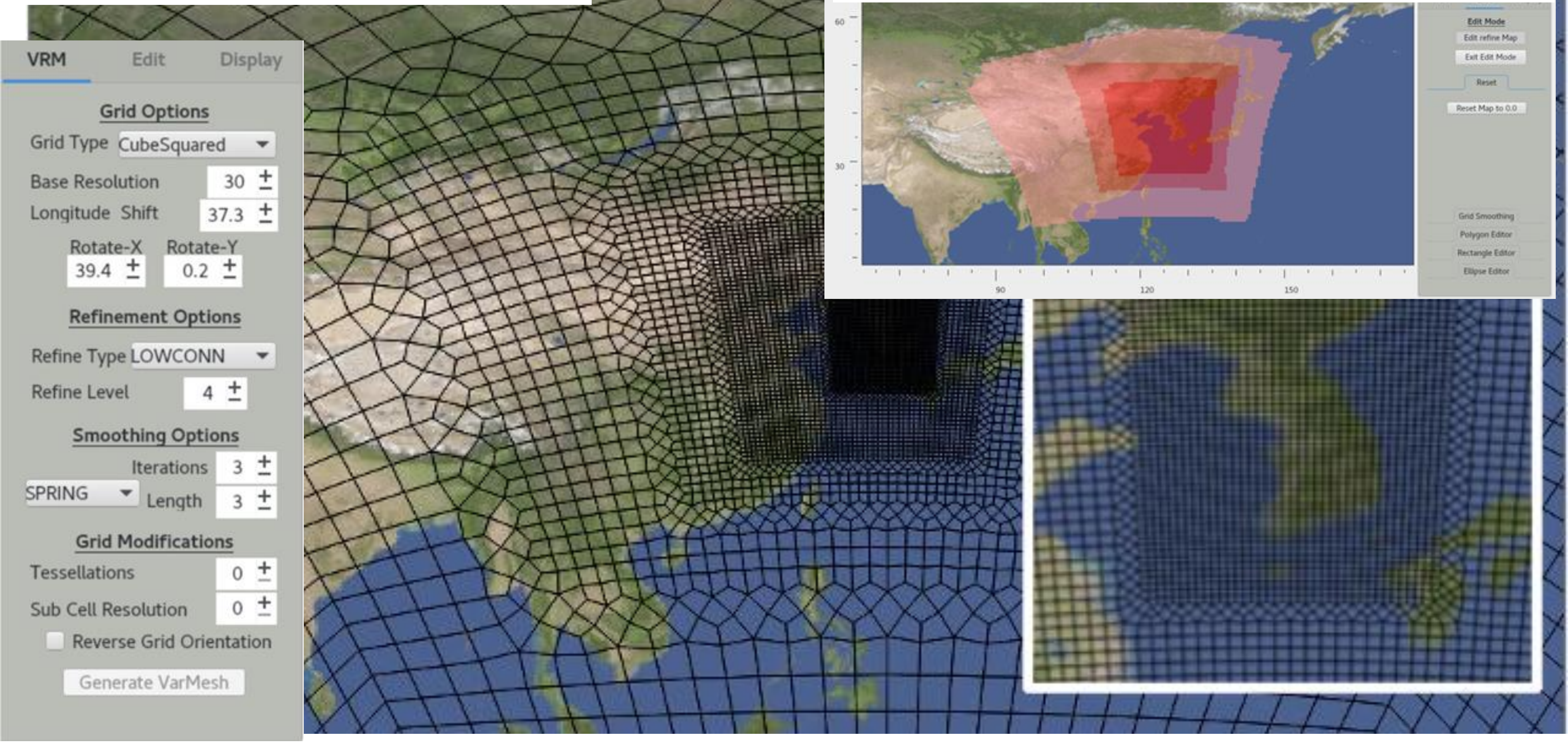
Each intermediate resolution has several cells on all sides

Transition grid cells are generally uniform

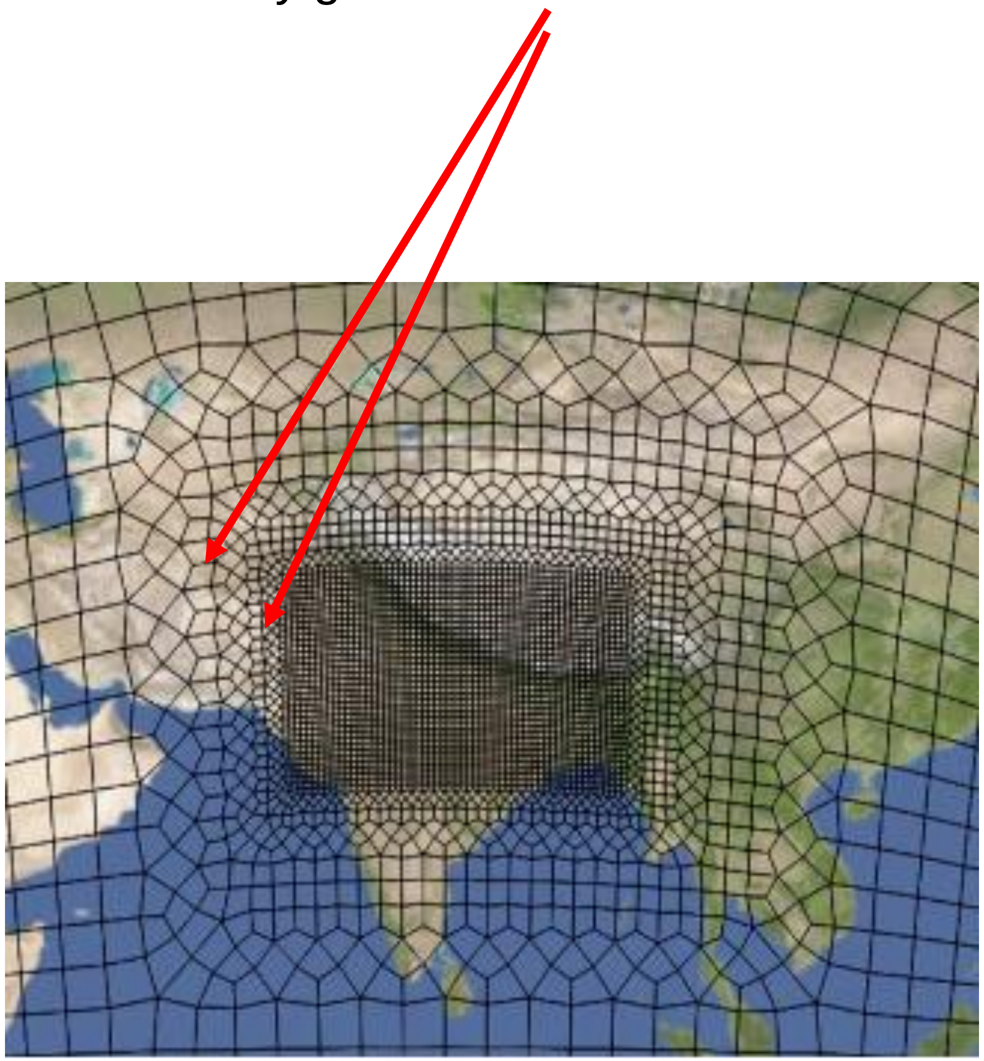


ne30x16, with 0.5, 0.25, 0.125 halos

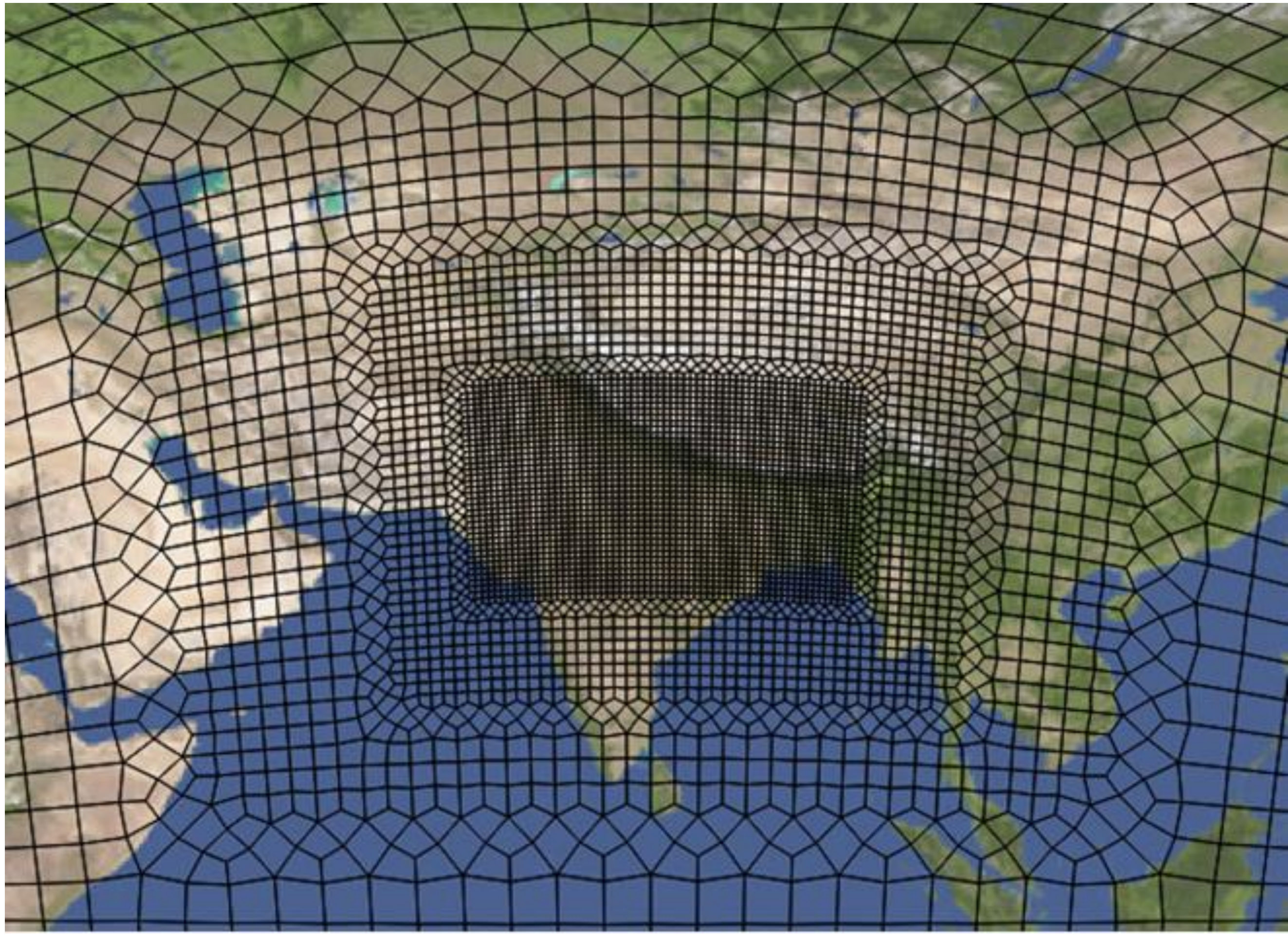
Although refinement map looks irregular, grid is ok.



Preliminary grid - uneven halo

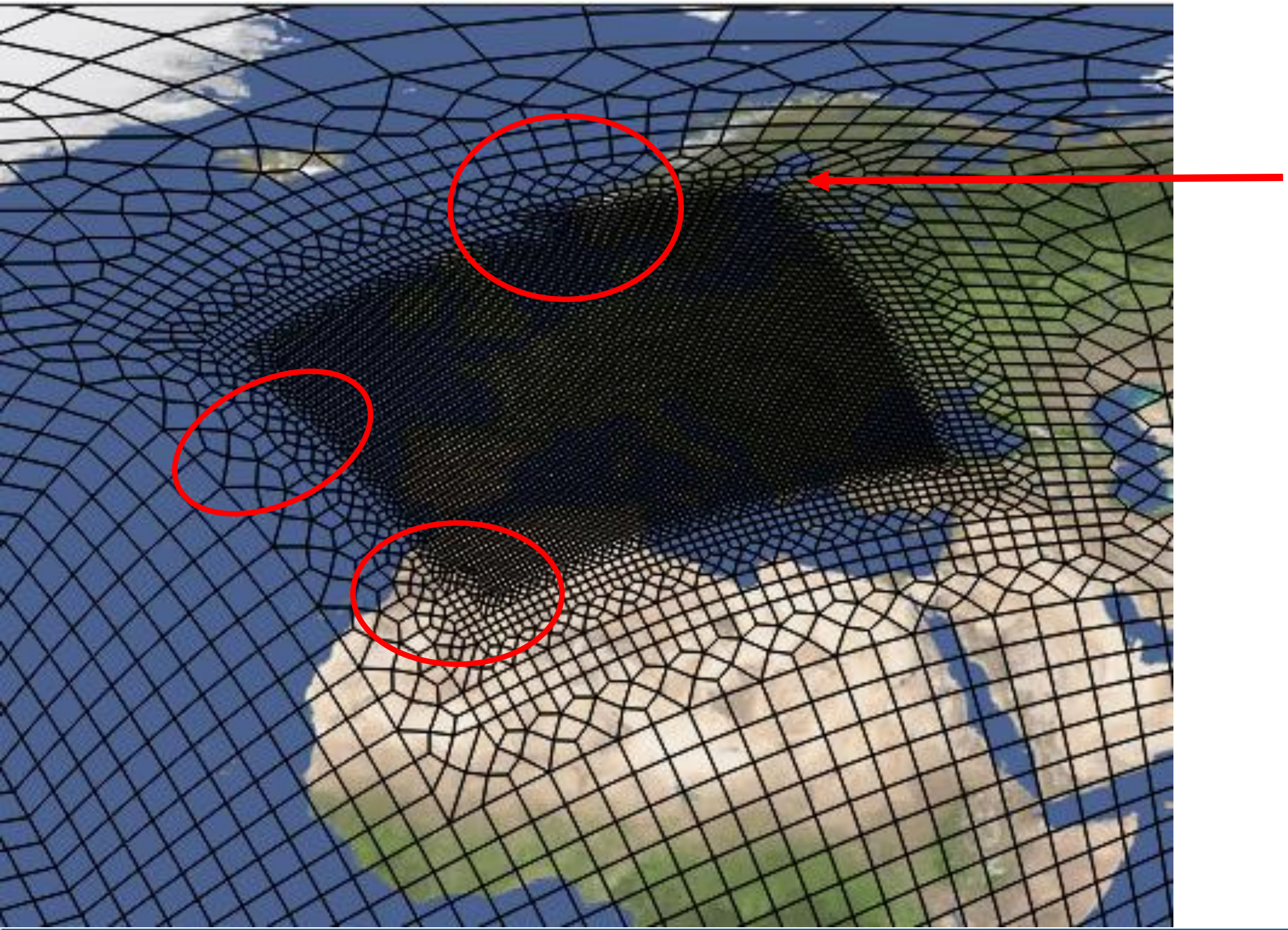


Final grid



examples of poor grids - convex edges, weird transition grids, …

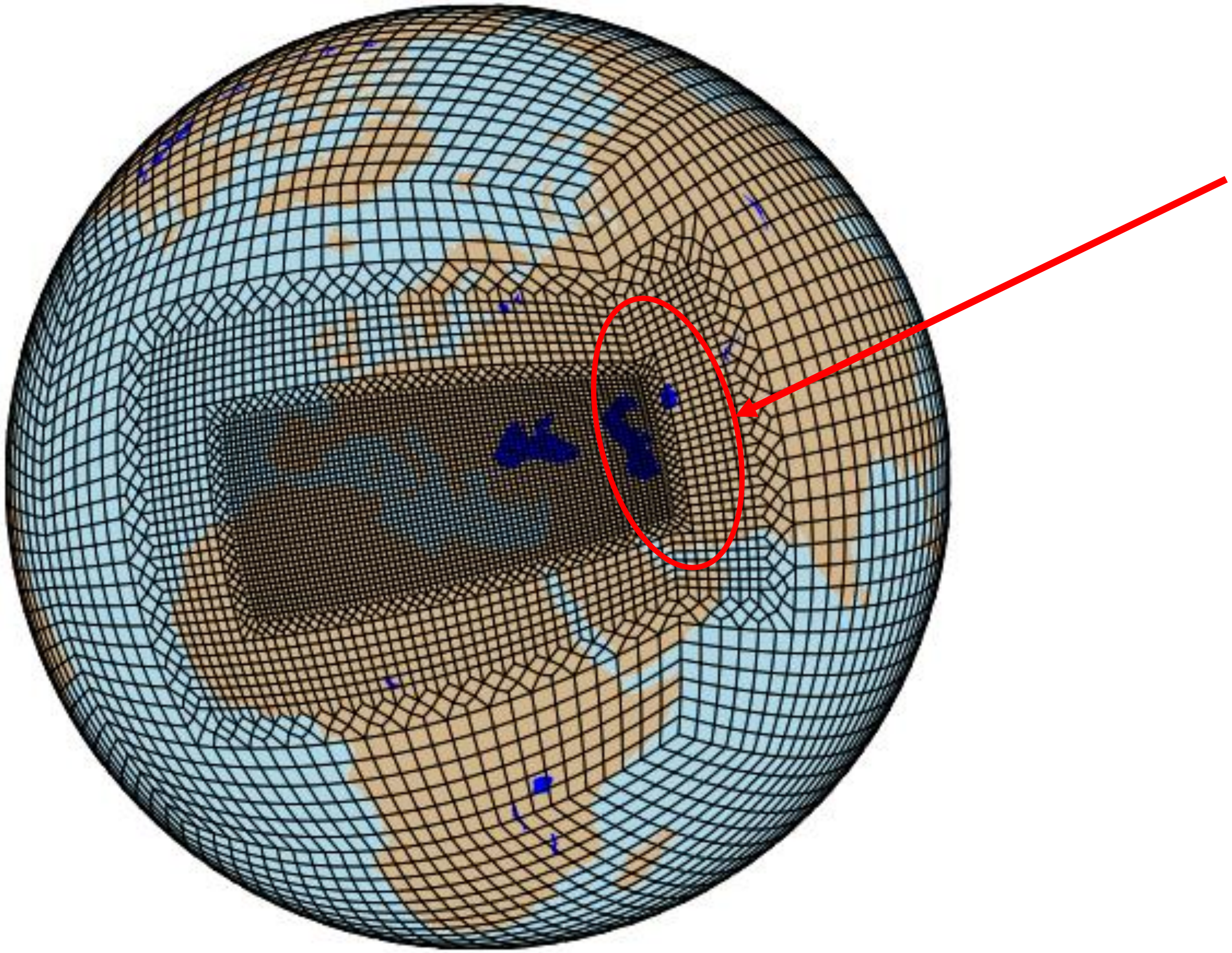
ne30x8, LOWCON, Spring: 3 iter. 0 length



- Irregular halo
- Bump on north side
- SW corner is convex
- Changing resolution across cube edge

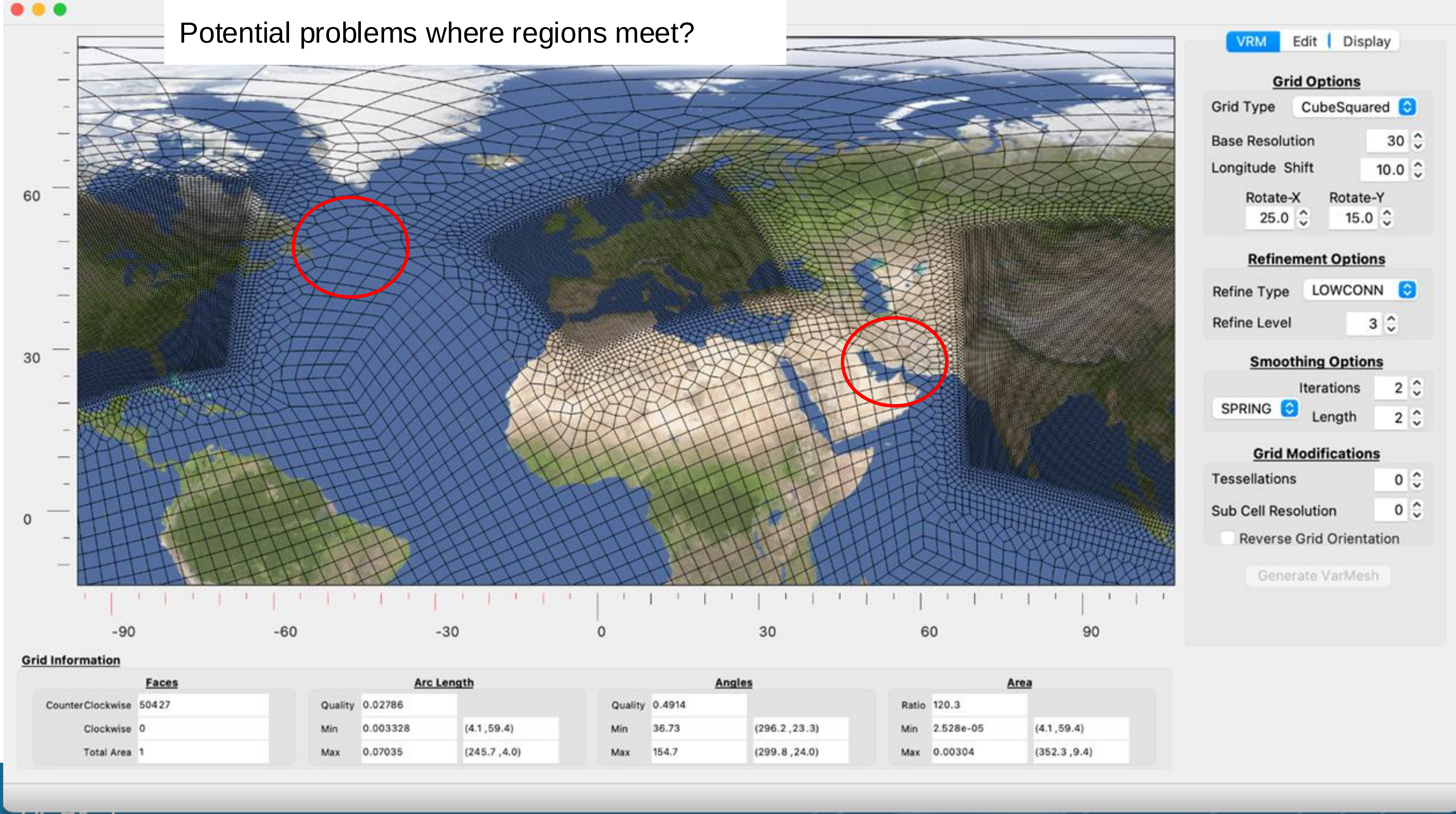
examples of poor grids - convex edges, weird transition grids, …

ne30x4, LOWCON

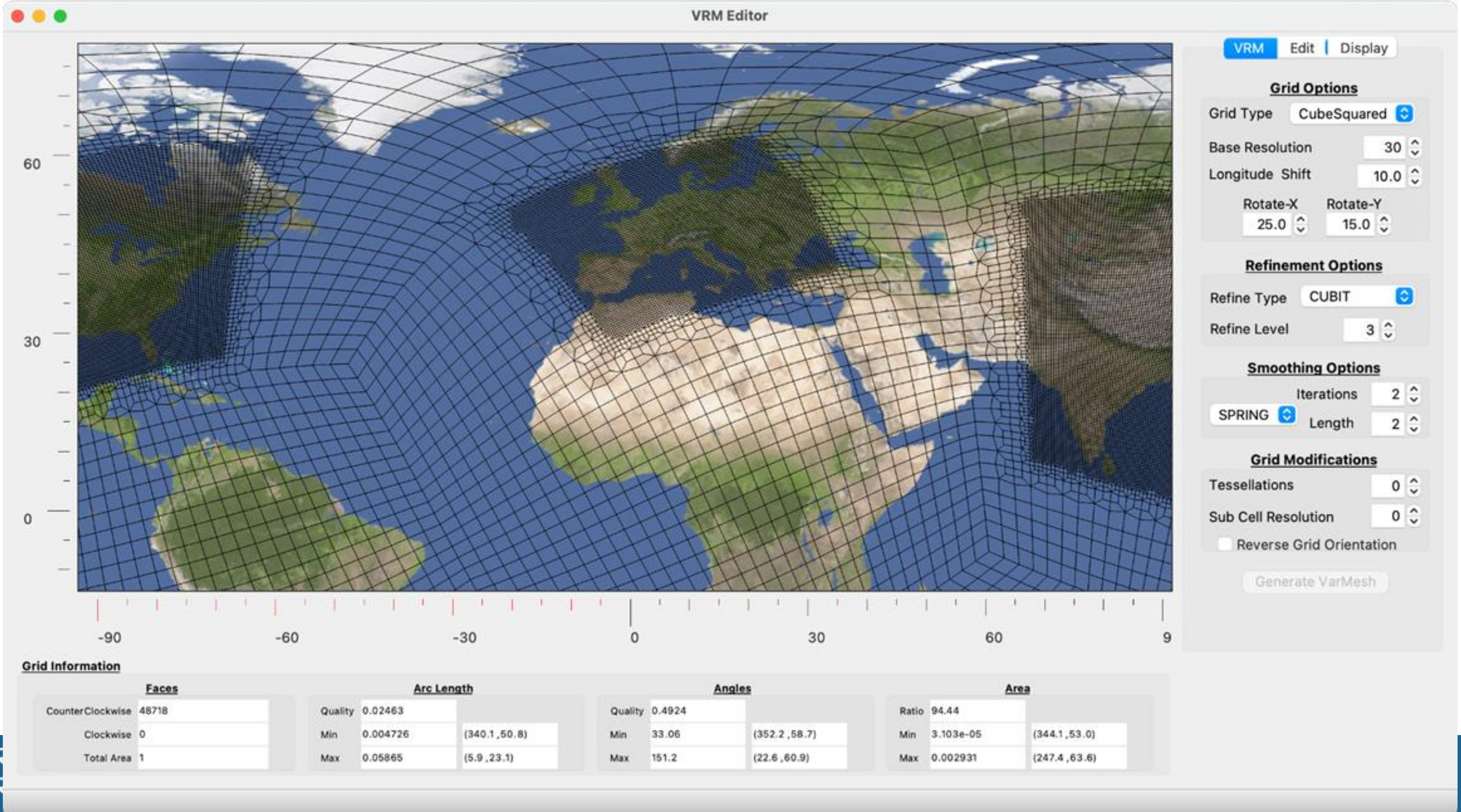


- Really nice buffer zones!
- But changing resolution across cube edge is a no-no!

Potential problems where regions meet?

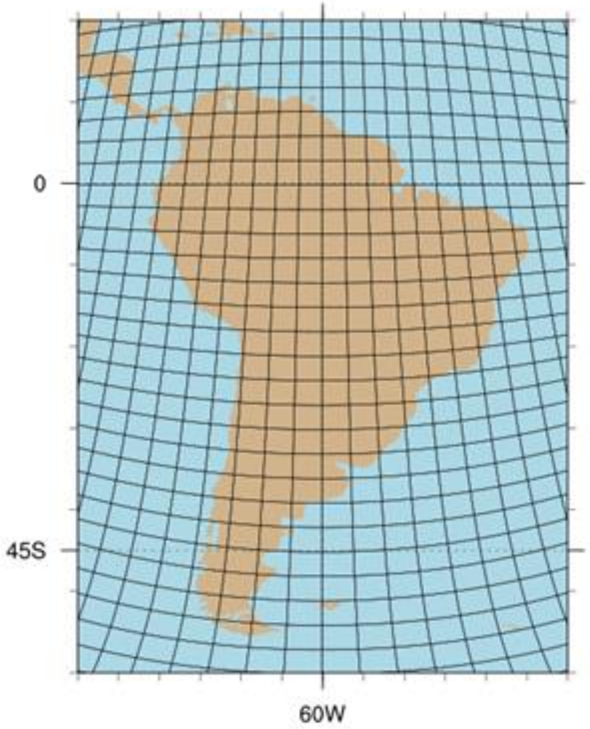


Same refinement map as previous, but with CUBIT - regions have a few ne30 grids separating them

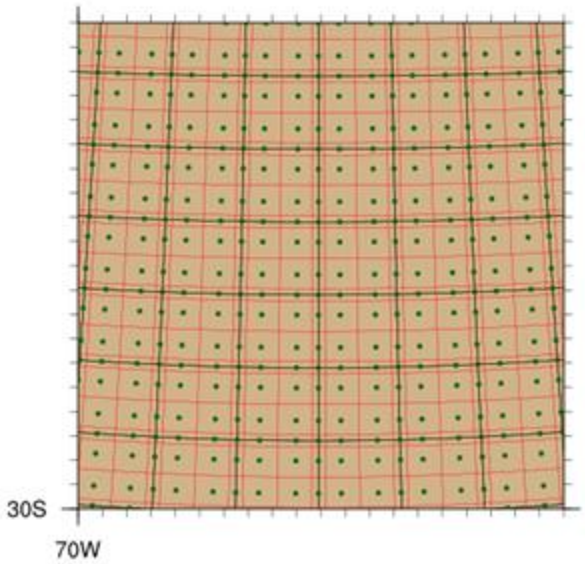


Creating a New Grid

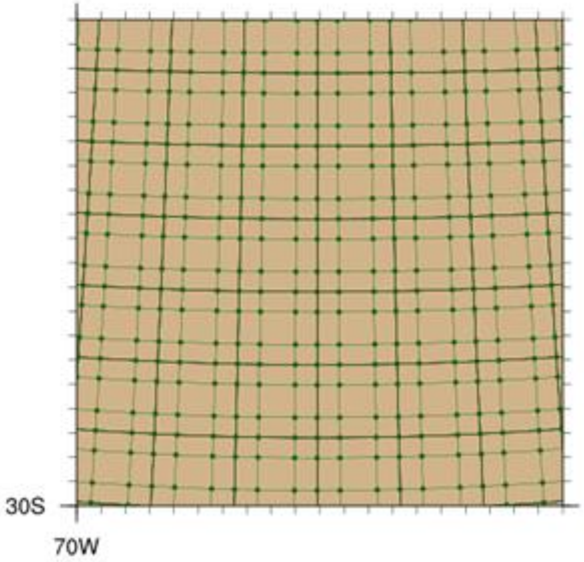
The Grids We Create:



EXODUS: contains the quadrilateral spectral elements



SCRIP: contains spectral element grid points, the polygonal region associated with each point, and the corresponding area weight for each point.

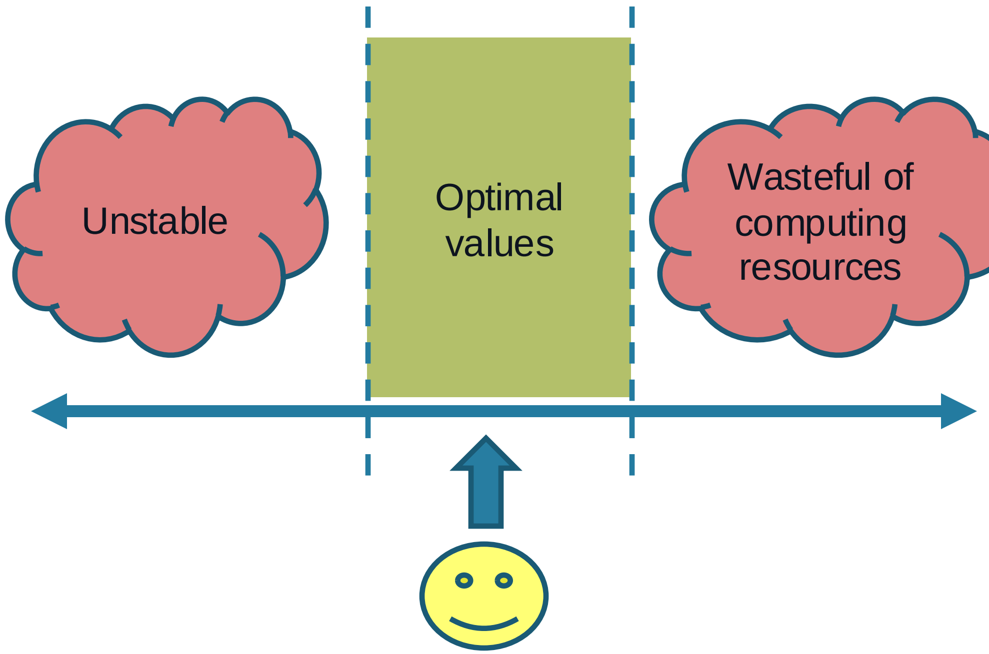


LATLON: contains Lat, Lon, and area values found in the history files. It also contains the connectivity information for each quad region useful for graphing.

MESH: An enhancement of the SCRIP file with additional information to optimize ESMF mapping functions.

Setting Initial Values For a Stable 5-Day Run

- For each new grid, users must establish default values for time stepping, damping, etc...
- Starting with a 5-Day run, the first goal is to obtain a stable run
- From there values should be refined to obtain valid results without being wasteful of computing resources.




```
pelayout  --->  more resolution = more processor/memory needs.
      |-> ATM_NCPL / timestep    ---  max limit for CFL
            |-> nsplit
            |       |-> Apply CAM forcing
            |       |-> rspllt
            |       |       |-> qsplit
            |       |       |       |-> Apply dyn RHS forcing
            |       |       |       |-> hypervis_subcycle_q
            |       |       |       |         |-> Apply hyper-viscosity
            |       |       |       |-------------
            |       |       |-> Advect tracers
            |       |       |-----------------
            |       |-> Apply vertical remap
            |       |-> Apply energy fixer
            |       |--------------
            |----------------
```

- The relevant parameters have a hierarchical relationship and affect different processes

Setting Initial Values For a Stable 5-Day Run (cont.)

- Begin with the PELAYOUT to ensure that the there are sufficient memory and processor resources
- Then establish the model physics timestep which is determined by ATM_NCPL
  dt_phys = 86400/ATM_NCPL
- Once these are established, then work on the remaining parameters.

Search for "dt_dyn" in the atm.log file:

```
Estimates for maximum stable and actual time-steps for different aspects of algorithm:
(assume max wind is 120.00000000m/s)
(assume max gravity wave speed is 342m/s)

* dt_dyn      (time-stepping dycore  ; u,v,T,dM) <     70.21s     25.00s
* dt_dyn_vis   (hyperviscosity)       ; u,v,T,dM) <    87.36s      8.33s
* dt_tracer_se  (time-stepping tracers ; q        ) <    60.92s     25.00s
* dt_tracer_vis (hyperviscosity tracers; q        ) <   218.39s     25.00s
* dt_remap (vertical remap dt)    100.00
* dt   (del2 sponge         ; u,v,T,dM) <    36.17s    25.00s
* dt   (del2 sponge         ; u,v,T,dM) <   123.66s    25.00s
* dt   (del2 sponge         ; u,v,T,dM) <   416.10s    25.00s

 tstep_type =          4
CAM dtime (dt_phys):            600.00
```

```
pelayout  --->  more resolution = more processor/memory needs.
      |->  ATM_NCPL / timestep    ---  max limit for CFL
            |-> nsplit
            |       |-> Apply CAM forcing
            |       |-> rspllt
            |       |       |-> qsplit
            |       |       |       |-> Apply dyn RHS forcing
            |       |       |       |-> hypervis_subcycle_q
            |       |       |       |         |-> Apply hyper-viscosity
            |       |       |       |------------
            |       |       |-> Advect tracers
            |       |       |-----------------
            |       |-> Apply vertical remap
            |       |-> Apply energy fixer
            |       |--------------
            |----------------
```

- The **atm.log** file has useful estimates as a guide to setting values.
- See Adam Herrington's document in the Docs directory for details on the process of establishing these values:

Docs/CAM-tsteps-inic-for-newgrids_v0.pdf

Settings for running a case

Once the users has established acceptable parameters values, the files in the REPO directory must be edited to establish them as the defaults.

- The default PELAYOUT and ATM_NCPL values need to be set for each case.

shell_commands

```
#----------------------------------------------------------------
# Set the default time step here:
#
#      ATM_NCPL=48    <--> 1800 sec timestep
#      ATM_NCPL=96    <-->  900 sec timestep
#      ATM_NCPL=144   <-->  600 sec timestep
#      ATM_NCPL=288   <-->  300 sec timestep
# notice a pattern?
#----------------------------------------------------------------
./xmlchange ATM_NCPL=144

#----------------------------------------
# Set the default pelayout:
#----------------------------------------
./xmlchange NTASKS=1080
```

- The remaining tuning parameters are set in user_nl_cam along with the default topography and initial model state.

user_nl_cam

```
&dyn_se_inparm
 se_mesh_file ='$(AMWG_DEMO_FEB22)/AMWG_REPO/
               ne0np4.SAMGRID01.ne30x4//grids/SAMGRID01_ne30x4_EXODUS.nc'
 se_hypervis_scaling    =  3.0D0
 se_hypervis_subcycle  =  3
 se_nsplit               =  6
 se_rsplit               =  4
 se_qsplit               =  1
 se_nu_top               =  1.0e5
 se_hypervis_power     =  0
 se_hypervis_subcycle_q =  1
 se_hypervis_subcycle_sponge =  1
 se_refined_mesh        = .true.
   . . . . . . . . .
   . . . . . . . . .
/
&cam_initfiles_nl
 bnd_topo      = '$(AMWG_DEMO_FEB22)/AMWG_REPO//ne0np4.SAMGRID01.ne30x4//topo/
                 topo_ne0np4.SAMGRID01.ne30x4_nc3000_Co060_Fi001_MulG_PF_RR_Nsw042_210207.nc'
 ncdata       = '$(AMWG_DEMO_FEB22)/AMWG_REPO//ne0np4.SAMGRID01.ne30x4//inic/
                 cami-mam4_0000-01-01_ne0np4.SAMGRID01.ne30x4_L32_c210207.nc'
/
```

- The user_nl_clm file contains the default fsurdat file for the grid.

user_nl_clm

```
!==================================================
! Comment out the fsurdat file you need to use:
!==================================================
! fsurdat = '$(AMWG_DEMO_FEB22)/AMWG_REPO//ne0np4.SAMGRID01.ne30x4//clm_surfdata_5_0/
!           surfdata_ne0np4.SAMGRID01.ne30x4_hist_16pfts_Irrig_CMIP6_simyr2000_c210207.nc'
fsurdat = '$(AMWG_DEMO_FEB22)/AMWG_REPO//ne0np4.SAMGRID01.ne30x4//clm_surfdata_5_0/
           surfdata_ne0np4.SAMGRID01.ne30x4_hist_16pfts_Irrig_CMIP6_simyr1850_c210207.nc'
!==================================================
```

Final steps for running MUSICAv0 (CAM-chem)

The interpolated IC file for the 5-Day test run will invariably be out of balance requiring shorter time steps at startup.
Once the initial tuning process is completed, a new IC file should be generated from model output, and the tuning values should be further optimized.

For scientifically meaningful runs, there are additional steps:

- The land model must be spun up from the initial default.
- Emissions data must be conservatively regridded for the new grid.
- Nudging data must be processed for the new grid
- Regrid a CAM-chem IC file (to get reasonable concentrations).
  If needed merge this with the spun-up CAM IC file (get T, Q, etc.)

Resources

VR tutorial:
https://github.com/ESMCI/Community_Mesh_Generation_Toolkit/tree/master/VRM_tools/Docs/ AMWG_FEB22_DEMO_FILES

MUSICA wiki page: https://wiki.ucar.edu/display/MUSICA/MUSICA+Home
https://wiki.ucar.edu/display/MUSICA/Generating+variable+resolution+grids
CAM-chem wiki page: https://wiki.ucar.edu/display/camchem/Home
CESM2 website: https://www.cesm.ucar.edu/models/cesm2/
CESM Tutorial: https://www.cesm.ucar.edu/events/tutorials/2022/coursework.html
CESM Forum: https://bb.cgd.ucar.edu/cesm/forums/cam-chem.154/

Creating a Variable Resolution Grid

Detailed instructions

Tools for creating a new Spectral Element variable resolution (VR) grid

Available in the Community Mesh Generation Toolkit

https://github.com/ESMCI/Community_Mesh_Generation_Toolkit

If not running on casper, get your own copy:

> *> git clone https://github.com/ESMCI/Community_Mesh_Generation_Toolkit.git*

If running on casper.ucar.edu, use:

/glade/work/emmons/tutorial_Nanjing/VRM_tools/VRM_Editor/src/VRM_Editor

Create Repository for New Grid

Naming Conventions

grid_name = ne0np4.*NAME*.ne30x*R*

where *NAME* is a unique name for your grid, ne30 is for the base resolution, *R is for the* refinement factor of the highest resolution region. All variable resolution grids with 4 GLL points must begin with ne0np4.

grid_label = *NAME*_ne30x*R*

In the examples below, a grid for a small refined region at ne30x8 (ne240, 1/8 deg) centered over Nanjing will be created, using *NAME*=Nanjing, *R*=8, so:

grid_name = **ne0np4.Nanjing.ne30x8 grid_label** = **Nanjing_ne30x8**

Replace these with appropriate names and resolution for your own grid.

Set up a repository for your grid

Create a directory in your /glade/work/$USER/ space for a repository of your created files ($REPO). For example:

$REPO = /glade/work/emmons/tutorial_Nanjing/ne0np4.Nanjing.ne30x8/

Make 'grids' directory

In your repository directory, create a 'grids' subdirectory.

Run VRM_Editor from your Repository ‘grids’ directory

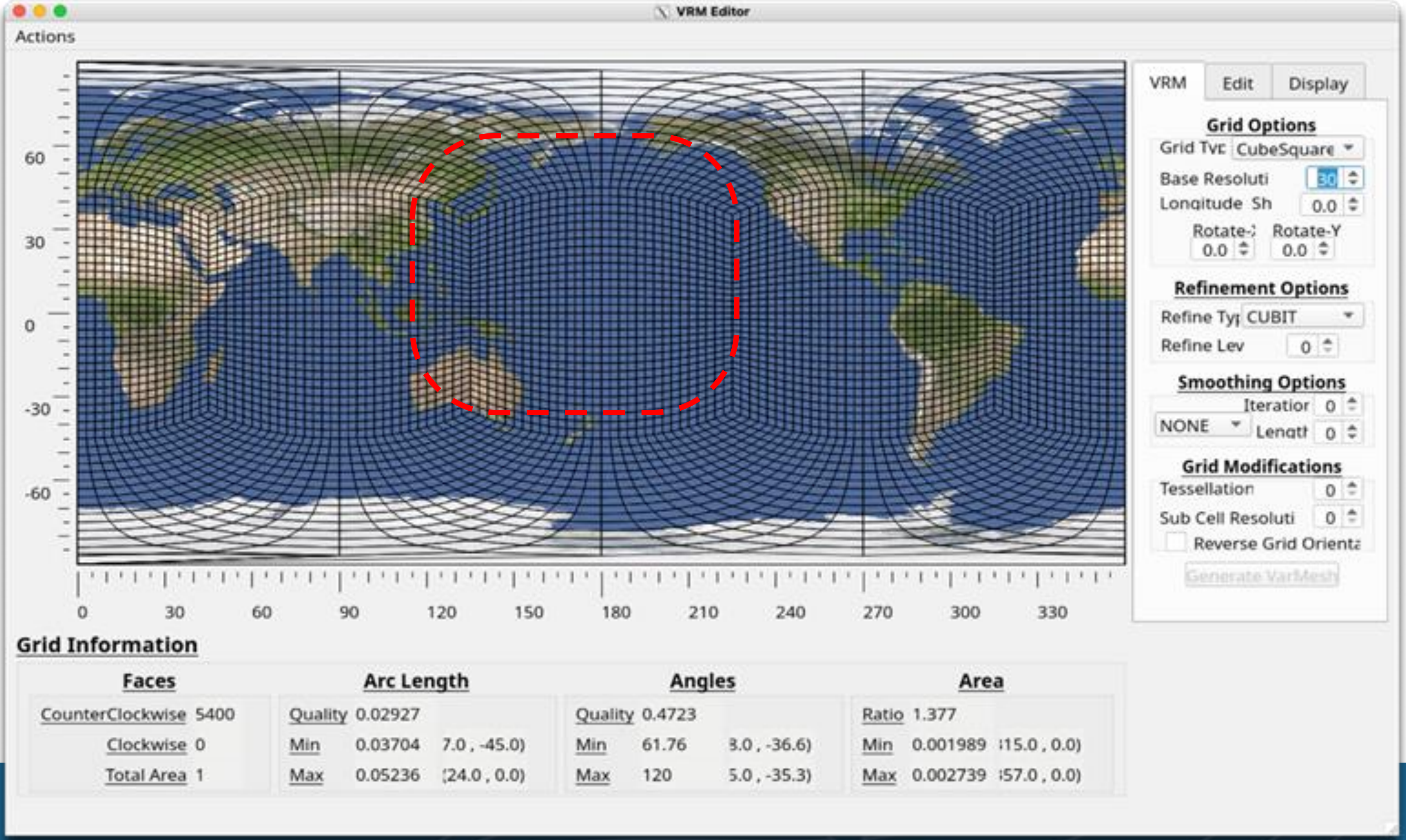
```
> cd $REPO/grids
> /glade/work/emmons/tutorial_Nanjing/VRM_tools/VRM_Editor/src/VRM_Editor
```

Set up the base grid

On the ‘VRM’ tab, select:

- Grid Type: CubeSquared
- Base Resolution: 30
- Click Generate VarMesh.

The red dashed line shows a cube face.



NCAR UCAR

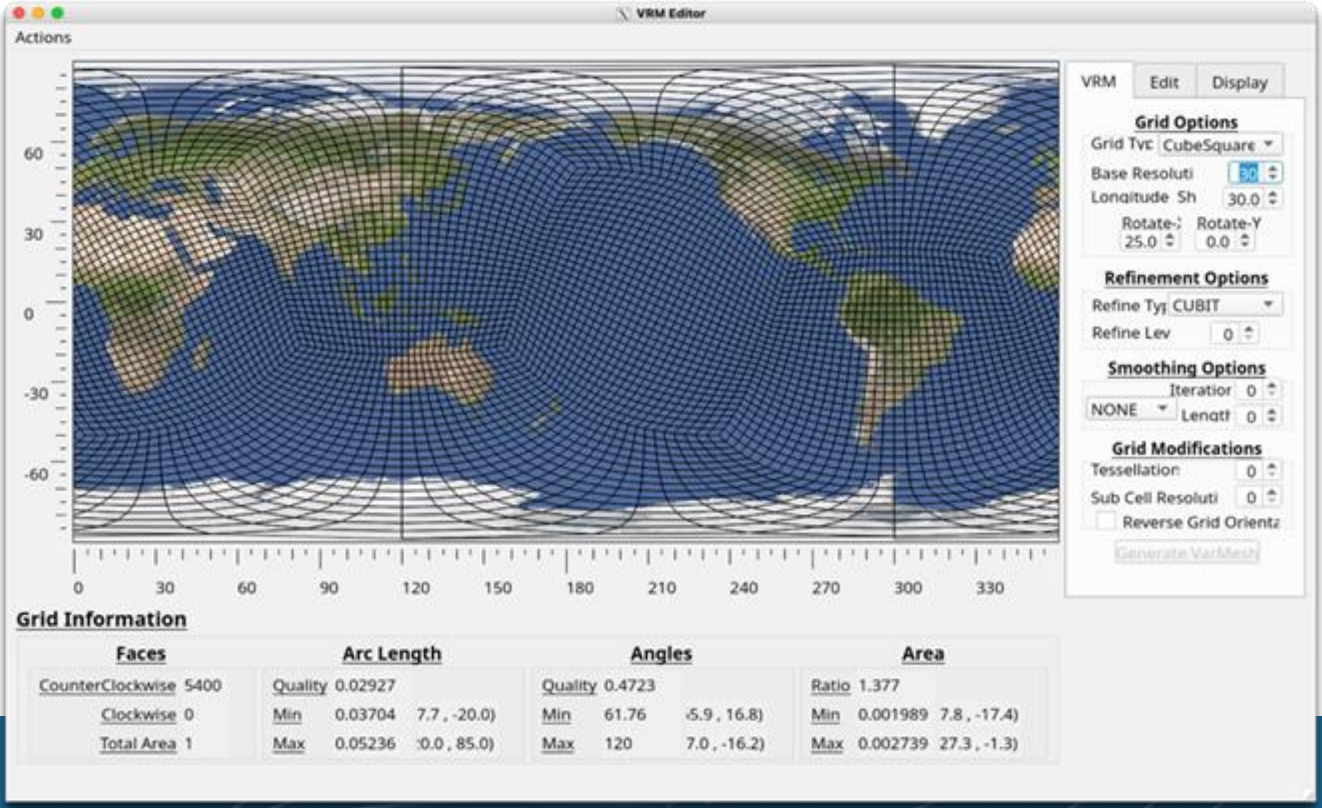
Shift face to region of interest

Adjust Longitude Shift, Rotate-X and Rotate-Y so that your region of interest is centered within a cube face.

You can zoom in/out by using the scroll button of your mouse. The right and left arrow keys move the map sideways; up/down to shift the map N/S.

For example, set Longitude Shift to 30 and Rotate-X to 25, then click Generate VarMesh, to produce grid shown below.

Keep track of the Longitude Shift, Rotate-X and Rotate-Y values you settle on - you will need them later.



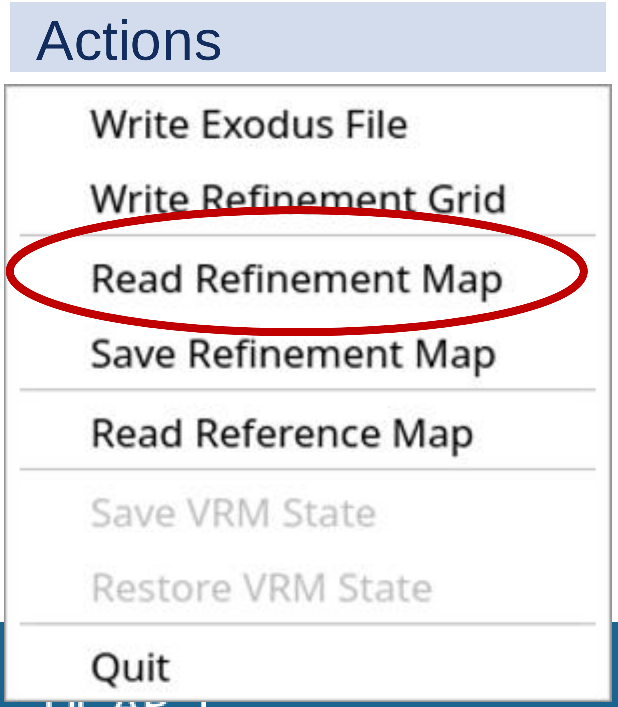
Create refinement region

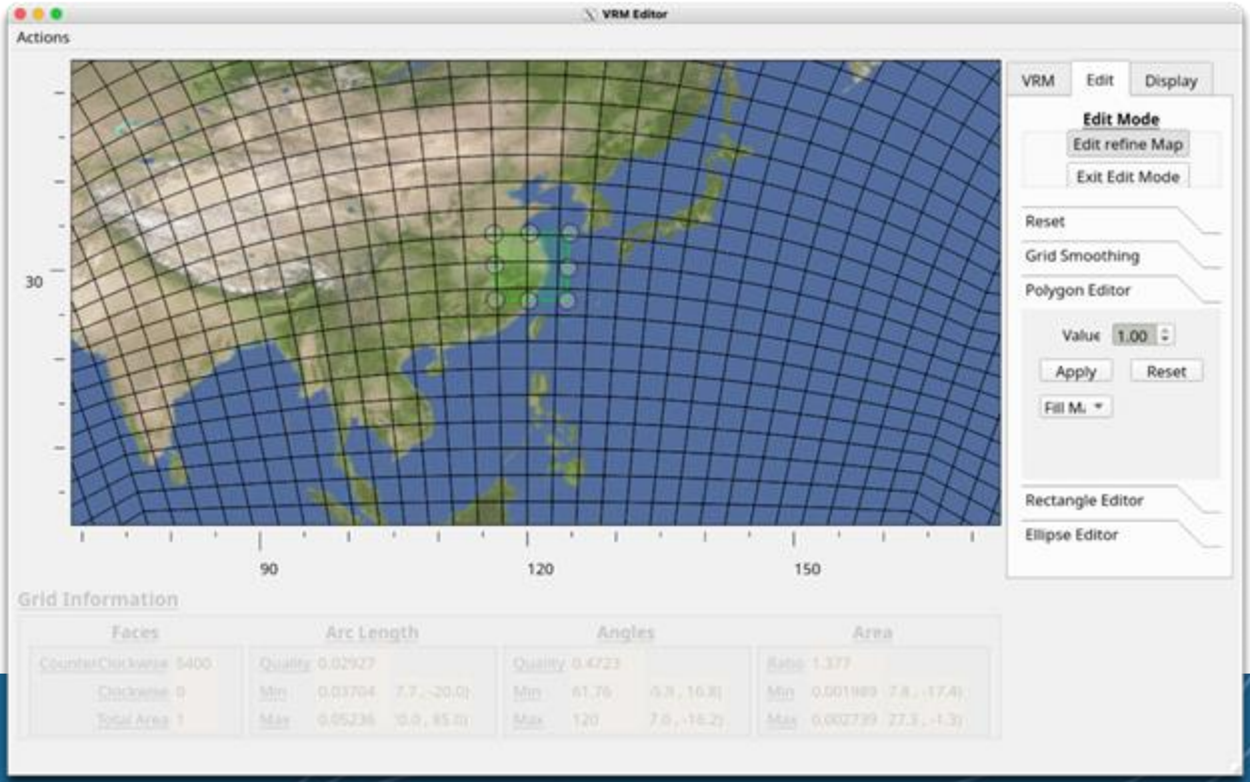
The default resolution of the Editor map is a bit coarse, so read in a higher resolution map. In the "Actions" menu (upper left), select Read Refinement Map.

Open /glade/work/emmons/Community_Mesh_Generation_Toolkit_Nov23/VRM_tools/VRM_Editor/src/REFMAP_1440x720.nc. For easier access, copy this file to your working ('grids') directory.

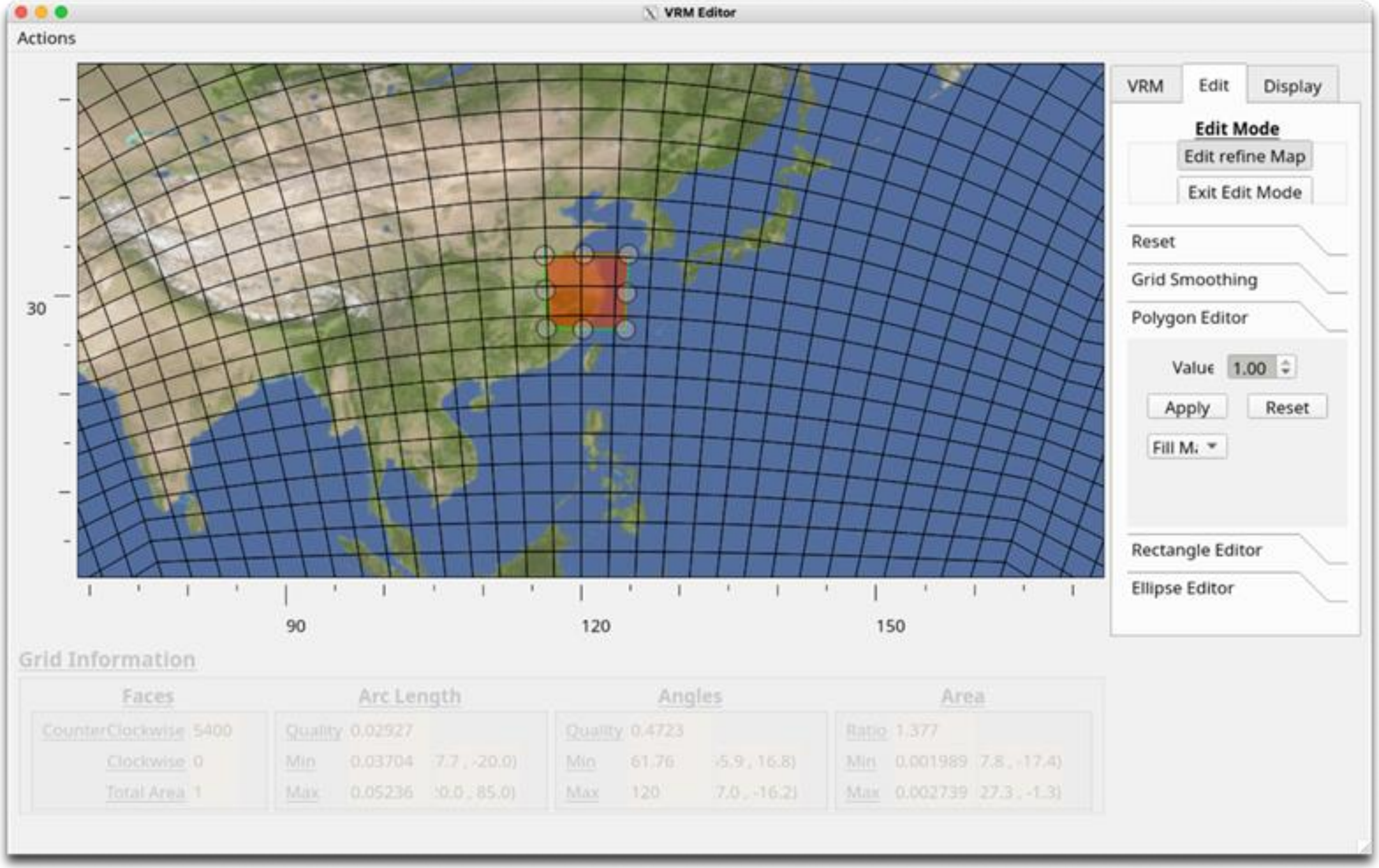
On the 'Edit' tab, click 'Edit refine Map'.

Select 'Polygon Editor' - a green box appears over the Pacific. Drag and adjust the 8 points to create a refinement region. Use the scroll button of your mouse to zoom in and out, and the arrow keys to shift the map. Try to have the polygon borders be parallel to the grid lines.





Click Apply (with Value =1.0)



To save this, click 'Exit Edit Mode' and 'Yes' to save.

Go to 'Display' tab, check Refinement Map to see the refined region as a red box.

Go back to ‘VRM’ tab. For best results (in many cases), the following settings are recommended:

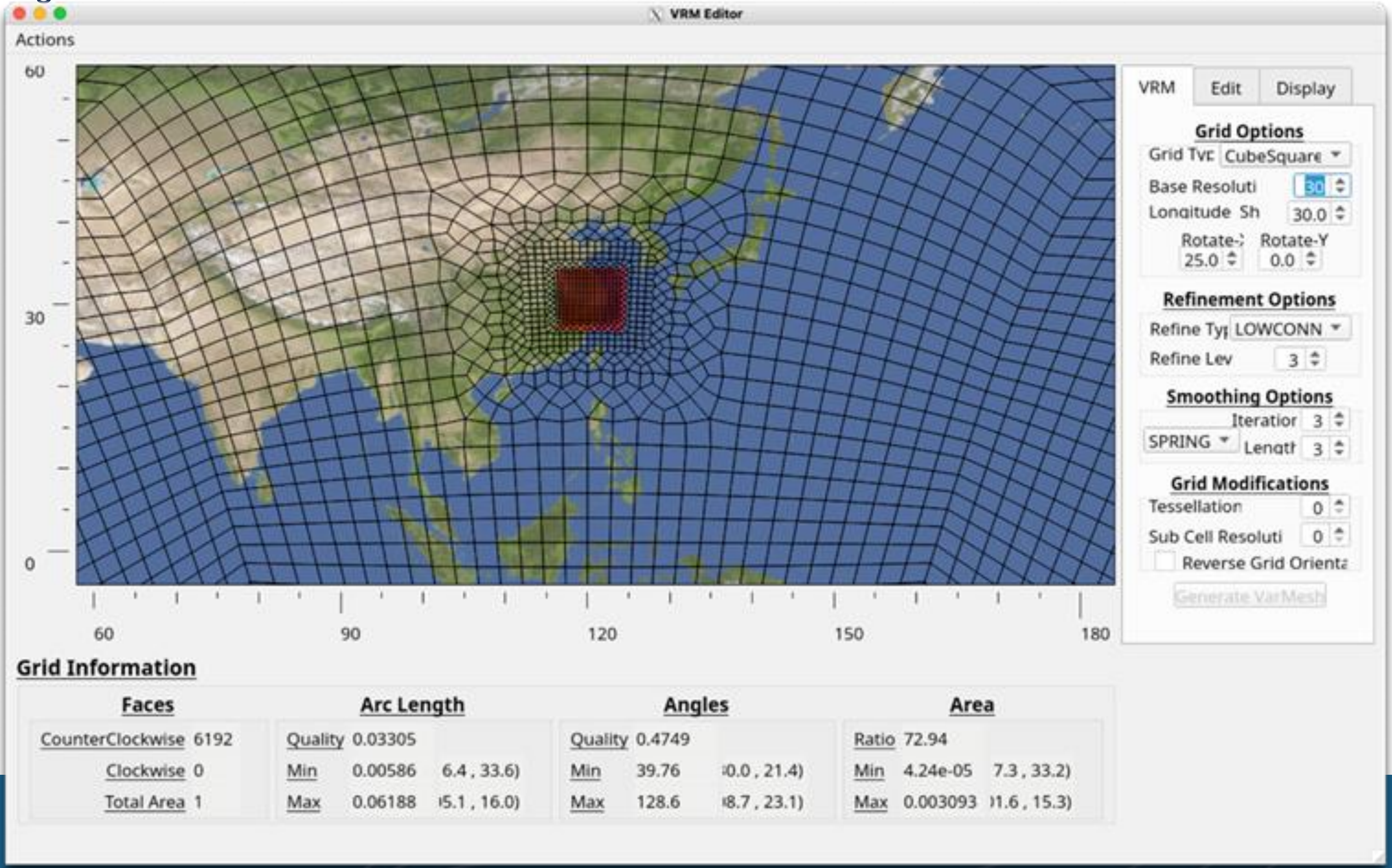
- Refine Type = LOWCONN
- Select Refine Level (1 = ~0.5 deg, 2=0.25 deg, 3=⅛ deg=14km, …). Resolution in the refined region is the base grid resolution divided by 2^(refinement level)
- Smoothing Options: Spring, with Iterations = 3, Length = 3

Click 'Generate VarMesh'.

This case has Refine Lev = 3, to give a refined mesh with 1/8 degree resolution at center.

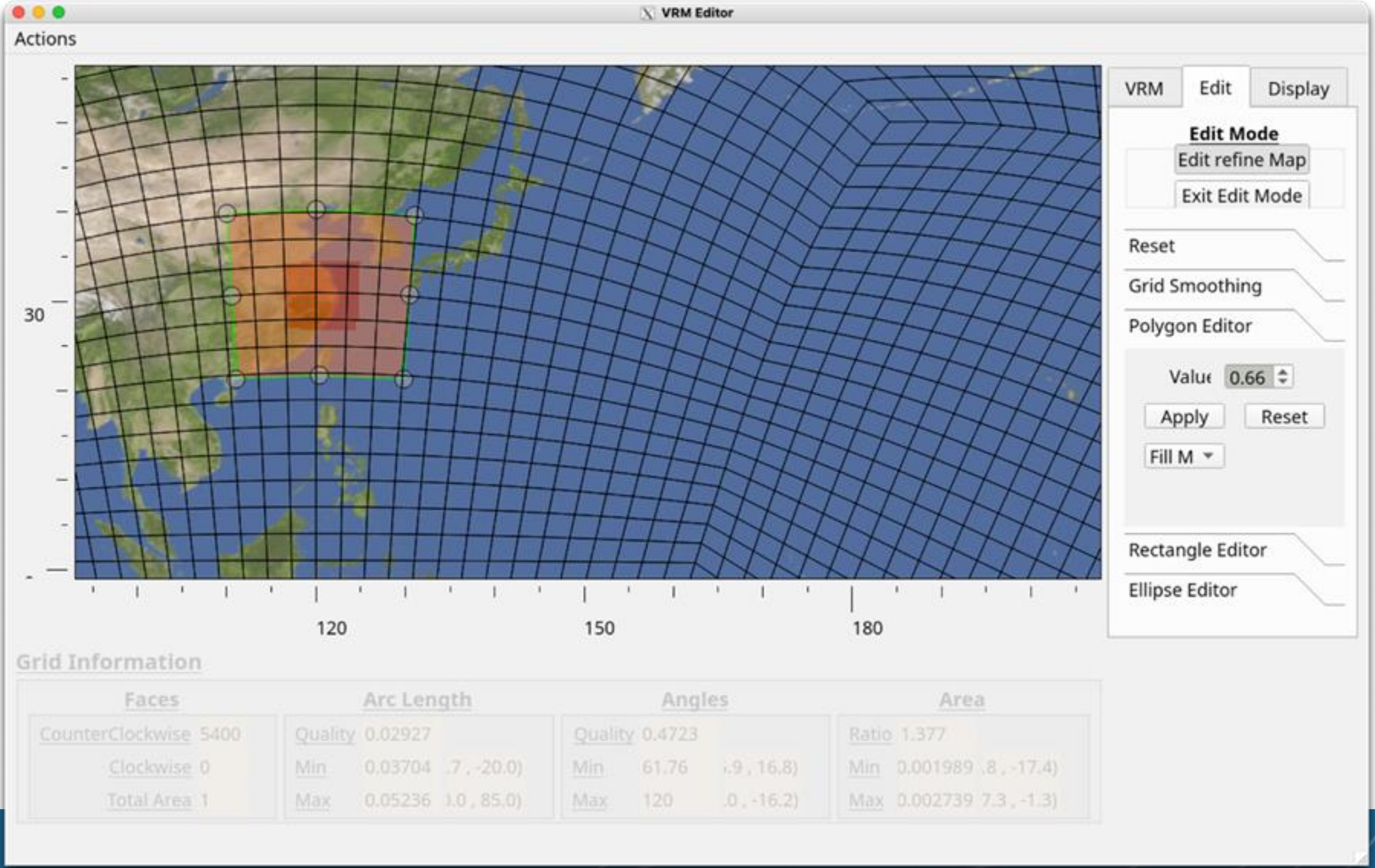
The LOWCONN setting uses templates that span 2x2 base elements to transition between resolutions, and the SPRING smoothing rounds out the element shapes to reduce sharp angles. There may be situations when other settings give better results.

Uneven edges might be removed with small adjustments (0.1) to Longitude Shift or Rotate-X, -Y values.

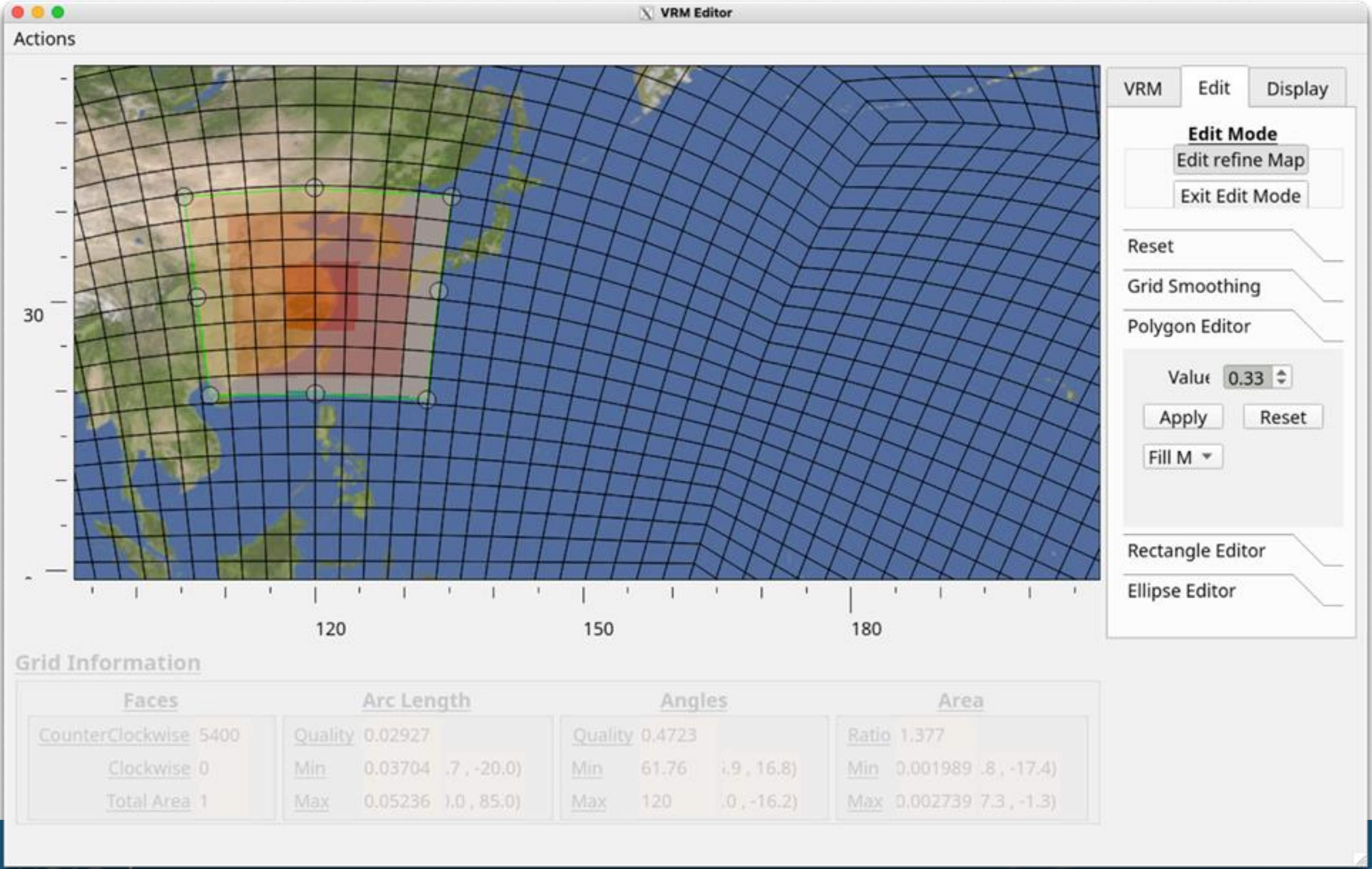


It is best to have a couple of rows at each intermediate resolution. This can be done by making a 'halo' in the Editor: make the polygon a bit larger than the refined region, and set the value at the appropriate fraction.

For an ne30 base grid with Refine Level=3 (ne240), the halo region surrounding the finest refinement (at ne120 resolution) should have a Value=0.66.

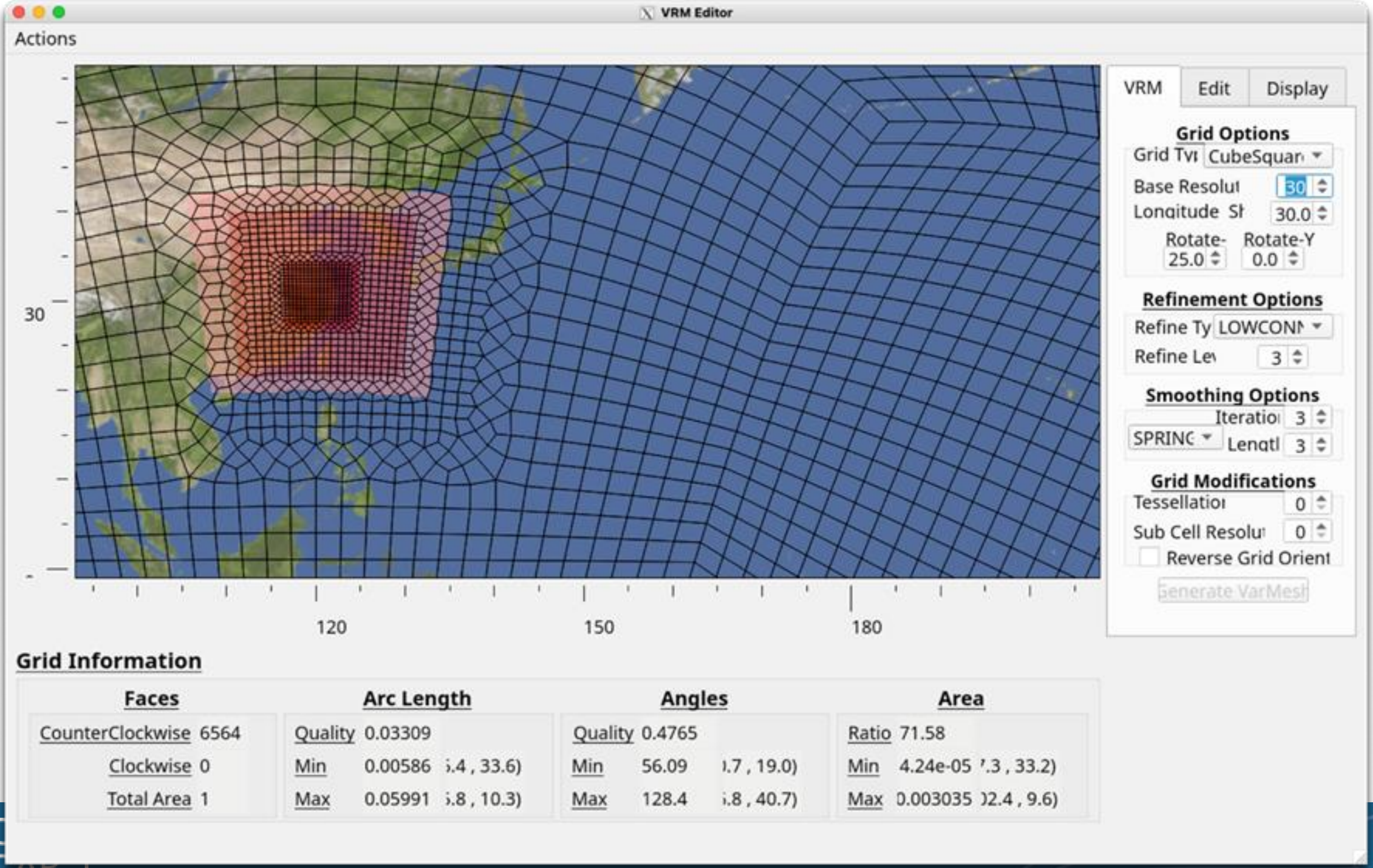


Then a second halo, at ne60 resolution, can be created with Value=0.33.



If making a refined region with the finest resolution ne120 (ne30x4), then make only one halo with Value=0.5.

When satisfied with the halos, Exit Edit Mode, Yes to save, and go back to VRM tab. With the same settings as above (LOWCONN, Refine lev =3, etc.), click Generate VarMesh:



The halos can be adjusted repeatedly in the Edit menu until satisfied with the grid.

Once the grid is at least close to what you want proceed to saving the grid in the next steps.

Save the Refinement Map - under the Actions menu. This writes a netcdf file of the map of refinement values (0 for no refinement, 1 for maximum refinement). Save the files as something like: REFMAP_Nanjing_ne30x8.nc.

Save frequently as the VRM Editor on the Mac tends to crash. You can then Read the refinement later and start adjusting the grid from that point.

Also, make note of any Longitude Shift and Rotate-X, -Y values as those are not saved in the Refinement map file.

If you are happy with your grid,
under Actions:
Write Exodus File - give it a name like
Nanjing_ne30x8_EXODUS.nc.

If you want to make further manual edits to your grid, do not save the EXODUS file.

Write Refinement Grid - for manual editing of refinement region and halos

If you have trouble making a grid with smooth borders and transitions around the refinement region, you can save the grid as a text file and manually edit it.

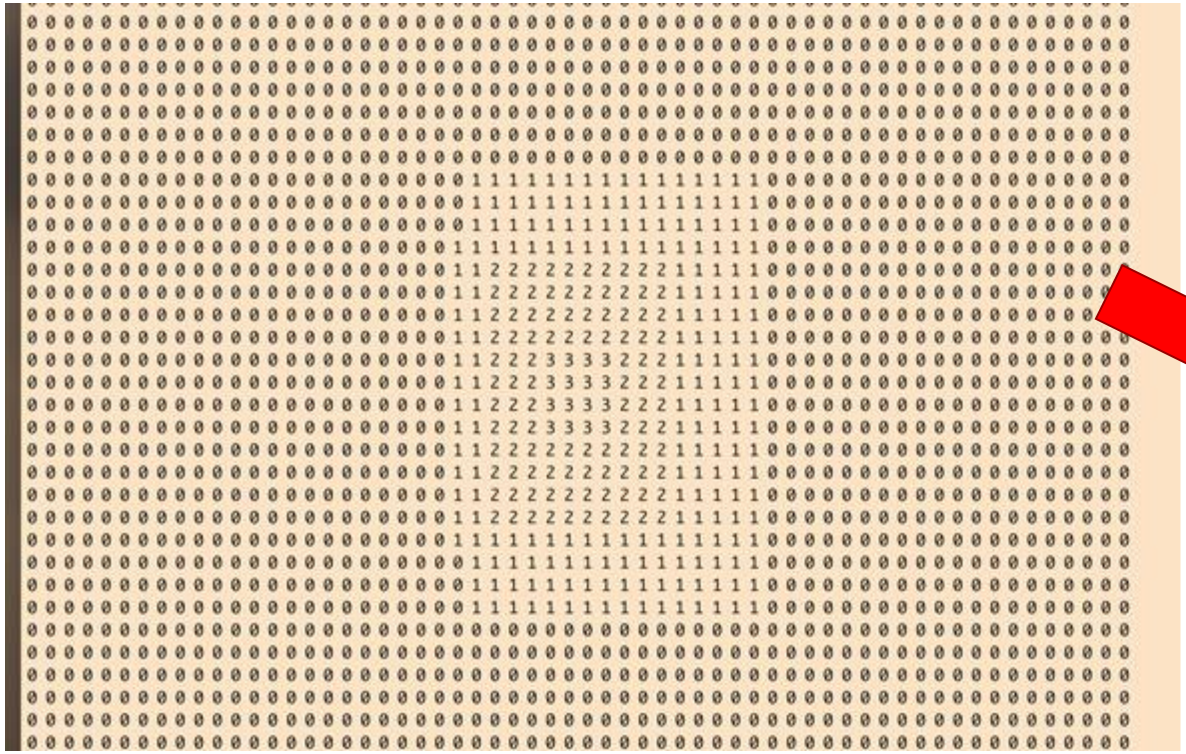
Once you have a rough version of your grid, save a Refinement Grid - in the Actions menu, select 'Write Refinement Grid'.

This is a text file similar to the Refinement Map, so name it something like:
RefGrid_Nanjing_ne30x8.dat

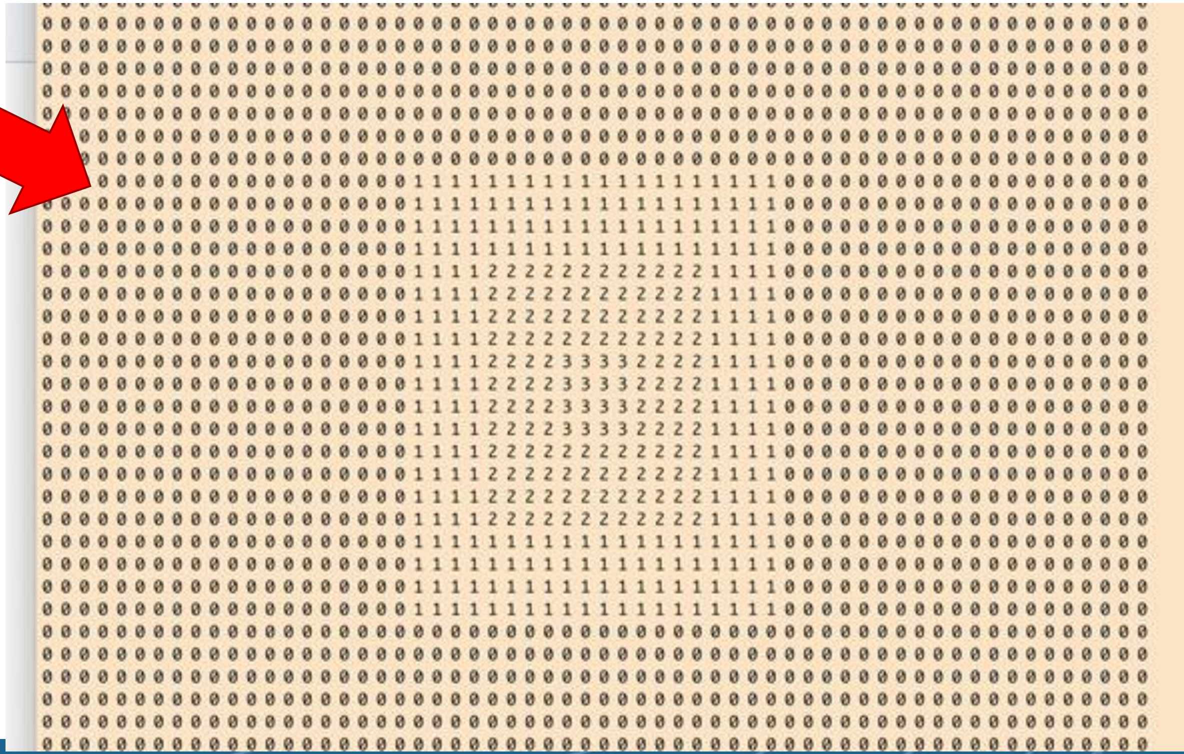
Record the Longitude Shift, Rotate-X and Rotate-Y values.

Then edit the RefGrid file to have straight edges for each refine level. For a refinement factor of 8 (refine level=3) the file has 0 for no refinement and 1, 2, 3, etc. for each refine level.

This example shows irregular borders, and different width halos:



After editing, a clean grid template looks like this:



Use Create_VRMgrid to create EXODUS file from Refinement Grid

The command line ‘Create_VRMgrid’ will create an EXODUS file from the updated Refinement Grid file.

```
> /glade/work/emmons/tutorial_Nanjing/VRM_tools/VRM_Editor/src/Create_VRMgrid

--refine_type "LOWCONN" {or "CUBIT"}

--grid_type "CubeSquared"

--resolution 30

--refine_level {refinement [1,2,3 …]}

--smooth_type "SPRING" --smooth_dist 3 --smooth_iter 3

--x_rotate {xrot_value} --y_rotate {yrot_value} --lon_shift {lonshift_value}

--refine_file REFMAP_{yourLabel}_{resolution}.nc

--refine_cube RefGrid_{yourLabel}_{resolution}.dat

--output {yourLabel}_{resolution}_EXODUS.nc
```

(be sure this is all on one line)

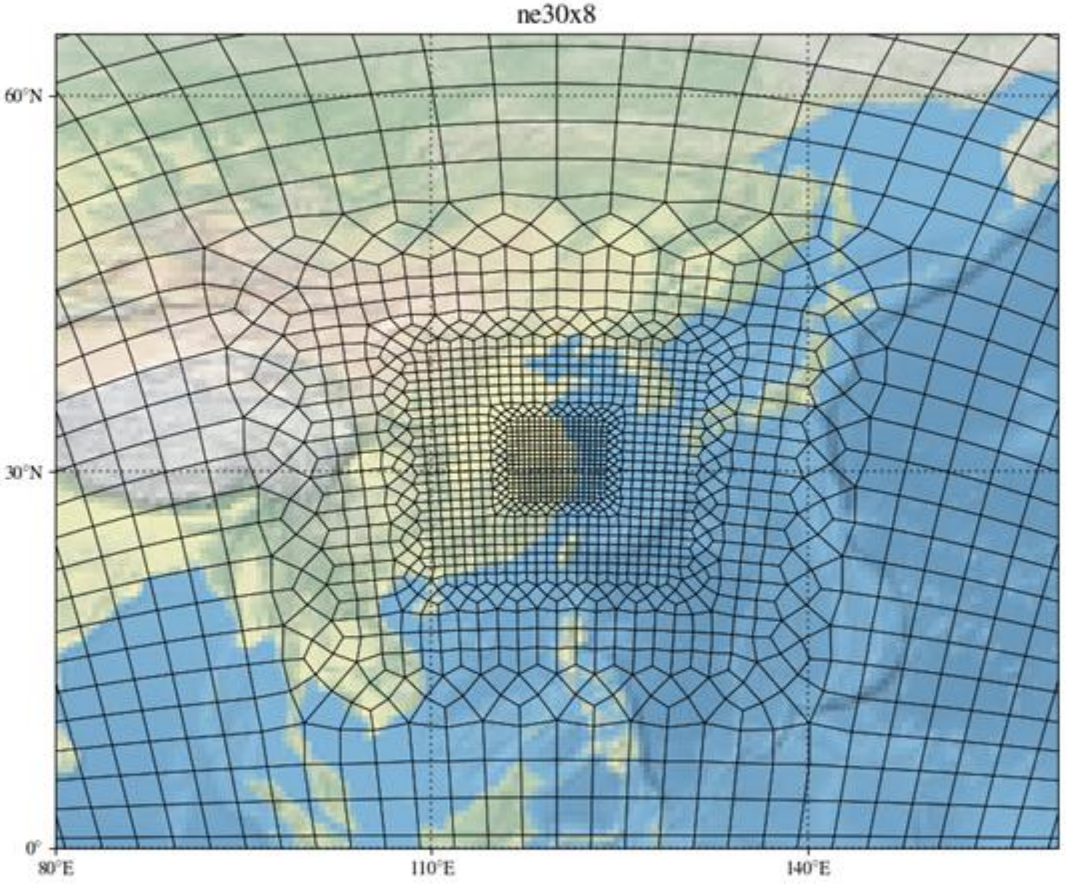
For this example:

/glade/derecho/scratch/emmons/nanjing_musica_tutorial>
/glade/work/emmons/tutorial_Nanjing/VRM_tools/VRM_Editor/src/Create_VRMgrid --refine_type "LOWCONN" --grid_type "CubeSquared" --resolution 30 --refine_level 3 --smooth_type "SPRING" --smooth_dist 3 --smooth_iter 3 --x_rotate 25 --y_rotate 0 --lon_shift 30 --refine_file REFMAP_Nanjing_ne30x8.nc --refine_cube RefGrid_Nanjing_ne30x8.dat --output Nanjing_ne30x8_EXODUS.nc

Plot EXODUS file with

/glade/u/home/emmons/tutorial_Nanjing_notebooks/plotting/

Plot_exodus_grid.ipynb



NCAR UCAR

Create additional grid files

Create SCRIP and LATLON grid files from EXODUS file

On casper:

```
> cd $REPO/grids
```

```
> cp /glade/work/emmons/tutorial_Nanjing/VRM_tools/VRM_ControlVolumes/src/input.nl input-Nanjing.nl
```

Edit input-Nanjing_ne30x8.nl to have your path to $REPO/grids and the grid name. Then run Gen_ControlVolumes:

```
> /glade/work/emmons/tutorial_Nanjing/VRM_tools/VRM_ControlVolumes/src/Gen_ControlVolumes.exe input-Nanjing.nl > LOG_Nanjing
```

This produces Nanjing_ne30x8_np4_SCRIP.nc and Nanjing_ne30x8_np4_LATLON.nc

Examine the LATLON file to get the number of columns in your new grid:

```
> ncdump -h Nanjing_ne30x8_np4_LATLON.nc
```

This prints (among other things):

ncol = 60482 ;

You will need this value for several of the following steps.

Create ESMF mesh file from SCRIP file

On ***casper***, load needed modules:

```
> module load mpi-serial/2.3.0
> module load esmf/8.5.0
```

To find the path to ESMF_Scrip2Unstruct, run:

```
> module show esmf/8.5.0
```

Find the listing for the "PATH" directory:

"PATH","/glade/u/apps/casper/23.10/spack/opt/spack/esmf/8.5.0/mpi-serial/2.3.0/oneapi/2023.2.1/dfkx/bin"

In the directory with your SCRIP file ($REPO/grids), run (be sure all on one line):

```
> /glade/u/apps/casper/23.10/spack/opt/spack/esmf/8.5.0/mpi-
serial/2.3.0/oneapi/2023.2.1/dfkx/bin/ESMF_Scrip2Unstruct
Nanjing_ne30x8_np4_SCRIP.nc Nanjing_ne30x8_np4_MESH.nc 0
```

Note the 0 at the end

This creates the ESMF mesh file: Nanjing_ne30x8_np4_MESH.nc

Create CESM input files for new grid

Follow instructions at:
https://wiki.ucar.edu/display/MUSICA/Custom+Grid+in+CESM3
#CustomGridinCESM3-InputfilesGenerateCESMinputfilesonnewgrid

Regrid Emissions

- You can run the model without regridding the emissions for preliminary tests. In this case the model will interpolate values from the default grid. (NON-CONSERVATIVE)
- Prior to running the model for meaningful results, the emissions should be conservatively regridded

- NCL scripts for regridding:

https://github.com/NCAR/IPT/tree/master/Emissions

Separate scripts for anthropogenic (CMIP6, CAMS) and biomass burning (QFED, FINN) inventories

- Python function for regridding:

https://ncar.github.io/CAM-chem/examples/functions/Regridding.html

Available in: PyPi package Vivaldi-A:

https://ncar.github.io/CAM-chem/examples/functions.html

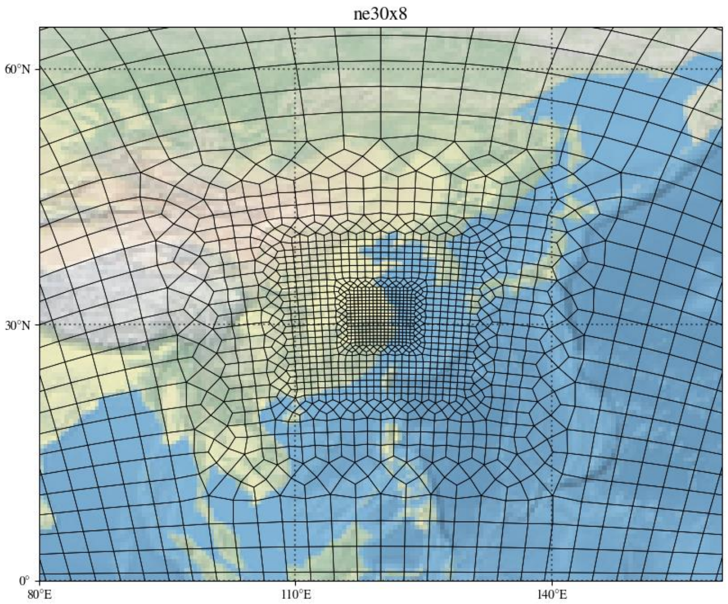
Regridding Meteorological Data for Nudging

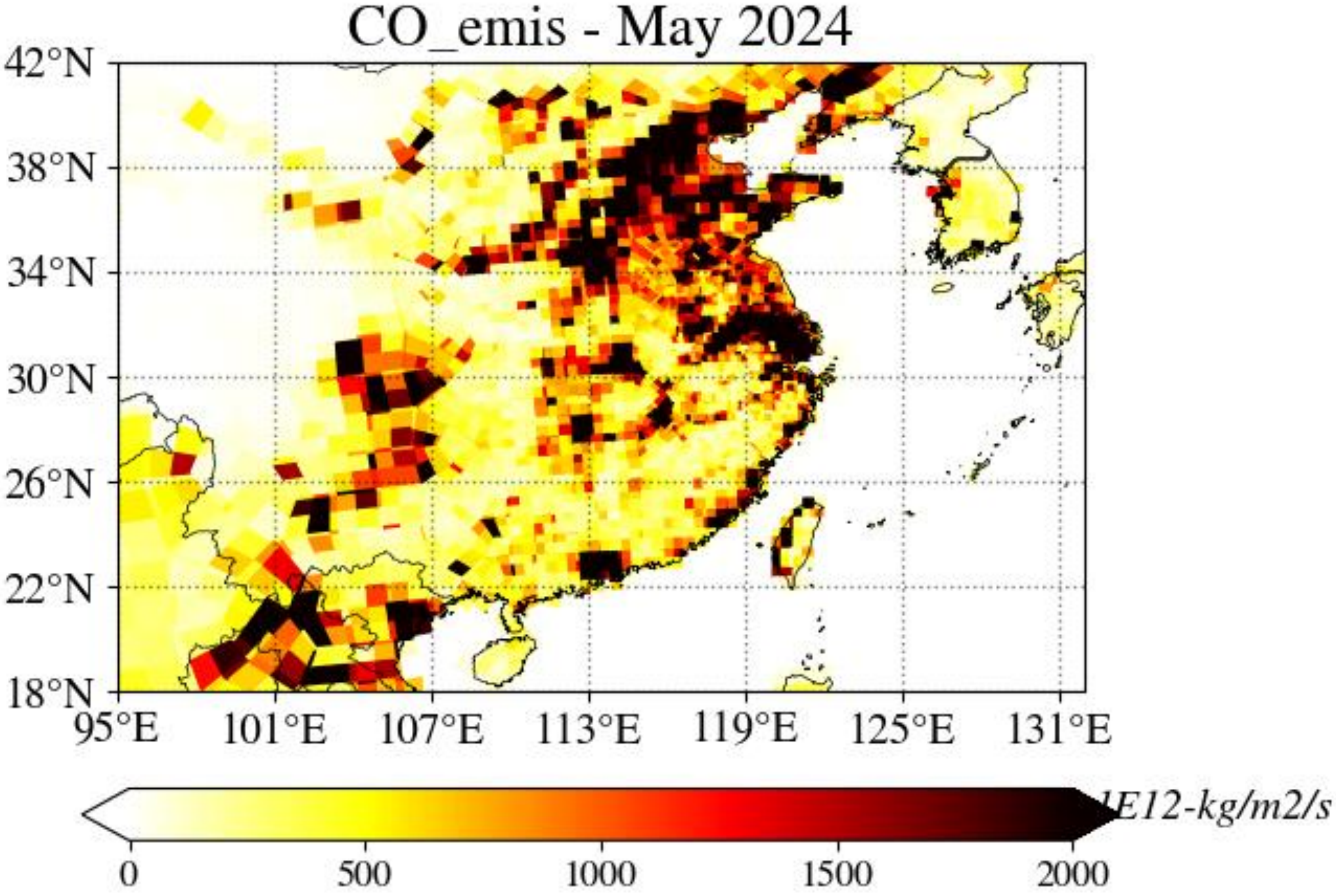
- Programs for creating nudging data can be found in the directory:

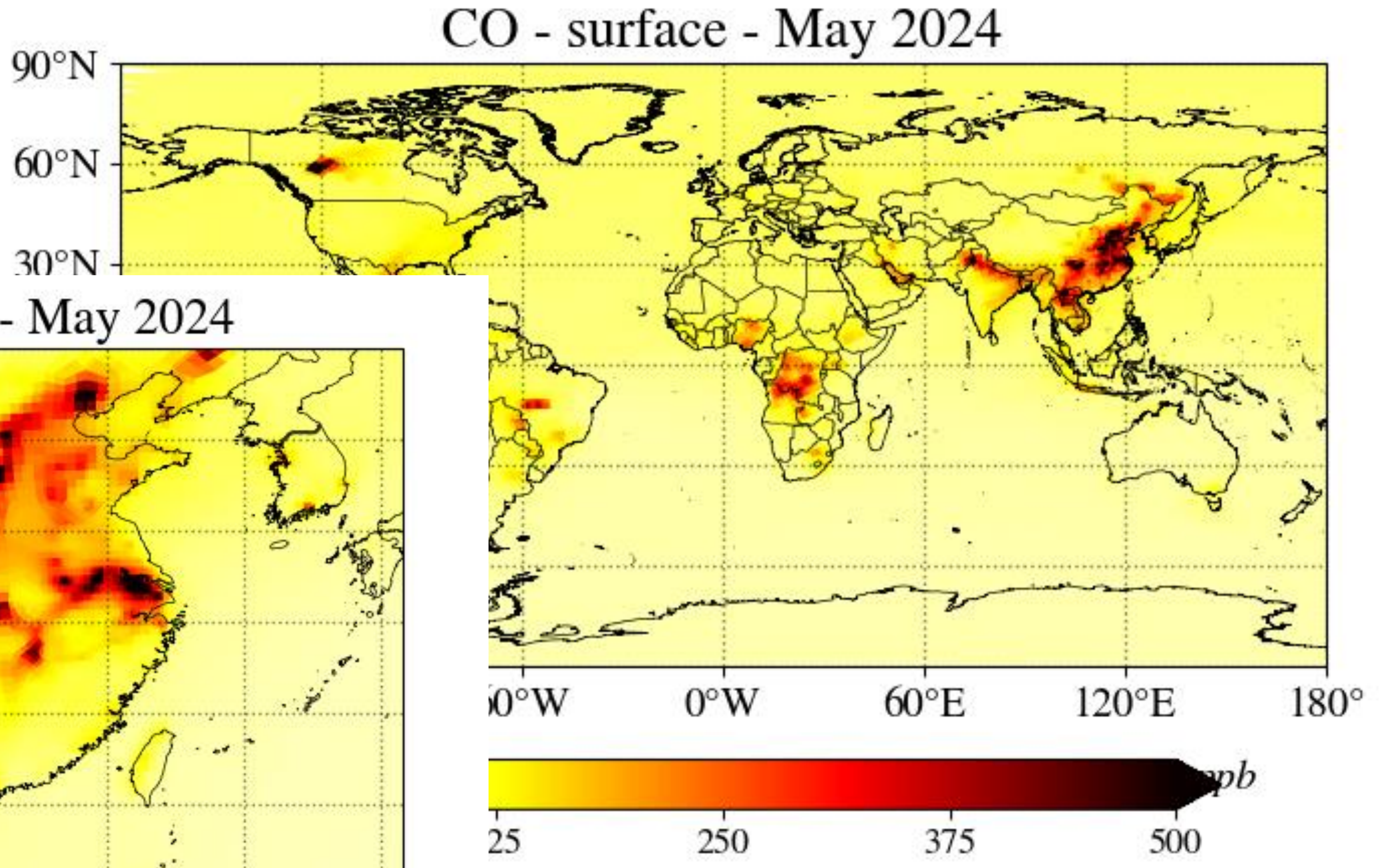
$IPT/Meterological_Reanalysis_Data/Spectral_element_dycore
https://github.com/NCAR/IPT/tree/master/Meterological_Reanalysis_Data/Spectral_element_dycore

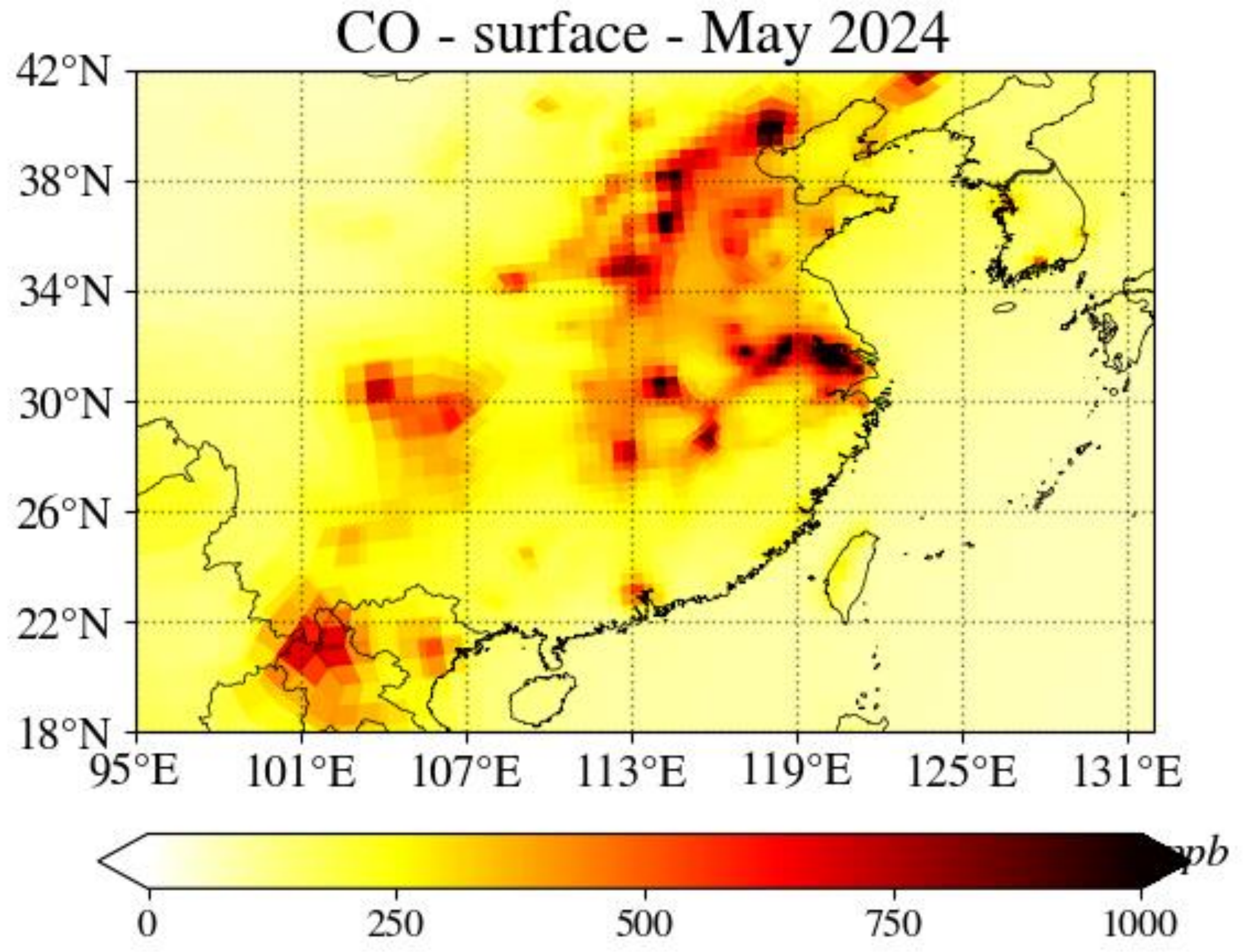
- Each directory has scripts used to process data for a different resolution. The Gen_Data_NEWGRID directory is a template that can be used to process data for a new VR grid.

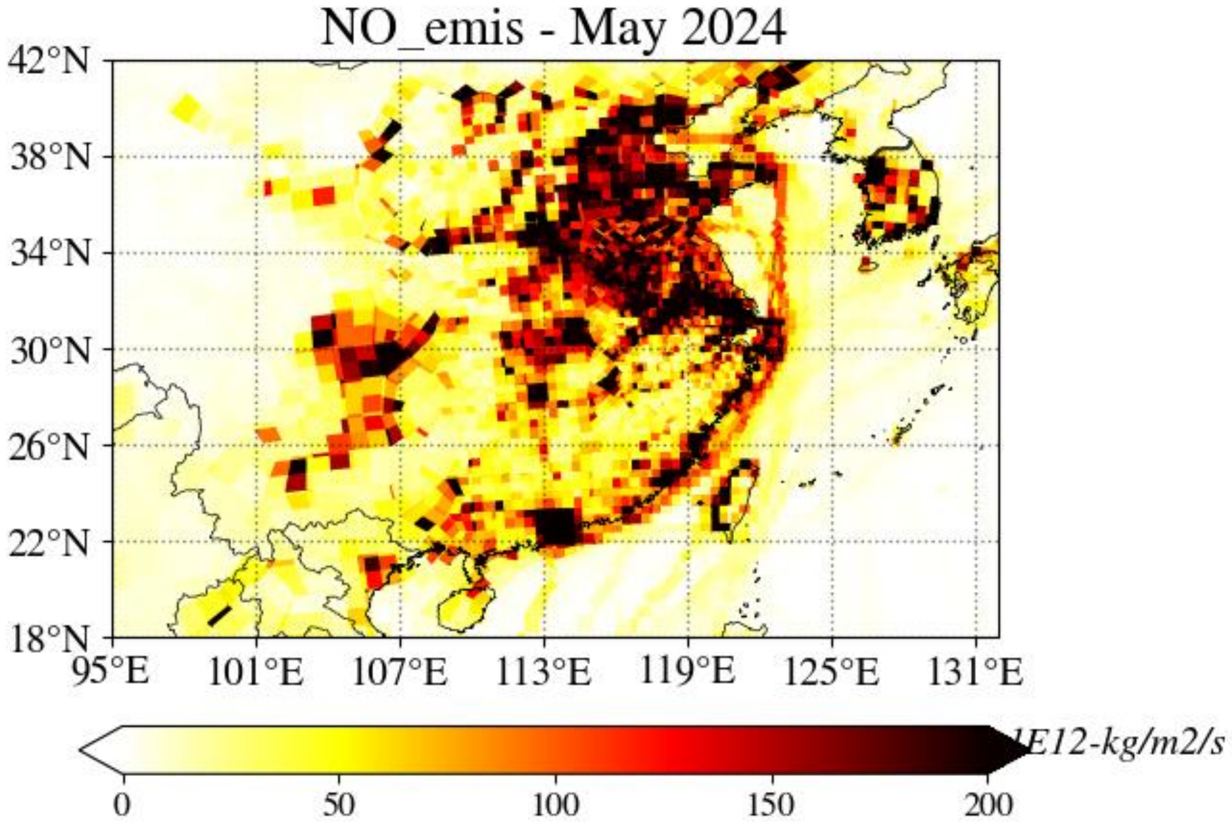
- If you use MERRA2, then the files from one of these directories can be modified to use your new grid. If you use a different reanalysis product, then you can use the README as a guide to make changes for that product.

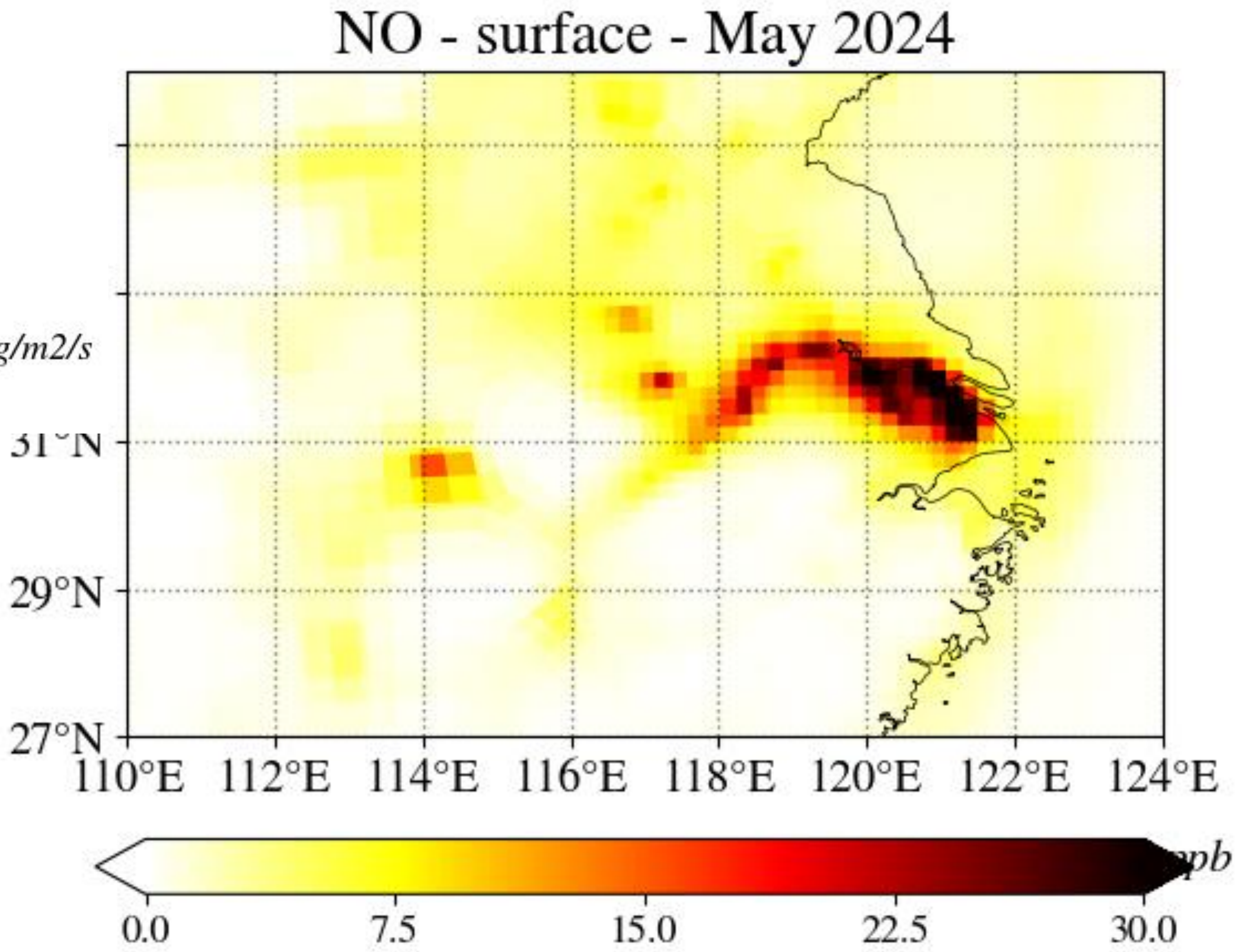

ne30x8
60°N
30°N
0°
80°E
110°E
140°E


CO_emis - May 2024
42°N
38°N
34°N
30°N
26°N
22°N
18°N
95°E
101°E
107°E
113°E
119°E
125°E
131°E
1E12-kg/m2/s
0
500
1000
1500
2000

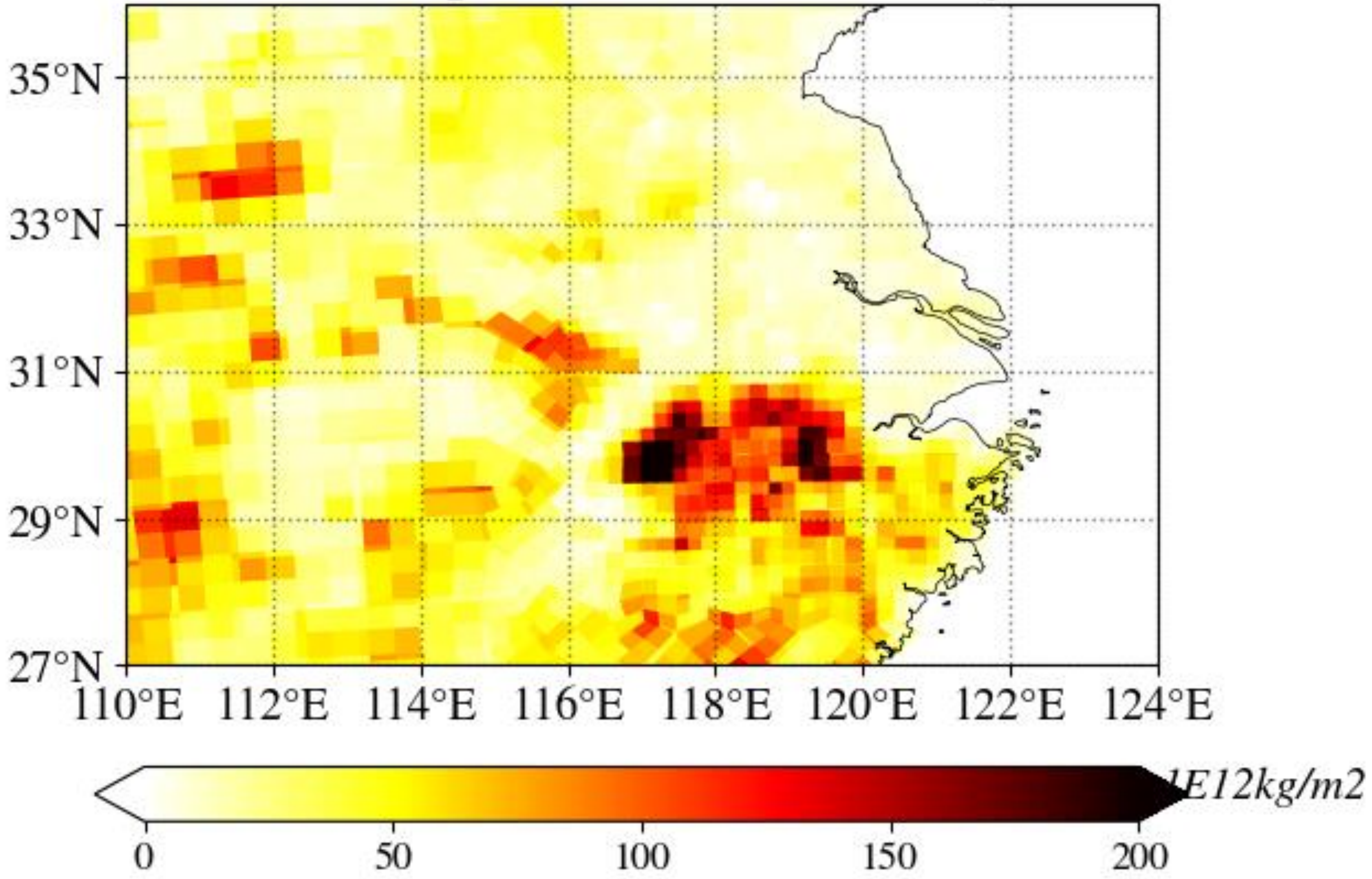

CO - surface - May 2024
90°N
60°N
30°N
0°W
60°E
120°E
180°
250
375
500


CO - surface - May 2024
42°N
38°N
34°N
30°N
26°N
22°N
18°N
95°E
101°E
107°E
113°E
119°E
125°E
131°E
ppb
0
250
500
750
1000

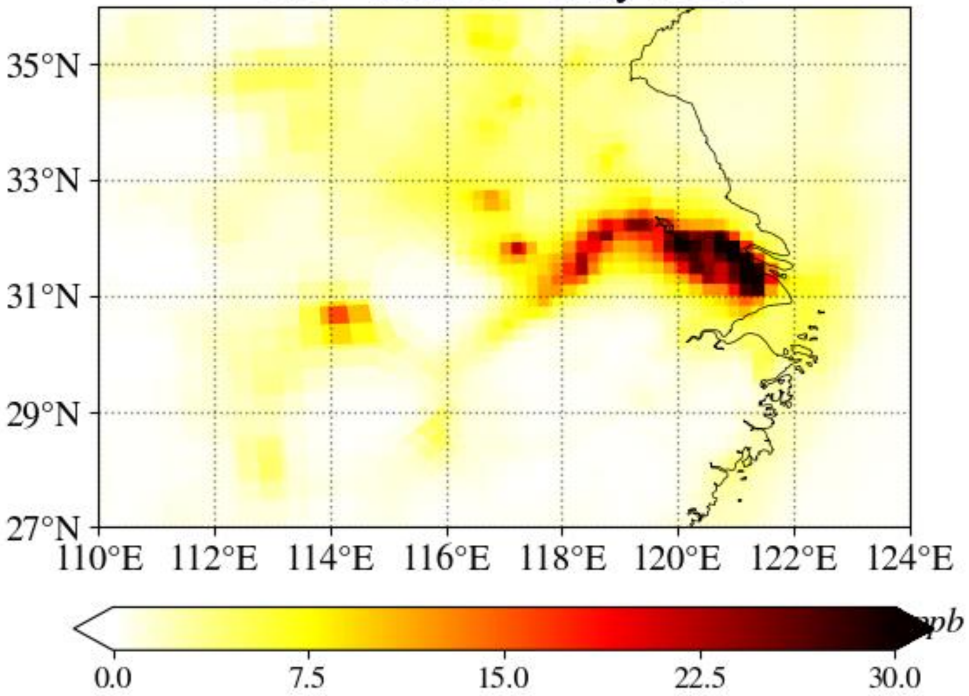

NO_emis - May 2024
42°N
38°N
34°N
30°N
26°N
22°N
18°N
95°E
101°E
107°E
113°E
119°E
125°E
131°E
1E12-kg/m2/s
0
50
100
150
200


NO - surface - May 2024
31°N
29°N
27°N
110°E
112°E
114°E
116°E
118°E
120°E
122°E
124°E
0.0
7.5
15.0
22.5
30.0

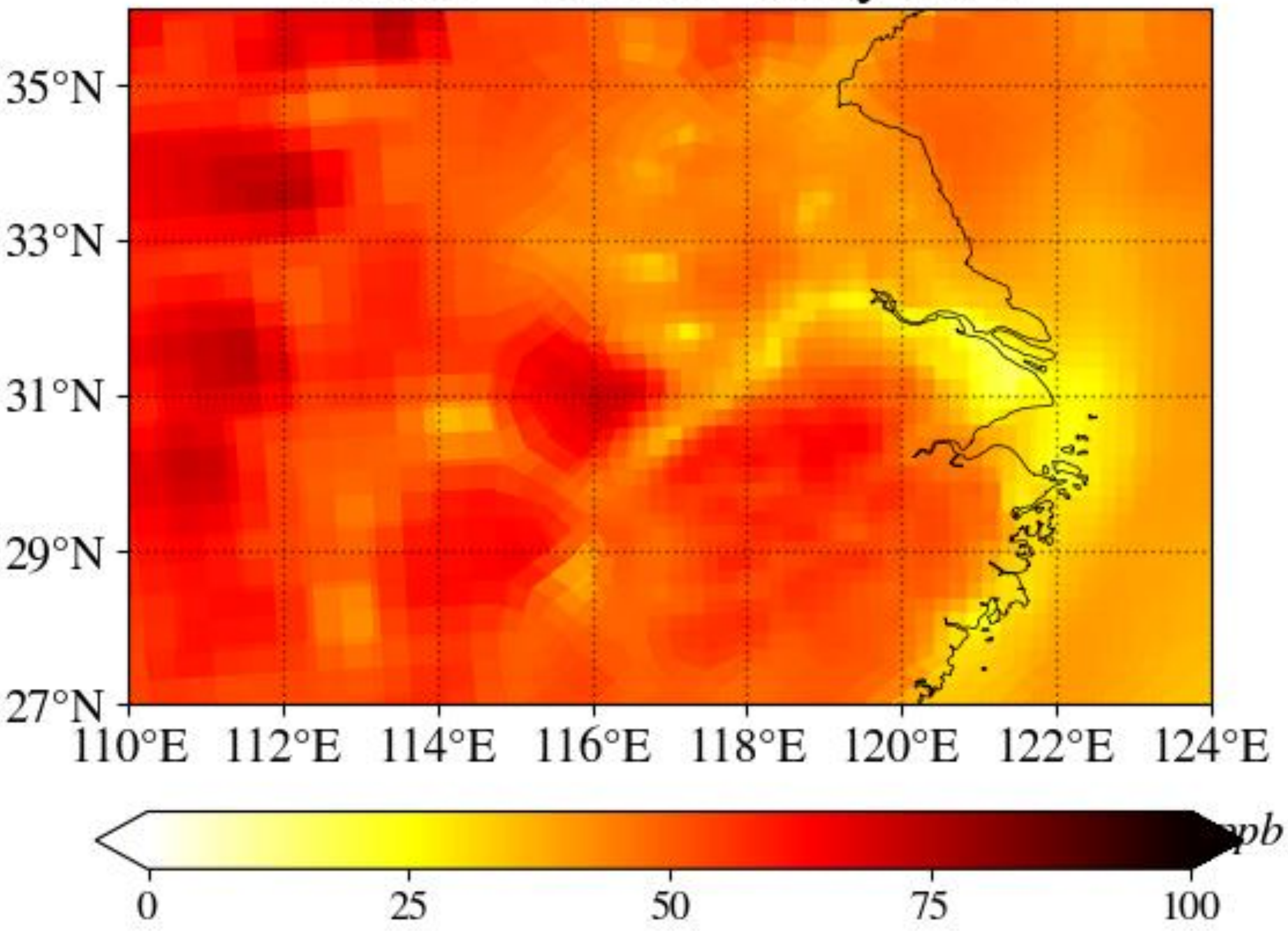
MEGAN Isoprene Emissions - May 2024

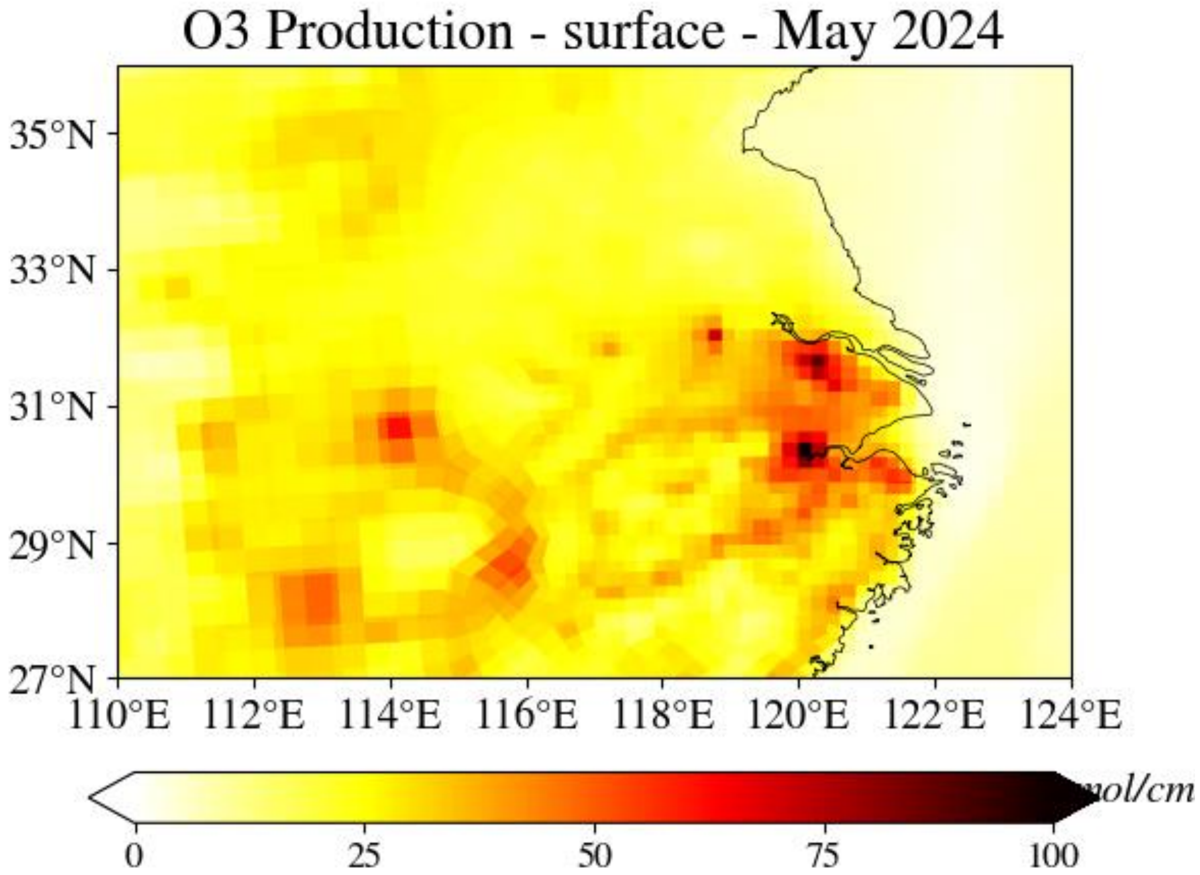


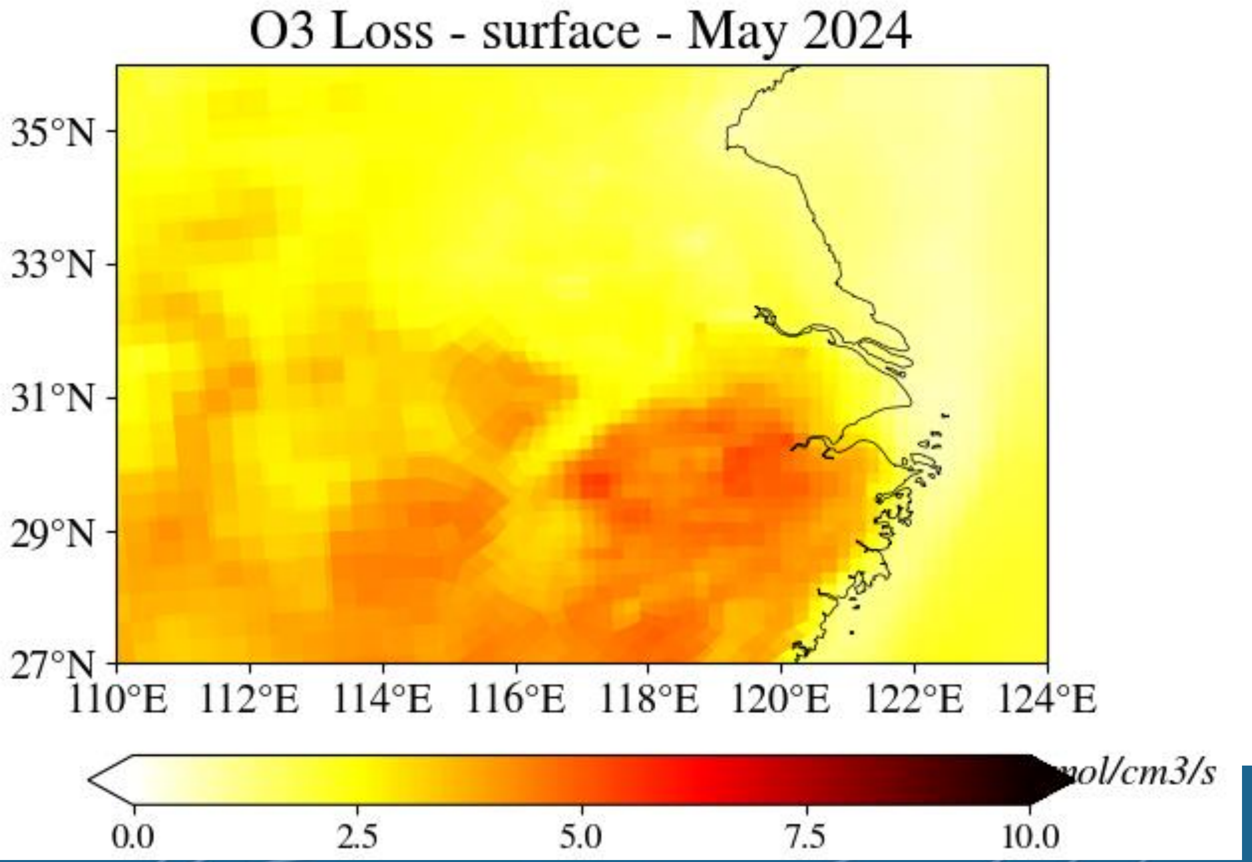
NO - surface - May 2024



Ozone - surface - May 2024




O3 Production - surface - May 2024
35°N
33°N
31°N
29°N
27°N
110°E
112°E
114°E
116°E
118°E
120°E
122°E
124°E
ol/cm
0
25
50
75
100


O3 Loss - surface - May 2024
35°N
33°N
31°N
29°N
27°N
110°E
112°E
114°E
116°E
118°E
120°E
122°E
124°E
ol/cm3/s
0.0
2.5
5.0
7.5
10.0